www.compassis.com

Turbulence Handbook

Environment for multi-physics simulation, including fluid dynamics, turbulence, advection of species, structural mechanics, free surface and user defined PDE solvers.

Tdyn
CFD+HT

Version
14.0.0

Compass Ingeniería y Sistemas http://www.compassis.com
Tel.: +34 932 181 989 - Fax.: +34 933 969 746 - E: info@compassis.com - C/ Tuset 8, 7o 2a, Barcelona 08006 (Spain)

http://www.compassis.com
info@compassis.com
June 2017

Table of Contents

1 Introduction

We can define turbulence as that state of fluid motion which is characterized by seemingly chaotic property changes, including rapid variation of pressure and velocity in space and time, but specially apparently random three-dimensional vorticity evolution.

Turbulence causes the formation of eddies of many different length scales. At low speeds the flow is laminar, i.e., the flow is smooth (though it may involve large scale vortices) but as the speed increases, at some point the transition is made to turbulent flow. Once turbulence is developed, large vortices are contracted along an orthogonal direction to the vorticity, and later stretched according to the vorticity direction to be able to conserve angular moment. This phenomena is known as the vortex stretching mechanism. The vortex stretching mechanism is responsible for the local amplification of vorticity intensity, as well as, for the formation of smaller and smaller scale structures in the flow. This phenomenon implies therefore energy transfer from the greater scales to the smaller ones, process known as the energy cascade.

Most of the kinetic energy of the turbulent motion is contained in the large scale structures. The energy cascade phenomenon (an inertial and essentially inviscid mechanism) works transfering energy from large scale structures to smaller scale structures. This process continues, creating smaller and smaller structures which produces an infinite hierarchy of eddies, where vortexes absorb energy from the greater eddies in which they are contained. Eventually this process creates structures that are small enough that molecular diffusion becomes important and viscous dissipation of energy finally takes place. The scale at which this happens is the so-called Kolmogorov length scale.

The complexity of this phenomenon makes that understanding of turbulence, its quantification, prediction, simulation and control has turned into one of the most complex and important problems in science and engineering.

Most flows of practical engineering interest are turbulent, and turbulence usually dominates the behaviour of the flow. Turbulence plays a crucial part in the determination of many relevant engineering parameters, such as frictional drag, heat transfer, flow separation, transition from laminar to turbulent flow, thickness of boundary layers, extent of secondary flows, and spreading of jets and wakes. When turbulence is present, it usually dominates all other flow phenomena and results in increasing energy dissipation, mixing, heat transfer, and drag.

Turbulent energy transport phenomena are usually described by a turbulent viscosity coefficient. This turbulent viscosity coefficient is defined in a phenomenological sense, by analogy with the molecular viscosity, but it does not have a true physical meaning, being dependent on the flow conditions, and not a property of the fluid, itself. In addition, the turbulent viscosity concept assumes a constitutive relation between a turbulent flux and the gradient of a mean variable similar to the relation between flux and gradient that exists for molecular transport. In the best case, this assumption is only an approximation.

Nevertheless, the turbulent viscosity is the most practical approach for quantitative analysis of turbulent flows, and many models have been postulated to calculate it.

2 Turbulence modeling

It is accepted that Navier-Stokes equations, used to describe the behaviour of viscous fluids, describe properly the turbulent phenomena. Consequently, considering the enormous capacity of actual computers, it is possible to consider that high precision numerical simulations of the Navier-Stokes equations can solve the problem of turbulence. Unfortunately, with the current capacity of computing power, the attempts of direct numerical simulation of Navier-Stokes equations have been limited to low Reynolds numbers (Re) or/and simple geometries. This type of direct simulations are usually known by its acronym DNS. The reason for this limited success of the DNS is explained by means of the heuristic estimator of Kolmogorov, $0(Re^{9/4})$, of the necessary degrees of freedom to simulate a flow to a certain Reynolds number. Despite the current advance of the computation technology, this estimator indicates that the possibility of using DNS for flows with high Reynolds numbers in practical applications is still surely distant.

From its beginnings the attempts of simulating turbulence have been focused on models based on the average in time or in space of magnitudes involved in the problem (velocity, pressure,...) originating the models of turbulence associated with the RANS equations (Reynolds Averaged Navier-Stokes) like k-ε, k-ω, ... These models have been widely used in engineering as an alternative to the impossibility to overcome the difficulties of DNS. Most of the current methods for the simulation of this phenomenon are based on heuristic or empirical hypotheses. This fact is explained by the lack of a mathematical theory for turbulence deduced from the Navier-Stokes equations.

In recent years a significant progress has been carried out in the development of new turbulence models based on the fact that not the entire range of scales of the flow is interesting for the majority of engineering applications. In this type of applications information contained in "the large scales" of the flow is enough to analyse magnitudes of interest as velocity, temperature,... Therefore, the idea that the global flow behaviour can be correctly approximated without the necessity to approximate the smaller scales correctly, can be seen as a possible great advance in the modelling of turbulence. This fact has originated the design of turbulence models that describe the interaction of small scales with large scales. These models are commonly known as Large Eddy Simulation models (LES).

Methodologies used to simulate turbulent flows, RANS or LES approaches, are based on the same concept: unability to simulate a turbulent flow using a finite discretization in time and space. Turbulence models introduce additional information (impossible to be captured by the approximation technique used in the simulation) to obtain physically coherent solutions. On the other side, numerical methods used for the integration of partial differential equations (PDE) need to be modified in order to able to reproduce solutions that present very high localized gradients. These modifications, known as stabilization techniques, make possible to capture these sharp and localized changes of the solution.

A relevant aspect of LES theory is the close relationship between the mathematical properties of LES models and the numerical methods used for their implementation. LES models are numerical schemes to solve problems as solution uniqueness, existence of a maximum principle, convergence to suitable solutions, or convergence in graph norm. In last years it is more and more common the idea in the scientific community, especially in the numerical community, that turbulence models and stabilization techniques play a very similar role. Implicit Large Eddy Simulation models (ILES) are LES type-models based on the extension of stabilizations techniques, that allow the regularization of the Navier-Stokes equations.

The turbulent states that can be encountered across the whole range of industrially relevant flows are rich, complex and varied. After one century of intensive theoretical and experimental research, it is accepted that there is no single turbulence model, that can span these states and that there is no generally valid universal model of turbulence.

Tdyn incorporates a variety of different models and options available for simulating turbulent flows. The choice of which turbulence model to use and the interpretation of its performance (i.e. establishing bounds on key predicted parameters) is a far-from-trivial matter. Next points should be considered to select one model.

- That model should be chosen, for a particular application, which has been shown to generate the most correct predictions, by comparison with reliable experimental data for similar situations. The greater the departure from the conditions of the experiments, whether through time-dependence, property variations, three-dimensionality, or roughness, the less certainly reliable become the predictions of the tested model.
- The more complex (and perhaps therefore physically realistic) the model, the finer must be the computational grid, and the greater the expense. Therefore, in the circumstances arising in engineering, the choice must usually be made of the most practicable model rather than of that favoured by researchers.

In order to help in the selection of the turbulence model to be used in every case, the general features and broad limitations of different classes of model will be discussed next. Further information about turbulence modelling can be found in the references.

Remarks:

Turbulence modelling options are available in RANSOL, HEATRANS and ADVECT modules.

RANSE models

Turbulent flows contain many unsteady eddies covering a range of sizes and time scales. Tdyn includes a module (RANSOL) able to perform turbulent flow simulations by solving the stabilised RANS equations. RANS equations are developed from the time-dependent three-dimensional Navier-Stokes equations that are averaged in such manner, that unsteady structures of small sizes in space and time are eliminated and become expressed by the mean effects on the flow through the so-called Reynolds or turbulent stresses. These stresses have to be interpreted in terms of averaged variables to close the system of equations. This requires the construction of a mathematical model known as a turbulence model.

The RANSE turbulence modelling approach is based on the concept of a dynamic turbulent viscosity, μ_T. This relates the turbulent stresses appearing in the RANS equations to the averaged velocity gradients (i.e. the rate of strain) in direct analogy to the classical interpretation of viscous stresses in laminar flow by means of the fluid viscosity, μ. For example. in a shear layer where the dominant velocity gradient is $\partial u/\partial y$ (u is the averaged velocity in the principal direction of flow and y is the cross-stream co-ordinate) the turbulent shear stress is given as $\mu_T \cdot \partial u/\partial y$.

The viscous analogy can be extended to the interpretation of the turbulent energy fluxes using the so-called Reynolds extended analogy. From dimensional considerations μ_T/ρ is proportional to V·L, where V is a velocity scale and L is a length scale of the larger turbulent motions (called the mixing length in the so-called mixing length models). Both the velocity scale V and the length scale L are determined by the state of turbulence. The different turbulence models implement physical considerations to determine V and L fields depending on the flow characteristics.

Classes of RANSE turbulence models available in Tdyn

Zero-equation models

The simplest prescription of V and L is done in with the so-called algebraic (or zero-equation) class of models. These assume, that V and L can be related by algebraic equations to the local properties of the flow. This is fairly straightforward for simple flows but can often be difficult in geometrically complex configurations.

Algebraic models of turbulence have the virtue of simplicity and are widely used with considerable success for simple shear flows such as attached boundary layers, jets and wakes. For more complex flows where the state of turbulence is not locally determined but related to the upstream history of the flow a more sophisticated prescription is required.

One-equation models

The one-equation models attempt to improve on the zero-equation models by using an eddy viscosity that no longer depends purely on the local flow conditions but takes into account where the flow has come from, i.e. upon the flow history.

In these models, V is identified with $k^{1/2}$, where k is the kinetic energy per unit mass of fluid arising from the turbulent fluctuations in velocity around the averaged velocity. A transport equation for k can be derived from the Navier-Stokes equations and it is the single transport equation in the one-equation model.

In shear-layer type flows, and especially in regions close to a wall, it is often possible to algebraically prescribe L with reasonable confidence. In geometrically complex configurations it is difficult to prescribe the L field, because it is dependent on non-local quantities, such as a boundary layer thickness, displacement thickness, etc., and it introduces similar uncertainties as in an algebraic turbulence model.

In order to circumvent above mentioned limitations, Spalart and Allmaras [3] have devised an alternative formulation of an one-equation model appropriate for different types of flows, which determines the turbulent viscosity directly from a single transport equation for μ_T.

Two-equation models

For general applications, it is usual to solve two separate transport equations to determine V and L, giving rise to the name two-equation model. In combination with the transport equation for k, an additional transport equation is solved for a quantity, which determines the length scale L. This class of models is the best known and the most widely used in industrial applications since it is the simplest, level of closure which does not require geometry or flow regime dependent input.

