

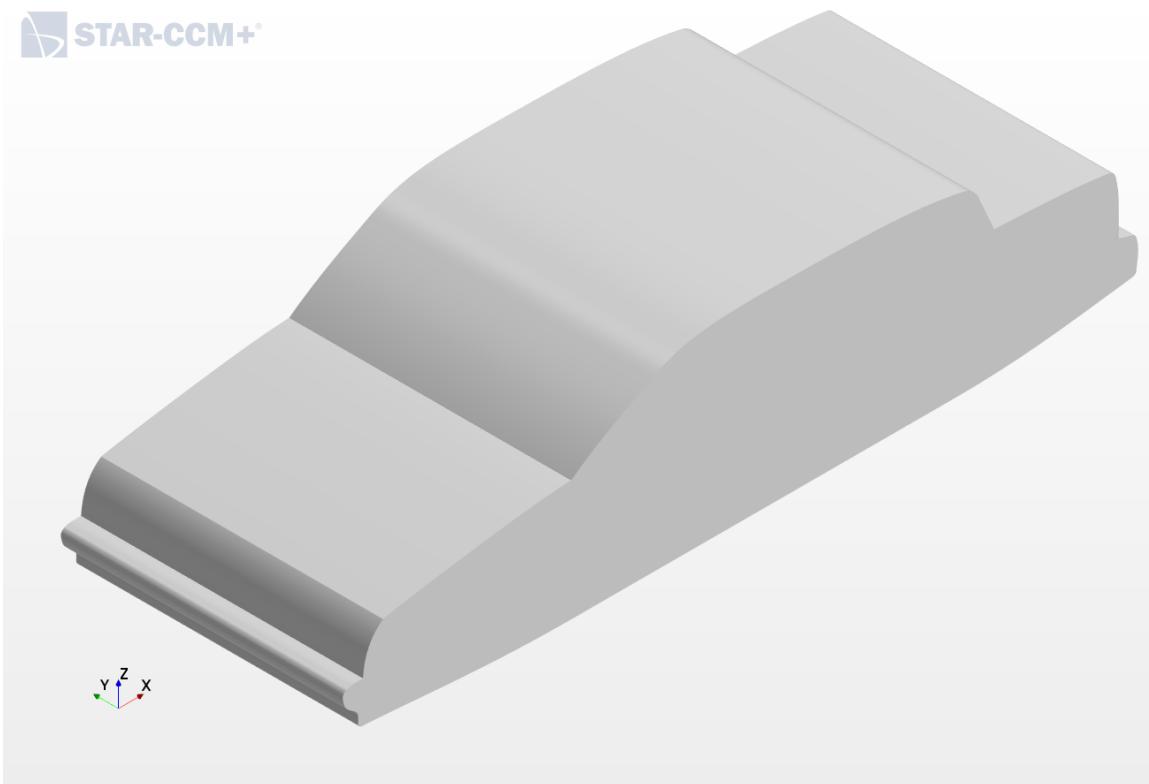
## Tutorial 1a : Simulation of Unsteady Incompressible Flow around a 3D Car

### **1. Introduction**

The main objectives of this tutorial are :

- to build a mesh around a simple shape geometry without using the surface wrapper ;
- to simulate the unsteady incompressible flow around this geometry.

The considered geometry is a simplified 3D car shown below.



*Car geometry*

After this tutorial, you will be able to :

- import an existing geometry
- create a computation domain
- generate a polyhedral mesh with prism layers (without using the surface wrapper)
- define the solver for unsteady incompressible flow
- refine the mesh in some parts of the computation domain
- 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\_car.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 car 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 car to obtain the computation domain. We choose « Create New Part » :

Import Mode = Create New Part

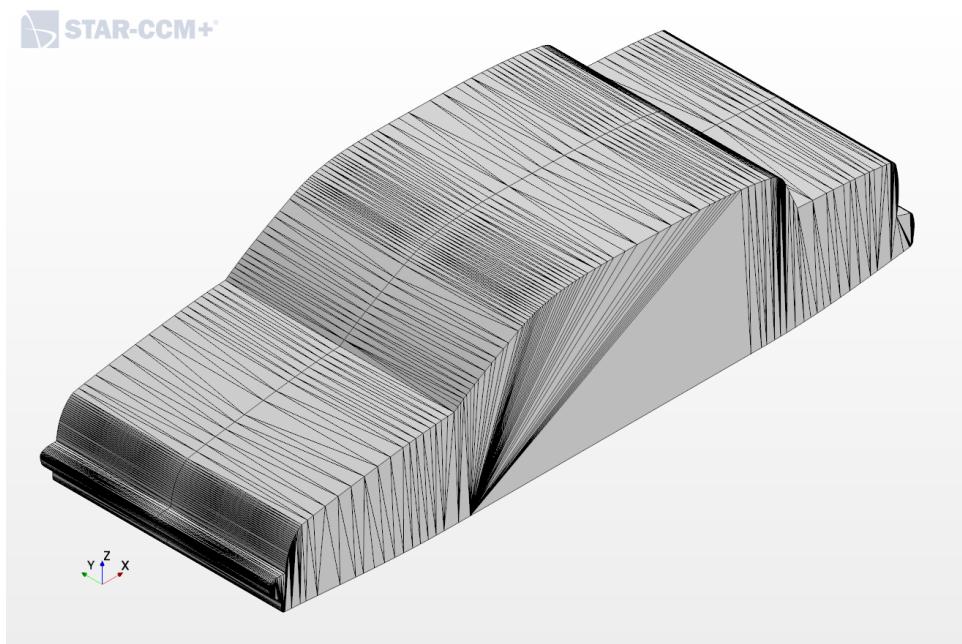
Units = mm

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

Click « OK »

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

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\_car » composed of only one surface named « catia ». You can rename this surface :

Right click on « catia » -> Rename, and rename this surface « car »

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. Here we do not need to split. The whole surface will have the same mesh and physical conditions.

Below « Surfaces », we find « Curves » where feature curves are defined. However, when we import surfaces from a stl file, the feature curves are not imported. It is important to extract them, without that Star CCM+ will generate curved surfaces where feature curves should be present. To extract those feature curves :

Right click on « Curves » -> Compute Part Curves -> Verify « car » is chosen and click « Apply ». Red feature curves are generated.

Click « Close »

Note we had no feature curve, because we imported the geometry from a stl file. We should not meet that problem with iges or step files.

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

We create a block whose boundary conditions are far (10 car length) from the investigated geometry, except for the ground :

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

Define the Maximum and Minimum Coordinates :

Corner 1 : X = -45.0m ; Y = -46.0m ; Z = 0.0m

Corner 2 : X = 50.0m ; Y = 46.0m ; Z = 46.0m

Create

**Click « Close »**

« Block » was created in Parts. If we go to Parts -> Block -> Surfaces, there is only one surface called « Block Surface ». However we need several surfaces to define the different boundary conditions (inlet, outlet, ground ...). To split the surface :

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

In the window « Split Part Surfaces by Angle » :

Verify « Block Surface » is chosen

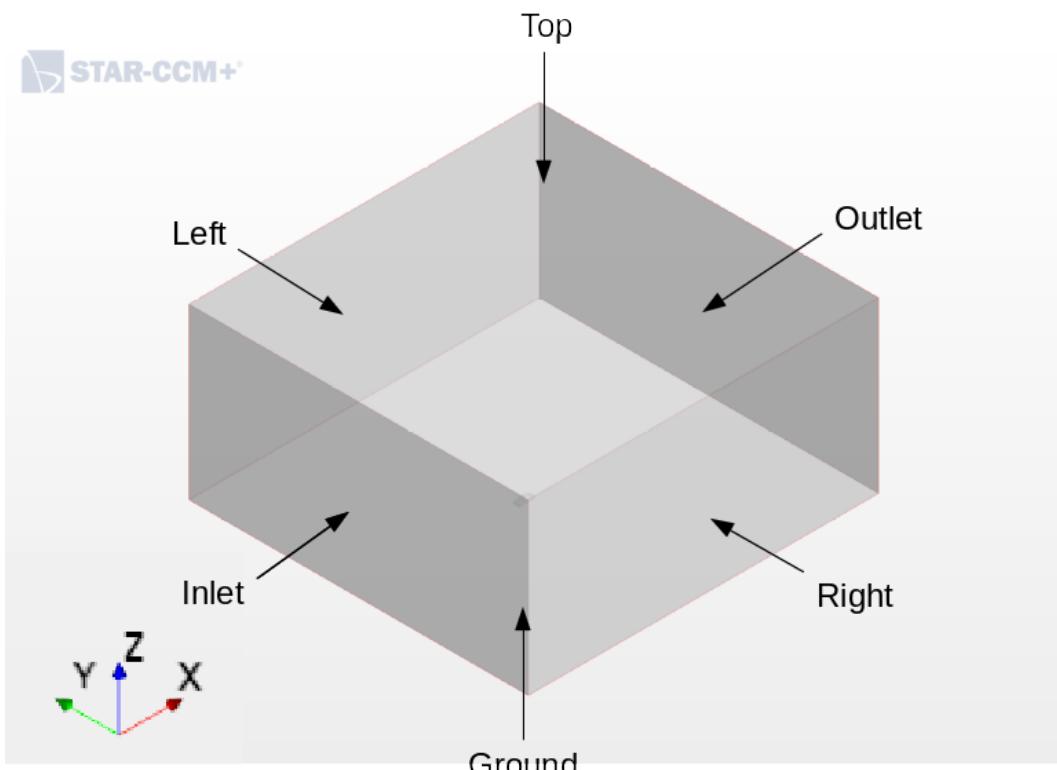
Define an angle of 89.0 degrees

Click « OK »

Since « Block surface » was splitted where the angle is above 89 degrees, we have now 6 surfaces named « Block Surface » followed by a number.

