

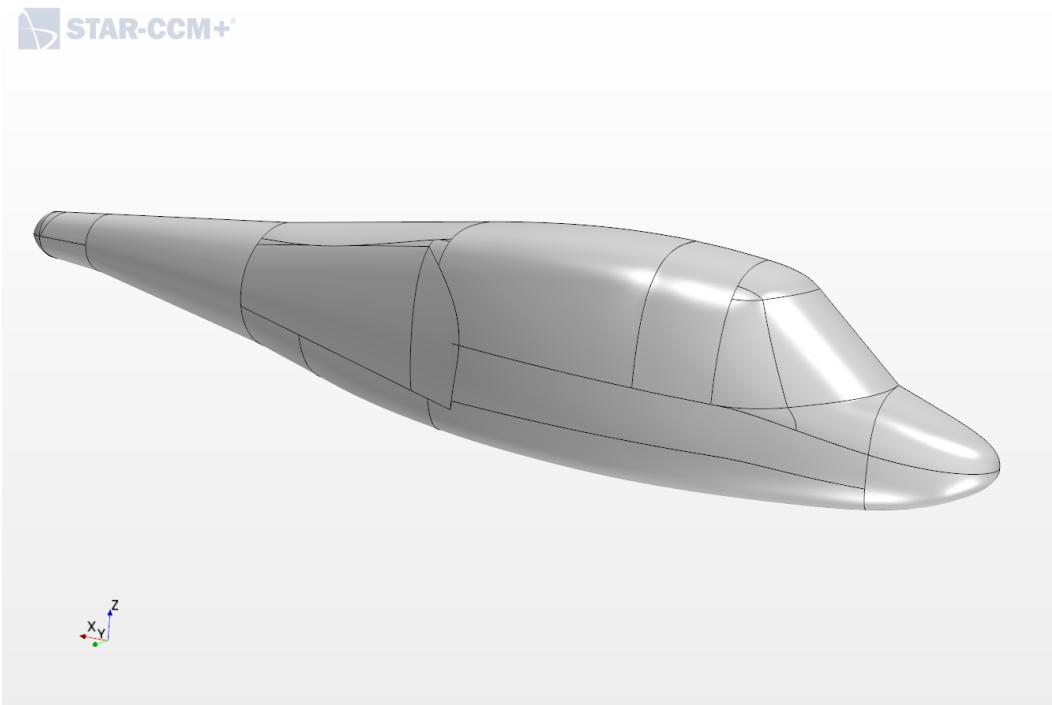
Tutorial 1b : Simulation of Steady Compressible Flow around a 3D Helicopter

1. Introduction

The main objectives of this tutorial are :

- to build a mesh by using the surface wrapper ;
- to simulate the steady compressible flow around the geometry.

The considered geometry is a 3D half helicopter without rotor as shown below.



Helicopter geometry

After this tutorial, you will be able to :

- import an existing geometry
- create a computation domain
- generate a trimmed mesh with prism layers (by using the surface wrapper)
- refine the mesh in some parts of the computation domain
- define the solver for steady compressible flow
- create isosurface, and scalar or vector scenes for post-processing

2. Mesh Setup

2.1 Starting a new simulation

To start Star CCM+ (with Linux), write the two following commands in the terminal:

module load starccm/13.04

starccm

When Star CCM+ is opened, start a new simulation :

File -> New ...

Start a parallel session with 8 cores to accelerate the simulation

Process options = Parallel on Local Host

Compute Processes = 8

Click « OK »

2.2 Importing the geometry

Import the geometry file :

File -> Import -> Import Surface Mesh ...

Choose the file named « 3D_heli.stl »

It seems strange to choose « Import Mesh », whereas we want to import a geometry. The reason is Star CCM+ tessellates all surfaces into triangles and creates a first surface mesh. However this mesh cannot be directly used for CFD and will be remeshed further.

Note the surfaces in a stl file are already tessellated.

In the « import Surface Options », we can choose between two import modes : « Create New Part » and « Create New Region ». You can choose this last option only if the geometry you import can be meshed directly, without doing operations with this geometry to create the computation domain. On the contrary, if some operations (subtraction, merging, splitting ...) are needed with the geometry before meshing (ex : to create the computation domain), choose « Create New Part ».

For this tutorial, the geometry we import only contains the helicopter surfaces, and we have no computation domain. The computation domain is a volume in which the fluid flows. We need to create a block and to subtract the helicopter to obtain the computation domain. We choose « Create New Part » :

Import Mode = Create New Part

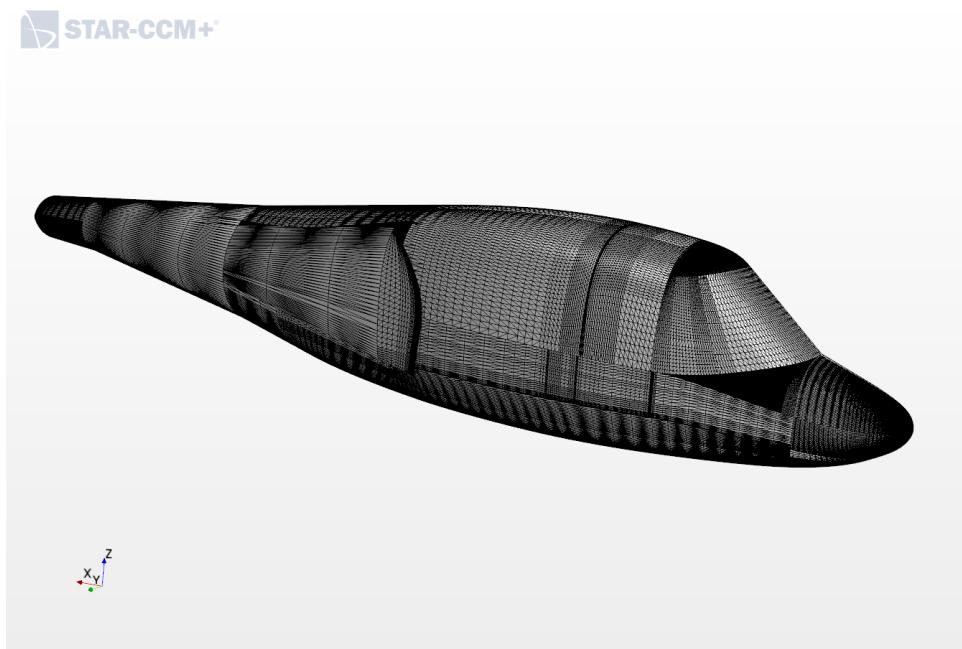
Units = mm

We chose Units = mm, because all dimensions were defined in mm in the cad software to create the helicopter geometry.

Click « OK »

Star CCM+ imported the geometry and created a Geometry Scene to display the helicopter.

To visualize the tessellation, you can click on the icon « Show All Meshes » in the top middle of the window. It is obvious this first surface mesh cannot be directly used without modifications.



Tessellation of the imported geometry

2.3 Creating the Computation Domain

If you go to Geometry -> Parts, you can see a part named « 3D_heli » composed of several surfaces named « cad ». You can combine these surfaces in only one surface and rename it :
 Select all « cad » surfaces then right click -> Combine
 Right click on the resulting surface -> Rename, and rename this surface « heli »

Note that, if needed, we can have several surfaces to define different meshing or physical conditions. For example, we can define a finer mesh for the trailing edge of an airfoil.
 We need only one surface, since the whole helicopter surface will have the same mesh and physical conditions.

For the moment, we have only the helicopter geometry. We need to create a block and to subtract the helicopter to the block to obtain the computation domain.

We create a block whose boundary conditions are far (10 helicopter length) from the investigated geometry :

Go on Geometry -> « Parts » and right click -> New Shape Part -> Block

Define the Maximum and Minimum Coordinates :

Corner 1 : X = -26.0m ; Y = 0.0m ; Z = -26.0m

Corner 2 : X = 26.0m ; Y = 26.0m ; Z = 26.0m

Create

Click « Close »

« Block » was created in Parts.

Rename that Block « Domain » (right click on « Block » and « Rename »)

If we go to Parts -> Domain -> Surfaces, there is only one surface called « Block Surface ». However we need several surfaces to define the different boundary conditions. We will use a « Free Stream » boundary condition type for the far-field conditions. This particular boundary condition is very often used for external aerodynamics with compressible flow. It needs only one surface for the whole far-field conditions : inlet, outlet and side surfaces.

Note this type of boundary condition cannot be used with constant density gas, and it is difficult to converge with low Mach number. That explains why we use it for compressible flow.
 Moreover the « velocity inlet » boundary condition cannot be used with « ideal gas » and works only with constant density gas.

To split the surface :

Select « Block Surface » then right click -> Split by Patch

Select the Patch in the Plane Y = 0.0 m (= surface in contact with the helicopter, it should be patch 28 with version 13.04)

Click « Create » then « Close »

We have now two surfaces for the Block :

- one for the symmetry plane (in contact with the helicopter)
- one for the far-field conditions (all other surfaces)

We rename the surfaces :

Select the surface in contact with the helicopter and right click -> Rename, and rename the surface « Sym ».

Right click on the other surface -> Rename, and rename this surface « FF ».

The block is now ready for the subtraction with the helicopter :

Select « Domain » AND « 3D_heli » and right click -> Boolean -> Subtract ...

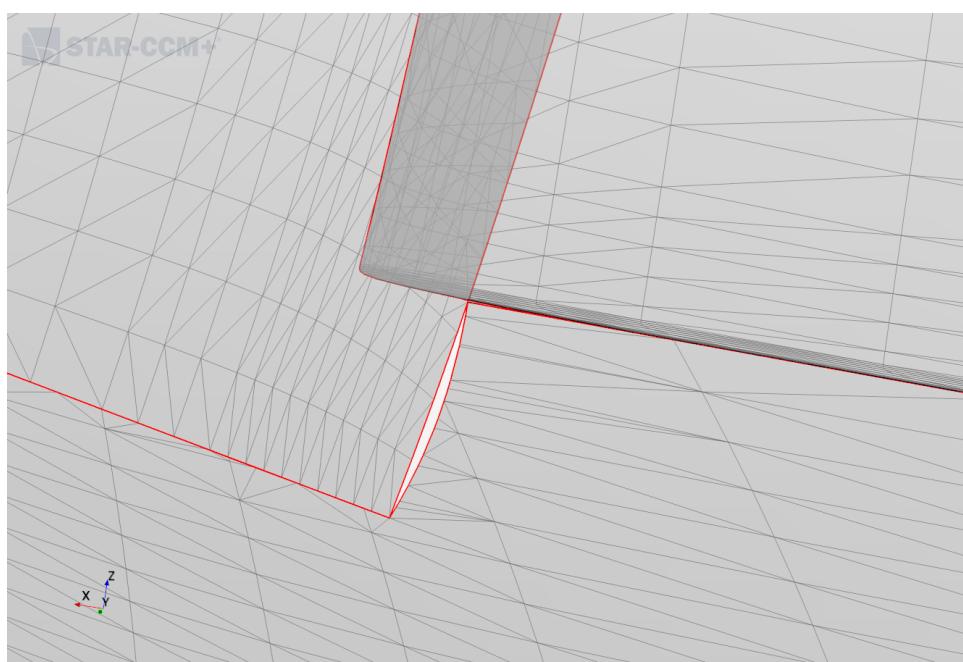
In the « Subtract Parts » window :

Target Part = Domain

Click « OK »

A warning message is opening : « Part 3D_heli is not closed and manifold ».

The two parts were not subtracted because the helicopter surface is not a clean surface. There are holes and overlapping faces as shown below. It happens very often with complex shape geometries.



Overlapping faces (dark grey) and a hole (white) for the helicopter surface

To overcome the problem, we will use the surface wrapper. It is a tool which « cleans » the surfaces to find a closed volume for the computation domain.

The next step is to assign these two geometries to a region. A region is a volume in which we can define meshing and physics models.

In Geometry -> Parts, select « 3D_heli » and « Domain » then right click -> Assign Parts to Regions ...

Verify « 3D_heli » and « Domain » are chosen in « Parts »

Choose « Create a Boundary for Each Part Surface » (and not « Create a Boundary for Each Part »)

Click « Apply » then « Close »

This last option is very important. If you chose « Create a Boundary for Each Part », there would be only one boundary condition for the whole computation domain. It would not be possible to define separately the far-field, the wall and the symmetry plane.

If you go to Regions, you can see a « Region » was created after the last operation « Assign Parts to Regions ». Inside this Region, there are several boundaries with the names defined before : 3D_heli.heli, Domain.FF and Domain.Sym.