The most popular version of two equation models is the k-ε model, where ε is the rate at which turbulent energy is dissipated to smaller eddies (the so-called turbulent dissipation). A modelled transport equation for ε is solved and then L is determined as $C_\mu \cdot k^{3/2}/\varepsilon$ where C_μ is usually taken as a constant. The second most widely used type of two-equation model is the k-ω model, where ω is the specific dissipation of energy. A modelled transport equation for ω is solved and L is then determined from $k^{1/2}/\omega$.

Tdyn choice of RANSE turbulence models

- Mixing_Length

Zero-equation turbulence model based in the Prandtl hypothesis, where the turbulence length scale (L) must be given in the EddyLen Field entry (of the Tdyn Initial Data window) and the velocity scale is evaluated with the following formula:

$$V = L \cdot \sqrt{2\varepsilon_{ij} \cdot \varepsilon_{ij}}$$

where ε_{ij} are the components of the velocity gradient tensor.

Turbulence length scale L can be defined explicitly in some practical cases, based on experimental data or theoretical considerations. Probably the most well known formulation for L is based on the assumption that the characteristic lenght L is proportional to the distance to the wall. If we also make use of the fact that as the edge of the boundary layer is reached, the eddy size tends to be proportional to a fraction of the thickness of the boundary layer. Thus:

$$L = L_0 \cdot \left(\frac{\kappa d}{\kappa d + L_0} \right)$$

where κ is the von Karman constant (equal to 0.41), d is the distance from the wall, and L_0 is the asymptotic value of L towards the edge of the boundary layer. However, above assumption doesn't predict accurately the behaviour of the flow close to the wall in many cases. Different formulations can be found in the literature to define L for turbulent channel flows and other simple flows.

One of the most widely known formulation for L is that given by Nikuradse [1], based on different experimental analyses in pipes and other simple flow problems:

$$L = R \cdot \left(0.14 - 0.018 \cdot \left(1 - \frac{y}{R}\right)^2 - 0.06 \cdot \left(1 - \frac{y}{R}\right)^4\right)$$

where R is the radius of the cilindrical pipe or the height of the flow in open flows.

This formulation is very useful for modelling turbulent flows in pipes and require to insert the following formula in the EddyLen Field (R sould be substituyed by its corresponding value in every case):

```
R*[0.14-0.018*(1-ds/R)^2-0.06*(1-ds/R)^4]
```

Mixing lenght model is a simple and robust model that can be used in simple cases where an experimental field for the turbulence length scale is available. For example, it is highly efficient for analysis of flow in pipes in combination with Nikuradse formulation.

- Kinetic_Energy

Prandtl's one equation (k) model for turbulent flows with integration to the wall, where the turbulence length scale (L) must be given explicitly in the EddyLen Field entry.

The velocity scale V is identified with $k^{1/2}$, where k is the kinetic energy per unit mass of fluid. This model solves a transport equation for k. It is very adequate for simple flows where the turbulent length scale can be defined algebraically.

This model improves the solution given by the mixing length model in complex flows, but still requires to specify the length scale field.

- K_Energy_Two_Layers

Prandtl's one equation (k) model for turbulent flows, where the turbulence length scale (L) is given in the EddyLen Field entry. The implementation of this model includes an improvement of the treatment of the boundary layer flow, equivalent to the low-Reynolds modification of the two equations models, see k-ε high Reynolds model below for further information).

- Spalart_Allmaras

One equation model for turbulent flows with integration to the wall (i.e. low-Reynolds model, see k-ε high Reynolds model below for further information). The aim of this model is to improve the predictions obtained with algebraic mixing-length models to develop a local model for complex flows, and to provide a simpler alternative to two-equations turbulence models.

The model uses the distance to the nearest wall in its formulation, and provides smooth laminar-turbulent transition capabilities. It does not require as fine a grid resolution in wall-bounded flows as two-equations turbulence models, and it shows good convergence in simpler flows.

The empirical results used in the development of the model were mixing layers, wakes and flat-plate boundary layer flows. That is why in these kind of problems, the model gives very accurate predictions of the flow. It also shows improvements in the prediction of flows with adverse pressure gradients compared to k-ε and k-ω models. Furthermore, this model is quite successful in practical turbulent flows in external airfoil applications. However, it does not give good predictions in jet flows.

- K_E_High_Reynolds

Two-equation k-ε model for turbulent flows. The k-ε model is the most widely known and extensively used two equations turbulence model. Still today, this model joint to Law of the Wall functions, remains the workhorse of the industrial computation. k-ε model was originally developed to improve the mixing-length model and to avoid the algebraic prescription of the turbulent length scale in complex flows.

In this model, transport equations are solved for two scalar properties of the turbulence. The k equation is a model for the turbulent kinetic energy, and the ε-equation is a model for the dissipation rate of turbulent kinetic energy. L is determined as $C_\mu \cdot k^{3/2}/\varepsilon$ where C_μ is taken as a constant.

In wall-attached boundary layers, the normal gradients of the flow variables become large as the wall distance reduces to zero. As the wall is approached, turbulent fluctuations are suppressed and eventually viscous effects become important in the region known as the viscous sub-layer. This modified turbulence structure means that many standard turbulence models are not valid all the way through to the wall. That is why, high-Reynolds k-ε is valid for the turbulent flow region, but fail in the viscous sub-layer close to the wall. Therefore high-Reynolds k-ε should be used in combination with a law of the wall. On the other hand, various so-called low-Reynolds version of the k-ε model have been proposed. Tdyn implements several of this type of models (see k-ε two-layers, Lam-Bremhorst and Launder-Sharma models).

Below, a list of recommendations for usign the different implementarions of the k-ε models as well as major weaknesses in practical applications, associated with these models in combination with the use of law of the wall (see Near wall modelling -pag. 13-) are considered.

- This model gives good results for free-shear-layer flows with relatively small pressure gradients. For wall bounded flows, the model gives good agreement with experimental results for zero and small mean pressure gradients, but is less accurate for large adverse pressure gradients.

- The turbulent kinetic energy is over-predicted in regions of flow impingement and re-attachment leading to poor prediction of the development of boundary layer flow around leading edges and bluff bodies. The high turbulence levels predicted upstream of a stagnation point are transported around the body and the real boundary layer development is swamped by this effect. The problems depend on the free-stream values of k and ε and do not occur in all cases.
- Highly swirling flows are often poorly predicted due to the complex strain fields. Regions of recirculation in a swirling flow are often under-estimated.
- Mixing is often poorly predicted in flows with strong buoyancy effects or high streamline curvature.
- Flow separation from surfaces under the action of adverse pressure gradients is often poorly predicted. The real flow is likely to be much closer to separation (or more separated) than the calculations suggest. The SST version of the k-ω model can offer a considerable improvement.
- Flow recovery following a re-attachment is often poorly predicted. If possible, avoid the use of wall functions in these regions.
- The far-field spreading rates of round jets are predicted incorrectly.
- Turbulence driven secondary flows in straight ducts of non-circular cross section are not predicted at all.
- Laminar and transitional regions of flow cannot be accurately solved with the k-ε model. It should only be used for fully developed turbulent flows.To overcome above mentioned deficiencies, different improvements of the model have been developed and presented next.

- K_E_Two_Layers

Two-equation k-ε model for turbulent flows with integration to the wall.

As explained above, high-Reynolds k-ε is valid for the turbulent flow region, but fail in the viscous sub-layer close to the wall. In order to solve this limitation, the high-Reynolds k-ε can be used in the interior of the flow and coupled to a one-equation model which is used to resolve just the wall region.

k-ε two layers model is a low-Reynolds implementation of the k-ε model that uses the high-Re model only away from the wall in the fully turbulent region, and in the near-wall layer, where the viscosity effects are important, the turbulence is resolved with a one-equation model involving a length-scale prescription.

This model improves the flow prediction in the boundary layer, but requires a very fine mesh in that area to accurately solve the boundary layer. This model is not adequate to be used in combination with law of the wall approximations.

- K_E_Lam_Bremhorst

Two-equation k-ε model for turbulent flows with integration to the wall. The model implemented is based on the description done in [3] and [5].

k-ε Lam-Bremhorst model is a low-Reynolds version of the k-ε model that have been proposed to remove the boundary layer modeling limitation of the high-Reynolds version. This model improves the flow prediction in the boundary layer, but requires a very fine mesh in that area to perform properly. This model is not adequate to be used in combination with law of the wall approximations.

- K_E_Launder_Sharma

Two-equation k-ε model for turbulent flows with integration to the wall. The model implemented is based on the description done in [9].

k-ε Launder-Sharma model is a low-Reynolds version of the k-ε model that have been proposed to remove the boundary layer modeling limitation of the high-Reynolds version. This model improves the flow prediction in the boundary layer, but requires a very fine mesh in that area to perform properly. This model is not adequate to be used in combination with law of the wall approximations.

- K_Omega

Two equation k-ω model for turbulent flows with integration to the wall (low-Reynolds model). The model implemented is based on the description done in [2].

This model was developed in parallel with the k-ε model as an alternative to define the eddy viscosity function. Two convective transport equations are solved for the turbulent kinetic enery and its specific dissipation rate, k and ω, respectively. L is then determined as $k^{1/2}/\omega$.

The k-ω model performs very well close to walls in boundary layer flows, particularly under strong adverse pressure gradients. However it is very sensitive to the free stream value of ω and unless great care is taken in setting this value, spurious results can be obtained in both boundary layer flows and free shear flows.

The k-ε model is less sensitive to free stream values but is often inadequate in adverse pressure gradients. Tdyn incorporates a variant of this model (the k-ω SST model) that tries to circumvent this problem by retaining the properties of k-ω close to the wall and gradually blending into the k-ε model away from the wall. This model has been shown to eliminate the free stream sensitivity problem without sacrificing the k-ω near wall performance.

- K_Omega_SST

Two-equation model for turbulent flows with integration to the wall (low-Reynolds model), expressed in terms of a k-ω model formulation. The k-ω shear-stress-transport (SST) model combines several desirable elements of standard k-ε and k-ω models. The two major features of this model are a zonal weighting of model coefficients and a limitation on the growth of the eddy viscosity in rapidly strained flows. The zonal modeling uses the k-ω model near solid walls and a standard k-ε model near boundary layer edges and in free-shear layers. This switching is achieved with a blending function of the model coefficients. The SST model also modifies the eddy viscosity prediction, improving the prediction of flows with strong adverse pressure gradients and separation.

The performance of standard two-equation turbulence models deteriorates when the turbulence structure is no longer close to local equilibrium. That is why these models generaly underpredict the retardation and separation of the boundary layer due to adverse pressure gradients. This is a serious deficiency, leading to a understimation of the effects of viscous-inviscid interaction which in many cases results in too optimistic performance estimation for aerodynamic bodies. In that cases, the SST variation of the k-ω model leads to marked improvements in performance for non-equilibrium boundary layer regions such as those found close to separation. However, such modifications should not be viewed as a universal cure. For example SST is less able to deal with flow recovery following re-attachment.