We rename these 6 surfaces as shown on the figure below. To rename a surface :

Select the surface and right click -> Rename, and rename the surface with the name given on the figure below.



*Names for the 6 block surfaces*

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

Select « Block » and « 3D\_car » and right click -> Boolean -> Subtract ...

In the « Subtract Parts » window :

Target Part = Block

Click « OK »

A part called « Subtract » was created. It is the result of the previous subtraction. If we have a look on the Surfaces inside « Subtract », we have all the surfaces defined before. We can rename this part :

Select « Subtract » and right click -> Rename, rename the part « Domain »

Note that we can subtract these two parts because they are closed volumes. If one of the volumes was not correctly closed, the subtraction would not be possible. In that case, refer to another tutorial dealing with the « Surface Wrapper » (Tutorial 1b).

We have now the geometry for the computation domain. We just need to assign this geometry to a region. A region is a volume in which we can define meshing and physics models.

Select « Domain » and right click -> Assign Parts to Regions ...

Verify that only « Domain » is chosen in « Parts »

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

Click « Apply »

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 an inlet, an outlet and walls.

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 : Domain, Inlet, Domain.Outlet, Domain.car ...

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 surface import create 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 be generated by the subtraction of the block and the car. We do not need to select this model.

In « Surface Mesh » :

Select Surface Remesher

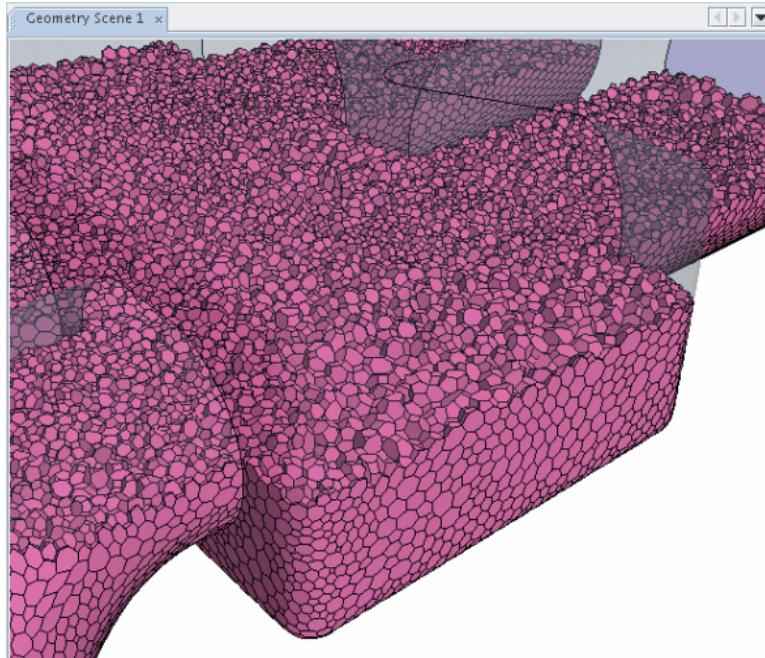
Do not select Surface Wrapper !

For the « Volume Mesh » :

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

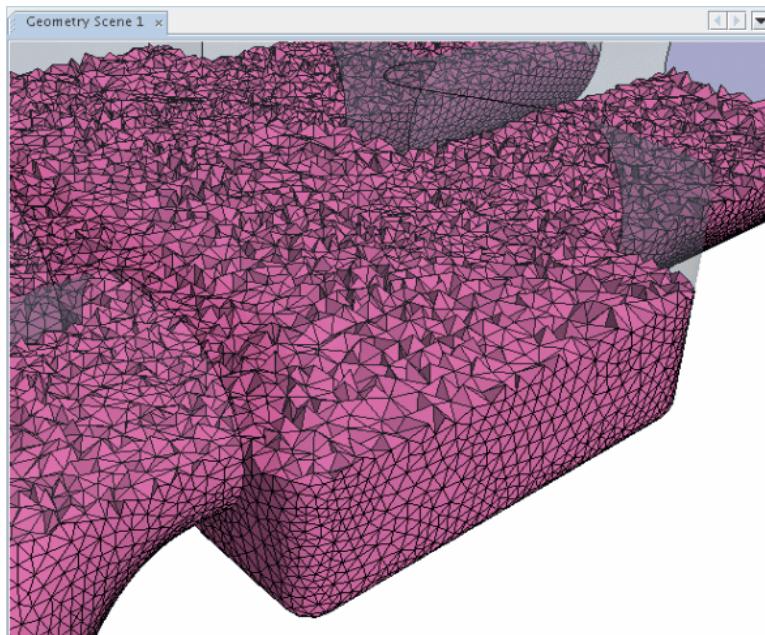
of following models.

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



*Example of polyhedral mesh*

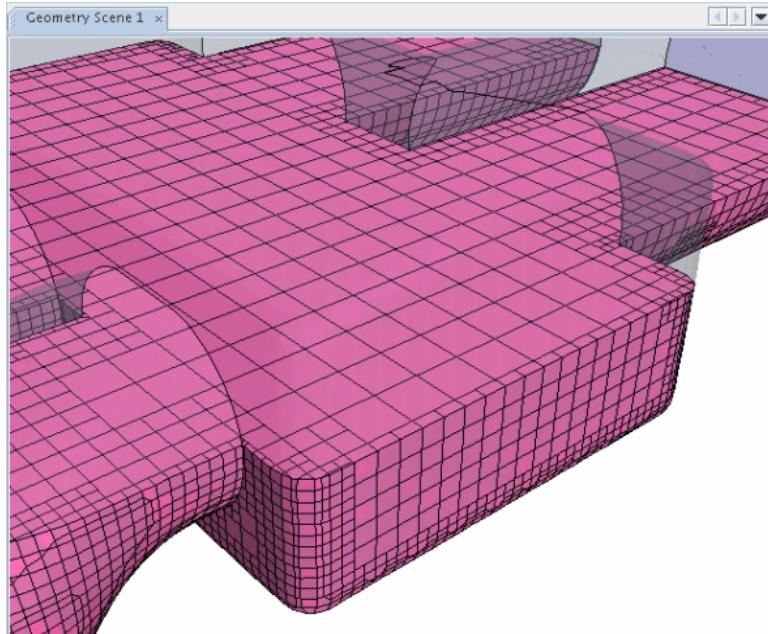
- « Tetrahedral Mesher », this model uses tetrahedrals to fill the volume. Compared to polyhedral mesher, the latter converges faster and better captures gradients.



*Example of tetrahedral mesh*

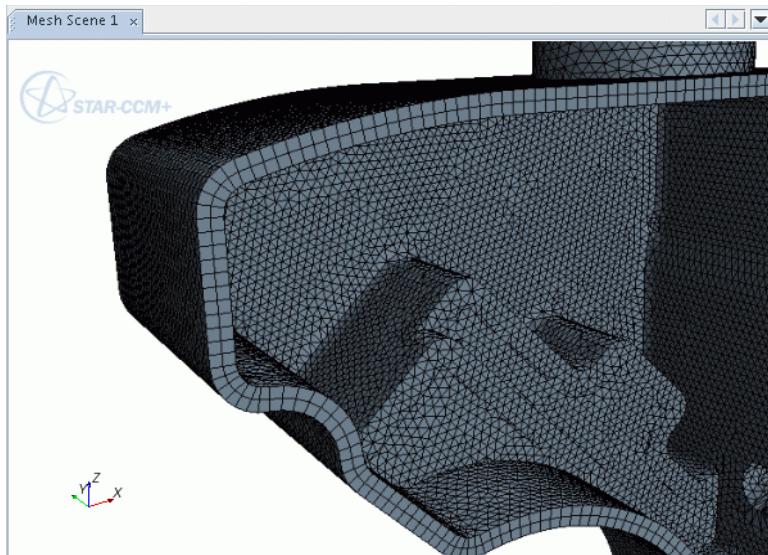
- « Trimmer », this model uses predominantly hexahedral cells to fill the volume. But cells are cut by the surface of the boundary conditions. Moreover cell refinement is obtained by splitting by 2 for each direction. Compared to polyhedral mesher, it easier to fill small gaps with trimmed cells, and

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.

