Cell size at the wall

Hydraulic diameter

The definition of the hydraulic diameter for an internal flow is the following: **D_h=4S/P,** with S standing for the flow surface (surface of the section the fluid flows through) and P the wetted perimeter (perimeter of S). For a pipe with a circular section, the hydraulic diameter is equal to the usual diameter.

Non-dimensional distance

One generally uses the non-dimensional quantity $y^+ = yu^*/\nu$ to define the mesh size close to the wall. In this relation, y is the distance of the centre of the mesh cells to the wall (m), u* the friction velocity at the wall (m/s) and ν the molecular kinematic viscosity of the fluid (m^2/s). The friction velocity at the wall u* represents the wall shear stress τ_p and is defined as $u^*=(\tau_p/\rho)^{1/2}$.

Although the exact calculation of y^+ is not possible *a priori*, one may obtain a reasonable estimation for internal flows, using a friction velocity of 5 to 10% of the characteristic value of the velocity. The value of 5% corresponds to the turbulent flow in a smooth pipe long enough for turbulence to be perfectly developed (in this case, Moody's diagram yields approximately $u^*=(0.02/8)^{1/2}$). As a conclusion, one may retain:

$y^+ = yu^*/\nu$, using a very rough approximation, y^+ is of the order of 0.05 yU/ν to 0.10 yU/ν.

Recommendations

Mesh refinement at the wall should take into account the turbulence model selected. The following rules can be used as a guide however they should be adapted thanks to the recommendations regarding the configuration provided further down.

- **For LES, it is advisable to use a mesh as fine as possible**, when the wall phenomena dominate the flow:
    - In the direction normal to the wall: the first cell centre should be such that **y^+<1,** with a very progressive coarsening towards the core of the flow: the ratio of two successive cell sizes should not be larger than **1.1** (i.e. one should use at least 5 to 10 cells below y^+=20).
    - In the main flow direction: the size Δx of the cells should be such that **Δx^+< 40** and the ratio of two successive cell sizes should not be larger than **1.1 or 1.2** (or in general < 2).
    - In the direction transverse to the flow and parallel to the wall: the size Δz of the cells should be such that **Δz^+< 20** and the ratio of two successive cell sizes should not be larger than **1.1 or 1.2** if no other information is available.
- **For the v2-f "phi model" and the EBRSM model ("low Reynolds" models), it is compulsory** to resolve the boundary layer:
    - In the direction normal to the wall: it is compulsory that the first cell centre be such that **y^+< 1** and **at least 5 to 10 cells should be placed below y^+=20.**
- For the k-omega, the cell size at the wall is of little importance:
    - In the direction normal to the wall: it is advised, however, to build the mesh so that there are no cells below y+ = 1 (otherwise the results may become sensitive to the mesh size because of the asymptotic behaviour of the variable omega)**.**
- **For the other RANS models – k-epsilon and RSM – ("high Reynolds" models), resolution of the** the viscous sublayer layer should be avoided:
    - the mesh, in the direction normal to the wall, should be prepared so that **y^+> 20,**

preferably around 30 and it is desirable to build the mesh so that there is **at least 5 to 10 cells below y=0.1 D_h**

The simulation log file ("listing") provides useful information to check the adequacy of the mesh for a given model. It is also a good idea to use post-processing to visualize the y^+ values that *Code_Saturne* generates (no attention should be paid to the zero values that may appear locally - these merely indicate the presence of impinging or detachment regions). These features are necessary if the user is to modify the mesh appropriately.

The origin of the limitations to the wall cell size

The rules that are proposed above come from the fact that essentially two approaches may be used to deal with the wall region:

- either a fine discretization of the boundary layer, associated with a physical model that can represent a boundary layer (either a "low Reynolds" approach or "direct numerical simulation")
- or a discretization such that the boundary layer is entirely contained in the first cell adjacent to the wall by way of a "wall-function", i.e. an analytical expression that accounts for the variations of the physical quantities across the boundary layer (most often under the hypothesis that the flow is fully developed and 2D on an infinite flat plate) and thus the physical model used for the second and subsequent grid cells does not have to model a boundary layer.

The low Reynolds RANS models are designed to take into account the physical quantities across the whole boundary layer. However, this requires that the discretization is fine enough to resolve the large near wall values of the gradients. This is the reason why one tries to ensure that y^+ is of the order of 1.

The high Reynolds RANS models, on the contrary, produce incorrect results in the viscous sublayer (y^+<10 or 20) because they are not designed to resolve boundary layers: wall-functions are a solution to this problem but they are usually applied *only in the cells adjacent to the wall*. Hence, it is compulsory to make sure that the rest of the mesh is also outside the viscous sublayer. That is the reason why one tries to ensure that y^+ is of the order of 30 at the centre of the first cell adjacent to the wall.

The use of scalable wall-functions for the RANS models may alleviate the problems if the local y^+ drops below y^+<30. Of course, if the mesh is so fine that several layers of cells are located inside the viscous sublayer (ie a "low Reynolds" mesh), the difficulty remains because the physical model is not designed to deal with boundary layers.

One may also note that a tetrahedral grid will cause problems at the wall. In *Code_Saturne* the boundary conditions are applied at the cell faces however tetrahedral grids contain cells which have a vertex on the boundaries but no faces on the boundary. This situation will create local perturbations as the boundary condition gets applied to some cells whose face touches the boundary while not to others whose vertex touches the boundary. This is one of the reasons why the walls should be meshed with a layer of hexahedra or prisms where it is desirable to capture the boundary layer accurately. Moreover, it is generally considered that the maximal width of a developed boundary layer is 0.1 Dh for internal flows. If it is important to capture the boundary layer, it should be discretized with several cells: this is the reason why it is advisable to use at least 5 to 10 cells inside the region extending 0.1 D_h from the wall (from a practical point of view, this is generally a constraint for high Reynolds models only).

For LES and if the influence of the wall is of great importance(heat transfer, thermal fatigue, vibration…), it is reasonable to assume that one wishes to obtain accurate local or transient data at the wall and that one does not wish to use a wall-function based on restrictive hypotheses. However, it is not always possible to avoid the use of a wall-function, because of the CPU cost or because of the mesh size. This is the reason why *Code_Saturne* activates a wall-function by default for LES. If the mesh is fine enough, its influence vanishes and the LES becomes a direct numerical simulation.