- K_KT

Two-equation k-kτ model for turbulent flows with integration to the wall (low-Reynolds model). The model implemented is based on the description done in [10]. It is based on the derivation of an equation for the time scale of turbulence (τ), starting from an equation for the autocorrelation of the turbulence velocity under the assumption of quasistationarity.

In this model, transport equations are solved for k and kτ. Then, L is determined as $C_\mu \cdot k\tau$ where C_μ is taken as a constant.

The main advantage of this model is that kτ is defined on the walls (by contrast ε and ω are singular) and therefore the boundary conditions on the solid walls for this model are more accurate.

LES models

As explained in previous sections, LES turbulence modeling concept is based on the fact that not the entire range of scales of the flow is interesting for the majority of engineering applications. In practical applications, information contained in the "large scales" of the flow is enough to analyse magnitudes of interest. Therefore, the idea that the global flow behaviour can be correctly approximated without the necessity to approximate the smaller scales correctly, can be seen as a possible great advance in the modelling of turbulence. This fact has originated the design of turbulence models that filters the small scales phenomena, allowing to directly calculate the global flow behaviour at large scales. These models are commonly known as Large Eddy Simulation models (LES).

These models are becoming more and more popular and have shown their capability to accurately model complex flows. The main drawback of these models is that in most of the practical cases, they need a finer mesh than the more standard RANSE models to give accurate enough results. Actually, the boundary layer decay of the eddy kinetic energy cannot be accurately capture by this models, unless the mesh definition is close to the DNS requirements.

For this reason, LES models implemented in Tdyn incorporates specific techniques to improve their behaviour in boundary layers.

Tdyn choice of LES turbulence models

- Smagorinsky

Basic large eddy simulation (LES) turbulence model.

This is probably the most popular LES model [6]. Smagorinsky model adds to the stress tensor a nonlinear viscous term depending on an ad hoc fixed small length scale.

While Smagorinsky turbulence model is remarkable by its capacity to reproduce the energy spectra [19], it has has one main disadvantage: the artificial dissipation does not disappear in the vicinity of contour walls (where the velocity is fixed), where it is well known that turbulence vanishes. This drawback is corrected in the Tdyn implementation by including an eddy viscosity damping in the boundary layer area.

Smagorinsky model is sometimes considered as an algebraic or zero-equation models, beause it assumes that the turbulent viscosity is related by algebraic equations to the local properties of the flow. This is fairly straightforward for simple flows but implies a larger mesh density in geometrically complex configurations. The model implemented in Tdyn includes a correction for increasing the accuracy in boundary layer resolution.

- ILES

Implicit LES model based on Finite Increment Calculus formulation.

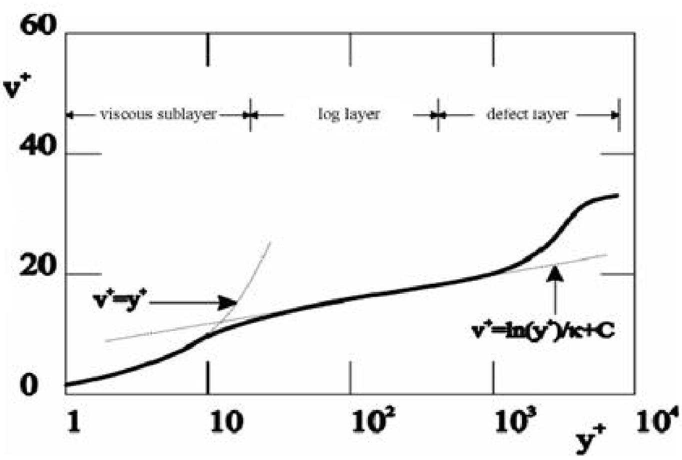
The Finite Calculus Method (FIC) is based in invoking the balance of fluxes in a fluid domain of finite size. This introduces naturally additional terms in the classical differential equations of momentum and mass balance of infinitesimal fluid mechanics, which are function of characteristic length dimensions related to the element size in the discretized problem.

ILES model implements the matrix stabilization terms introduced by the FIC/FEM formulation that allow to model accurately high Re number flows [7,8,11]. The model implemented in Tdyn includes a correction for increasing the accuracy in boundary layer resolution.

3 Near wall modelling

In wall-attached boundary layers, the normal gradients in same flow variables become large as the wall distance reduces to zero. A large number of mesh points packed close to the wall is required to resolve these gradients. In many practical engineering applications the CPU and RAM memory requirements to solve the necessary mesh are too high.

Furthermore, as the wall is approached, turbulent fluctuations are suppressed and eventually viscous effects become important in the region known as the viscous sub-layer (see figure below). This modified turbulence structure means that many standard turbulence models (those so-called high Reynolds models, see RANSE models -pag. 4- for further information) are not valid all the way through to the wall. Thus special wall modelling procedures are required.



Typical velocity distribution in a turbulent boundary layer

In order to overcome the above-mentioned limitations, Tdyn incorporates several advanced boundary condition models. In these models, the near-wall region is not explicitly resolved with the numerical model, but is bridged using the so-called law-of-the-wall functions. In order to construct these functions the region close to the wall is characterized in terms of variables rendered dimensionless with respect to conditions at the wall. These dimensionless variables are defined in terms of the wall friction velocity u_T that is defined as $(\tau_w/\rho)^{1/2}$ where τ_W is the wall shear stress.

Let y be the normal distance from the wall and let U be time-averaged velocity parallel to the wall, then the dimensionless velocity, U^+ and dimensionless wall distance, y^+ are defined as:

$$U^+=U/u_T$$

$$y^+=y\cdot\rho\cdot u_T/\mu$$

If the flow close to the wall is determined by conditions at the wall, then U^+ can be expected to be a universal (wall) function of y^+ up to some limiting value of y^+. This is indeed observed in practice, with a linear relationship between U^+ and y^+ in the viscous sub-layer (see figure above), and a logarithmic relationship in the layers adjacent to this (so-called log-layer). For rough walls, this law of the wall must be modified by scaling y on the equivalent roughness height, z_o (y^+ is replaced by y/z_o) and also by adjusting the coefficients.

The y^+-limit of validity of the law of the wall depends on external factors such as pressure gradient and the penetration of far field influences. In some circumstances the range of validity may also be affected by local influences such as buoyancy forces if there is strong heat transfer through the wall.

The law of the wall implementation of Tdyn is given by the following function:

$$U^+=2.5\cdot\ln(1+\kappa\cdot y^+)+7.8\cdot(1-e^{-y+/11}-y^+/11\cdot e^{-0.33\cdot y+})$$

known as Reichardt's extended law of the wall. This formulation is considered to be valid between $y^+=0$ and $y^+=300$ (about $y=0.1\delta$, being δ the boundary layer thickness).

In many practical cases, the boundary layer thickness can be estimated from the following analytical-based formulae, obtained for a flat plate:

- Laminar flow ($10^3 < Re < 10^6$)

$(\delta/x)\approx(5.0/Re_x^{1/2})$

- Turbulent flow ($10^6 < Re$)

$(\delta/x)\approx(0.16/Re_x^{1/7})$

Where Re is the characteristic Reynolds number of the problem, and Re_x is the local Reynolds number, based on the distance x to the leading edge of the plate,

$$Re_x=\rho Ux/\mu$$

The standard wall functions are valid for smooth walls, but can be modified to take into account roughness effects by adjustment of the constants in the law of the wall. If a rough wall is being modelled the wall distance in the law of the wall is non-dimensionalised with an equivalent roughness height. Tdyn includes a specific Law of the Wall model to take into account this effect (see RoughWall field). The law of the wall used in this case is given by:

$$U+=1/\kappa\cdot\ln(y^+\cdot S_R/100)+8.4$$

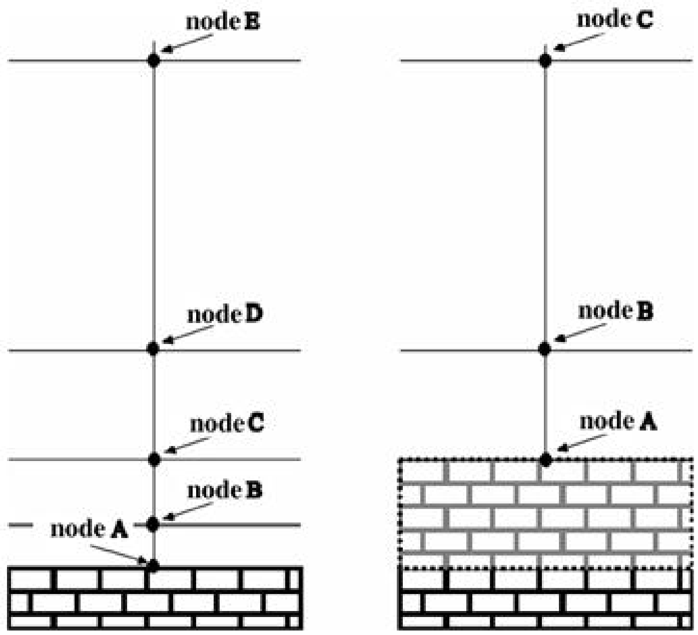
where

$$S_R = (50/k_R^+)^2, \text{ if } k_R^+ < 25$$

$$S_R = 100/K_R^+, \text{ if } k_R^+ \geq 25$$

These universal functions can be used to relate flow variables at the first computational mesh point displaced some distance y from the wall (point C in the left figure below) directly to the wall shear stress without resolving the structure in between. Indeed, these functions can be used to calculate the shear stress (traction) in the closest points to the wall (points A, B, C of the figure at the left). This value can be used as boundary condition for the solution of the fluid flow in the rest of the points (points C, D, E of the figure at the left) using the standard numerical scheme of the fluid solver.

In conclusion, the law of the wall implementation consist of solving the closest points to the wall (A,B,C in the left figure below) using the analytical (universal, but approximate) formulae defined above, while the standard numerical scheme is used for the rest (C,D,E). In order to match the two approaches, the adequate boundary condition is prescribed in the numerical scheme (at point C).



Real domain up to the wall (left), law of the wall approximation (right).

Standard law of the wall functions are one of the biggest sources of misconceptions in turbulent flow computations, even for experienced users. Their purpose is to bridge the extremely thin viscous layer near the surface. They do not free the user from the need to adequately resolve the turbulent portion of the boundary layer.

The calculated flow must be consistent or nearly consistent with the assumptions made in arriving at the law-of-the-wall function equations, i.e. an attached two-dimensional Couette flow with small pressure gradients, local equilibrium of turbulence (production rate of k equals its dissipation rate) and a constant near-wall stress layer. Applying wall functions outside this application range, e.g. for general three-dimensional flows, separating flows, swirling flows and flows along spinning walls, may lead to inaccuracies. However, in many practical cases law-of-the-wall boundary conditions are accurate enough in the presence of separated regions and/or strong three-dimensional flows.