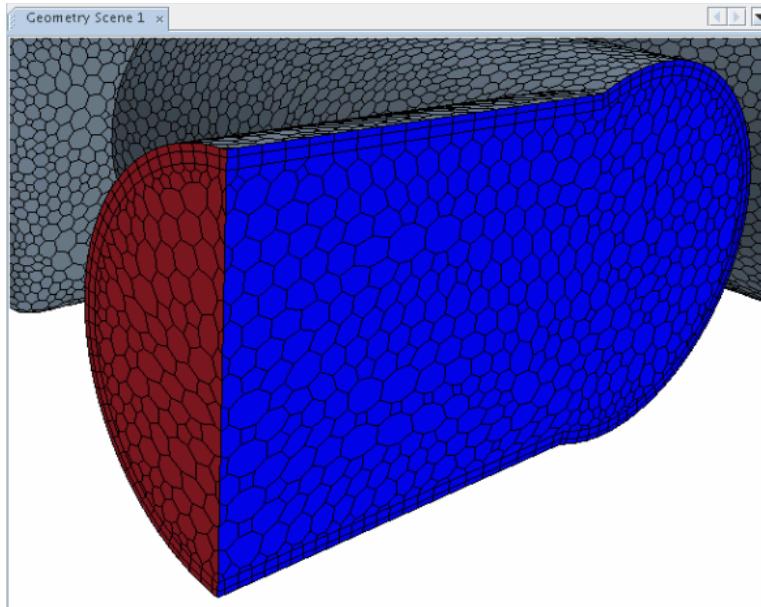


*Example of thin mesh : 2 layers are generated from the surface to mesh the volume thickness.*

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

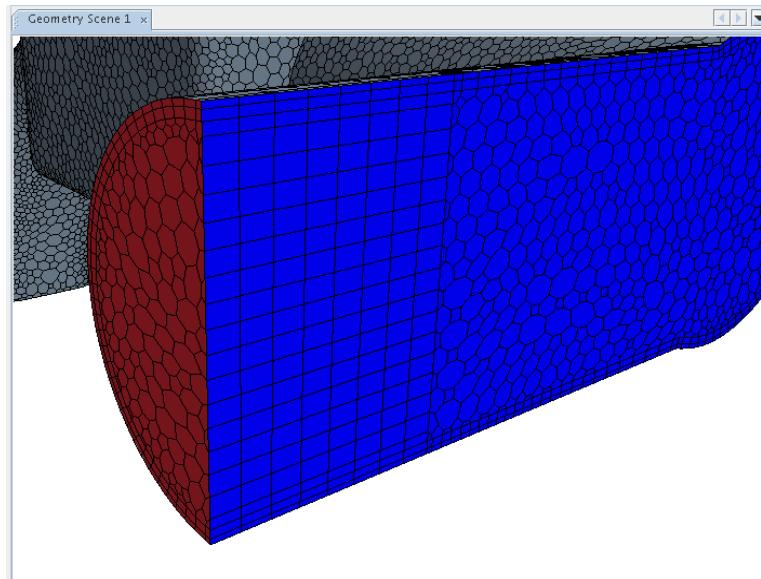
- « Prism Layer Mesher », this model generates prism layers at walls. Thin cells are extruded from the surface mesh to the volume mesh (polyhedral, tetrahedral or trimmed cells). This model is very

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.



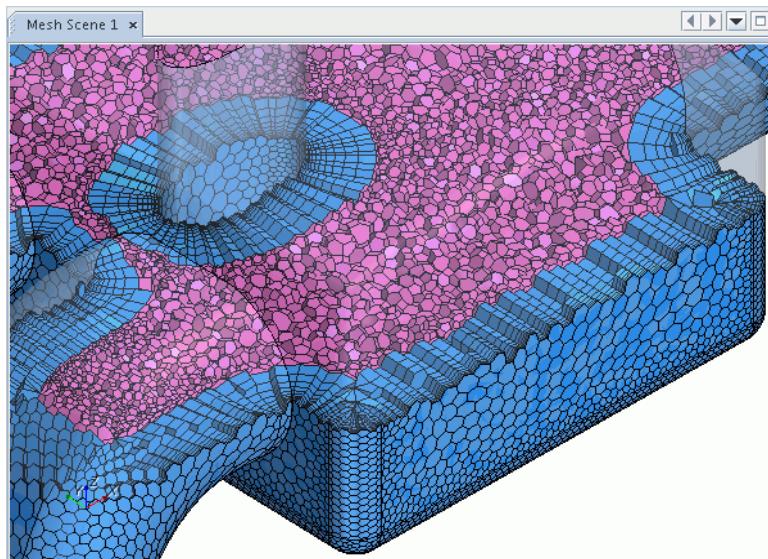
*Example of extruder mesh : 8 layers are extruded, outside the volume, mesh from the inlet.*

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 « Polyhedral Mesher » with « Prism Layer Mesher ». In « Volume Mesh » :

Select « Polyhedral Mesher », 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.

- « 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 ».
- « Tet/Poly Density », here we interest on the « Growth factor ». Its default value, equal to 1, is low and the volume mesh size often increases too slowly. A value of 1.3 often gives a good compromise between the increase of mesh size and the global number of cells.

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 ».

Note that there is no parameter to control the maximum volume mesh size (only available with the trimmed mesher). The reason is, for a tetrahedral or polyhedral mesh, the volume mesh size will never exceed the maximum surface mesh size.

For our case, we set the following parameters (press « Enter » on your keyboard after every value change) :

Base Size = 1.0m  
 Number of Prism Layers = 29  
 Prism Layer Stretching = 1.25  
 Prism Layer Thickness -> Size Type = Absolute  
 Prism Layer Thickness -> Absolute Size = 7.23e-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 = 1.5 %

Surface Size -> Relative Target Size -> Percentage of Base = 5.0 %

Tet/Poly density -> Growth Factor = 1.3

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/m<sup>3</sup>
- Viscosity = 1.85508e-5 kg/(m.s)
- Boundary length = 4.5 m (assumed to be equal to the car length)
- Desired Y+ value = 1

The output are :

- Reynolds number = 8.6e+6
- Estimated wall distance = 1.4e-5 m
- Thickness of the first prism layer = **2.8e-5 m** (twice the estimated wall distance)
- Thickness of the overall prism layers = **6.83e-2 m** (estimated thickness of the turbulent boundary layer)

Number of Layers	Layer thickness [m]	Overall layer thickness [m]
1	<b>2.80E-05</b>	2.80E-05
2	3.50E-05	6.30E-05
3	4.38E-05	1.07E-04
4	5.47E-05	1.61E-04
5	6.84E-05	2.30E-04
6	8.54E-05	3.15E-04
7	1.07E-04	4.22E-04
8	1.34E-04	5.56E-04
9	1.67E-04	7.22E-04
10	2.09E-04	9.31E-04
11	2.61E-04	1.19E-03
12	3.26E-04	1.52E-03
13	4.07E-04	1.93E-03
14	5.09E-04	2.43E-03
15	6.37E-04	3.07E-03
16	7.96E-04	3.87E-03
17	9.95E-04	4.86E-03
18	1.24E-03	6.11E-03
19	1.55E-03	7.66E-03
20	1.94E-03	9.60E-03
21	2.43E-03	1.20E-02
22	3.04E-03	1.51E-02
23	3.79E-03	1.89E-02
24	4.74E-03	2.36E-02
25	5.93E-03	2.95E-02
26	7.41E-03	3.69E-02
27	9.26E-03	4.62E-02
28	1.16E-02	5.78E-02
29	1.45E-02	<b>7.23E-02</b>

*Calculation of each prism layer thickness and overall prism layer thickness*

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

We can notice the thickness of the first prism layer was set **2.8e-5 m**, as found before. All following layers have a thickness equal to that of the previous one multiplied by the prisms layer stretching (=1.25). We need at least 29 layers to reach an overall layer thickness equal or greater than **6.83e-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 for some parts of the geometry like far-field surfaces (Inlet, Outlet, Top, Ground, Left and Right).

If we have a look in **Regions -> Region -> Boundaries -> Domain.Inlet -> 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.Inlet**).