If you have only one Boundary, that means you chose « Create a Boundary for Each Part » instead of « Create a Boundary for Each Part Surface » for the previous operation. In that case, you can delete your region and do again the last operation with the correct option.

2.4 Defining the Mesh Models

We need to define a Mesh Continuum with different mesh models and parameters to mesh the Region. To create a Mesh Continuum :

Select Continua and right click -> New -> Mesh Continuum

A Mesh Continuum named « Mesh 1 » is created.

In « Mesh 1 », double click on Models

In the « Mesh 1 Model Selection », we can choose several models for the surface and volume meshes.

For the « Surface Mesh » :

- « Surface Remesher », although this model is optional, it must be chosen for any case. Because the tessellation at the geometry import creates a first surface mesh, which is not correct for volume meshing (the volume mesh starts from the surface mesh). This first surface mesh must be remeshed.
- « Surface Wrapper », this model must be used if no closed volume cannot be generated for the computation domain. In our case, this volume could not be generated by the subtraction of the block and the helicopter. We need to select this model.

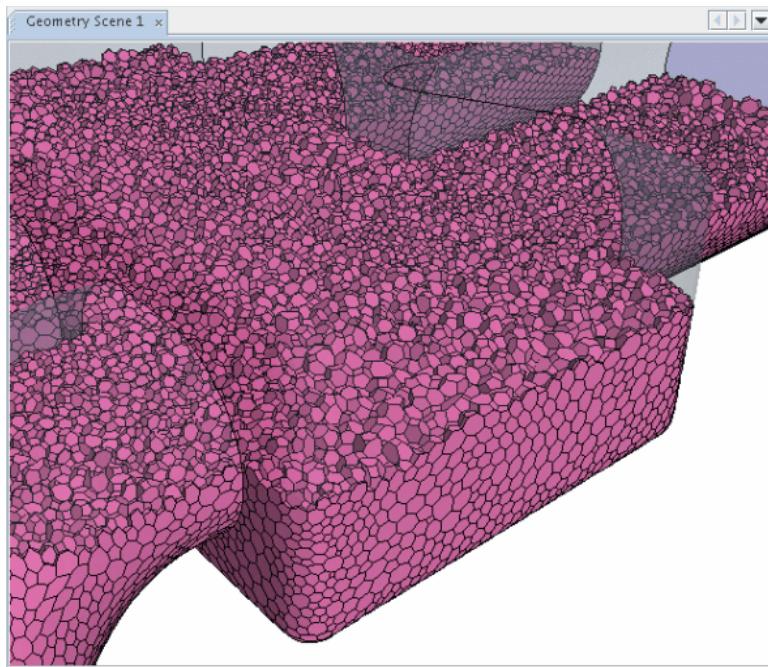
In « Surface Mesh » :

Select « Surface Remesher » AND the « Surface Wrapper » !

For the « Volume Mesh » :

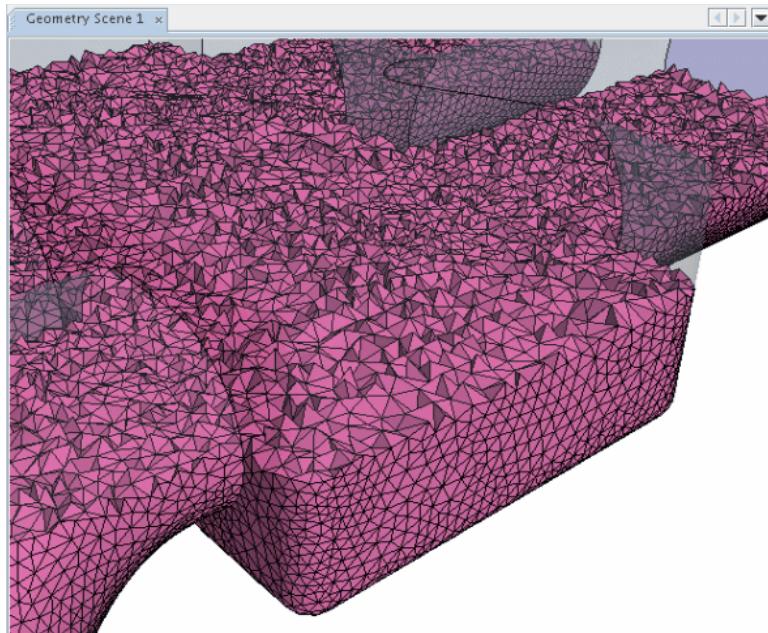
- « Advancing Layer Mesher », this model will be explained further, because it uses a combination of following models.

- « Polyhedral Mesher », this model uses polyhedrals to fill the volume.



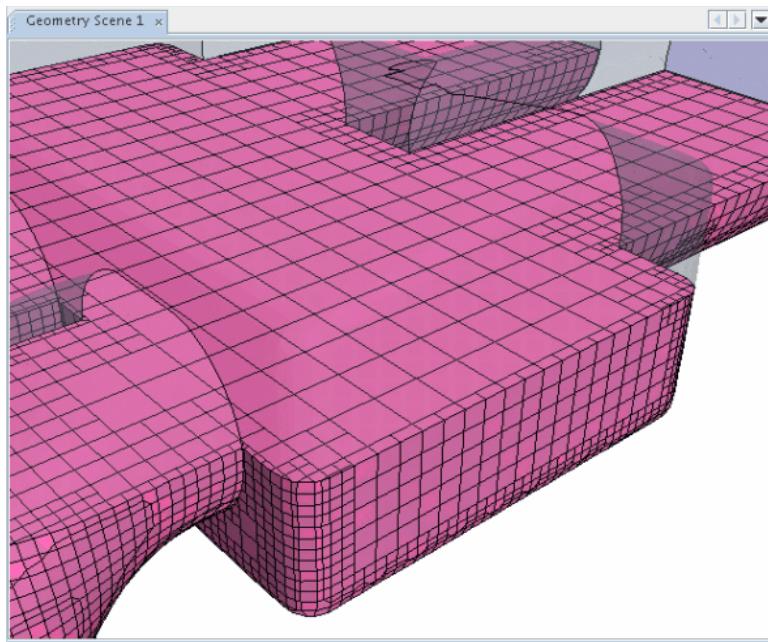
Example of polyhedral mesh

- « Tetrahedral Mesher », this model uses tetrahedral to fill the volume. Compared to polyhedral mesher, the latter converges faster and better captures gradients.



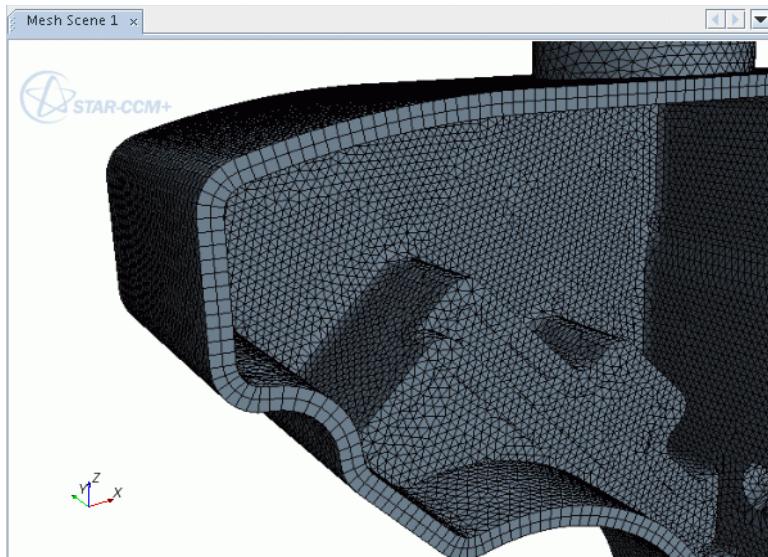
Example of tetrahedral mesh

- « Trimmer », this model uses predominantly hexahedral cells to fill the volume. But cells are cut by the surface of the boundary conditions. Moreover cell refinement is obtained by splitting by 2 for each direction. Compared to polyhedral mesher, it easier to fill small gaps with trimmed cells, and we can better control the increase of the cell size. This kind of cells is not recommended for direct aeroacoustic simulations. The increase or decrease in the cell size by multiplying or dividing the size by 2 generates reflections of acoustic waves.



Example of trimmed mesh

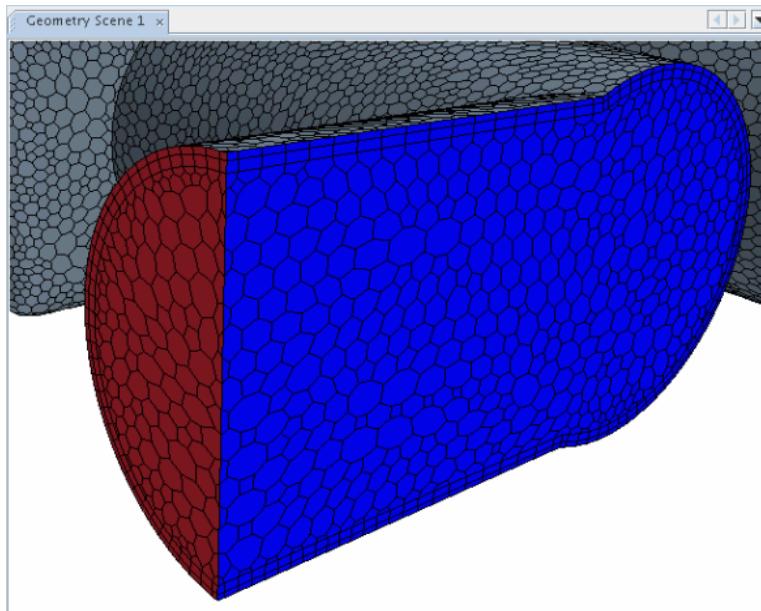
- « Thin Mesher », this model can be selected to mesh volumes with a small thickness. Prismatic cells are generated from the surface mesh (triangles or polyhedrals). This model is very useful to mesh volumes with a constant thickness for thermal analysis.



Example of thin mesh : 2 layers are generated from the surface to mesh the volume thickness.

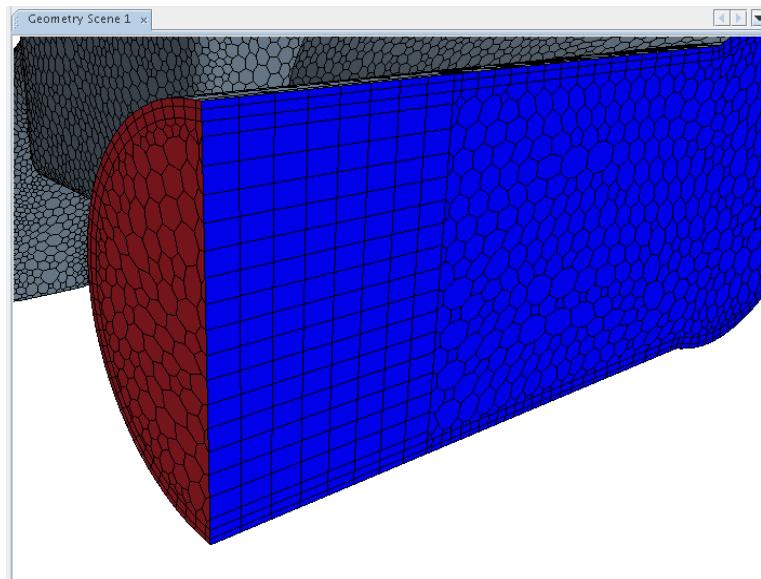
If one of these last models is selected, other optional models are available. Only two are described below :

- « Prism Layer Mesher », this model generates prism layers at walls. Thin cells are extruded from the surface mesh to the volume mesh (polyhedral, tetrahedral or trimmed cells). This model is very often used to capture boundary layers.



Example of prism layer mesh : 2 layers are extruded, inside the volume mesh, from walls.

- « Extruder », once the volume mesh is built, this model generates additional meshes by extruding surface meshes. It can be useful to extrude inlets or outlets, or to generate porous or mass source media.



