Mesh generation

Introduction

The capability of *Code_Saturne* to import any mesh (any type of element, non-conformal meshes…) has been developed to help users during the mesh generation stage.

However, the best suited meshes for the numerical schemes employed in *Code_Saturne* are conformal meshes made of:

- **cubes** with edges aligned with the streamlines or, if that is not possible, made of bricks (i.e. orthogonal hexahedra) with a small aspect ratio and edges aligned with the streamlines (Figure 1)
- **equilateral tetrahedra** for flows without any privileged direction

For RANS computations, at least, a good mesh consisting of tetrahedra (a fine mesh of equilateral tetrahedra) is a better choice than a mesh of hexahedra containing non-orthogonal cells.

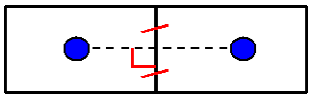

Figure 1: an orthogonal mesh

General advice

When these simple rules cannot be applied to the whole mesh, the following criteria should be respected as much as possible – keeping in mind that they are not absolute criteria stemming from mathematical demonstrations but rather practical advice based on experience:

- **Align the mesh with the streamlines or with the stratifications** (i.e. with the expected isolines of the important quantities): in particular, it is advisable to create 1 to 5 layers of hexahedra or of orthogonal prisms at the walls (i.e. prisms with edges aligned with the normal to the wall).
- **Enforce the following geometrical criteria**, as much as possible (remembering that difficulties may not necessarily appear on low quality cells if the quantities are uniform in these regions):
  - **warping**: angles of 5° or more should be avoided (Figure 2):
    - Dividing the warped faces into triangles (to eliminate the warping) does not necessarily improve the precision (even if the local truncation error is *a priori* reduced) and may deteriorate other quality criteria. However, dividing the warped faces into triangles is compulsory for Lagrangian computations (to avoid the risk of losing particles).
    - Creating non-conformal meshes may lead to warped faces.

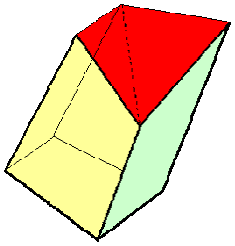

Figure 2: a warped face, coloured in red

  - **aspect ratio** L/h (Figure 3 illustrates in 2D the 3D criterion which is the ratio of the

larger to the smaller characteristic length of the cell):

- optimal value: 1
- maximal value: 10, and up to 100 for cells aligned with the streamlines (or even more, depending on the physical phenomena considered)

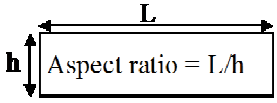

Figure 3: aspect ratio L/h

o **centring deviation**: for a given face of a cell, it is the distance between the centre of the face F and the point O, defined as the intersection of the plane of the face with the line defined by the centres of the neighbouring cells (Figure 4, Figure 5) :

- optimal value: 0
- maximal value: try to keep the point O within the face
- creating non-conformal meshes and refining the mesh may modify the value of the centring deviation (Figure 6)

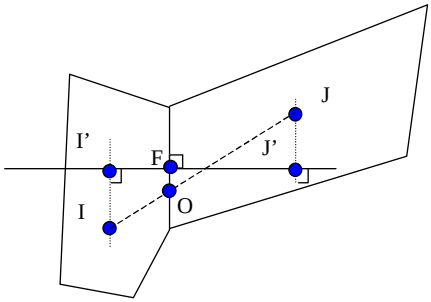

Figure 4: remarkable points

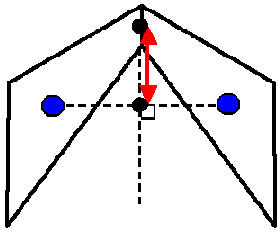

Figure 5: centring deviation (intersection outside of the face)

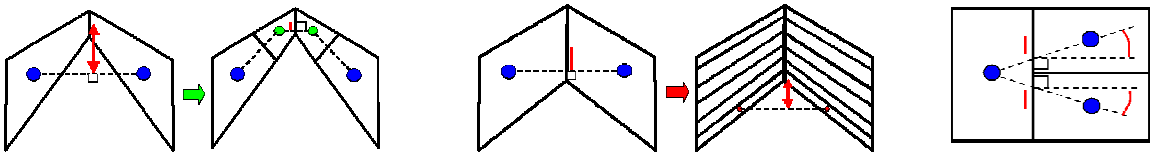

Figure 6: examples of modification of the centring deviation: improvement due to refinement (left), degradation due to refinement (centre), degradation on a non-conformal mesh (right)

o **non-orthogonality angle**: angle between the normal to a face and, for internal faces, the line joining the centre of the neighbouring cells or, for boundary faces, the line joining the centre of the face to the centre of the neighbouring cell (Figure 7)

- optimal value: 0
- usual values: lower than 15° for hexahedra and lower than 45° for tetrahedra

- maximal value: 80°; a large non-orthogonality angle is more easily handled by the solver when it is located away from regions where the gradients of the variables are large (for example non-orthogonality should be avoided near walls). Creating non-conformal meshes may generate very large angles of non-orthogonality. The maximal value that is supported depends on the physical phenomena that are considered.