We define different surface sizes for Inlet, Outlet, Top, Ground, Left and Right :

**Go to Regions -> Region -> Boundaries**

**Select the 6 boundaries (with Ctrl) Inlet, Outlet, Top, Ground, Left and Right 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 6 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 « Ground » is close to « car », whose minimum size is defined in those « Reference Values » (i.e. 15mm)
- the target size is the greatest size we want for the far field boundaries. 2.5 m 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 = 1.5 %**

**Relative Target Size -> Percentage of Base = 250.0 %**

**Click on « Close »**

The 6 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.Inlet » and « Domain.outlet » if we change their type respectively to « Velocity Inlet » and « Pressure Outlet » :

**Go to Regions -> Region -> Boundaries -> Domain.Inlet, change « Type » to « Velocity Inlet »**

**Go to Regions -> Region -> Boundaries -> Domain.Outlet, change « Type » to « Pressure Outlet »**

No prism layer will be generated for « Inlet » and « Outlet », since they are no more walls.

However « Top », « Left » and « Right » will be physically defined as slip walls (and without

boundary layer), and prism layers will be generated by default during the volume mesh generation. To disable that prism layer generation :

Go to Regions -> Region -> Boundaries -> Select the 3 surfaces « Top », « Left » and « Right » and right click -> Edit

Mesh Conditions -> Customize Prism Mesh -> choose « Disable »

No prism layer will be generated for « Top », « Left » and « Right », even they are walls.

Concerning « Ground », we want prism layers but they are not important for that boundary (no flow detachment). We will generate less prism layers for that surface :

Go to Regions -> Region -> Boundaries -> Domain.Ground -> Mesh Conditions -> Customize Prism Mesh -> Specify Custom Values

Go to Regions -> Region -> Boundaries -> Domain.Ground -> Mesh Values

Number of Prism Layers = 10

Prism Layer Stretching = 1.25

Prism Layer Thickness -> Size Type = Absolute Size and Absolute Size = 0.1m

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\_car.sim »

## 2.7 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 52 486.

To see resulting surface mesh :

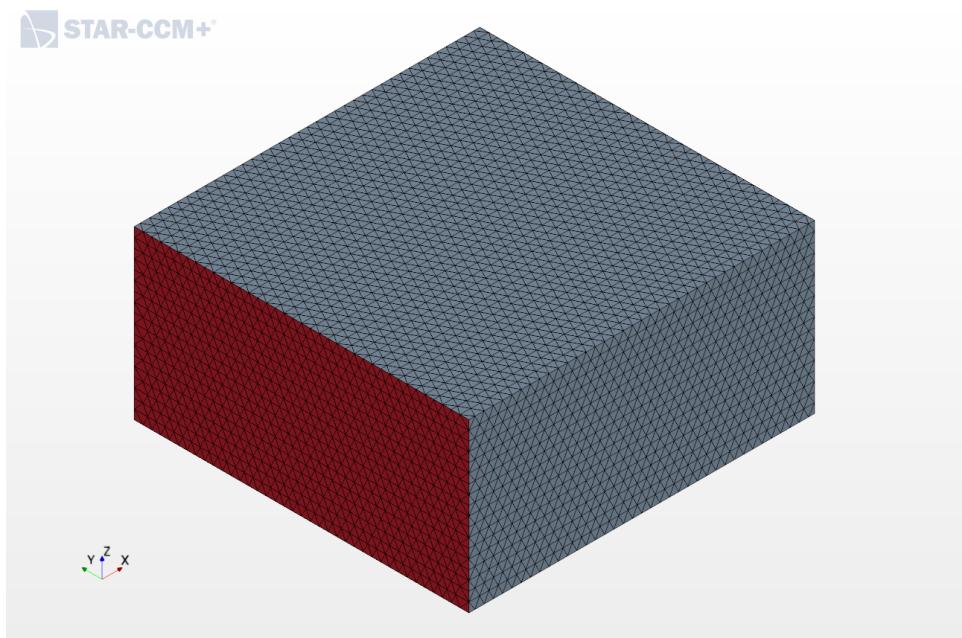
Go to « Scenes » -> Geometry Scene 1 -> Displayers

Double click on « Outline 1 » and « Part Curve 1 » to hide them

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

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

We can see now the generated surface mesh for the whole computation domain.

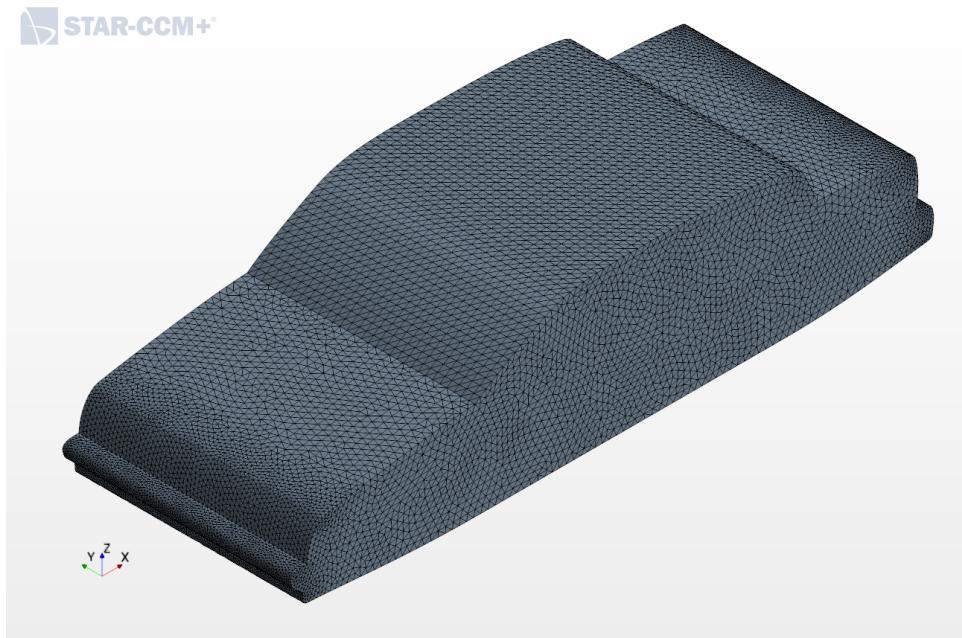


*Surface Mesh for the computation domain*

Go again into « Parts » in « Geometry 1 » and only select the Boundaries « Domain.car » in « Region ».

We can see now the generated surface mesh for the car.

Note that, compared to imported surface mesh, the surface was remeshed correctly by the « Surface Remesher ».



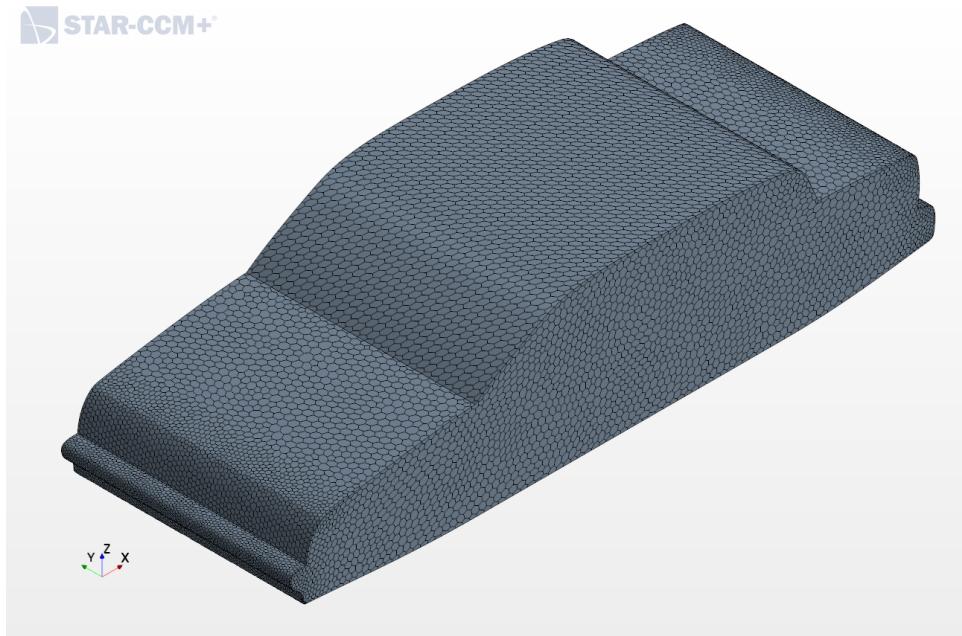
*Surface Mesh for the car*

## 2.8 Generating the Volume Mesh

Go to Mesh and click on « Generate Volume Mesh »