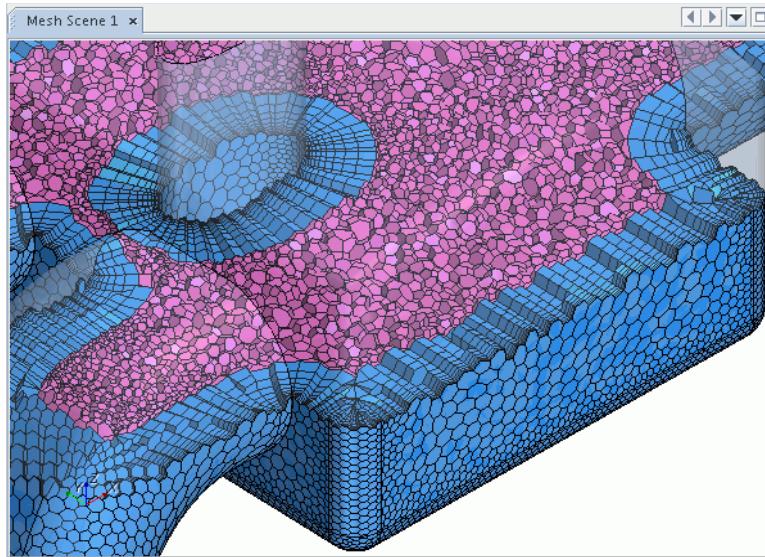
Example of extruder mesh : 8 layers are extruded, outside the volume, mesh from the inlet.

Now we can describe the « Advancing Layer Mesher », which is similar to the combination of the « Polyhedral Mesher » and « Prism Layer Mesher », but with some improvements.

If we use the combination of any volume mesher with the « Prism Layer Mesher », the overall prism layer thickness will be decreased or deleted in corners to avoid generating bad quality cells. That can be problematic with complex shape geometries, because the boundary layers are no more captured in many parts of the computation domain.

The « Advancing Layer Mesher » combines polyhedral cells in the core mesh and prism layers with a constant thickness, even in corners. In that case, some cells in the prism layers are no more extrusion of the surface mesh, and triangular cells are added or replaces prismatic cells in the prim

layers.



Example of advanced layer mesh : the prism layer thickness (in blue) remains constant, even in corners.

In our case, we will use the « Trimmer » with « Prism Layer Mesher ». In « Volume Mesh » :

Select « Trimmer », then « Prism Layer Mesher »

Click « Close »

We do not need additional models.

2.5 Defining the Mesh Reference Values

We need to define the reference values for the mesh. These values will be set by default to the whole computation domain.

There are two ways to define each mesh size in Star CCM+ :

- « Relative to Base », the size is defined with a « Pourcentage of Base » relative to a « Base Size »
- « Absolute », the size is defined with an absolute size, and does not depend on the « Base Size »

Go to Continua -> Mesh 1 -> Reference Values

Some of these reference values are now described :

- « Base Size », as explained above, mesh sizes can be defined by a percentage of base relative to this « Base size ». This parameter can be useful for mesh convergence studies. Instead of changing a lot of mesh sizes, we can only change this base size and all other sizes relative to this base size will be also modified.

- « Maximum Cell Size », it is maximum size for the volume mesh.

- « Maximum Core/Prism Transition Ratio », if enabled, the size of the surface mesh will be adapted to obtain the same size at the transition between the trimmed mesh (core mesh) and the last prism layer. That avoid huge growth rates between these two kind of cells. However that can increase considerably the number of cells.

- « Number of prims layers », it is the number of prisms layers for the « Prism Layer Mesher ». We will explain further how to calculate this number.
 - « Prism Layer stretching », it is the growth factor from a layer to the following one, the first layer (the thinnest one) being in contact with the wall. It must not exceed 1.25.
 - « Prism Layer Thickness », it is the overall thickness for all the prism layers. We will explain further how to calculate this thickness. It is by default defined relative to the « Base Size ». We advise to define an absolute value, because the boundary layer thickness should not depend on the mesh size, when we change the « Base Size ».
 - « Surface curvature », it is the number of points for the discretization of a circle. This number is 36 (1 point every 10°) by default. This parameter is used to refine the mesh on curved surfaces.
 - « Surface Growth Rate », it is the maximal growth rate between two face areas of the surface mesh. The default value of 1.3 should not be exceeded. A value of 1.1 seems better.
 - « Surface Proximity », the « # Points in gap » defines the minimum number of cells (excluding the prism layers) we want between two nearby surfaces, if the distance between those surfaces is above the « Search Floor ». The default value for the search floor is 0.0m, that means this « Surface Proximity » tool will not be used. We need to define a strictly positive value for the search floor to enable this « Surface Proximity » tool.
 - « Surface Size », there are two parameters : the « Minimum Size » and the « Target Size ». It is obvious the « Target Size » must be defined strictly greater than the « Minimum Size ». The surface mesher will always try to build a surface mesh with the target size, unless there is an other surface mesh parameter (surface curvature, growth rate or proximity) constraining the mesher to decrease locally the surface mesh size. However, this size can be lower than the « Minimum Size ».
 - « Template Growth Rate », it defines the number of layers before increasing the mesh size. There are two options. The « Default Growth Rate » defines the growth rate for the surface and volume mesh. It is possible to define a different growth rate for surfaces with the « Boundary Growth Rate ».
 - « Wrapper Scale Factor », it defines the percentage of the « Surface Size » (defined before) which must be used for the « Surface Wrapper ». The latter uses the parameters of the « Surface Size » to wrap a surface. However there is always a loss of precision during the surface wrapper, and differences can appear between the original surface and the « wrapped surface ». A solution could be to decrease the « Surface Size ». But we need sometimes to decrease strongly this size to obtain a correct wrapped surface, that considerably increases the number of cells of the volume mesh. The « Wrapper Scale Factor » can be used to decrease the « Surface Size » only during the surface wrapping to obtain a correct wrapped surface.
- Note that if this factor is too low, the surface wrapper can find leaks where there are holes, and it will not find a closed volume.

We can guess they are sometimes conflicts between these parameters. For example, when we want to refine the surface mesh at the leading edge of an airfoil, we can increase the « Surface Curvature ». However, if the « Minimum Surface Size » is already reached, the increase of the « Surface Curvature » will have no change on the surface mesh. In that case, we could decrease first the « Minimum Surface Size » before increasing the « Surface Curvature ».

We define the following parameters (press « Enter » on your keyboard after every value change) :

Base Size = 0.09m

Maximum Cell Size = 2500 %

Maximum Core/Prism Transition Ratio -> Limit Cell Size by Prism Layer Thickness = Enable, then

Size Thickness Ratio = 2

Number of Prism Layers = 32

Prism Layer Stretching = 1.25

Prism Layer Thickness -> Size Type = Absolute

Prism Layer Thickness -> Absolute Size = 3.83e-2 m

Surface Curvature -> # Pts/circle = 36.0

Surface Growth Rate = 1.1

Surface Proximity = (do not change default values)

Surface Size -> Relative Minimum Size -> Percentage of Base = 6.25 %

Surface Size -> Relative Target Size -> Percentage of Base = 100 %

Template Growth Rate -> Default Growth Rate = Custom, Custom Default Growth Rate = 3

Template Growth Rate -> Boundary Growth Rate = Custom, Custom Boundary Growth Rate = 4

Wrapper Scale Factor = 33 %

Note we defined the « Base Size » as the size we want for the Surface Target Size. Then the « Percentage of Base » for the « Relative Target Size » was defined as 100 % of the « Base Size ».

The « Percentage of Base » for the « Relative Minimum Size » was defined as 6.25 % of the « Base Size ». The reason is $6.25\% = 100\% / 2^4$. The trimmer mesher refines the mesh by splitting cells by 2 for each direction. For the minimum size, starting from the target size, we want four refinements. That explains the percentage of $6.25\% = 100\% / 2^4$.

Now we will explain how we found the 3 values for the prism layer parameters. First of all, we need to calculate :

- the thickness of the first prism layer (also called « Wall Layer ») ;
- the overall thickness of all prism layers.

To calculate these two thicknesses, please refer to the file « Mesh.pdf »

If we consider the following parameters :

- Freestream velocity = 30 m/s
- Density = 1.18415 kg/m³
- Viscosity = 1.85508e-5 kg/(m.s)
- Boundary length = 2.5 m (assumed to be equal to the helicopter length)
- Desired Y+ value = 1

The output are :

- Reynolds number = 1.9e+7
- Estimated wall distance = 3.8e-6 m
- Thickness of the first prism layer = **7.6e-6 m** (twice the estimated wall distance)
- Thickness of the overall prism layers = **3.24e-2 m** (estimated thickness of the turbulent boundary layer)

To find the parameters for Star CCM+, we can use an excel file named « Prism_layers.xls ». As shown above, we have 3 columns :

- the number of the layer

- the thickness of this layer (equal to that of the previous one multiplied by the prism layer stretching = 1.25 here)
- the sum of the thicknesses of this layer and all previous ones

Number of Layers	Layer thickness [m]	Overall layer thickness [m]
1	7.60E-06	7.60E-06
2	9.50E-06	1.71E-05
3	1.19E-05	2.90E-05
4	1.48E-05	4.38E-05
5	1.86E-05	6.24E-05
6	2.32E-05	8.56E-05
7	2.90E-05	1.15E-04
8	3.62E-05	1.51E-04
9	4.53E-05	1.96E-04
10	5.66E-05	2.53E-04
11	7.08E-05	3.24E-04
12	8.85E-05	4.12E-04
13	1.11E-04	5.23E-04
14	1.38E-04	6.61E-04
15	1.73E-04	8.34E-04
16	2.16E-04	1.05E-03
17	2.70E-04	1.32E-03
18	3.38E-04	1.66E-03
19	4.22E-04	2.08E-03
20	5.27E-04	2.61E-03
21	6.59E-04	3.27E-03
22	8.24E-04	4.09E-03
23	1.03E-03	5.12E-03
24	1.29E-03	6.41E-03
25	1.61E-03	8.02E-03
26	2.01E-03	1.00E-02
27	2.51E-03	1.25E-02
28	3.14E-03	1.57E-02
29	3.93E-03	1.96E-02
30	4.91E-03	2.45E-02
31	6.14E-03	3.07E-02
32	7.67E-03	3.83E-02

Calculation of each prism layer thickness and overall prism layer thickness

We can notice the thickness of the first prism layer was set **7.6e-6 m**, as found before. All following layers have a thickness equal to that of the previous one multiplied by the prism layer stretching (=1.25). We need at least 32 layers to reach an overall layer thickness equal or greater than **3.24e-2 m.**

2.6 Defining the Boundary Mesh Conditions and Values

All reference values are now defined and they will be used as default values for the whole computation domain. However we need different sizes in some parts of the geometry like the far-field surface or the symmetry plane.

If we have a look in **Regions -> Region -> Boundaries -> Domain.FF -> Mesh Conditions**, we can see there are similar parameters, as defined before for the mesh reference values. Now we can customize different parameters only for the selected boundary (here Domain.FF).

We define different surface sizes for Domain.FF and Domain.sym :

Go to Regions -> Region -> Boundaries

Select the 2 boundaries (with Ctrl) Domain.FF and Domain.Sym and right click on any selected boundary -> Edit

In the « Multiple Objects » Window, go to « Mesh Conditions » -> « Custom Surface Size » and enable « Custom Surface Size »

« Mesh Values » appeared below. We can define a minimum and a target sizes for the 2 selected boundaries. These sizes must cover a large range of values :

- the minimum size must be the minimum one in « Reference Values », because the boundary « Domain.Sym » is in contact with « 3D_heli.heli » whose minimum size is defined in the « Reference Values » (i.e. 6.25% of the Base Size)
- the target size is the greatest size we want for the far field boundaries. 2.25 m = 2 500% of the Base Size seems to be a reasonable value for our case.

Still in the « Multiple Objects » Window, go to « Mesh Values » -> Surface Size -> Relative Minimum Size -> Percentage of Base = 6.25 %

Relative Target Size -> Percentage of Base = 2 500 %

Click on « Close »

The 2 boundaries have now surface sizes different from those in « Reference values ».

Concerning the prism layers, they will be generated for each boundary with « Wall » type, which is the default type for every boundary. This is not a problem for « Domain.FF » and « Domain.Sym » if we change their type respectively to « Free Stream » and « Symmetry Plane » :

Go to Regions -> Region -> Boundaries -> Domain.FF, change « Type » to « Free Stream »

Go to Regions -> Region -> Boundaries -> Domain.Sym, change « Type » to « Symmetry Plane »

No prism layer will be generated for « Domain.FF » and « Domain.Sym », since they are no more walls.

2.7 Defining the Mesh Refinement in the Wake

We refine finally the mesh in the wake. We need to create a « Volumetric Control » to define a volume mesh size inside a box. To create this box :

Go to « Geometry » and right click on « Parts » -> New Shape Part -> Block

Corner 1 = [1.3, 0.0, -0.2] m, m, m

Corner 2 = [5.0, 0.2, 0.2] m, m, m

Click « Create »

Rename that block « Volume control » (right click and « Rename »)

To create the volumetric control :

Go to « Continua » -> Mesh 1 -> right click on « Volumetric control » -> New « Volumetric Control 1 » is created.