Furthermore, the following constraints should be observed when using law of the wall functions:

- The value of y^+ at the first mesh point (not of the wall, B point in the right figure above) remains within the limit of validity of the wall functions. It is recommended to keep the first point of the mesh below $y^+ < 100$. In many cases, a sensible approach to satisfy the logarithmic profile assumption and to resolve the boundary layer is to place the first node of the mesh (not in the wall) in the range $20 < y^+ < 30$. This procedure offers the best chances to resolve the turbulent portion of the boundary layer.
- Law of the wall functions do not free the user from the need to adequately resolve the turbulent portion of the boundary layer. An adequate resolution means at least eight to ten points in the turbulent boundary layer, if the accurate prediction of boundary layer effects, like wall friction, heat transfer and separation is intended.
- In addition a lower limit on y^+, which ensures that the first point (not in the wall) does not fall into the viscous sub-layer, should be considered. If a global analysis mesh with y^+ below the viscous layer limit can be run (what can be done in the case of analysis with moderate or low Reynolds numbers), then an accurate enough direct solution of the boundary layer, could be obtained (see Vfix Wall boundary type).
- To generate a grid with a pre-specified y^+ distribution is difficult, as y^+ depends on the solution, which is not known during the grid generation phase. As many engineers compute similar flow in similar geometries at similar Reynolds numbers, previous computations can serve as a guideline. In other cases, a simple first guess based in simple flows could be used (i.e. flat plate data).
- Check the resolution of the boundary layer. If boundary layer effects are important, it is recommended to check the resolution of the boundary layer after the computation. This can be achieved by a plot of the ratio between the turbulent and the molecular viscosity, which is high inside the boundary layer. Adequate boundary layer resolution requires several points in the layer.

In many practical cases, above recommendations can not be accomplished. If this is the case, some deviation from real phenomena can be expected.

Wall boundary conditions

Tdyn offers a choice of different sophisticated wall boundary conditions (BoundType option). The most relevant for the turbulent modeling are presented next:

- InvisWall

Impose a slipping boundary condition (i.e. wall normal velocity component will be zero). This condition is adequate for inviscid flows or for those cases, where the boundary layer phenomena can be neglected.

- V_fixWall

Impose the null velocity condition on the boundary (i.e. velocity on the wall will be zero). This condition is used to explicitly resolve the near-wall region with the numerical model. In order to accurately solve the boundary layer using this condition, a global analysis mesh with y^+ (of the first mesh node out the wall) below the viscous layer limit have to be used. In most of the engineering applications, this is only practicable for analysis in a regime with moderate or low Reynolds number.

Applying this condition outside its application range may lead to large inaccuracies in the resolution of the boundary layer and in the evaluation of the friction (viscous) forces.

- DeltaWall

Extended law of the wall condition is applied on the boundary at the wall distance y (i.e. the fluid stress, traction, given by the law of the wall at a wall distance y, will be applied as boundary condition in the fluid solver). The wall distance must be inserted in the field Delta. It can be calculated form the analytical-based formulas for the boundary layer thickness presented in Near wall modelling -pag. 13- section.

The implementation of this boundary condition, based on the Reichardt's extended law of the wall, is presented in Near wall modelling -pag. 13- section. This formulation is considered to be valid between y^+=0 and y^+=300 (about y=0.1δ, being δ the boundary layer thickness).

- RoughWall

Law of the wall condition, taking wall roughness into account, is applied at the wall distance y (the fluid stress, traction, given by the law of the wall at a wall distance y will be applied as boundary condition in the fluid solver). The wall distance must be inserted in the field Delta.

The implementation of this boundary condition, based on the law of the wall with roughness, is explained in Near wall modelling -pag. 13- section. This formulation is considered to be valid between y^+=30 and y^+=100 (about y=0.03δ, being δ the boundary layer thickness).

- YplusWall

Extended Law of the wall condition is applied on the boundary at the non-dimensional wall distance y^+. (i.e. the fluid stress, traction, given by the law of the wall at a wall distance y^+, will be applied as boundary condition in the fluid solver). The non-dimensional wall distance must be inserted in the field Yplus.

This is a simplified (linear) implementation of the boundary condition, based on the Reichardt's extended law of the wall, presented in Near wall modelling -pag. 13- section. This formulation is considered to be valid between y^+=30 and y^+=100 (about y=0.03δ, being δ the boundary layer thickness).

- Cw_U2Wall

A traction given by $C_W \cdot V^2$, where C_W is a constant and V the fluid velocity, is imposed on the boundary. The constant C_W must be inserted in the field Cw.

- ITTC Wall

Extended Law of the wall condition is applied on the boundary at the non-dimensional wall distance y^+ (i.e. the fluid stress -traction- given by the law of the wall at a non-dimensional wall distance y^+ will be applied as boundary condition in the fluid solver). The non-dimensional wall distance must be inserted in the field Yplus.

This boundary condition is similar to YplusWall, but it is corrected based on numerical experiments, to match the friction force results to those predicted by the ITTC 57 friction law. This makes this implementation very useful for naval towing-tank-test analyses.

The boundary layer formulation used in this case, is considered to be valid between $y^+=30$ and $y^+=100$ (about $y=0.03\delta$, being δ the boundary layer thickness).

4 Guidelines on turbulence modeling

Initial and boundary conditions

The iterative solution methods used in Tdyn calculate the flow from an initial estimate of the flow field. The initial guess can influence the convergence process and, in some cases, the converged solution itself. Therefore, it is necessary to set initially a valid solution of the flow equations. Tdyn allows the user to define an starting-up period to adjust the initial flow field defined by the user to a valid solution of the flow equations.

It is important to remark that LES models implemented in Tdyn are insensitive to initial and boundary conditions and therefore do not need any definition of initial fields of the turbulence parameters or imposition of boundary conditions for the turbulence.

Note that Tdyn uses by default the initial conditions to automatically set the boundary conditions of the problem. This is coherent with the fact that the initial conditions must be compatible with the boundary conditions. Therefore, in most of the problems is not necessary to define any specific boundary condition for the turbulence problem, being enough to specify the initial quatities for the turbulence parameters.

The initial turbulence quantities that must be set in Tdyn are the eddy kinetic energy k (EddyKener Field in the Initial Data window) and the length scale of the larger turbulent motions L (EddyLength Field in the Initial Data window). Furthermore, L field is used in the one-equation models to evaluate the turbulent viscosity (see RANSE models -pag. 4-). If possible, these quanties should be defined from experimental data available in similar cases. If there are no experimental data available, the values have to be specified using sensible engineering assumptions (see below), and the influence of the choice should be examined by sensitivity tests with different simulations. In most of the cases, an uniform field for these turbulence parameters, compatible with the boundary conditions, can be used.

Usually, the appropiate value of k for an application is specified through a turbulence intensity level TIL, which is defined by the ratio of the fluctuating component of the velocity (u') to the mean velocity V:

$$\mathrm{TIL} = \frac{u'}{V}$$

$$u' = \sqrt{\frac{1}{3} \cdot \left(u'^2_x + u'^2_y + u'^2_y\right)} = \sqrt{\frac{2}{3} \cdot k}$$

$$k = \frac{3}{2} \cdot \left(\mathrm{TIL} \cdot V\right)^2$$

In external flows over aircrafts, cars or submarines the turbulence level is well below 1%, typically a value of TIL = 0.003 (0.3%) can be selected. In atmospheric boundary layer flows the level can be two orders of magnitude higher (TIL = 0.30 (30%)) and details of the actual boundary layer profiles are needed (i.e. a decaying profile of k in the boundary layer have to be defined). For internal flows iside complex geometries like heat exchangers and rotating machinery, the turbulence level of TIL = 0.05 to 0.15 (5 to 15%) is usually appropriate. For flows in not-so-complex devices like large pipes, ventilation flows etc. or low speed flows (low Reynolds number), the turbulence intensity is typically between 1% and 5%.

In fully developed pipe flow, the turbulence intensity at the core can be estimated as:

$$TIL = 0.16 \cdot Re_d^{-0.125}$$

Where Re_d is the Reynolds number based on the pipe hydraulic diameter D_h. The hydraulic diameter of a duct is usually defined as (A is the cross-section area and P is the perimeter of the section):

$$D_h = 4 \cdot \frac{A}{P}$$

Appropiate values of the turbulent length scale, obtained from experimental data or analytical considerations, are available for simple flows. For example, the classical formulation for L profiles in a fully developed pipe or channel flows (see RANSE models -pag. 4- section) gives a characteristic L value at the core of $L = 0.07 \cdot D_h$.

This gives a reference for the evaluation of the length scale for internal flows. A constant value of length scale derived from a characteristic geometrical feature can be used. In many cases, a value of 1 to 10% of the hydraulic diameter is a reasonable guess.

In other cases, the value of L can be determined from a sensible ratio of turbulent and molecular viscosity μ_T/μ. Note that $\mu_T \sim \rho \cdot L \cdot k^{1/2}$ and therefore $L \sim \mu_T/(\rho \cdot k^{1/2})$.

For external flows with remote boundary layers, a value determined from the assumption that the ratio of turbulent and molecular viscosity is between 1 and 10 is a reasonable guess. In any case, it is recommended to check the consistency of the definitions of k and L by making a plot of the ratio of turbulent to molecular viscosity μ_T/μ.

If more sophisticated distributions of turbulence variables are used, check their consistency with the velocity profile. An inconsistent formulation may lead to an immediate unrealistic reduction of the turbulence quantities after the inlet.

In cases where problems arise with the initial quantities or boundary conditions, the inflow boundary should be moved sufficiently far from the region of interest so that a natural inlet boundary layer can develop.

About mesh requirements

The mesh to be used in the analyses must be adequate to accurately solve the boundary layer. When no law of the wall boundary is used (see V_fixWall option in the Tdyn reference manual), a basic requirement is that at least two points of the mesh must be within the viscous sublayer.