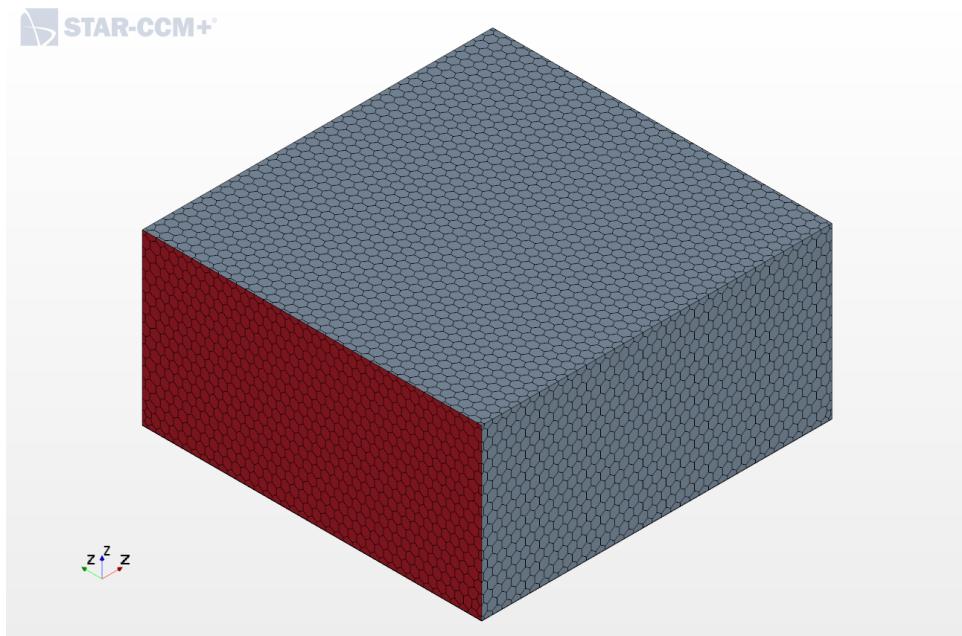
After a few minutes, the volume mesh is generated. The final number of cells should be 682 134.  
To see resulting volume mesh :

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



*Volume mesh at the car surfaces*

Go into « Parts » in « Geometry 1 » and select all boundaries in « Region ». We can see the volume mesh at the surfaces of the computation domain.



*Volume mesh at the surfaces of the computation domain*

## 2.9 Volume Mesh Diagnostics

We check the mesh quality :

Go to « Mesh » -> Diagnostics

Verify « Region » is enabled and click « OK »

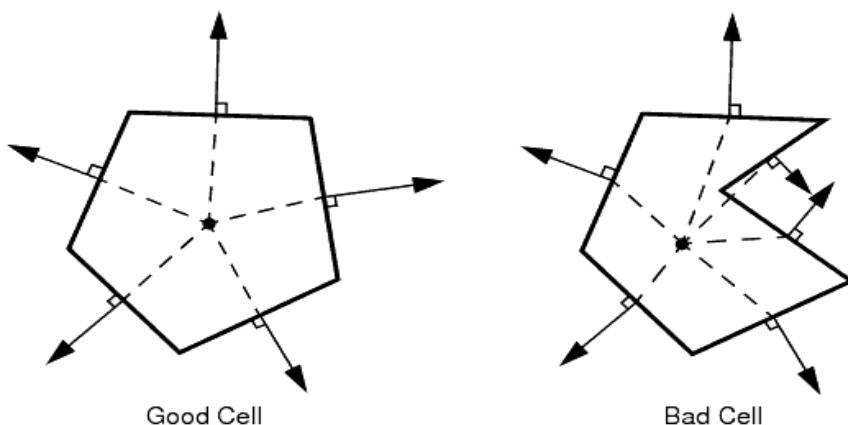
The report should be :

```
-----  
--- Computing statistics in Region: Region  
-----  
-> ENTITY COUNT:  
# Cells: 682134  
# Faces: 3042763  
# Verts: 1826336  
-> EXTENTS:  
x: [-4.5000e+01, 5.0000e+01 ] m  
y: [-4.6000e+01, 4.6000e+01 ] m  
z: [0.0000e+00 , 4.6000e+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      682134 100.000%  
-> VOLUME CHANGE STATISTICS:  
Minimum Volume Change: 7.374624e-03  
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      30  0.004%  
1e-02 <= Volume Change < 1e-01      8883  1.302%  
1e-01 <= Volume Change <= 1e+00      673221 98.693%
```

#### *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.



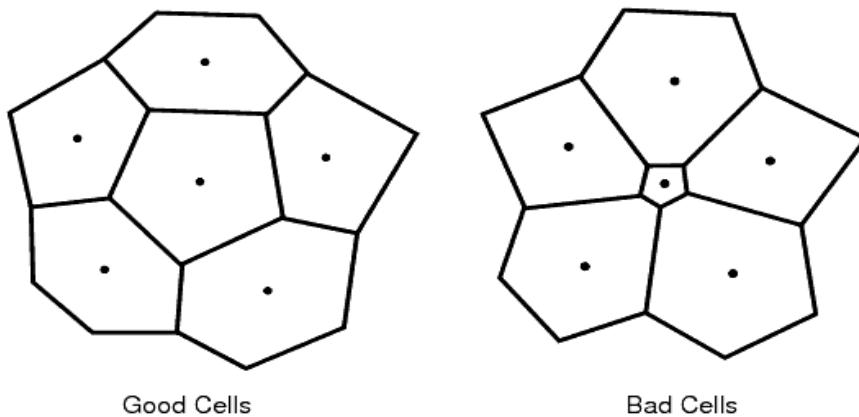
In a good quality cell, the face normals point outwards, away from the cell centroid. In a cell with bad face validity, one or more of the face normals point inwards, towards the cell centroid.

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 7.0e-3.

According to this Mesh Diagnostics Report, the mesh is correct, even if some cells have a volume change below 0.01.

However that 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 saw only the volume mesh on surfaces (car 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)

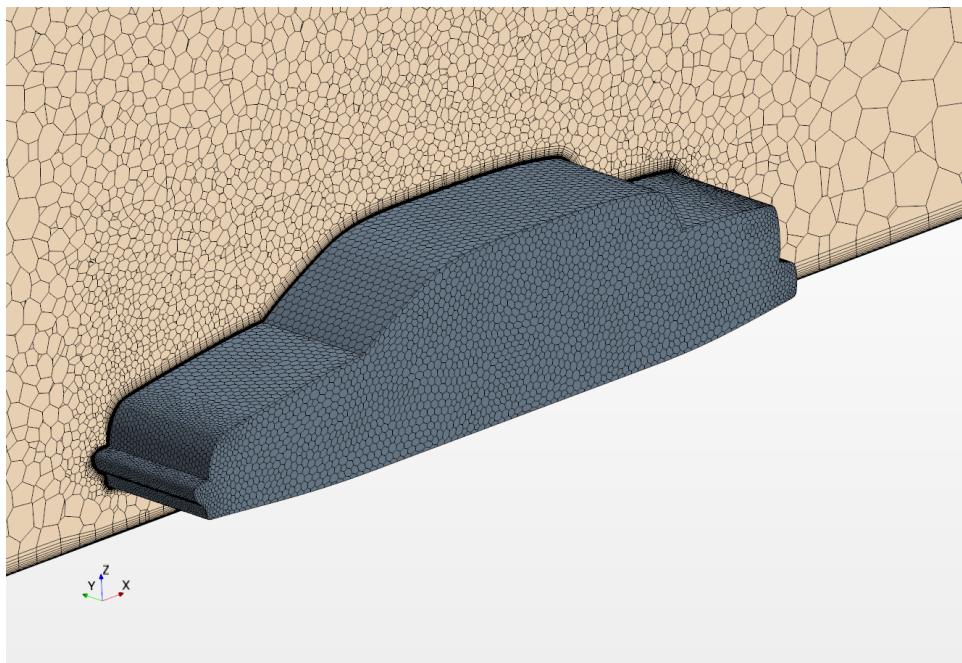
Enter [0.0, 1.0, 0.0] m in « Normal » (normal to the plane)

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 Y=0.0m »

A plane, whose equation is  $Y = 0.0$  m, appeared in « Geometry Scene 1 ». It clearly shows the size of volume mesh increases when we go away from the car. The mesh could be too coarse to model properly the wake. If needed, we will refine the mesh downstream the car during the simulation.



*Volume mesh splitted by the plane  $Y = 0.0$  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 = Segregated Flow (recommended for incompressible flow)

Equation of State = Constant Density (valid only for incompressible & isothermal 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 incompressible flow, it is recommended to use the « Segregated Flow » solver, whereas the « Coupled Flow » solver is recommended for compressible flow.

For incompressible isothermal flow, we can use a constant density for the gas phase. The « Ideal Gas » equation could be also chosen, but it would increase the computation time for the same final result. The « Ideal Gas » equation must be used for different cases :

- compressible flow
- aerothermal analysis
- multi-species gas mixture

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

We can notice we have the possibility here to change the gas properties. We do not change the default values.

Go to « Continua » -> Physics 1 -> Models -> Segregated 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.

However, the aspect ratio for the first prism layer at the car wall, equal to 1 786, is high. It was calculated by dividing the target surface size (= 0.05m) by the thickness of the first layer (= 2.8e-5m). High aspect ratios for the first layer (generally above 1 000) often leads to the divergence, if we initialize the simulation with the far-field velocity. After a few iterations, very high values are computed for the velocity in the first layer, and the simulation diverges.