In « Volumetric Control 1 » Properties -> Parts, select the part « Volumetric Control »

In « Mesh Conditions » -> Trimmer, enable « Customize Anisotropic Size »

In « Mesh Values » -> Trimmer Anisotropic Size -> Enable « Custom Y Size » AND « Custom Z Size »

In « Trimmer Anisotropic Size », change « Relative Y Size » AND « Relative Z Size » to Percentage of Base = 12.5 %

We can see the interest of the trimmer mesher which can generate meshes with anisotropic sizes, contrary to polyhedral and tetrahedral meshers.

The meshing setup is now completely defined. You can save your simulation file before generating the mesh :

Go to « File » -> Save as ... and save you simulation as « 3D_heli.sim »

2.8 Generating the Surface Mesh

Go to Mesh

We can see we have two options for the mesh generation :

- « Generate Surface Mesh », only generates the surface mesh ;
- « Generate Volume Mesh », generates first the surface mesh, then the volume mesh from the surface mesh.

Before starting the volume mesh generation, make sure the surface mesh is correct !

Click on « Generate Surface Mesh »

After a few seconds, the surface mesh is generated. The final number of faces should be 28 922.

This surface mesh was generated in two steps :

- the surface wrapper, which generated a closed volume
- the surface remesher, which remeshed the wrapped surface with the mesh parameters

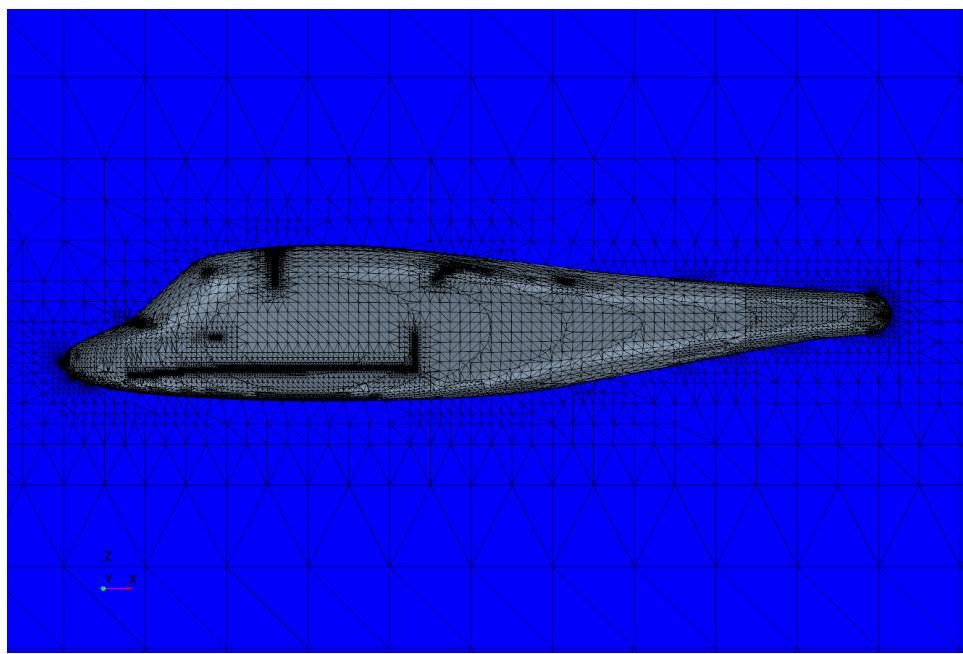
To display first the wrapped surface mesh :

Go to « Scenes » -> Geometry Scene 1 -> Displayers

Double click on « Outline 1 » to hide it

On « Geometry 1 », change « Representation » to «Wrapped Surface »

Go into « Parts » in « Geometry 1 », deselect all « Parts » and select all « Boundaries » in « Region »

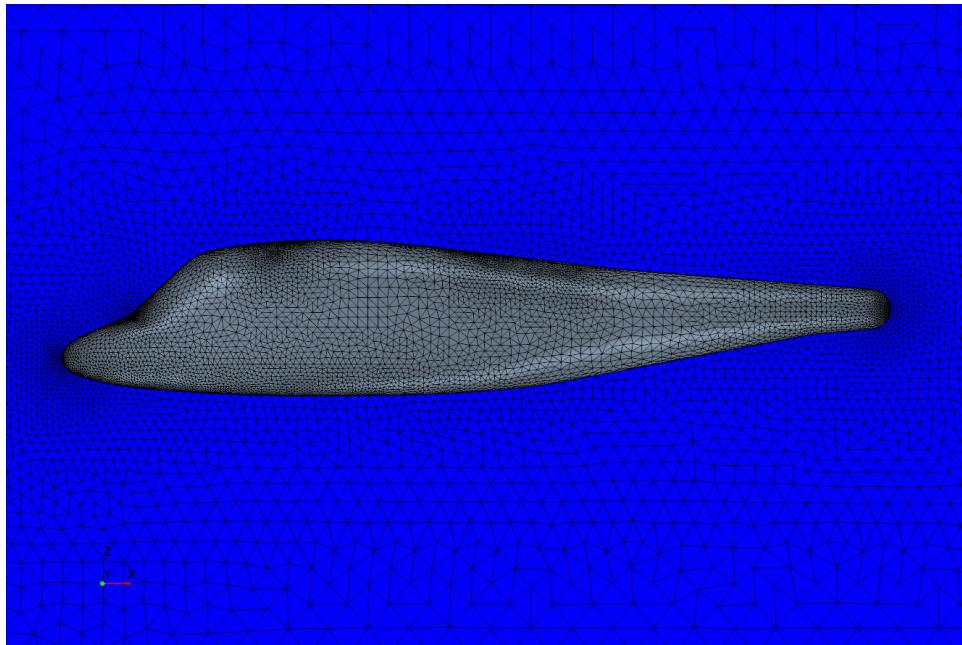


Wrapped Surface Mesh

We can see now the wrapped surface mesh. It clearly shows the wrapper surface closed the volume with a very fine surface mesh where we had problems of connection between faces.

We display now the remeshed surface mesh :

Go to « Scenes » -> Geometry Scene 1 -> Displayers -> Geometry 1 and change « Representation » to « Remeshed Surface »



Remeshed Surface Mesh

We have now the remeshed surface with the correct sizes for the surface mesh.

2.9 Generating the Volume Mesh

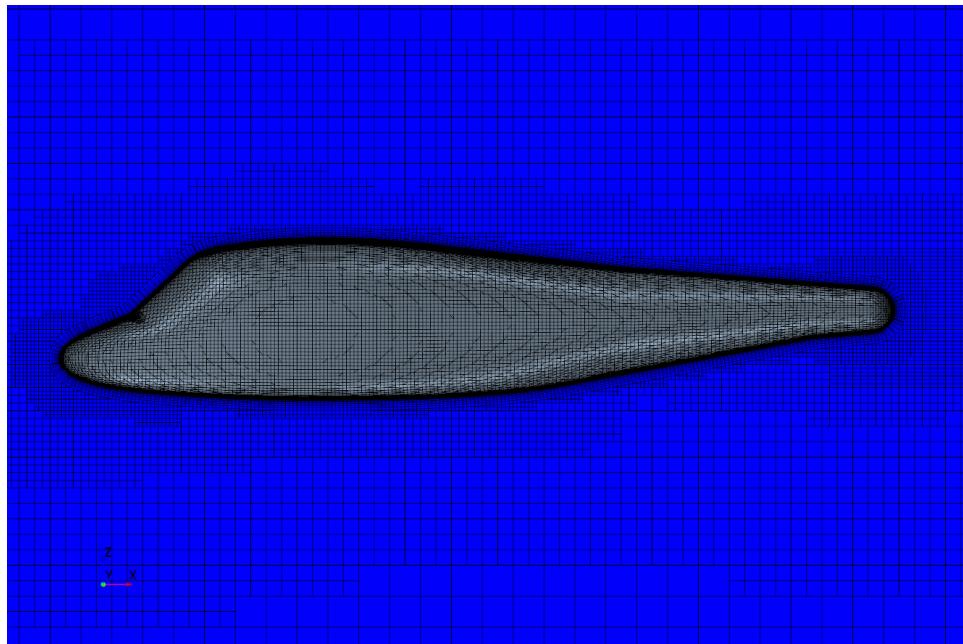
Go to Mesh and click on « Generate Volume Mesh »

After a few seconds, the volume mesh is generated. The final number of cells should be 673 262.

To see resulting volume mesh :

Go to « Scenes » -> Geometry Scene 1 -> Displayers -> Geometry 1 and change « Representation » to « Volume Mesh »

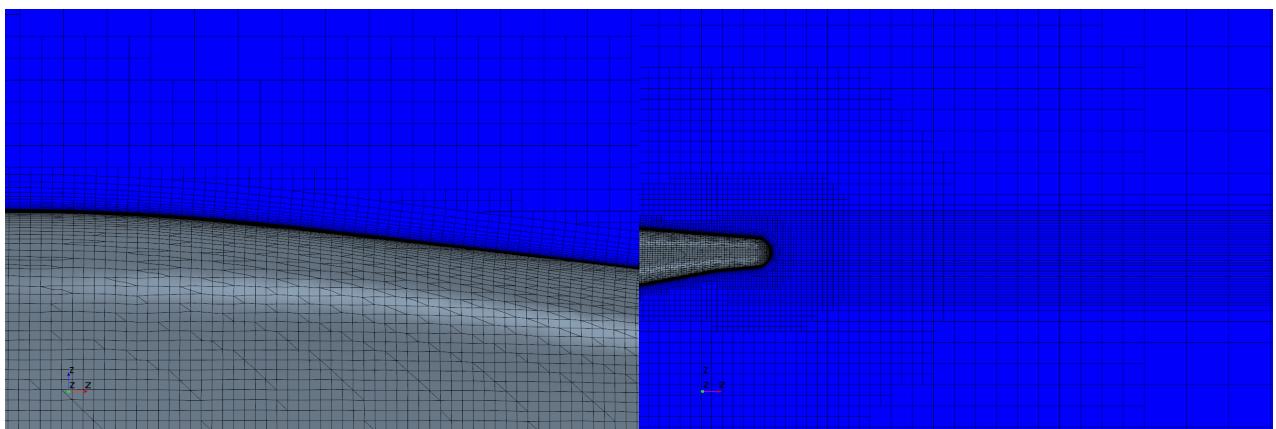
We can observe the trimmed mesh with prisms layers.



Volume mesh

We can notice :

- as defined in the reference values, the « Maximum Core/Prism/Transition Ratio » (the ratio between the size of the trimmed cells and the thickness of the last prism layer) is limited to 2 ;
- as defined in the volumetric control, the mesh in the wake was refined only in the Y and Z directions, and we obtained an isotropic trimmed mesh.



Transition between the core mesh (trimmed mesh) and the prism layers limited to 2 (left) and anisotropic trimmed mesh (refined only in Y and Z directions) in the wake.

We check the mesh quality :

Go to « Mesh » -> Diagnostics

Verify « Region » is enabled and click « OK »

The report should be :

--- Computing statistics in Region: Region

-> ENTITY COUNT:

Cells: 673262

Faces: 1967104

Verts: 666258

-> EXTENTS:

x: [-2.6000e+01, 2.6000e+01] m

y: [-5.5511e-17, 2.6000e+01] m

z: [-2.6000e+01, 2.6000e+01] m

Setting pro-STAR cell IDs on Region starting at 1

-> MESH VALIDITY:

Mesh is topologically valid and has no negative volume cells.