The thickness of the viscous sublayer can be estimated as ($y^+ \le 5$):

$$\delta_v = 5 \cdot \frac{\mu}{\sqrt{\rho \cdot \tau_w}}$$

An average value of the wall stress τ_w can be estimated from the friction factor C_f, that is defined by the following relation (V_r is the inlet or reference velocity):

$$\tau_w = \frac{1}{2} \cdot \rho \cdot V_r^2 \cdot C_f$$

Different analythical or experimental-based formulation of the friction factor are available:

- Colebrook-White law, that estimates the friction factor in ducts for fully developed turbulent flow (Re_d is the Reynolds number based on the hydraulic diameter):

$$\frac{1}{\sqrt{4 \cdot C_f}} = -2 \cdot \log_{10}\left(\frac{2.51}{Re_d \cdot \sqrt{4 \cdot C_f}}\right)$$

- Hagen-Poiseuille solution of the laminar flow equations, that estimates the friction factor in circular ducts:

$$Cf = \frac{16}{Re_d}$$

- ITTC-57 law, that estimates the friction factor of a flat plate (turbulent flow) and that it is commonly used to predict the global friction factor in ships and submarines (Re number is based on the length of the ship):

$$Cf = \frac{0.075}{(\log_{10}(Re) - 2)^2}$$

- Blasius solution of the laminar flow equations ($1000 < Re_L < 2000$) for a flat plate, gives a way of the local friction factor as (Re_x is the Reynolds number, calculated with the distance x to the leading edge of the plate):

$$Cf = \frac{0.664}{\sqrt{Re_x}}$$

- Kestin and Persen obtained an expression for the local friction factor in a flat plate (turbulent flow):

$$Cf = \frac{0.455}{(0.06 \cdot \ln(Re_x))^2}$$

Above formulae, allow to estimate a value for the thickness of the first mesh element in the boundary layer:

$$h_0 = 3.53 \cdot \frac{\mu}{\rho \cdot V_r \sqrt{C_f}}$$

Furthermore, it is recommended to keep the aspect ratio of those elements below 250. Therefore, the recommended maximum element size on the surface of the wall/body is:

$$h_s = 250 \cdot h_0$$

On the other hand, the mesh used in the analysis should have enough elements in the boundary layer area to accurately integrate the equations. It is recommended to place at least eight elements within the boundary layer. This can be verified by estimating the boundary layer thickness. For example, in the case of a turbulent flow about a flat plate, the boundary layer thickness δ can be estimated by the well-known 1/7 power law (x is the distance to the leading edge of the plate):

$$\frac{\delta}{x} = \frac{0.16}{Re^{1/7}}$$

Other consideration to take is that the last element of the boundary layer mesh should have a similar thickness (at least 1/2 ratio) that the element next to it (which thickness is approximately h_s). This must be checked when defining the characteristics of the boundary layer mesh.

Finally, in case of using a law of the wall simplification, the thickness of the first element in the boundary layer can be increased. However, it is recommended to keep this value in the range $30 < y^+ < 60$. This gives a reference for the thickness of the first element size (k ranging from 42.4 to 84.7):

$$h_0 = k \cdot \frac{\mu}{\rho \cdot V_r \sqrt{C_f}}$$

Laminar and transitional flows

The distinction between laminar, transitional and turbulenf flow is difficult. Sometimes the flow appears in different states depending on the area of the analysis domain that it is observed. The general problem of the transition from laminar to turbulent flow, that is the computation of the onset of transition as welll as the length of the transition zone is a subject of fundamental research.

From a practical point of view, the simplest way around this problem is to calculate the flow as a turbulent one. If a model with a low-Reynolds behaviour is used, then the turbulent kinetic energy is approximately zero in the nominal laminar flow regimes, and in this case, the models are able to predict a transitional behaviour. However, much care has to be taken, as this does not ensure that the physics of the transition is respected by the turbulence model. The physics can be very complex and highly different according to the type of transition.

If available, it is recommended to use experimental data to check whether the flow contains extensive regions of laminar or transistion flow, which could be incorrectly estimated by the high-Reynolds κ-ε model with wall functions, or poorly predicted by the low-Reynolds models.

Tdyn allows to intervent in the code to switch the turbulence model on or off at predetermined locations determined by experimental information. This intervention can be done by means of turbulence boundary conditions or using the Tdyn-Tcl interface.

Turbulence in heat and mass transfer

When turbulence is present, it usually dominates all other flow phenomena and results in increasing the effectiveness of the heat and mass transfer. A similar analogy as that used in the kinetic energy transport, based on the turbulent viscosity coefficient, is used in the case of the heat and mass transfer.

The so-called heat transfer eddy diffusivity is specified in Tdyn by the turbulent Prandtl number, Pr (that should be set in the Modules data window, HEATRANS page). Pr is a non-dimensional term defined as the ratio between the turbulent kinematic viscosity and the heat transfer eddy diffusivity. It is useful for solving the heat transfer problem of turbulent boundary layer flows. The simplest model for Pr is the Reynolds analogy, which yields a turbulent Prandtl number of 1. From experimental data, Pr has an average value of 0.85, but ranges from 0.7 to 0.9 depending on the Prandtl number of the fluid in question.

Regarding the mass transfer of substances, the turbulent mass diffusion can be specified in Tdyn by setting the value of the Schmidt number for every species, Sc (analog to Pr). Sc is a dimensionless number equal to the ratio of the eddy kinematic viscosity to the eddy mass diffusion (that should be set in the Modules data window, ADVECT page).

Final recommendation of turbulence modeling

In the following, several general recommendation on turbulence modeling are listed.

- A symmetric computation is often carried out in order to reduce the computing time and memory requirements. But, there are many applications where the geometry is symmetric but the resulting flow is asymmetric. This can be an important factor in predicting the realistic dynamical behaviour of the fluid flow. In that cases, the steady solution that could be obtained is spurious and in contradiction to the physics of the problem. A general recommendation to verify the validity of steady state solutions is to swicth to transient mode once it is obtained, to check if the steady solution remains stable.
- Ensure that low numerical and convergence errors have been achieved in turbulent flow simulations. The relevance of turbulence modelling only becomes significant in CFD simulations where other sources of error, in particular the numerical and convergence errors, have been removed or properly controlled. No proper evaluation of the merits of different turbulence models can be made unless the discretisation error of the numerical algorithm is known, and grid sensitivity studies become crucial far all turbulent flow computations. If possible, it is recommended to run every analysis for at least two different meshes with slightly different mesh sizes, to check the sensitivity of the results to the mesh density.
- If possible, examine the effect and sensitivity of results to the turbulence model by changing the turbulence model being used.
- When using a particular turbulence model, check the published literature with regard to the known weaknesses of the model. The weaknesses of the k-ε model, which is the most commonly used model in industrial applications, are listed in RANSE models -pag. 4- section, together with same indications of possible palliative actions that might be fruitfully considered.
- Decide whether to use a law-of-the-wall boundary condition, in which the near-wall region is bridged with wall functions (see Near wall modelling -pag. 13-). This decision will be based on available resources and the requirements for resolution of the boundary layer. The validity of the wall function approach or the use of a low Reynolds number model should be examined for the flow configuration under study. Law-of-the-wall boundary conditions are accurate enough in the presence of separated regions and/or strong three-dimensional flows.
- The turbulent states that can be encountered across the whole range of industrially relevant flows are rich, complex and varied. After one century of intensive theoretical and experimental research, it is accepted that there is no single turbulence model, that can span these states and that there is no generally valid universal model of turbulence. The choice of which turbulence model to use and the interpretation of its performance (i.e. establishing bounds on key predicted parameters) is a far-from-trivial matter. Next points should be considered to select one model.
- That turbulence model should be chosen, for a particular application, which has been shown to generate the most correct predictions, by comparison with reliable experimental data for similar situations. The greater the departure from the conditions of the experiments, whether through time-dependence, property variations, three-dimensionality, or roughness, the less certainly reliable become the predictions of

the tested model.

- The more complex (and perhaps therefore physically realistic) the model, the finer must be the computational grid, and the greater the expense. Therefore, in the circumstances arising in engineering, the choice must usually be made of the most practicable model

Application example

Analysis of the turbulent flow in an axisymmetric pipe

This example shows the analysis of a fluid flowing through a circular pipe of constant cross-section. The pipe diameter is D=0.2 m and the length L=8 m. The inlet velocity of the flow V_{in}=1 m/s. Consider the velocity to be constant over the inlet cross-section. The fluid exhausts into the ambient atmosphere which is at a relative pressure of 0.0 Pa.

Taking a density ρ=1 kg/m^3 and viscosity μ= 10^{-5} kg/ms, the Reynolds number Re based on the pipe diameter is:

$$Re=\frac{\rho \cdot V_{in} \cdot D}{\mu}=20000$$

Since the problem has symmetry about the pipe axis, the axisymmetric module of Tdyn will be used for the analysis.

For further information about how to run this example using Tdyn, please refer to the Tdyn Tutorials documentation.

Mesh characteristics

The mesh must be adequate to accurately solve the boundary layer. A basic requirement is that the first element of the mesh must be within the viscous sublayer (see Guidelines on turbulence modeling -pag. 19-).

The thickness of the viscous sublayer can be estimated as ($y^+ \leq 5$):

$$\delta_v=5 \cdot \frac{\mu}{\sqrt{\rho \cdot \tau_w}}$$

And the wall stress τ_w can be estimated using the Colebrook-White law, that estimates the friction factor in circular ducts:

$$\frac{1}{\sqrt{4 \cdot C_f}}=-2 \cdot \log_{10}\left(\frac{2.51}{Re \cdot \sqrt{4 \cdot C_f}}\right)$$

From the above formula, C_f is 0.0065, and from the definition of the friction factor (see Guidelines on turbulence modeling -pag. 19-) the wall stress can be finally calculated as:

$$\tau_w = \frac{1}{2} \cdot \rho \cdot V_m^2 \cdot Cf = 0.0033 \frac{Kg}{m \cdot s^2}$$

Finally,

$$\delta_v = 5 \cdot \frac{\mu}{\sqrt{\rho \cdot \tau_w}} = 8.7 \cdot 10^{-4} m$$

And therefore, the second grid point of the mesh must be located within that distance.

Turbulence parameters

The turbulence parameters in pipes can be calculated as follows (see Guidelines on turbulence modeling -pag. 19-):

$$TIL = 0.16 \cdot Re^{-\frac{1}{8}} = 5\%$$

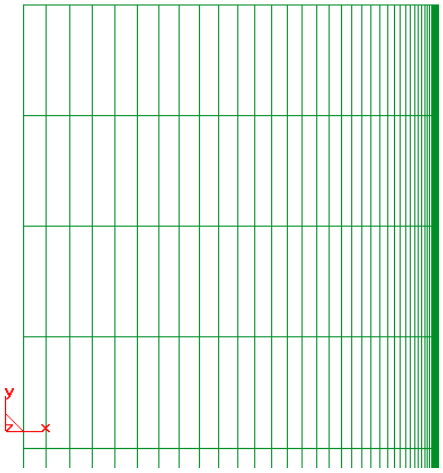
$$K_t = \frac{3}{2} \cdot (TIL \cdot V_{in})^2 = 0.0032 \frac{m^2}{s^2}$$

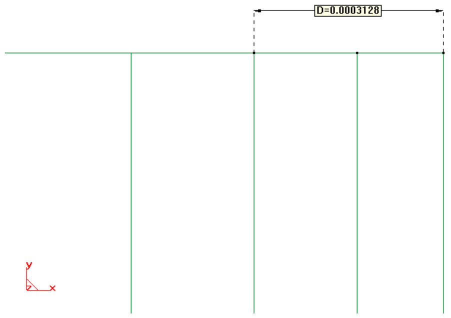
$$L_t = 0.01 \cdot D = 0.002\ m$$

Results

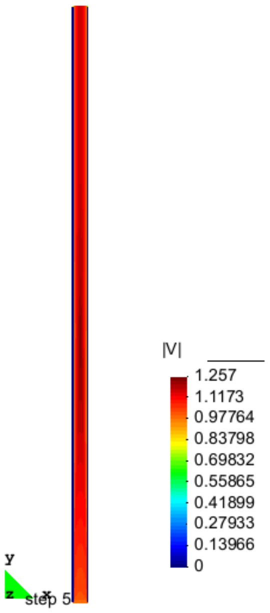
The following results have been obtained for a 2D axisymmetric analysis using the k-ε turbulence model.