To overcome that problem, we can initialize the simulation without velocity.

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

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

Turbulence Intensity = 0.01 (= 1%)

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

Turbulent Viscosity Scale = 1 m/s (it is advised to define a velocity representative of flow field inside the computation domain)

Turbulent Viscosity Ratio = 10

Velocity = [0.0, 0.0, 0.0] m/s (as explained before, we initialize without velocity, because of the high aspect ratio for the first layers)

### 3.4 Defining the Physics Boundary Conditions

We need to define the boundary conditions, which are the physical conditions (velocity, pressure, 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.Inlet »

**Go to « Regions » -> Region -> Boundaries -> Domain.Inlet -> Physics Conditions -> Flow Direction Specification**

We notice the default flow direction for a « Velocity Inlet » boundary type is normal to the boundary. We can also give components.

**Do not change the method (Boundary-Normal) for our case !**

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

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

**Do not change this value !**

**Go to « Regions » -> Region -> Boundaries -> Domain.Inlet -> Physics Values -> Velocity Magnitude**

The default value is 1.0 m/s.

**Change this value to 30.0 m/s !**

We define the conditions for « Domain.Outlet »

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

The default value is 0.0 Pa. We remind this pressure is a gauge pressure relative to the « Reference Pressure » (=101 325 Pa).

**Do not change this value !**

The other parameters concerning the turbulence are « backflow » parameters. This means these parameters will be applied to the outlet only where there is backflow (i.e. air flows into the computation domain instead of flowing outside). Backflow can occur during the computation, especially at the beginning. If there is still backflow when the simulation is converged, the computation domain could be too small.

As a summary, we defined :

- the velocity and turbulence for the inlet
- the pressure for the outlet

All remaining boundaries are by default no-slip static walls. We need to change these boundaries :

- « Domain.Ground » will have the far-field velocity
- « Domain.Top », « Domain.Left » and « Domain.Right » will be slip walls

Go to « Regions » -> Region -> Boundaries -> Domain.Ground -> Physics Conditions -> Tangential Velocity Specification

Method = Vector

Go to « Regions » -> Region -> Boundaries -> Domain.Ground -> Physics Values -> Relative Velocity -> Value = [30.0, 0.0, 0.0] m/s

The ground velocity is now equal to the far-field velocity.

Go to « Regions » -> Region -> Boundaries -> Domain.Top -> Physics Conditions -> Shear Stress Specification

Method = Slip

Do this last operation also for « Domain.Left » and « Domain.Right ».

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 :

- Car drag coefficient
- Car 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 « Domain.car » (the surface on which we calculate the drag)

Reference Area = 1.944 m<sup>2</sup> (the projection of the car surface on a frontal plane)

Reference Density = 1.18415 kg/m<sup>3</sup> (air density was defined in the physics models)

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 = 30.0 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 = [3.0, 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 Velocity »

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

Go to « Scalar 1 » -> Parts and Select « Plane Section Y=0.0m » in « Derived Parts »

The plane is displayed without function field.

In « Scalar Field » -> Function, select « Velocity Magnitude ».

The field of velocity magnitude 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 :

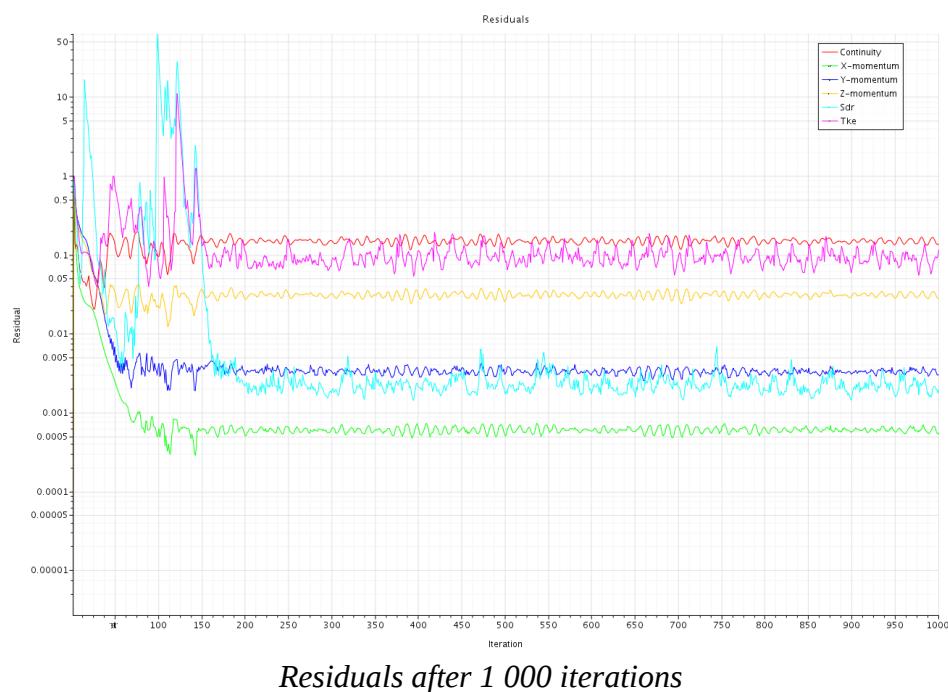
**Go to « Solution » and Click « Run »**

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 Velocity »

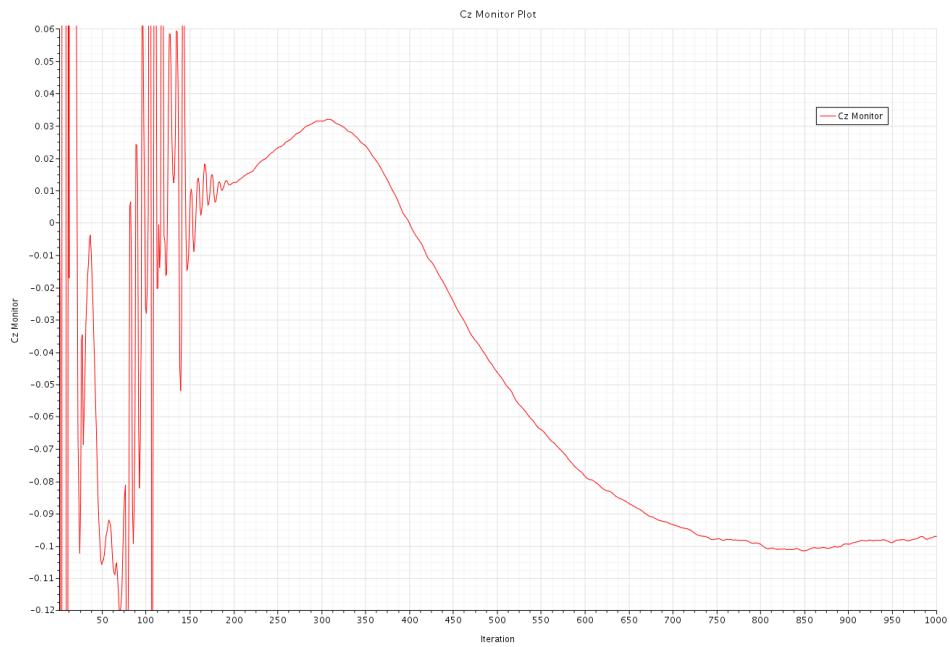
Thousands of iterations are generally needed to converge. But after 1 000 iterations, we can guess the simulation should not converge, because the « Residual monitor plot » shows that many residuals stabilize around high values (residual values below  $10^{-3}$  were expected for the convergence).



We often obtain that behavior with residuals for two cases :

- because of a coarse or bad quality mesh, or errors in the solver or boundary conditions setup ;
- because the flow is actually unsteady and it is impossible to find a steady solution, even with a steady solver.

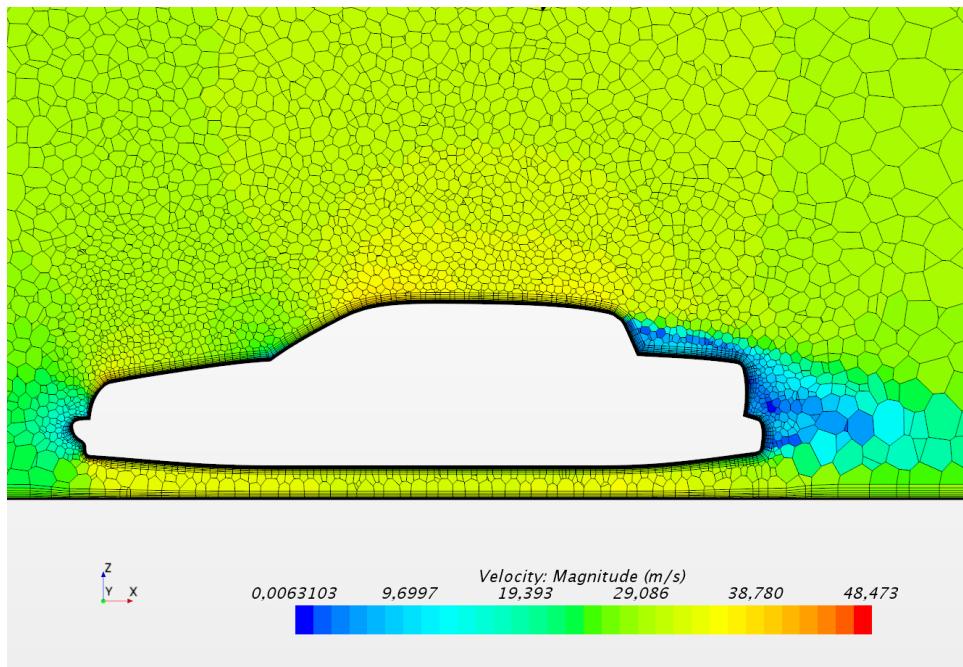
With other monitor plots ( $C_d$ ,  $C_z$  and Pressure-wake), we observe instabilities for the 500 first iterations, then these monitors seem to converge to a constant value.