-> FACE VALIDITY STATISTICS:

Minimum Face Validity: 1.000000e+00

Maximum Face Validity: 1.000000e+00

Face Validity < 0.50	0	0.000%
----------------------	---	--------

0.50 <= Face Validity < 0.60	0	0.000%
------------------------------	---	--------

0.60 <= Face Validity < 0.70	0	0.000%
------------------------------	---	--------

0.70 <= Face Validity < 0.80	0	0.000%
------------------------------	---	--------

0.80 <= Face Validity < 0.90	0	0.000%
------------------------------	---	--------

0.90 <= Face Validity < 0.95	0	0.000%
------------------------------	---	--------

0.95 <= Face Validity < 1.00	0	0.000%
------------------------------	---	--------

1.00 <= Face Validity	673262	100.000%
-----------------------	--------	----------

-> VOLUME CHANGE STATISTICS:

Minimum Volume Change: 1.111481e-02

Maximum Volume Change: 1.000000e+00

Volume Change < 0e+00	0	0.000%
-----------------------	---	--------

0e+00 <= Volume Change < 1e-06	0	0.000%
--------------------------------	---	--------

1e-06 <= Volume Change < 1e-05	0	0.000%
--------------------------------	---	--------

1e-05 <= Volume Change < 1e-04	0	0.000%
--------------------------------	---	--------

1e-04 <= Volume Change < 1e-03	0	0.000%
--------------------------------	---	--------

1e-03 <= Volume Change < 1e-02	0	0.000%
--------------------------------	---	--------

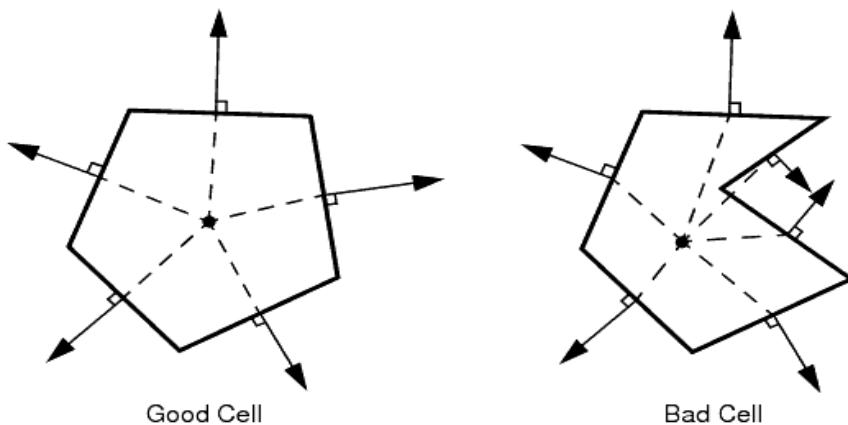
1e-02 <= Volume Change < 1e-01	3161	0.470%
--------------------------------	------	--------

1e-01 <= Volume Change <= 1e+00	670101	99.530%
---------------------------------	--------	---------

Mesh Diagnostics Report

First we can notice we have no negative volume cell. Only one negative volume cell will make the simulation unstable and it will diverge very fast.

Then we can check the face validity. The face validity is an area-weighted measure of the correctness of the face normals relative to their attached cell centroid.



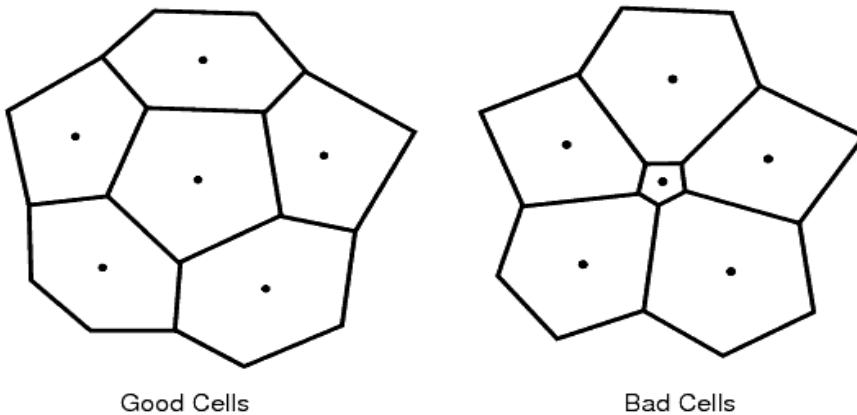
In a good quality cell, the face normals point outwards, away from the cell centroid. In a cell with bad face validity, one or more of the face normals point inwards, towards the cell centroid.

A face validity of 1.0 means all face normals are correctly pointing away from the cell centroid. Values below 1.0 mean that some of the cell faces have normals pointing inward towards the cell centroid, indicating some form of concavity. Values of below 0.5 signify a negative volume cell. Cells with a face validity below 1.0 are considered bad.

The simulation will have problems to converge if only one cell has a face validity below 0.9.

All cells for our mesh have a value of 1.0.

The last mesh quality criterion is the volume change. The volume change metric describes the ratio of the volume of a cell to that of its largest neighbor.



A value of 1.0 indicates that the cell has a volume equal to or higher than its neighbors. A large jump in volume from one cell to another can cause potential inaccuracies and instability in the solvers.

Cells with a volume change of 0.01 or lower are considered bad cells. They can be generated between the last prism layer and the core mesh, that is very difficult to prevent.

The simulation will have problems to converge if only one cell has a volume change below 1.0e-4.

There is no cell in our mesh with a value below 1.0e-2.

According to this Mesh Diagnostics Report, the mesh is correct.

However it does not mean the mesh is fine enough to model correctly all physical phenomena we want to simulate. A mesh convergence study is often necessary.

For the moment, we observed only the volume mesh on surfaces (helicopter and computation domain surfaces). We create now a plane to see a split of the volume mesh :

Go on « Derived Parts » and right click -> New Part -> Section -> Plane

Verify « Region » is selected in « Input Parts » (to specify the plane splits the region named « Region »)

Enter [0.0, 0.0, 0.0] m in « Origin » (origin of the plane)

Enter [1.0, 0.0, 0.0] m in « Normal » (normal to the plane)

Verify « New Geometry Displayer » is selected (to create a displayer which displays the plane in « Geometry Scene »)

Click on « Create » then « Close »

Rename that plane (right click on the Plane section and choose « Rename ») « Plane Section X=0.0m »

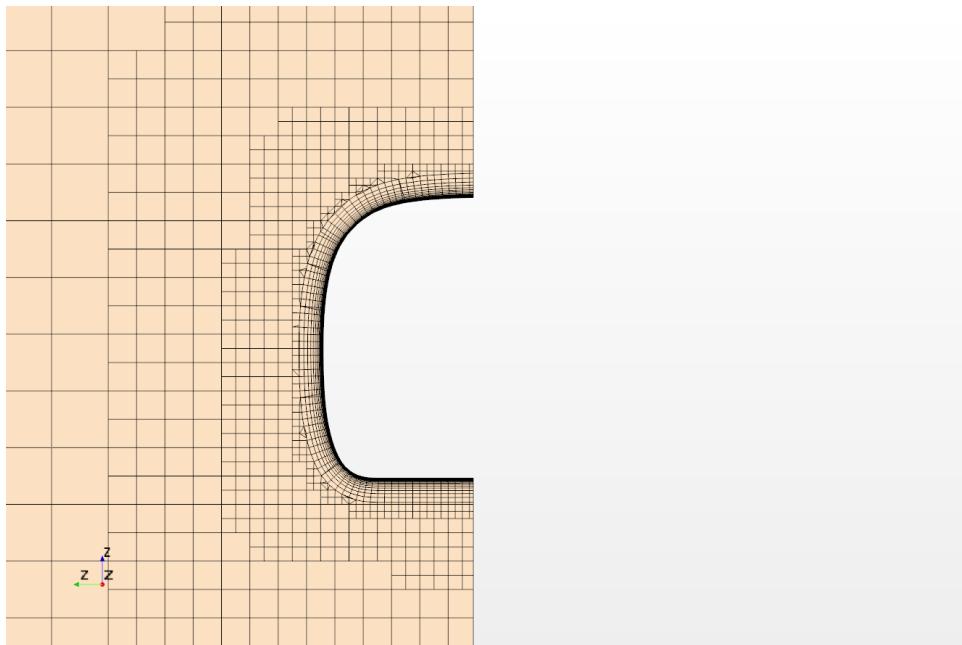
To display the plane X = 0.0m :

Go to « Scenes » -> Geometry Scene 1 -> Displayers

We can see a displayer « Section Geometry 1 » was created for the plane X = 0.0m

Double click on « Geometry 1 » to hide it

It clearly shows the size of volume mesh increases when we go away from the helicopter. We can also observe the transition between the core mesh and the prism layers.



Volume mesh splitted by the plane $X = 0.0\text{ m}$

Save the simulation !

3. Physics Setup

3.1 Defining the Physics Continuum

In « Continua », a physics continuum named « Physics 1 » was created after the volume mesh generation. We can select now all physical models :

Go to « Continua » -> Physics 1 -> double click on « Models »

A « Physics 1 Model Selection » window is opening. Select :

Space = Three Dimensional (already selected)

Time = Steady

Material = Gas

Flow = Coupled Flow (recommended for compressible flow)

Equation of State = Ideal Gas (never use constant density for compressible flow)

Viscous = Turbulent

Reynolds Averaged Turbulence = K-Omega Turbulence (Spalart-Allmaras and K-Omega STT are recommended for external Aerodynamics)

Click on « Close »

In the case of compressible flow, it is recommended to use the « Coupled Flow » solver, whereas the « Segregated Flow » solver is recommended for incompressible flow.

For compressible flow, we must use at least « ideal Gas » for gas density. The « Constant Density » option cannot take into account the gas compressibility.

Note the Energy equation is solved when « Ideal Gas » is selected.

Go to « Continua » -> Physics 1 -> Models -> Gas -> Air -> Material Properties

We can notice we have the possibility here to change the gas properties and the turbulent Prandtl number. They are constant by default.

We do not change the default values.

Go to « Continua » -> Physics 1 -> Models -> Coupled Flow

We can notice the default spatial discretisation scheme is « 2nd-order ».

We do not change the default parameters.

3.2 Defining the Physics Reference Values

Go to « Continua » -> Physics 1 -> Reference Values

The interesting parameter in our case is the « Reference Pressure ». The default value is the standard atmospheric pressure (101 325 Pa).

It is important to note that « Pressure » (i.e. the static pressure) and « Total Pressure » are defined and calculated in star CCM+ with gauge values relative to that « Reference Pressure ». We obtain the following equations respectively for the static and total pressures :

$$\text{Absolute Pressure} = \text{Reference Pressure} + \text{Pressure}$$

$$\text{Absolute Total Pressure} = \text{Reference Pressure} + \text{Total Pressure}$$

For incompressible flow or compressible flow with low pressure fluctuations, it is recommended to choose a value of « Reference Pressure » to have fluctuations of pressure around a mean value of 0.

In our case, we will define :

- a « Reference Pressure » of 101 325 Pa
- initial and boundary pressures (explained further) of 0 Pa

The computed values for the pressure at the end of the simulation will vary slightly around 0.

We do not need to change the default value of 101 325 Pa for « Reference Pressure ».

If the simulation were calculated at high altitude conditions with an atmospheric pressure of 25 000 Pa, we would define :

- a « Reference Pressure » of 25 000 Pa
- initial and boundary pressures (explained further) of 0 Pa

The computed values for the pressure at the end of the simulation would also vary slightly around 0.

For high compressible flow with high pressure fluctuations (ex : presence of shock waves), it is recommended to set the value of « Reference Pressure » to 0, and the pressures and the absolute pressures are equal. If it were our case, we would define :

- a « Reference Pressure » of 0 Pa
- initial and boundary pressures (explained further) of 101 325 Pa

The computed values for the pressure at the end of the simulation would vary strongly around 101 325 Pa.

3.3 Defining the Physics Initial Conditions

We need to define the initial flow fields (of velocity, pressure, turbulence ...) in the whole computation domain.

For external aerodynamics, the main part of the computation domain has fields very close to the far-field conditions. The best solution to converge faster is to define identical boundary and initial conditions.

Go to « Continua » -> Physics 1 -> Initial Conditions

Pressure = 0 Pa (as explained previously, it is a gauge pressure relative to the reference pressure)

Static Temperature = 293.0 K

Turbulence Intensity = 0.01 (= 1%)

Turbulence Specification = Intensity + Viscosity Ratio (how we want to specify the turbulence)

Turbulent Viscosity Scale = 119.0 m/s (it is advised to define a velocity representative of flow field inside the computation domain)

Turbulent Viscosity Ratio = 10

Velocity = [119.0, 0.0, 0.0] m/s

3.4 Defining the Physics Boundary Conditions

We need to define the boundary conditions, which are the physical conditions (velocity, pressure, temperature, turbulence) at the boundaries of the computation domain.

There are often confusion between initial and boundary conditions :

- initial conditions are the flow fields we define for the whole computation domain only for the start of the simulation.
- boundary conditions are the flow fields we define for the boundaries of the computation domain throughout the simulation.

We define the conditions for « Domain.FF »

Go to « Regions » -> Region -> Boundaries -> Domain.FF -> Physics Conditions -> Flow Direction

Specification

We notice the default flow direction for a « Free Stream » boundary type is « Components », which is the better choice than « Normal to Boundary », because a « Free Stream » condition can be an inlet and an outlet.

Do not change the method (Components) for our case !

Go to « Regions » -> Region -> Boundaries -> Domain.FF -> Physics Conditions -> Turbulence Specification

We notice the default method is « Intensity + Viscosity Ratio ». We have also other methods (« K+omega » and « Intensity + Length Scale »).

Do not change the method (Intensity + Viscosity Ratio) for our case !

Go to « Regions » -> Region -> Boundaries -> Domain.FF -> Physics Values -> Flow Direction

The default components are [1.0, 0.0, 0.0].