The mesh used in this analysis is composed of 300 x 40 linear quad elements, being the second grid point at $3.1 \cdot 10^{-4}$ m from the wall.

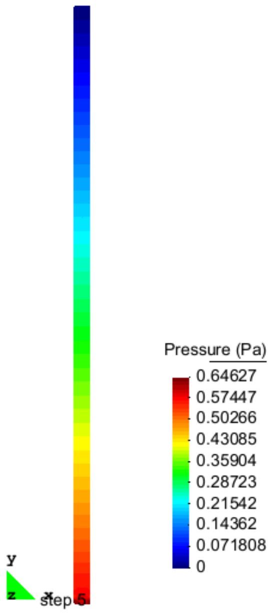




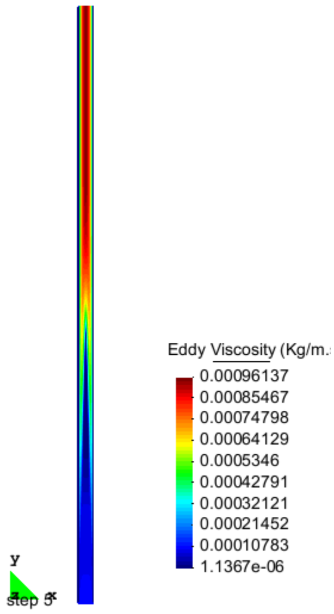
The following pictures show the resulting fields of velocity, pressure and eddy viscosity.



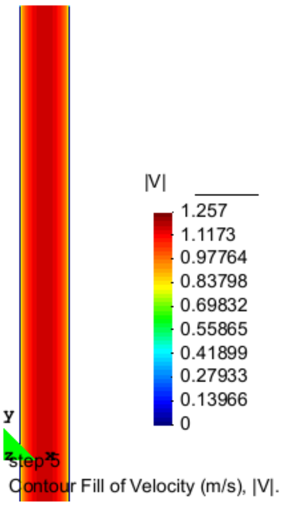
Contour Fill of Velocity (m/s), |V|.

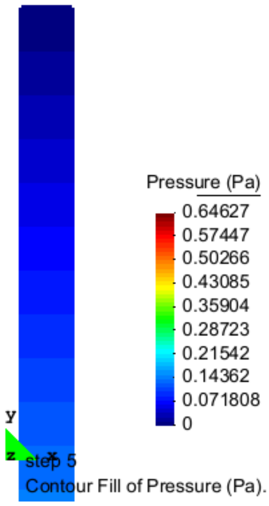


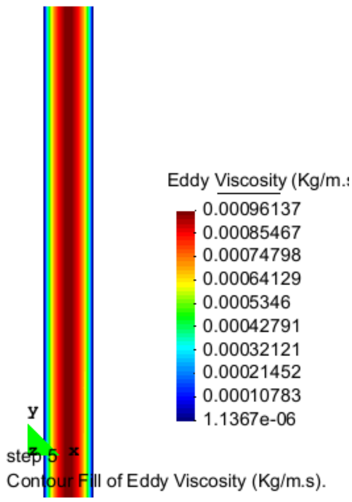
Contour Fill of Pressure (Pa).



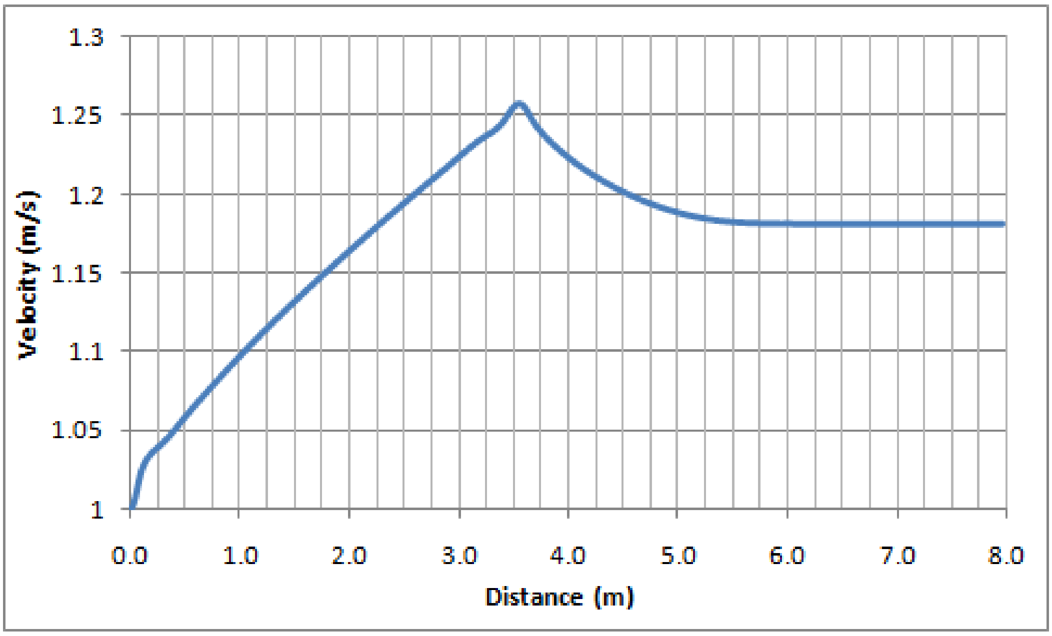
Contour Fill of Eddy Viscosity (Kg/m.s).





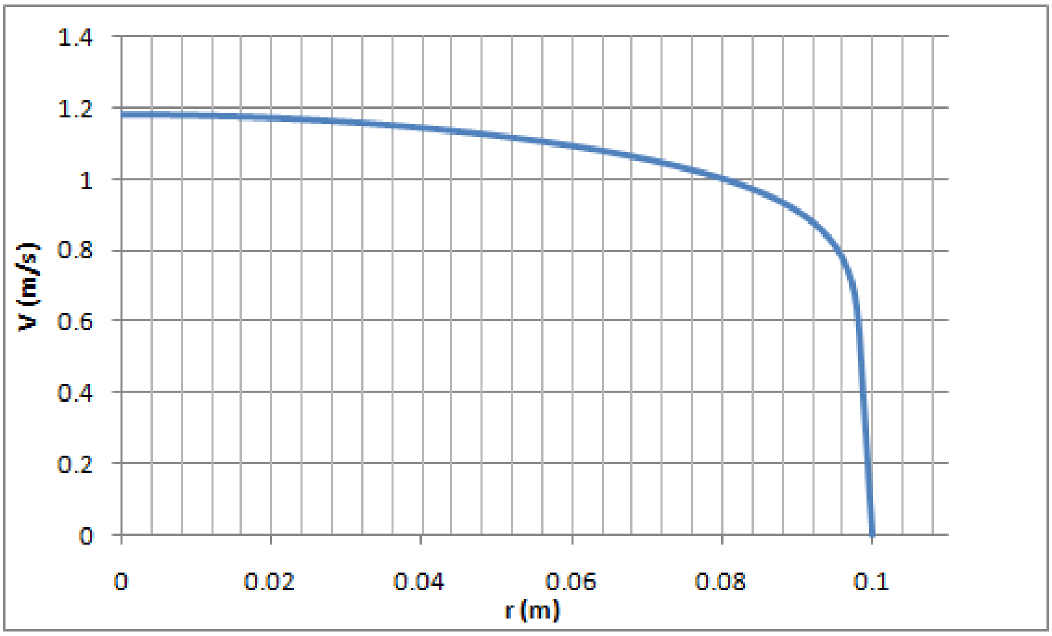


The resulting length of the transition region of the pipe necessary for fully developed flow is about 6 m. As can be seen in the following picture, the fluid does not accelerate once the flow is fully developed.



Axial velocity evolution in the centerline of the pipe

The following picture shows the axial velocity distribution in the outlet section.



Axial velocity distribution in the outlet section

Finally, the numerical evaluation of the friction factor C_f in the outlet section of the pipe gives a value of 0.0066, which is almost identical to the figure estimated by the Colebrook-White law (C_f = 0.0065).

Analysis of the turbulent flow in a 3D pipe

Another example is presented in what follows to illustrate the capabilities for modelling the fluid flow through an actual 3D circular pipe. In this case, pipe diameter is D=6 mm and pipe length is L=231.4 mm. The inlet velocity of the flow is V_{in}=60 m/s. As in the previous example, the velocity is considered to be constant over the inlet cross-section, and the fluid exhausts into the ambient atmosphere which is at a relative pressure of 0.0 Pa. Fluid properties ρ=1.17 kg/m^3 and μ= $1.8 \cdot 10^{-5}$ kg/ms correspond to those of air at 25 ºC.

