How to generate a mesh with *snappyHexMesh*

In this report it is explained how to generate a mesh using the *snappyHexMesh* tool of OpenFOAM. The version used here is OpenFOAM 3.0.1. *snappyHexMesh* is the mesh generator in OpenFOAM and it creates a castellated, non structured mesh around a body. The first step is to generate the initial block containing the far field planes (with the proper boundary condition) using the *blockMesh* utility. It is importat to follow the correct enumerations of vertices, as shown in figure 1, otherwise *snappyHexMesh* can not define properly the axes and is not able to understand what region it has to mesh.

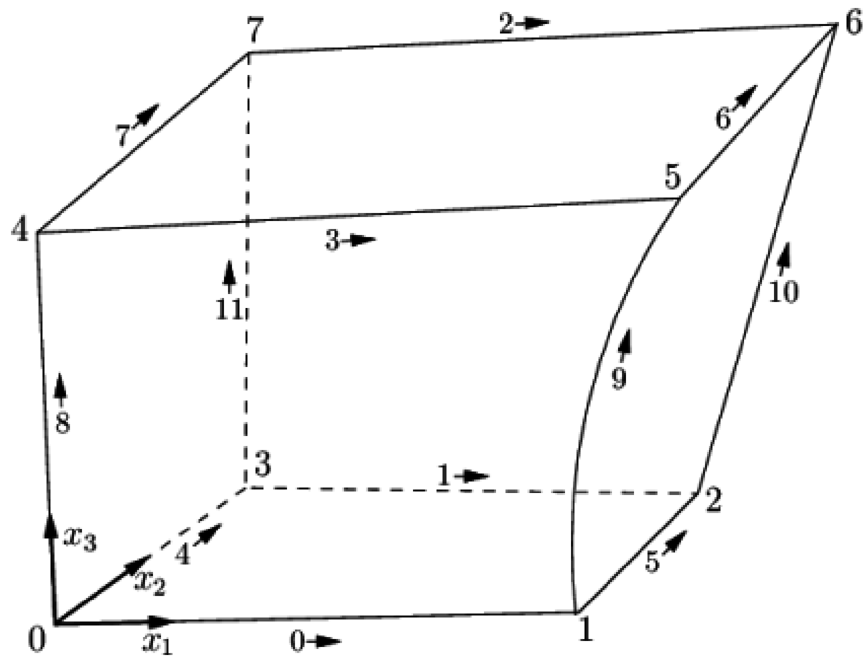

Figure 1: Generic single block generated by blockMesh. Note the right enumerations of vertices and the direction of travel of each edge.

By default all the meshes in OpenFOAM are 3D: if it is wanted to run a 2D case is possible to define only one cell in the third direction. After generated the single block, the second step is to call the *snappyHexMesh* utility. It takes the geometry from *costant/triSurface directory*, a .stl file whick can be generated by any CAD's software. In snappyHexMesh is possible to define different geometries in the first class too, all of them are reported in table 1 with the elements needed to define them properly.

Table 1: List of cell selection shapes.

Shape	Name	Parameters
Box	*searchableBox*	*min,max*
Cylinder	*searchableCylinder*	*point1,point2,radius*
Plane	*searchablePlane*	*point,normal*
Plate	*searchablePlate*	*origin,span*
Sphere	*searchableSphere*	*center,radius*
Plate	*searchableSurfaceCollection*	*geometries*

The *searchableSurfaceCollection* allows to combine, rotate, translate and scale differenct geometries that already exists.
After defining the geometries, *snappyHexMesh* follow three step:
-Creation of a castellated mesh using the entities defines;
-Snapping of the geometry in order to align the cells previously created with the geometry;
-Adding of prism layers to simulate the boundary layer.
Each of the previous steps can be activated by setting true at the beginning of the text *snappyHexMeshDict* file. This utility takes the .stl file and put it into the block previously created: the point on the surface are defined by intersections with the meshed block. After this operation the points on the surface are not displaced anymore, even if there are refining regions around it. From now there are two different regions: one out of the body and another one inside it: with the definition of *locationInMesh* point *snappyHexMesh* understands which region has to be meshed and which has to be eliminated. In my case, the body was an airfoil (NACA0012) and in order to better the analysis it was desirable to control the point on the surface, but as I said earlier after found the point on the surface by intersection *snappyHexMesh* can not move them anymore. The solution I found was to define the refinement regions first (two sphere around the leading and trailing edge and a box around all the airfoil), and then add the airfoil's geometry: the refinement regions, in fact, create, according to the level defined, more cells inside them and in this way when the geometry is added there are more point on the surface and since the sphere are centred on the leading and trailing edge a non uniform meshing in this regions. The last step the tool perform is to add layer: there are different parameters to control, but the most important are the number of layers and the *featureAngle*, which define what angle the layers collapses without extruding. *snappyHexMesh* make the layer high relative to cell size so it is shorter in the most refined region and larger in others: adding more layers is possible to have a uniform prism layer in proximity of the surface that grows far from it. The problem I found was that the prism layer doesn't end at the trailing edge or it ends in the center of the surface and starts again. Several meshes were created with different parameters: in this way it was recognized a depencende between the *featureAngle* and the number of layer. The exact set of parameters can be found in the code at the end of this report; meanwhile the steps of generation of the mesh are shown in the following figures.