Do not change these values !

Go to « Regions » -> Region -> Boundaries -> Domain.FF -> Physics Values -> Mach Number

The default value is 0.7.

Change this value to 0.35 !

This value will be used for air entering in the computation domain.

Go to « Regions » -> Region -> Boundaries -> Domain.FF -> Physics Values -> Pressure

The default value is 0.0. (because it is a gauge pressure relative to the reference pressure of 101 325.0 Pa)

Do not change this value !

This value will be used for air outgoing from the computation domain.

Go to « Regions » -> Region -> Boundaries -> Domain.FF -> Physics Values -> Static Temperature

The default value is 300.0 K.

Change this value to 293.0 K!

Go to « Regions » -> Region -> Boundaries -> Domain.FF -> Physics Values -> Turbulence Intensity

The default value is 0.01 (=1%). We define the same value for the initial conditions.

Do not change this value !

Go to « Regions » -> Region -> Boundaries -> Domain.FF -> Physics Values -> Turbulence Viscosity Ratio

The default value is 10. We define the same value for the initial conditions.

Do not change this value !

We do not need to change the default parameters for 3D_heli.heli (no-slip wall) and Domain.Sym (symmetry plane)

3.5 Defining the Solver

With the « Coupled Solver », we need to define the Courant Number (also called « CFL Number ») :

Go to « Solvers » -> Coupled Implicit

we can see the default value for the Courant Number with the steady coupled solver is 5.

In many cases, this value is too high for the first iterations, and we need to decrease this value to 1 before starting the simulation. After 100 or 200 iterations, we can increase the value to 5 to converge faster.

If the solver need to limit the change in some variables in the computation domain, or to divide the Courant Number by 10 (messages appear for these two cases), you need to decrease the Courant Number.

If you have a Courant Number of 1, and you still have these messages, check your mesh quality and solver settings !

In our case, we have a good mesh quality and the flow topology is quite simple (low compressible flow). We can start directly with the default value of 5.

Save the simulation !

4. Monitor et Scalar Scene Setup

The residual convergence is necessary for simulation convergence, but not sufficient !!

To ensure the simulation converges, we need to define additional monitors. A monitor allows following the evolution of parameters during the simulation.

Some examples of monitors are :

- difference of mass flow rate between inlet and outlet
- drag or/and lift coefficients for external flow
- overall total pressure drop for internal flow
- local velocity, pressure or temperature at a given point
- mean value of velocity, pressure or temperature through a surface
- ...

We will define 3 monitors :

- Helicopter drag coefficient
- Helicopter lift coefficient
- Pressure at a point in the wake

To create a monitor, we need to create a report first :

Go to « Reports » and right click -> New Report -> [Element Count ... Volume Uniformity] -> Force Coefficient

A report named « Force Coefficient 1 » is created.

Right click on « Force Coefficient 1 », select « Rename » and rename the report « Cd »

In the « Cd - Properties » window, we define :

Direction = [1.0, 0.0, 0.0] (the direction for the drag)

Force Option = Pressure + Shear (the force will be calculated from the pressure and viscosity effects)

Parts = in Regions, select « 3D_heli.heli » (the surface on which we calculate the drag)

Reference Area = 0.1068 m² (the projection of the half helicopter surface on a frontal plane)

Reference Density = 1.2 kg/m³ (air density at 239.0 K and 101 325 Pa)

Reference Pressure = 0.0 Pa (no link with reference pressure defined in reference conditions, no effect if we calculate forces on closed surfaces)

Reference Velocity = 119.0 m/s (Far-field velocity)

The report for the drag coefficient is now correctly defined. To create the monitor and the plot (to plot the monitor evolution) :

Go to « Reports », right click on « Cd » and choose « Create Monitor and Plot from Report »

The monitor and the plot are created.

In the same way, create a similar report, monitor and plot « Cz » with a direction [0.0, 0.0, 1.0].

Now we create the monitor for the pressure evolution in the wake. First we need to create the point :

Go on « Derived Parts » and right click -> New Part -> Probe -> Point ...

A point is created. To define this point, select in the « Point - Properties » Window :

Parts = Regions -> Region (to specify the point is in the Region)

Point = [1.7, 0.0, 0.5] m, m, m (point coordinates)

We create the report :

Go on « Reports » and right click -> [Element Count ... Volume Uniformity] -> Maximum

A report named « Maximum 1 » is created. The maximum report calculates the maximum value for different parts. If we select only one point, we extract the value on that point. We selected « Maximum », but we could also choose « Minimum ».

Go on « Maximum 1 » and right click, select « Rename » and rename the report « Pressure-wake »

In the « Pressure-wake - Properties » :

Parts = Derived Parts -> Point

Scalar Field Function = Pressure

To create the monitor and the plot :

Go to « Reports », right click on « Pressure-wake » and choose « Create Monitor and Plot from Report »

The monitor and the plot are created.

We have now the three monitors defined correctly.

Another way to check the convergence or to see where eventual instabilities occur is to create a « Scalar scene » which displays a field of velocity, pressure, vorticity ...

Create a « Scalar scene » :

Go on « Scenes » and right click -> New Scene -> Scalar

A new Scene named « Scalar Scene 1 » is created.

Rename that Scalar Scene :

Right click on « Scalar Scene 1 » -> Rename, and rename it « Scalar Scene Mach Number »

In that scene, in « Displayers », double click on « Outline 1 » to hide it

Go to « Scalar 1 » -> Parts and Select « Domain.sym » in « Regions »

The plane is displayed without function field.

In « Scalar Field » -> Function, select « Mach Number ».

The field of Mach Number is not displayed because there is no solution yet.

The simulation is now ready to run !!

Save the simulation !

5. Running the simulation and analysis of first results

Before starting the simulation, we need to define a « Stopping Criterion ». The simulation will stop when this criterion is reached.

Go to « Stopping Criteria » -> Maximum steps

A step is :

- an iteration for a steady simulation (our case here)
- a time step for an unsteady simulation

Do not change the number of steps :

Maximum steps = 1 000

To start the simulation :

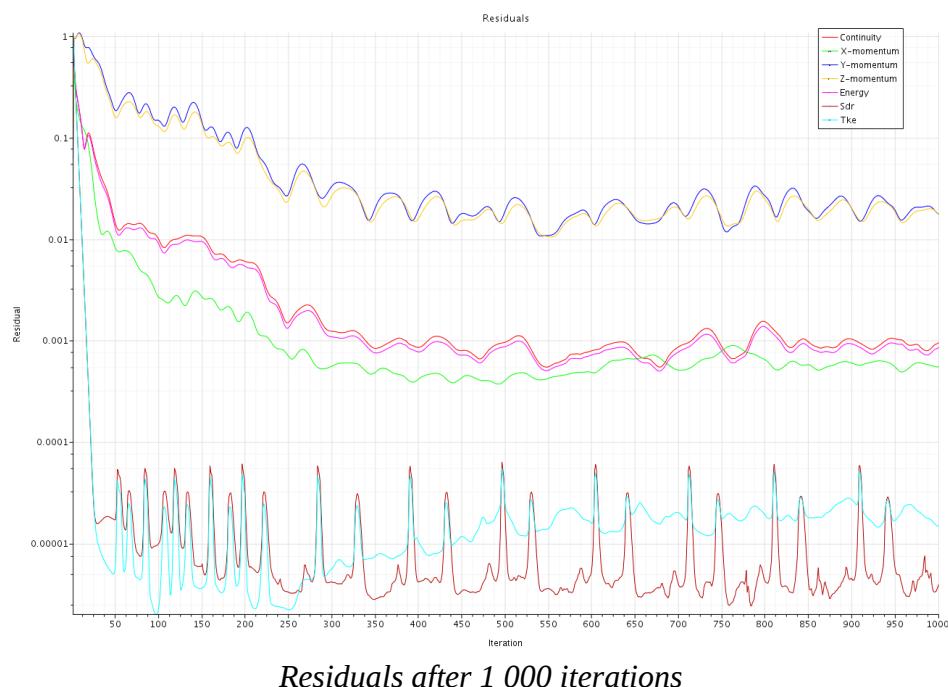
Go to « Solution » and Click « Run »

A window for the « Residuals » Plot is opening

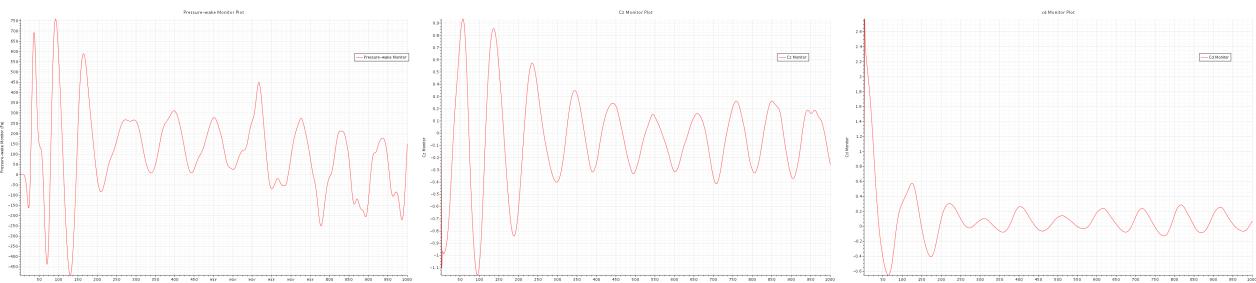
Are updated at each iteration :

- the « Residuals » monitor plot
- the « Cd » and « Cz » monitor plots
- the « Pressure-wake » monitor plot
- the « Scalar Scene Mach Number »

Thousands of iterations are generally needed to converge. But after 500 iterations, we can guess the simulation should not converge, because the « Residual monitor plot » shows that some residuals stabilize around high values (residual values below 10^{-3} were expected for the convergence).



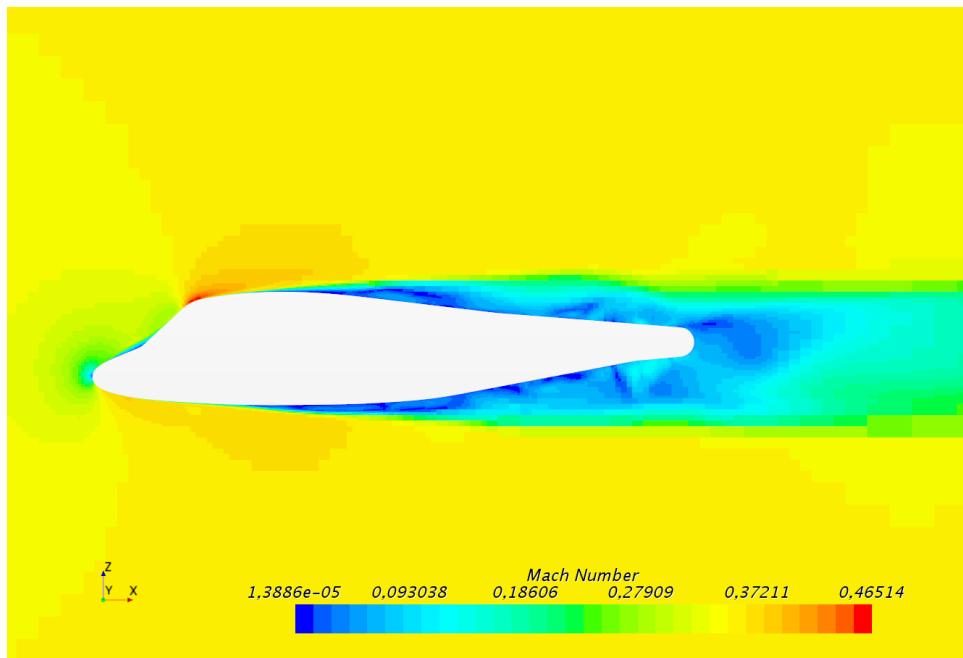
For all other monitor plots (Cd, Cz and Pressure-wake), we observe the amplitudes increase.



Cd, Cz and Pressure-wake Monitor Plots after 1 000 iterations

We visualize the flow to understand why we obtain this instability.

In the « Scalar Scene Mach Number », it appears clearly there are flow detachments which generate vortices and the flow instability. It is very surprising, because flow detachments were not expected in these parts of the geometry and they seem not physical.