The Reynolds number Re based on the pipe diameter is:

$$Re=\frac{\rho \cdot V_{in} \cdot D}{\mu}=23400$$

Mesh characteristics

As in the previous example, the thickness of the viscous sublayer is estimated using the friction factor C_f and the wall stress τ_w (see Guidelines on turbulence modeling -pag. 19-).

$$C_f = 0.00623$$

$$\tau_w = \frac{1}{2} \cdot \rho \cdot V_m^2 \cdot Cf = 13.12 \frac{Kg}{m \cdot s^2}$$

$$\delta_v = 5 \cdot \frac{\mu}{\sqrt{\rho \cdot \tau_w}} = 2.2971 \cdot 10^{-5} m$$

Turbulence parameters

The corresponding turbulence parameters result to be:

$$\mathrm{TIL} = 0.16 \cdot \mathrm{Re}^{-\frac{1}{8}} = 4.5\ \%$$

$$K_t = \frac{3}{2} \cdot (\mathrm{TIL} \cdot V_{\mathrm{in}})^2 = 11.18 \frac{m^2}{s^2}$$

$$L_t = 0.01 \cdot D = 6 \cdot 10^{-5}\ m$$

Results

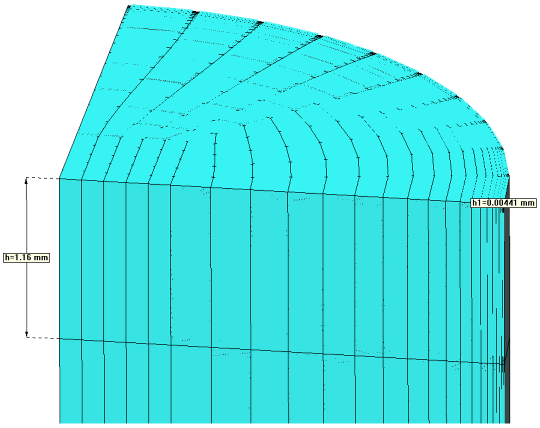
The results presented herein correspond to the analysis performed with both structured and unstructured 3D meshes using the k-ε turbulence model. The second grid point is located respectively at $1.1 \cdot 10^{-5}$ m and $2.2 \cdot 10^{-5}$ m from the wall, so that in both cases it lies within the calculated viscous sublayer distance. For the unstructured case, the 3D boundary layer mesh option was used with the following set of parameters:

first layer height = $7.68 \cdot 10^{-6}$ m

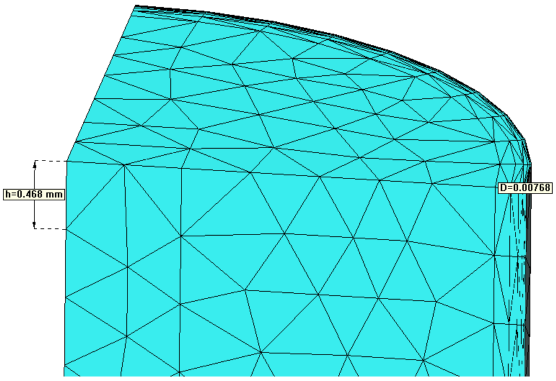
number of layers = 5

geometric grow factor = 1.9

Both meshes are presented in the following picture.

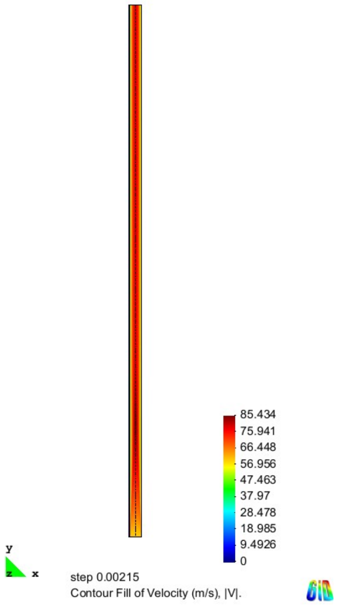


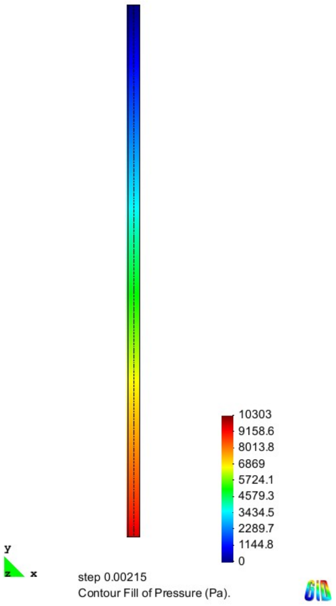
3D structured mesh (63450 elements / 51456 nodes)

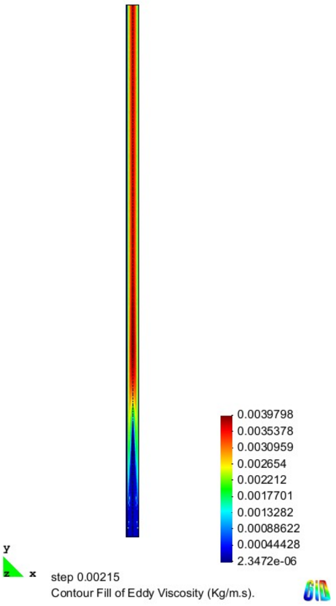


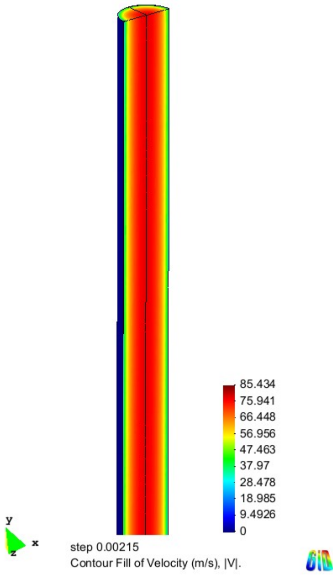
3D unstructured mesh (377460 elements / 64525 nodes)

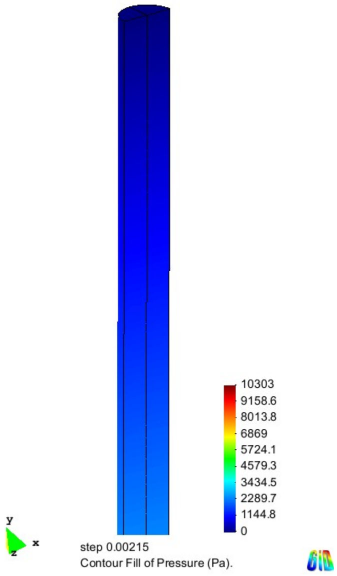
The following pictures show the resulting fields of velocity, pressure and eddy viscosity for the unstructured model. Details of the fields at the outlet section are also given. Similar results are obtained with the structured model.

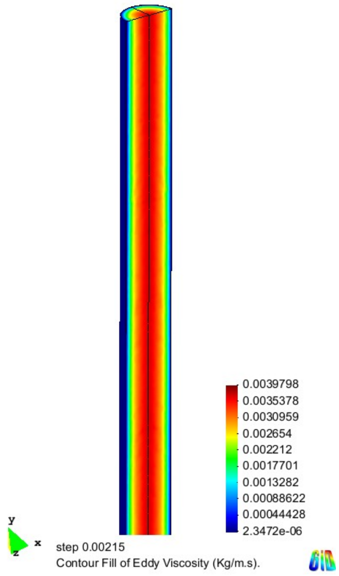




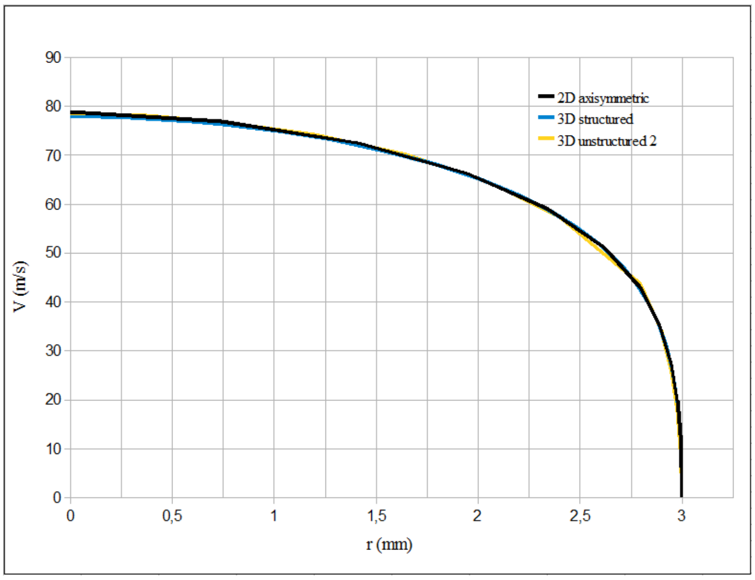








The following graph shows the axial velocity profile in the outlet section. Results obtained with a similar analysis using an axisymmetric model are also included for the sake of comparison.



Axial velocity profile at the outlet section

Practical guidelines within the whole range of Reynolds numbers

The application examples presented above correspond to a relatively low value of the Reynolds number. In those cases, it was possible to design a mesh that allowed the direct solution of the problem in the entire domain. This means that enough elements were located within the boundary layer so that large gradients of field variables (as the velocity for instance) could be accurately captured. Under this conditions a standard computation method can be used to solve the Navier-Stokes equations up to the wall. Unfortunately, this procedure will not be allways possible, in particular for large Reynolds numbers for which flow conditions become more demanding in terms of mesh resolution near the wall. Here, a general procedure will be outlined for modelling turbulent flow in a wide range of situations, providing different strategies depending on model demands (Reynolds number, mesh resolution...) and available computational capabilities.

General procedure

- First, for any given problem the finest possible mesh must be constructed fulfilling all usual quality mesh requirements but without exceeding the available computational capabilities of the computer at hand. In general mesh refinement close to body surfaces is necessary when trying to capture the solution close to the wall.
- If the resulting mesh has enough resolution and therefore it is possible to directly take into account the rapid variations in field variables that usually appear in near wall regions, the standard solution method can be used imposing the natural no-slip condition at the wall. Such a boundary condition can be imposed in Tdyn through the option
  Fluid Dyn. & Multi-phy. Data ▸ Conditions and Initial Data ▸ Fluid Flow ▸ Wall/Bodies ▸ Boundary Type ▸ VFixWall
- Contrarily if model size and Reynolds number are so large that the construction of a fine mesh to allow for direct solution of Navier-Stokes equations in the whole domain is not feasible, a suitable law of the wall must be applied to the body surface following the guidelines given in Near wall modelling -pag. 13-. In particular, the use of the DeltaWall condition is recommended.
  Fluid Dyn. & Multi-phy. Data ▸ Conditions and Initial Data ▸ Fluid Flow ▸ Wall/Bodies ▸ Boundary Type ▸ DeltaWall

Application to flow in pipes modelling in a wide range of Reynolds numbers

The procedure outlined above is applied here for the case of a turbulent flow in a pipe. The range $1\text{x}10^4 < \text{Re} < 1\text{x}10^9$ will be covered. For the sake of simplicity, the velocity, the pipe diameter and the fluid density remain constant so that the Reynolds number is increased just reducing the fluid viscosity.

$$v = 3.5\,\frac{m}{s}$$

$$\rho = 2.5\,\frac{\text{Kg}}{m^3}$$

$$D = 0.2\,m$$

$$L = 8.0\,m$$

Under this conditions, the Colebrock-White law can be used to estimate the value of the friction coefficient. Boundary layer mesh characteristics, turbulence and law of the wall parameters are further calculated as follows:

- First, the friction coefficient C_f is evaluated using the Colebrock-White law for any given Reynolds Number (Near wall modelling -pag. 13-).
- An approximated reference value of the wall shear stress is predicted using the equation $\tau_w = 1/2\left(\rho \cdot v^2\right) \cdot C_f$.
- Boundary layer thickness (δ) is estimated trough the following equation which is actually valid only for turbulent flows around flat plates $\delta = 0.16\left(L/\mathrm{Re}^{1/7}\right)$ being L the length of the pipe.
- On the other hand, viscous sublayer thickness (δ_v) is evaluated for an approximate value of $y^+ = 5$. Hence, $\delta_v = 5 \cdot \mu/\sqrt{\rho \cdot \tau_w}$.
- Assuming that at least one element would be necessary inside of the viscous sub-layer in order to capture the variation of field variables within this region, the minimum element thickness at the wall is taken to be $h_1 = \delta_v/3$. And finally, the length of the element at the wall is calculated assuming that the maximum element aspect ratio allowed for the numerical stability of the algorithms is 250. Hence $l = 250 \cdot h_1$.