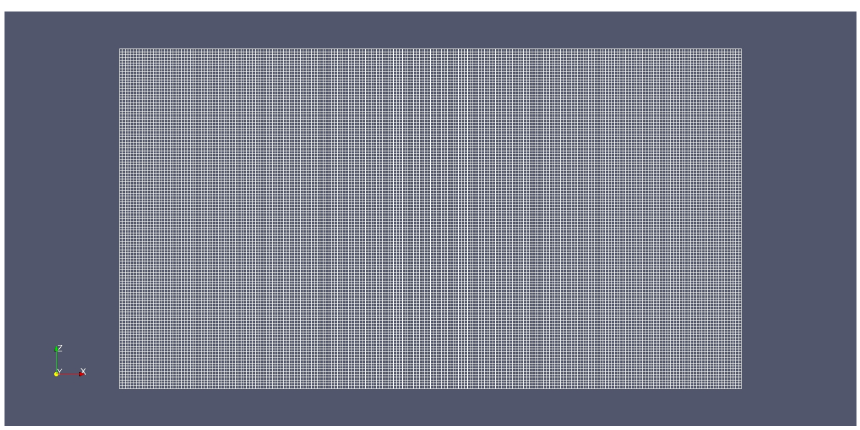

Figure 2: Initial block.

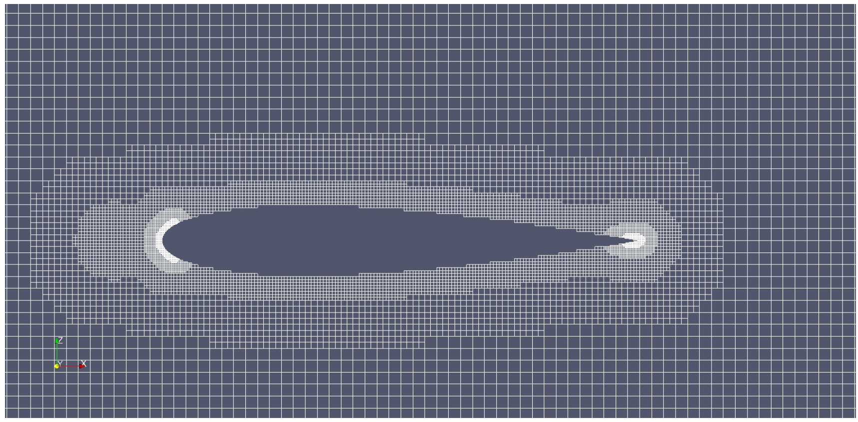

Figure 3: Castellated mesh around the airfoil; note the refinement regions.

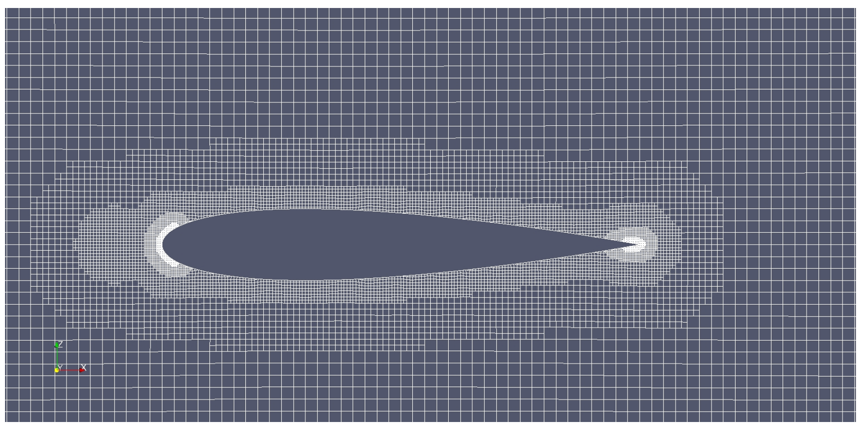

Figure 4: Snapped mesh around the airfoil.