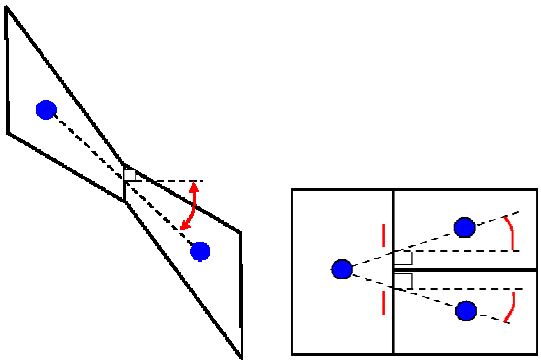

Figure 7: non-orthogonality angle (left) and effect of non-conformal meshes (right)

- **maximal weighting**: (|distance FJ'/ distance I'J'|) for internal faces, with I' and J' standing for the projection of I and J (centres of the neighbouring cells) on the line normal to their common face and containing the centre F of the face (Figure 4); the accuracy my decrease if the weighting is too large:
  - optimal value: 0.5
- **growth rate**: the ratio of the size of two successive cells in a given direction; one should in particular avoid situations where this ratio oscillates about 1 irregular mesh where cells are alternatively long and short as on Figure 8):
  - optimal value: 1
  - maximal value: 1.5 to 2

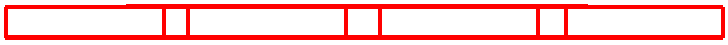

Figure 8: example of an irregularmesh to avoid

- **non-conformal meshes:** this mesh structure is created by joining two meshes along a common surface on which all the vertices or all the edges of the two initial meshes do not match. On the joining surface, edges of one of the original meshes cross faces of the other mesh and/or vertices of one of the original meshes fall inside a cell face of the other ("hanging nodes"). The conformal joining operation modifies the underlying structure so that such meshes[1] may be used (Figure 9).
  - Avoid conformal joining for LES (LES is sensitive to local modifications of the discretisation which can lead to unphysical energy in the flowfield).
  - If several successive layers of conformal joining is used to coarsen, it is advised to use a coarsening ratio of 2 cells / 3 cells rather than 1 cell / 2 cells (Figure 10). This is particularly important if the turbulence level is low and if the main direction of the flow is normal to the conformal joining surface (if the turbulence level is high, the mixing may help to eliminate the perturbations that could appear because of the checkerboard structure of the mesh).

[1] The "conformal joining" transforms a "non-conformal" association of meshes about a surface into a "conformal" composite mesh; the structure of the initial meshes is modified (faces are divided and vertices are added) so that the final composite mesh "conforms" to the following fundamental property that is used to apply collocated finite volume flux based techniques: "any internal face has exactly two neighbouring cells". Thus hexahedral meshes effectively become polyhedral meshes.

- For the other cases of conformal joining, the coarsening ratio should be kept of the order of 1 cell / 2 cell and below 1 cell / 5 cells (if it is essential to joins a single cell to more than 5 cells, it is advisable to do it in several steps/layers or to rethink the mesh structure; otherwise, it may be necessary to use an upwind convection scheme to stabilize the computation, bearing in mind that the accuracy of the results maybe be reduced and LES turbulence modelling will most probably be unreliable).
- If too many non-conformal joining interfaces are required, it may be wise to envisage an unstructured mesh consisting of tetrahedra.

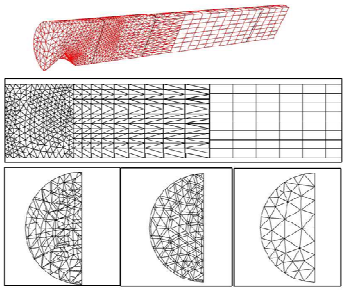

Figure 9: examples of non-conformal meshes

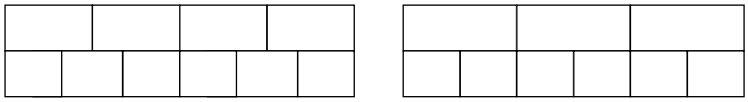

Figure 10: examples of non-conformal meshes – coarsening ratio 2 cells / 3 cells (left), 1 cell / 2 cells (right)

- **conformal joining:**
  - Avoid the conformal joining (!): the technique may be very useful if not other options are available, but it must not be considered as a “magic trick” to use everywhere (it may deteriorate the results).
  - Where possible choose plane interfaces in which the joining meshes map onto each other exactly[2] .
  - Place conformal joining interfaces that may produce non-orthogonal mesh cells as far as possible from the regions of interest (and away from regions with large gradients of the variables, in particular).
  - Identify the sets of faces to be joined together. Indeed, it is possible to let *Code_Saturne* decide which elements should be joined (on the basis of geometry-based criteria). However, it is advised to use “colours” to explicitly identify the faces that must be joined so as to speed up the process and to avoid the possibility for the code to choose to join faces that should not be joined.
  - As much as possible, differentiate between the faces associated with the each conformal joining interface. In doing so avoid the use of the colours already associated with the boundary conditions (this makes the completion of the joining process much easier to check).