Mach Number in the symmetry plane after 1 000 iterations

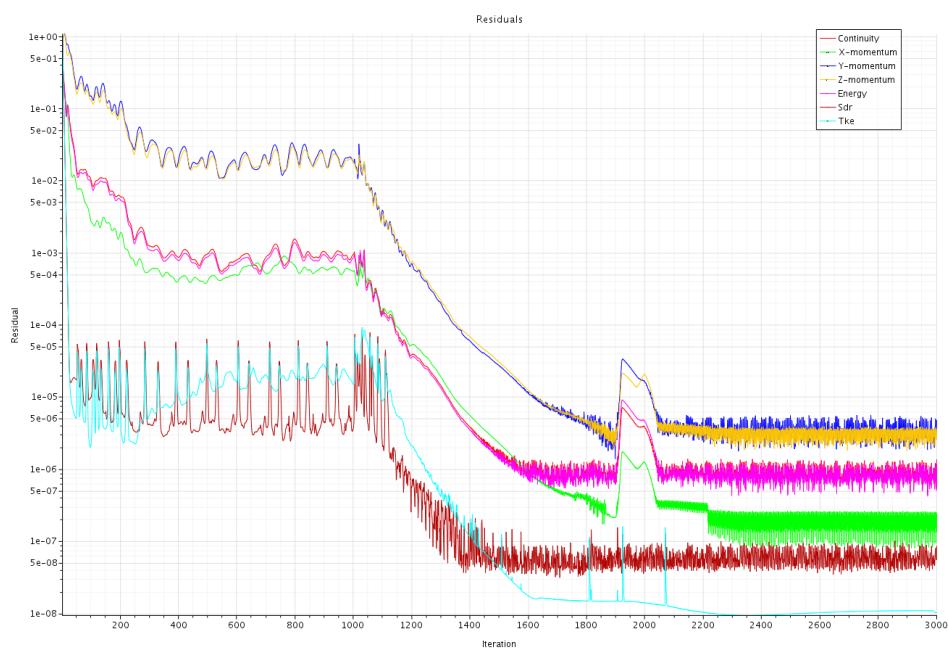
To converge easier, we increase the Courant Number :

Go to « Solvers » -> Coupled Implicit -> Courant Number = 20

We can run 2 000 more iterations :

Go to « Stopping Criteria » -> Maximum steps, Maximum steps = 3 000

Go to « Solution » and Click « Run »



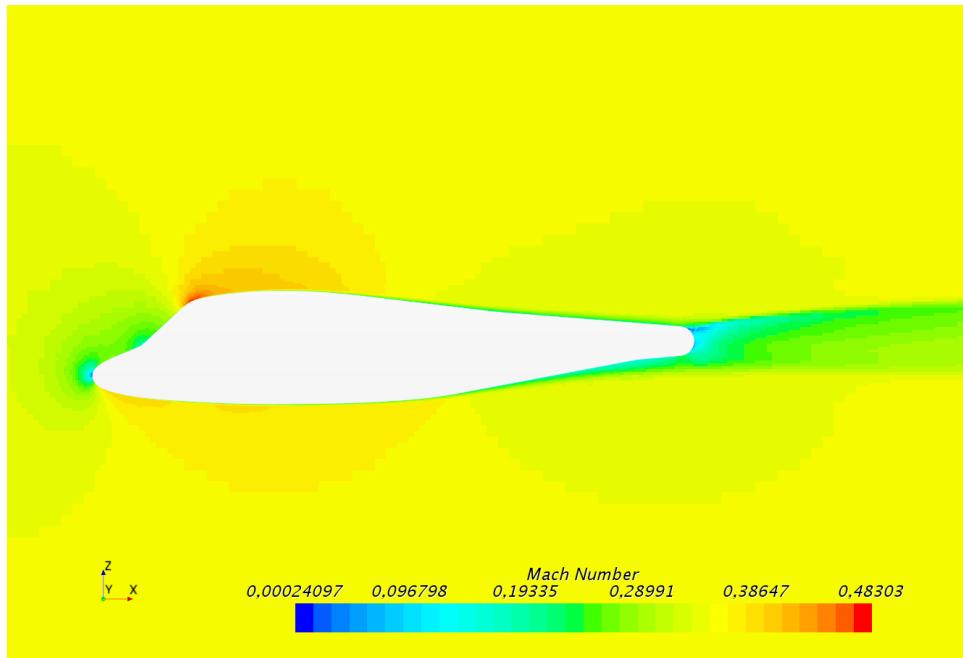
Residuals after 3 000 iterations

After 2 000 more iterations, the residuals strongly decreased to low values and, despite of the small oscillations, we can consider they converged. The other monitor plots (C_d , C_z and Pressure-wake) converge now to constant values.



C_d , C_z and Pressure-wake Monitor Plots after 3 000 iterations

The flow detachments, visible after the first 1 000 iterations, move clearly to the back of the helicopter and are more physical now. The increase of the Courant number helped to converge to a better solution.



Mach Number in the symmetry plane after 3 000 iterations

As a summary, we need in many cases to start with a Courant number of 1 for the computation stability. After 100 or 200 iterations, if the residuals tend to decrease, we can increase the Courant number to its default value of 5 to converge faster.

If the simulation does not converge because of flow instabilities, we have to analyze this flow and to wonder if instabilities are physical. If the origin is not a bad quality mesh or a wrong solver setup, we have 2 cases :

- if the flow instability seems physical, the flow is naturally unsteady and we need to change the steady solver to implicit unsteady solver ;
- if the flow instability seems not physical (ex : flow detachment where they should not occur), we can increase the Courant number to 10 or 20.

For this last case, many simulations can have problems to converge with such a Courant number. A solution is to change the steady solver to implicit unsteady solver, even if we expect a steady flow. That should help to converge to the correct steady solution.

6. Wall Y+

For the mesh generation, we calculated the thickness of the first prism layer to aim values around 1 for wall Y+ on the helicopter wall. We check if we obtain finally this value.

We create a scalar scene of wall Y+ :

Go on « Scenes » and right click -> New Scene -> Scalar

Rename that new Scalar Scene :

Right Click on « Scalar Scene 1 » -> Rename, and rename it « Scalar Scene Y+ »

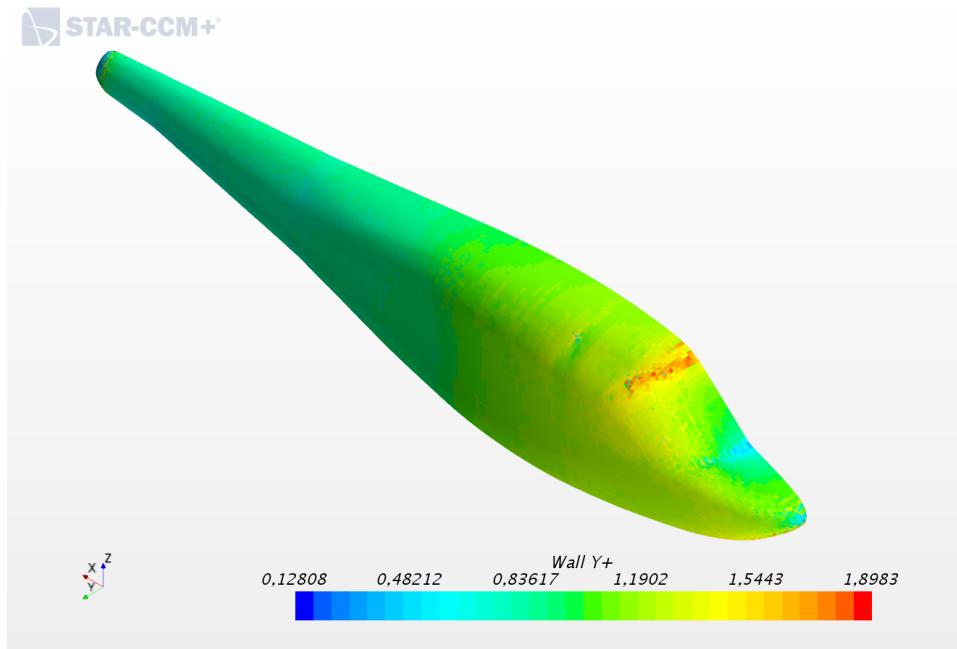
In that scene, in « Displayers », double click on « Outline 1 » to hide it

Go to « Scalar 1 » -> Parts

In Regions -> Region -> Boundaries and Select « 3D_heli.heli »

The helicopter wall is displayed without function field.

In « Scalar Field » -> Function, select « Wall Y+ ».



Wall Y+ for the helicopter wall

The mean value of wall Y+ on the helicopter wall is around 1. However we obtained greater values where we have flow acceleration. That is not a problem, as they are below a value of 5 (max = 1.9 in our case).

The thickness for the first prism layer was correctly calculated.

Note that for CFD simulations, it is always interesting (and very often mandatory) to investigate the mesh convergence.

That means we refine the mesh until this mesh refinement does not provide different results.

7. Post-processing

In this last part, some methods for the flow analysis are presented.

A) Contours

During the simulation, the setup for the scalar scene was detailed. We displayed contours of Mach or Courant numbers. Many others parameters like pressure, temperature, wall shear stress can also be displayed on planes or on the boundary conditions.

We can observe, for example, the pressure distribution on the helicopter wall :

Go on « Scenes » and right click -> New Scene -> Scalar

Rename that new Scalar Scene :

Right Click on « Scalar Scene 1 » -> Rename, and rename it « Scalar Scene Pressure »

In that scene, in « Displayers », double click on « Outline 1 » to hide it

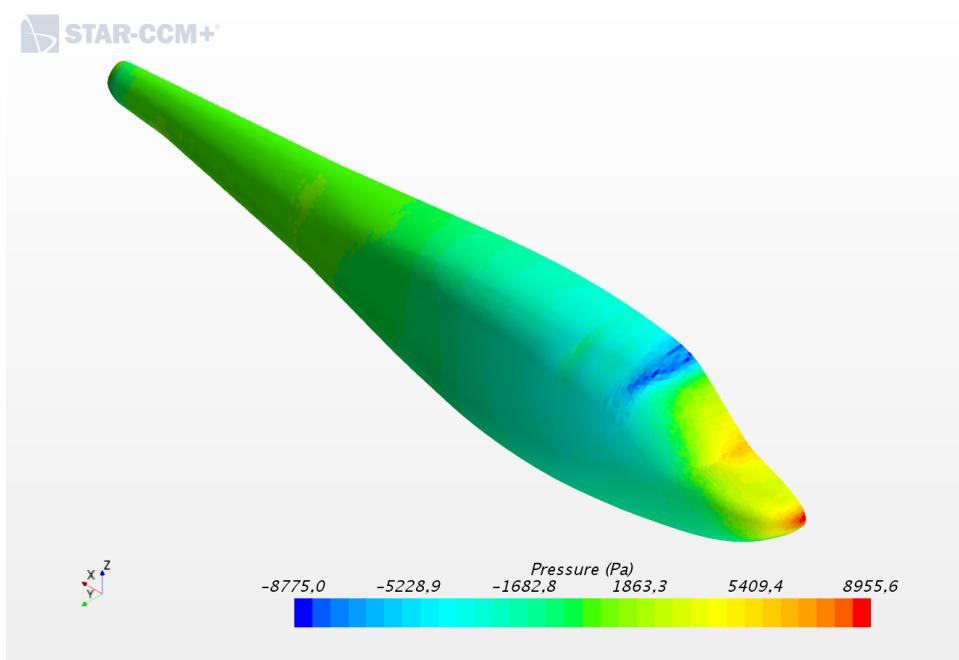
Go to « Scalar 1 » -> Parts

In Regions -> Region -> Boundaries and Select « 3D_heli.heli »

The helicopter wall is displayed without function field.

In « Scalar Field » -> Function, select « Pressure ».

We obtain the following pressure field on the helicopter.



Pressure on the helicopter wall

As expected, we obtained highest values of pressure at the stagnation point and low values where the airflow accelerates.