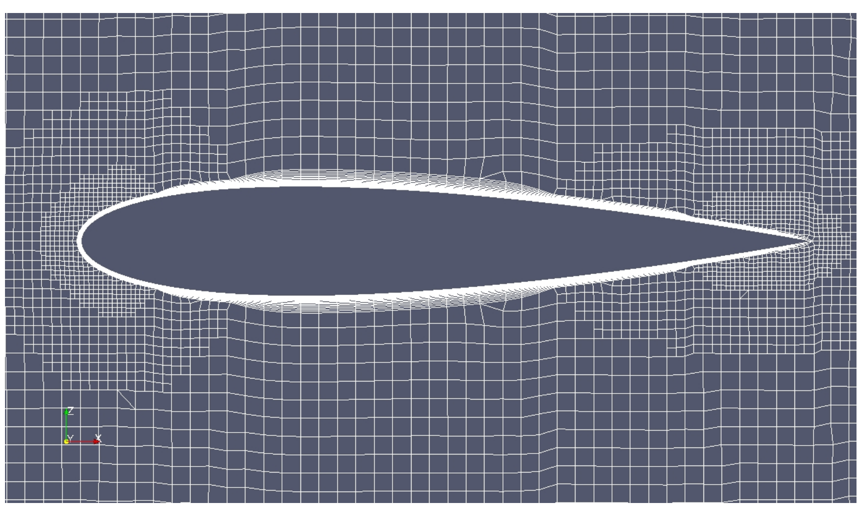

Figure 5: Adding of the prism layer.

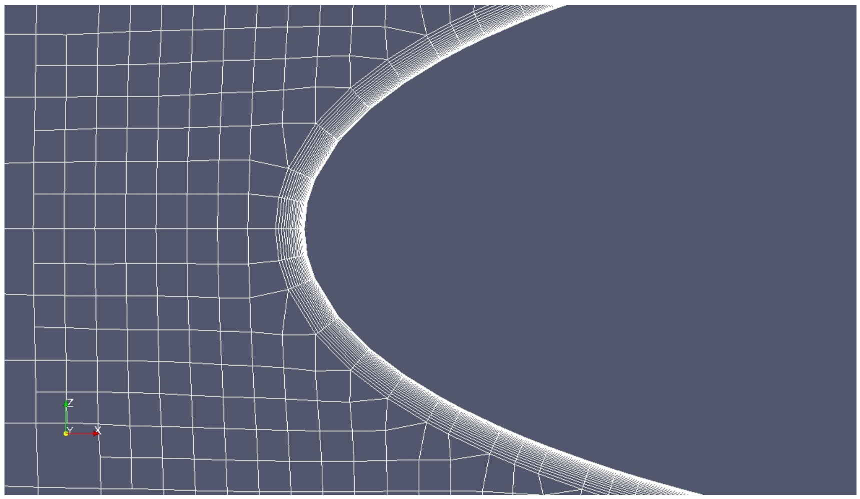

Figure 6: Particular of the leading edge.

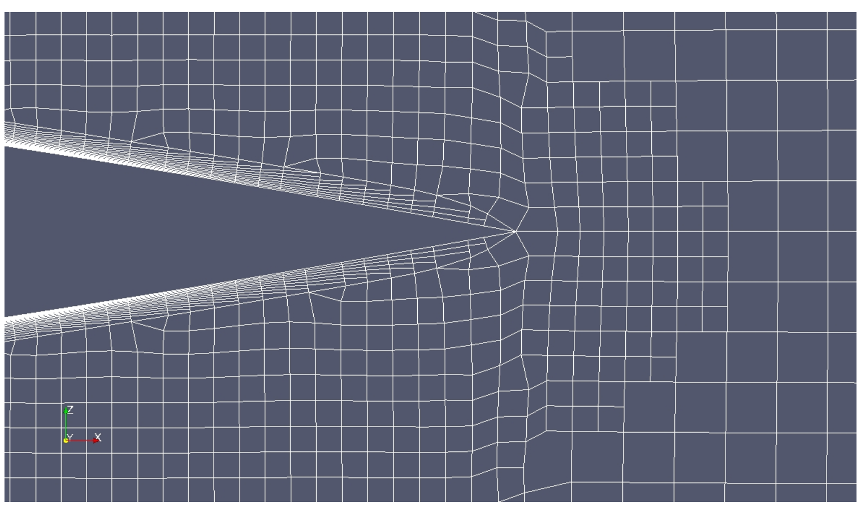

Figure 7: Particular of the trailing edge.