- **predefined mesh patterns:**
  - Use O-mesh for circular sections (the central pattern can be a square, a hexagon, an octagon…) and around obstacles.

[2] The meshes that should be joined should rest as exactly as possible on the same geometrical surface, with as little overlapping or chink as possible.

- Use pre-existing patterns for T-junctions and mixing grids (see existing studies).

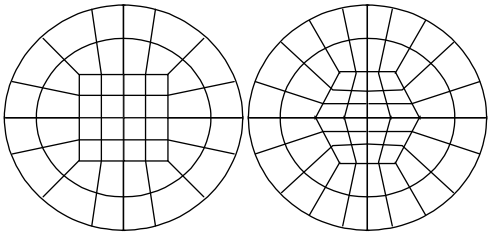

Figure 11: O-meshes

- **specify the cell size** from physical considerations:
  - Evaluate the size of the cells at the wall from the boundary layer thickness and from the constraints imposed by the selected turbulence model (see the dedicated section).
  - Evaluate the size of the cells in the core of the domain from the experience acquired through previous studies on similar geometries and from the size of the structures of the flow that shall be resolved.
  - Use at least 5 cells between two facing walls (with less than 5 cells, the fluid will go through, but the modelling will be too coarse to account for anything but for the mass conservation; if this situation is not local (associated with a singularity of the geometry), one should envisage to change the wall boundary into a slip boundary and to add a head loss source term accounting for the wall friction.
- **verify the conformal joining**:
  - Visualize the joined mesh to ensure that there is no face or portion of face that should have been joined and that has not. Such a face will have kept the “colour” it received before the joining stage: hence, it will be all the easier to visualize it as its colour will be different from that of other sets of boundary faces. It is important to understand that a face or a portion of face that should have but has not been joined becomes a boundary face (in particular, if it has the same colour as wall faces and if no specific action is taken by the user, it will be dealt with as a wall, and hence as an obstacle).
  - Check for possible un-joined portions of faces. It is possible to set a slip boundary condition on such residual portions of faces. However, it remains necessary to visualize them to make sure that they may not create any perturbation in the flow. For example, Figure 12 illustrates how portions of joined faces may remain and produce small steps (two meshes of the same circular-section pipe are considered; their refinement is different; they are joined along a cross section perpendicular to the pipe axis[3]).

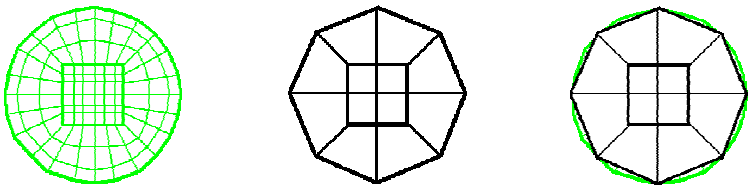

Figure 12: example of conformal joining potentially leading to residual boundary faces

Specific modelling

[3] More clearly, one may think of a mesh approximating the circular section by an octagon and a coarser mesh for which the refinement only allows to approximate the circular section by an hexagon: the joining of these two sections creates residual portions of faces that introduce irregularities of the surface if the code does not manage to detect that the vertices shall be displaced locally to avoid this artefact.

Some models require specific treatment:

- **Second-Order RANS models (Reynolds Stress Models)**: the structures that are resolved by these models are generally finer than those captured by first-order models (k-epsilon or k-omega) and advected further away during a longer periods of time. Indeed, second-order models are generally less diffusive, they account for secondary motion (corners, flow structures downstream a bend…), and the underlying system of equations has a "more convective" nature than the system of equations that is associated with first-order models (the latter is essentially based on the equilibrium between production and dissipation source terms). Hence, the level of refinement that is required to obtain a converged result with a first-order model is generally lower than with that required for a second-order model. Moreover, for a given coarse mesh, the results may be much worse with a second-order model than with a first-order model.
- **LES**: the mesh must be selected to resolve the large anisotropic structures, the model being designed to represent the smaller isotropic ones. To evaluate the cell size, one may – for want of something better – carry out a preliminary RANS calculation to evaluate the size of the turbulent structures:
    - The integral scale $L_T = \alpha\, k^{3/2}/\varepsilon$, with α approximately ranging from 0.1 to 0.3, provides the size of the large structures.
    - The Kolmogorov length-scale $\eta = (\nu^3/\varepsilon)^{1/4}$ provides the size of the smaller structures (the smaller vortices are immediately dissipated by the fluid viscosity).

  LES is theoretically applicable when $L_T >> \eta$ (i.e. when the "inertial zone" of the turbulent spectrum is established: usually for a sufficiently developed turbulence and with a turbulent Reynolds number $Re_t = (L_T/\eta)^{4/3}$ large enough, typically superior to 1000). One should select a mesh size of the order of (or smaller than) $L_T/10$ (with cells smaller than the Kolmogorov length-scale, the simulation effectively becomes a direct simulation). It is not always easy to respect this criterion, since the integral scale may be very small (for example: in a channel flow, the integral scale is proportional to the distance to the wall…).