*Cz Monitor Plot after 1 000 iterations*

The residuals do not converge, but the monitors tend to reach constant values. We need to check if the mesh is not too coarse in some parts of the computation domain.

In the « Scalar Scene Velocity », it appears clearly the mesh downstream the car is too coarse to model correctly the wake. We observe flow detachments and we can expect vortices, which are probably numerically dispersed, such as the flow instability. We need to refine the mesh in the wake.



*Velocity Magnitude in Plane  $Y = 0$  after 1 000 iterations*

## 6. Mesh refinement

We want to refine the mesh in the wake. Since the mesh is also coarse between the car and the ground, we will also increase the number of cells between these two surfaces.

Go to « Continua » -> Mesh 1 -> Reference Values -> Surface Proximity

In « Surface Proximity - Properties » :

Search Floor = 0.01 m

# Point in Gap = 5.0

That change will refine the mesh between the car and the ground.

We use now a « Volumetric Control » to define a volume mesh size inside a box in the wake of the car.

To create this box :

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

Corner 1 = [2.3, -0.9, 0.0] m, m, m

Corner 2 = [4.0, 0.9, 1.25] 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 » -> Polyhedral Mesher, enable « Customize Surface Remesher » AND  
« Customize Polyhedral Mesher »

In « Mesh Values » -> Custom Size -> Relative Size, Percentage of Base = 4.0

In the box « Volume control » the size for all faces and polyhedral cells will be 40mm.

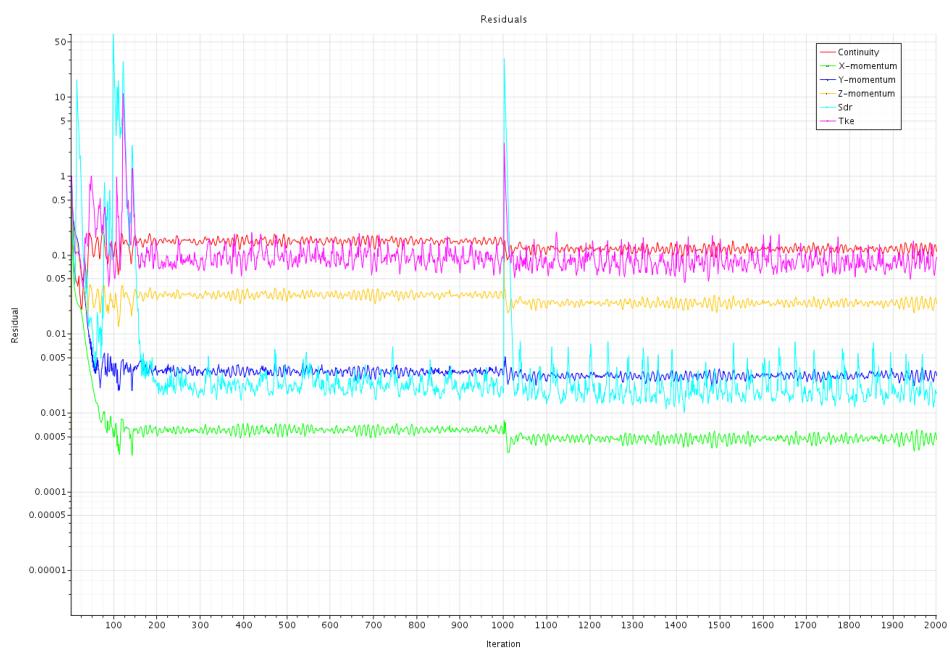
Go to « Mesh » -> Generate Volume Mesh

After a few minutes, a new volume mesh is generated with 1 035 477 cells. Note that the solution computed with the previous mesh was interpolated to the new one.

We can run 1 000 more iterations with the unsteady solver :

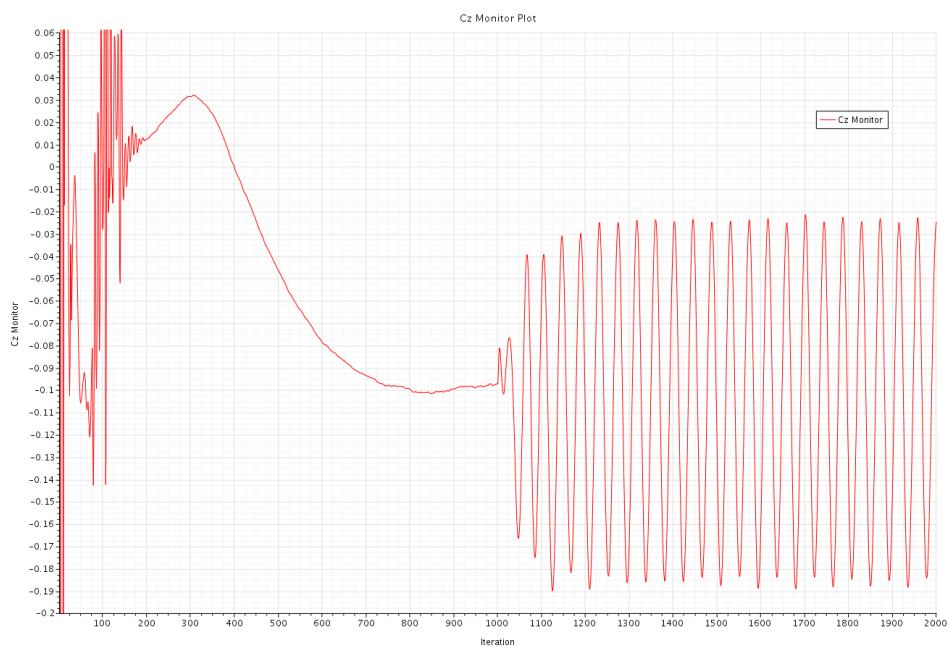
Go to « Stopping Criteria » -> Maximum steps, Maximum steps = 2 000

Go to « Solution » and Click « Run »

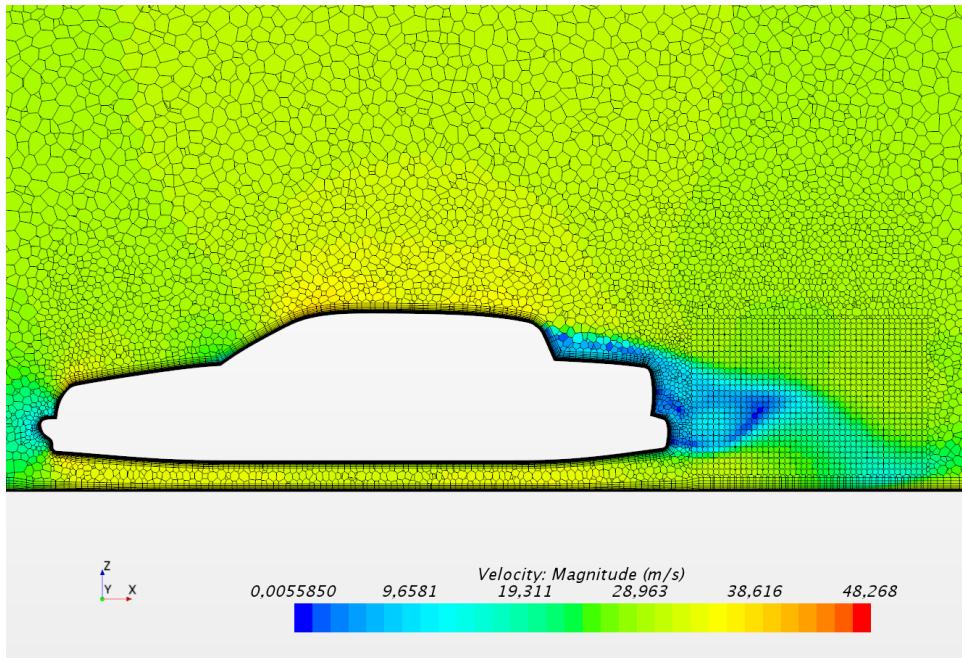


*Residuals after 2 000 iterations*

After 1 000 more iterations, the behavior of the residuals remains similar, contrary to the other monitor plots ( $C_x$ ,  $C_z$  and Pressure-wake) which oscillate now.