B) Plots

For a more detailed analysis of the pressure distribution, we can plot a pressure coefficient distribution. We create first the helicopter profile, which is the intersection of the helicopter wall and a plane $Y = 0$:

Go on « Derived Parts » and right click -> New Part -> Section -> Plane

In « Input Parts », disable Region and select, in Regions -> Region -> Boundaries, 3D_heli.heli (to specify the plane splits only the helicopter wall instead of the whole « Region »)

Enter [0.0, 1.0E-6, 0.0] m in « Origin » (origin of the plane)

We define Y = 1.0E-6 m because some points would be missing with Y = 0.0m

Enter [0.0, 1.0, 0.0] m in « Normal » (normal to the plane)

Verify « No Displayer » is chosen (we do not need to display that profile in a scene)

Click on « Create » then « Close »

Rename that plane (right click on the Plane section and choose « Rename ») « Heli profile »

We need to create the plot of the pressure coefficient on that helicopter profile :

Go on « Plots » and right click -> New Plot -> XY Plot

Rename (right click and rename) that new plot « Cp Heli Profile »

On « Cp Heli Profile », in the Properties window, Parts = Derived Parts -> Heli profile

In « Y Types » -> Y Type 1 -> Scalar Function -> Field Function = Pressure Coefficient

The pressure coefficient on the helicopter profile is now plotted. However, the values are very high, since a maximum value close to 1.0 was expected at the stagnation point. The reason is we need to define the « Reference Values » for the pressure coefficient :

Go to « Tools » -> Field Functions -> Pressure Coefficient

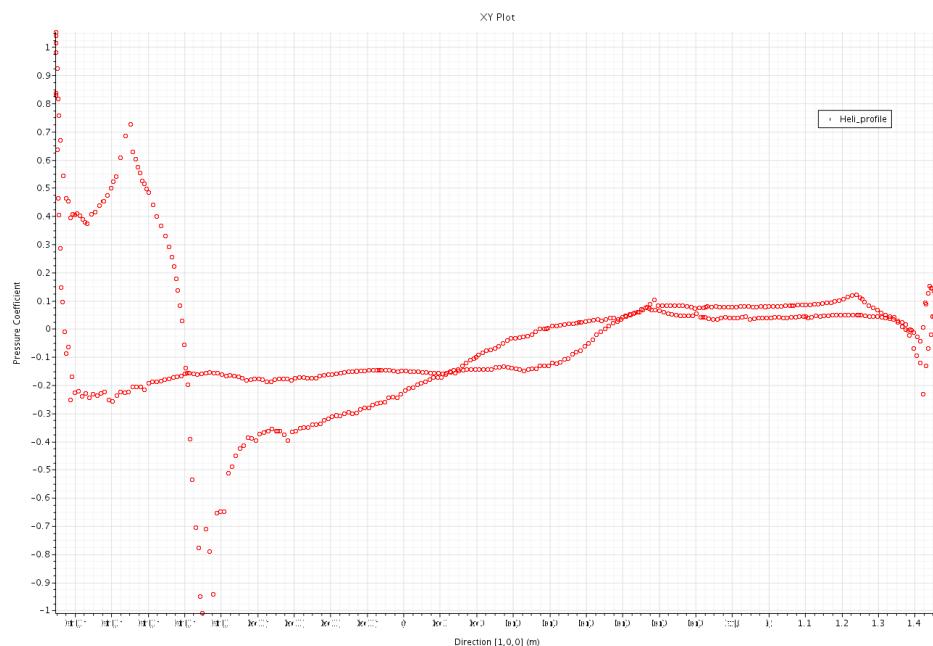
In « Pressure Coefficient - Properties » window, change the following parameters :

Reference Density = 1.2 kg/m³

Reference Pressure = 0.0 Pa (no link with reference pressure defined in reference conditions, no effect if we calculate forces on closed surfaces)

Reference Velocity = 119.0 m/s (Far-field velocity)

With the correct reference values, the maximum value (at stagnation point) is now close to 1.0.



Pressure coefficient on the helicopter profile

C) Isosurface

An isosurface is a surface on which a parameter is constant. The generation of isosurface of Q-Criterion allows the visualization of vortices.

Display the « Geometry Scene 1 » and display only the helicopter wall

Go on « Derived Parts » and right click -> New Part -> Isosurface

Verify « Region » is selected in « Input Parts » (to specify the isosurface is created in the region named « Region »)

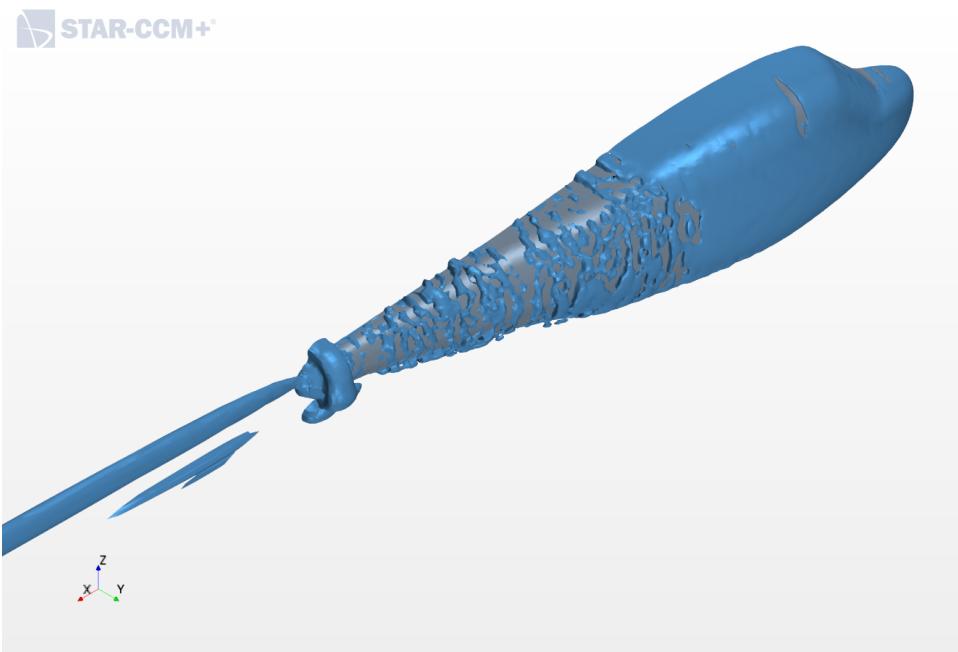
Scalar = Q-Criterion

Isovalue = $2\ 500\ s^{-2}$

Verify « New Geometry Displayer » is enabled (to create a displayer which displays the isosurface in « Geometry Scene 1 »)

Click « Create » then « Close »

The complex structures of the vortices downstream the helicopter can be visualized with the isosurface of Q-criterion = $2\ 500\ s^{-2}$ shown below.



Isosurface of Q-criterion = $2\ 500\ s^{-2}$

D) Line Integral Convolution

To visualize the flow separation, we can display wall shear stress vectors on the helicopter wall with the technique of line integral convolution :

Go on « Scenes » and right click -> New scene -> Vector

In « Displayers », double click on « Outline 1 » to hide it

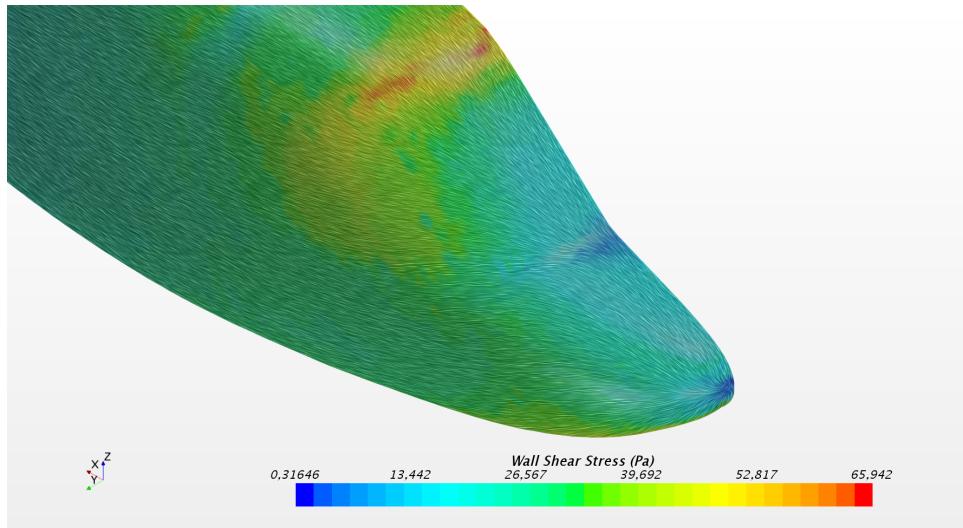
In « Vector 1 », change « Display Mode » from « Glyph » to « Line Integral Convolution »

Still in « Vector 1 », in « Parts » -> Regions -> Region -> Boundaries, select « 3D_heli.heli »

Still in « Vector 1 », in « Vector Field » -> change « Function » from « Velocity » to « Wall Shear Stress »

The line integral convolution is a technique to visualize a vector field with lines. We know that flow separation and reattachment occur where wall shear stress is null. On the following picture, we can locate these regions where :

- wall shear stress is null
- lines are joining and their direction change suddenly



Lines from wall shear stress vectors

E) Streamlines

Streamlines are generated from a seed, which can be a boundary of the computation domain or a derived part.

For internal aerodynamics, we can use the boundary conditions (for example : the inlet). That often generates streamlines suitable to understand the flow.

For external aerodynamics, far-field conditions are too far and too great to provide suitable streamlines. It is better to generate them from a derived parts created close to the investigated geometry.

In our case, we create a « Presentation Grid » (a matrix of probes) upstream the helicopter :

Go to « Derived Parts » and right click -> New Part -> Probe -> Presentation Grid ...

In the Edit Tool, the only way to create the presentation grid is to define a center and a direction. It is better to create any presentation grid and to modify it later by using the origin and the point 1 and 2 :

Verify « Region » is selected in « Input Parts »

Choose « No Displayer » in the « Display » window (we do not need to see that presentation Grid

Click « Create » then « Close »

Go to « Derived Parts » -> Presentation Grid

Modify the following parameters :

Origin = [-1.0, 0.0, -0.2] m, m, m

Point 1 = [-1.0, 0.1, -0.2] m, m, m

Point 2 = [-1.0, 0.0, 0.0] m, m, m

The presentation grid is now in front of the helicopter.

If needed, display the « Geometry Scene 1 » and hide (by double click) all displayers, except the helicopter surface.

The presentation grid will be used as a seed for the streamlines :

Go to « Derived Parts » and right click -> New Part -> Streamline ...

Input parts = Region (streamlines are generated in the region)

Seed Parts = Derived Parts -> Presentation Grid (streamlines are generated from the presentation grid)

Vector Field = Velocity (Streamlines are generated from velocity vectors)

Part-U Resolution =5 (to have 5 streamlines in the first direction)

Part-V Resolution =10 (to have 10 streamlines in the second direction)

Display = New Streamline Displayer (to display the streamlines)

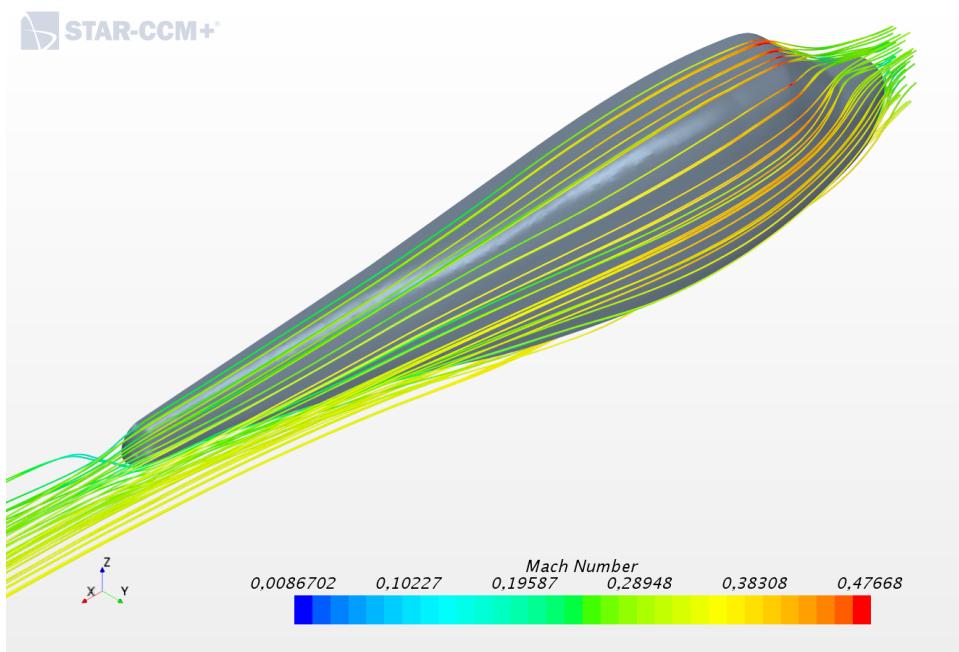
Click « Create » then « Close »

By default, the streamlines are lines (without thickness and difficult to see) and they are not coloured :

Go on « Scenes » -> Geometry Scene 1 -> Displayers -> Streamline Stream 1, modify in the properties window, Mode = Tubes (to give thickness to the streamlines)

In « Streamline Stream 1 », in Scalar Field, select « Mach Number » for Function (to colour the streamlines with Mach Number)

We have now streamlines generated from the presentation grid upstream the helicopter.



Streamlines generated from the presentation grid