The mesh characteristics obtained following the above procedure are summarized in the following table for the different values of Re under analysis. In all cases, the inlet velocity, density and pipe geometry remain the same and Re is varied just taking different, in some cases fictitious, values of fluid viscosity.

μ (Kg/m·s)	Re	C_f	τ_w (Pa)	δ (m)	δ_v (m)	h_1 (m)	l (m)
1.75×10^{-4}	$1 \times 10^{+4}$	6.45×10^{-3}	9.88×10^{-2}	3.11×10^{-1}	8.80×10^{-4}	2.93×10^{-4}	7.34×10^{-2}
1.75×10^{-5}	$1 \times 10^{+5}$	4.50×10^{-3}	6.89×10^{-2}	2.47×10^{-1}	2.11×10^{-4}	7.03×10^{-5}	1.76×10^{-2}
1.75×10^{-6}	$1 \times 10^{+6}$	2.90×10^{-3}	4.44×10^{-2}	1.78×10^{-1}	2.63×10^{-5}	8.75×10^{-6}	2.19×10^{-3}
1.75×10^{-7}	$1 \times 10^{+7}$	2.03×10^{-3}	3.11×10^{-2}	1.28×10^{-1}	3.14×10^{-6}	1.05×10^{-6}	2.62×10^{-4}
1.75×10^{-8}	$1 \times 10^{+8}$	1.49×10^{-3}	2.28×10^{-2}	9.21×10^{-2}	3.66×10^{-7}	1.22×10^{-7}	3.05×10^{-5}
1.75×10^{-9}	$1 \times 10^{+9}$	1.13×10^{-3}	1.73×10^{-2}	6.63×10^{-2}	4.21×10^{-8}	1.40×10^{-8}	3.51×10^{-6}

For the current application, we have established that an optimum mesh can be constructed with the characteristic parameters corresponding to a Reynolds number Re = $1x10^5$. The resulting mesh has a reasonable size and the model can be run in a reasonable computing time in both 2D and 3D configurations. Hence, we have taken this mesh as the reference one to be used for the simulation of all cases in the present study. In consequence, model cases with a Reynolds number lower than $1x10^5$ will be run without law of the wall just imposing the natural no-slip condition at the boundary and solving the Navier-Stokes equations within the whole domain. By contrast, model cases with a Reynolds number larger than 1e+5 will be simulated using a DeltaWall law of the wall (Wall boundary conditions -pag. 16-). Of course, the mesh and the corresponding limit value of Re for which we decided to switch from no-slip condition to law of the wall condition at the boundary is more or less arbitrary. Such a decision will depend in each particular case on the mesh requirements of the geometry at hand and on the computational time that can be assumed to be reasonable for each particular use.

As has been said, the mesh selected for the present analysis would be optimum for a Reynolds number Re = $1x10^5$. This results in a finite element mesh with the following characteristics and typical computing time:

2D axisymmetric cases:

- Number of nodes: 21636
- Number of elements (linear quadrilateral): 22270
- Characteristic computing time: 6000 seconds

3D cases:

- Number of nodes: 74421
- Number of elements (linear tetrahedra): 420065
- Characteristic computing time: 22000 seconds

For the mesh used in this case the first node out of the wall is located at a distance h_1 = 7.03e-5 m. Hence, from the definition of the dimensionless distance to the wall we can evaluate in each case the corresponding value of y^+ at the first node of the mesh (see Near wall modelling -pag. 13-).

$$y^{+} = \frac{h1 \cdot \sqrt{\tau_W \cdot \rho}}{\mu}$$

Re	ρ (Kg/m^3)	μ (Kg/m·s)	τ_w (Pa)	y+ (h_1 = 7.03e-5)
$1 \times 10^{+4}$	2.5	1.75×10^{-4}	9.88×10^{-2}	0.4
$1 \times 10^{+5}$	2.5	1.75×10^{-5}	6.89×10^{-2}	1.7
$1 \times 10^{+6}$	2.5	1.75×10^{-6}	4.44×10^{-2}	13.4
$1 \times 10^{+7}$	2.5	1.75×10^{-7}	3.11×10^{-2}	112
$1 \times 10^{+8}$	2.5	1.75×10^{-8}	2.28×10^{-2}	959
$1 \times 10^{+9}$	2.5	1.75×10^{-9}	1.73×10^{-2}	8355

Taking into account that the extended law of the wall used in the present analysis is considered to be valid in the range $0 < y^+ < 100$, we can see from the table above that the case corresponding to Re = 1e+7 lies in the limit of validity of the wall law in the context provided by the present mesh. Nevertheless good results can still be obtained for larger values of y^+ depending on the particular conditions of the flow.

Remark: The wall of the law approximation is consider to be accurate enough for practical purposes in the range $30 < y^+ < 100$.

As a rule of thumb, the following procedure can be followed to decide the conditions to be used and to determine pertinent values of the law of the wall parameter (y).

- If a mesh can be constructed so that the first node not in the wall lies well within the viscous sub-layer, solve the problem using the VFixWall condition at the boundary. In the present case, and according to the mesh that has been taken as optimal for the analysis, the VFixWall condition can be applied for Reynolds numbers Re ≤ $1\text{x}10^{+5}$.
- If the mesh is not accurate enough to resolve the viscous sublayer, the use of DeltaWall law is recommended. To this aim, it is possible to calculate the value of y+ that corresponds to the given mesh and for each flow condition by taking h_1 as the distance to the wall of the first node of the mesh. For the present case the results are reported in the table above. If the resulting value of y^+ lies within the strict range of validity of the law of the wall 0 < y+ < 100, we can be confident of using the DeltaWall condition setting the corresponding parameter $y=h_1$. For the present case the DeltaWall condition appears to be a feasible approach in the range $1\text{x}10^{+5} < \text{Re} < 1\text{x}10^{+7}$.
- Depending on the particular flow conditions, good results can still be obtained for larger values of y^+. Hence, for larger values of Re for which the constructed mesh does not provide an y^+ value within the theoretical range of validity, it is still recommended to use the DeltaWall condition, although careful revision of the results must be undertaken. Wall friction stress values predicted by the model may require special attention. In this cases, the selection of the DeltaWall parameter remains up to the user's criteria. If it is possible,

running the model using various values of y up to the limiting value y = 0.1 δ could help on elucidate the accuracy of the simulations. In particular it will show the sensitivity of the results to the value of the DeltaWall parameter.

To obtain accurate results, it is also necessary to use sensible values of turbulence parameters to initialize the turbulent fields and to impose realistic turbulence boundary conditions on the boundaries. Essentially, the turbulent kinetic energy and the turbulent characteristic length must be provided.

Fluid Dyn. & Multi-phy. Data ▸ Conditions and Initial Data ▸ Initial and Conditional Data ▸ Initial and Field Data

To this aim the following equations can be used:

$$K = \frac{3}{2}(\mathrm{TIL}\cdot v)^2$$

$$L = 0.01\, D$$

$$\mathrm{TIL} = 0.16\,(\mathrm{Re})^{-\frac{1}{8}}$$

being D the characteristic length of the problem (the pipe diameter in the present case).

The calculated values of K and L for all cases under analysis are reported in the following table:

Re	TIL	K (m^2/s^2)	L (m)
$1 \times 10^{+4}$	4.64×10^{-2}	3.96×10^{-2}	0.002
$1 \times 10^{+5}$	3.79×10^{-2}	2.65×10^{-2}	0.002
$1 \times 10^{+6}$	2.85×10^{-2}	1.49×10^{-2}	0.002
$1 \times 10^{+7}$	2.13×10^{-2}	8.37×10^{-3}	0.002
$1 \times 10^{+8}$	1.60×10^{-2}	4.70×10^{-3}	0.002
$1 \times 10^{+9}$	1.20×10^{-2}	2.65×10^{-3}	0.002

In the table below a summary of the results obtained for the case of the flow in pipes is presented. The values of the friction coefficient predicted by the simulations (average), using two turbulence models (Spalart Allmaras and k-ε) and the different boundary conditions in the wall, are compared with the experimental average given by the Colebrook-White line in the range $1\times10^{+4} \leq \mathrm{Re} \leq 1\times10^{+9}$.

Re	C_f Colebrook-White	C_f (simulations)
$1 \times 10^{+4}$	0.0065	0.0051
$1 \times 10^{+5}$	0.0045	0.0034
$1 \times 10^{+6}$	0.0029	0.0031
$1 \times 10^{+7}$	0.0020	0.0027
$1 \times 10^{+8}$	0.0015	0.0023
$1 \times 10^{+9}$	0.0011	0.0012

5 References

[1] Schlichting, H. Boundary layer theory. Mc. Graw Hill, New York (1969)

[2] D. C. Wilcox. Turbulence modeling for CFD. DCW Industries (2002)

[3] J. Bardina, P. Huang and T. Coakley. Turbulence Modelling Validation, Testing, and Development. NASA Technical Memorandum 110446 (1997)

[4] ERCOFTAC (European Research Community On Flow, Turbulence And Combustion), Best Practice Guidelines for Industrial Computational Fluid Dynamics, UK (2000)

[5] B. Launder and B. Sharma. Application of the Energy Dissipation model of Turbulence to the Calculation of Flow near a Spinning Disk. Letters in Heat and Mass Transfer, vol. 1 (2): 131-138 (1974)

[6] J. Smagorinsky. General circulation model of the atmosphere. Mon. Weather Rev., 91:99-164 (1963)

[7] E. Oñate, A. Valls, J. García. FIC/FEM formulation stabilizing terms for incompressible flows at low and high Reynolds numbers. Comput. Mech. 38: 440-455 (2006)

[8] E. Oñate, A. Valls and J. García. Computation of turbulent flows using a finite calculus-finite element formulation. Int. Jnl. Num. Meth. in Fluids. 54 (6-8): 609-637 (2007)

[9] Patel, V. C. and Rodi, W. and Scheuerer, G. Turbulence Models for Near-Wall and Low Reynolds Number Flows: A Review, AIAA Journal, 23 (9): 1308-1319 (1985)

[10] Zeiermann, S. and Wolfshtein, M. Turbulent time scale for turbulent-flow calculations, AIAA J., 24: 1606–1610 (1986)

[11] Models, Experiments and Computation in Turbulence, E. Castilla, E. Oñate J. M. Redondo eds., CIMNE, Barcelona (2007)

[12] F.M. White. Viscous Fluid Flow. McGraw-Hill, New York (1991)