*Cz Monitor Plot after 2 000 iterations*



*Velocity Magnitude in Plane  $Y = 0$  after 2 000 iterations*

The wake visible on the contours of velocity magnitude in plane  $Y = 0$  is now better discretized and became unsteady. That explains the fluctuations of  $C_x$ ,  $C_z$  and Pressure-wake.

It is not possible to converge to a steady solution, because the flow is naturally unsteady. We need to change the solver from steady to unsteady.

**In the case of natural unsteady flow, a current mistake is to « kill numerically » this instability to converge to a steady flow, and to assume that steady flow is the time averaged solution of the unsteady flow. That could be totally wrong !!**

## 7. Unsteady Solver

To change the solver from steady to unsteady :

Go to « Continua » -> Physics 1, and double click on « Models »  
 In « Physics 1 Model Selection », Disable « Steady » on the right  
 Then choose « Implicit Unsteady », on the left, in « Time »  
 Click « Close »

First we have to define a time step. Two criteria must be taken into account :

- the « Convective Courant Number » ; an implicit unsteady solver can support value of 100 for the Courant Number, but the average value must be around 1. If we consider the car surface target size (= 50mm) and the far-field velocity of 30m/s, the time step should be 1.67 ms.

- the highest frequency of the instability ; it is difficult to evaluate the highest frequency for such a complex flow. In some cases, for example the vortex shedding from a spherical cylinder, it is possible to calculate the frequency, and thus the period, from the Strouhal number. Since we need at least 20 points for the discretization of that period, we divide it by 20 to find the time step.  
 For example, if we assume the car is a cylinder, whose diameter is equal to the car height (= 1.05m), and considering a Strouhal equal to 0.22, the frequency of the vortex shedding would be 6.3 Hz. If we divide the resulting period (= 1/6.3 Hz = 0.159s) by 20, we finally find 8 ms.

As we cannot estimate accurately the frequency of the flow instability, we choose the time step of 1.67ms calculated from the convective Courant Number.

Go to « Solvers » -> Implicit Unsteady  
 Temporal Discretization = 2nd-Order  
 Time-Step = 0.00167s

We have also to define new « Stopping Criteria » :

Go to « Stopping Criteria » -> Maximum Inner Iterations

With Implicit Unsteady Solver, we need several « inner iterations » to converge « in the time step » before going to the following one. The default value of 5 inner iterations is low. It is not sufficient to converge in the time step, and can lead to diverge of the simulation. 10 inner iterations seems to be a minimum acceptable value. 20 inner iterations is often used.

Maximum inner iterations = 20

The next two stopping criteria are linked if we have a constant time step (our case here) : the « Maximum Physical Time » and the « Maximum Steps ». With the unsteady solver, a step is no more an iteration, but a time step. The maximum physical time is equal to the maximum steps multiplied by the time step. We can disable the « Maximum Physical Time » and use the « Maximum Steps ».

In « Maximum Physical Time », disable « Enabled » (this stopping criterion will not be considered)  
 In « Maximum Steps », Maximum Steps = 1 000 (to run 1.67 second of physical time).

To save all Monitors at every time step instead of every iteration, we change the trigger :

Go to Monitors, and do a multiple selections (with Ctrl) with Cd, Cz and Pressure-wake Monitors, right click -> Edit ...  
 Trigger = Time-step  
 Click « Close »

You can also update the plots at every time step, and change the bottom axis from « iterations » to

« Physical Time » :

Go to Plots, and do a multiple selections (with Ctrl) with Cd, Cz and Pressure-wake Monitor Plots, right click -> Edit ...

X-Axis Monitor = Physical Time

-> Update -> Trigger = Time-step

Click « Close »

Since this unsteady simulation is long (a few hours), we save automatically the simulation every 100 time steps :

Go to « File » -> Auto Save ...

In « Auto Save » window -> Max Autosaved Files = 2 (to keep the 2 last saved files)

-> Update -> enable « Enabled » (to enable the Auto Save)

Trigger = Time Step

-> Time-Step Frequency -> Frequency = 100 (to save every 100 time steps)

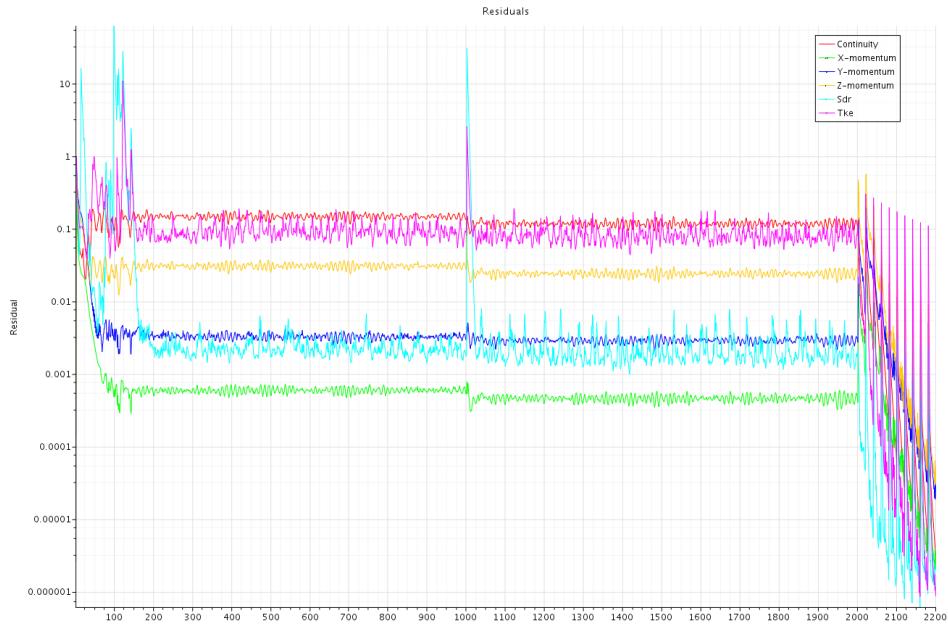
Click « Close »

We start the simulation :

Go to « Simulation » and Click « Run »

After 10 time steps, we can observe a different behavior for the residuals (residuals from 2 000 to 2 200). It is a classical behavior for residuals with the unsteady solver.

They globally decrease with peaks every 20 iterations. The reason is the unsteady solver converges during the 20 inner iterations of a time step. After those 20 inner iterations (corresponding to the end of the time step) the unsteady solver goes to the following time step and a peak appears. The unsteady solver converges again during the 20 inner iterations of the time step ...

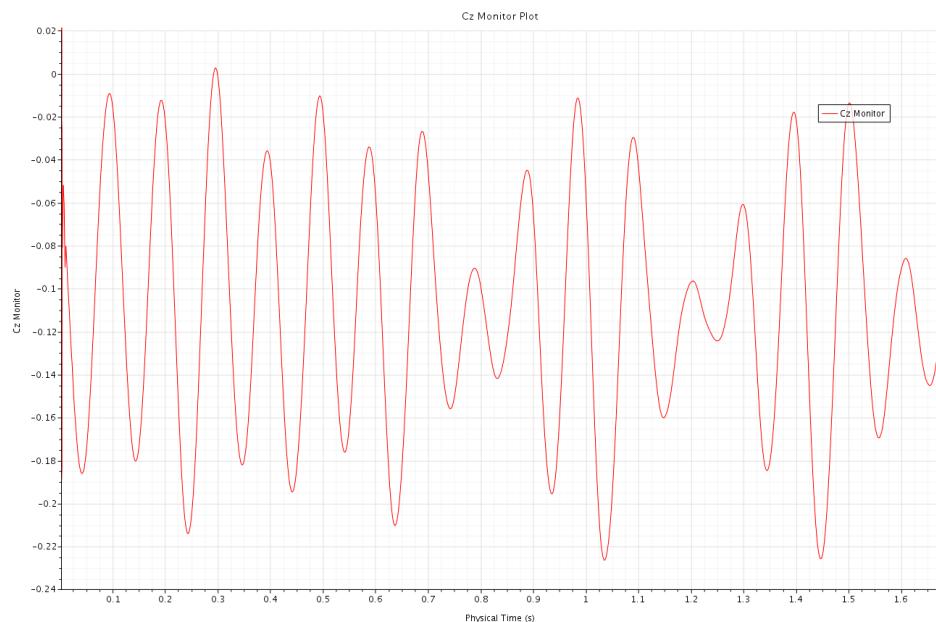


*Residuals after 2 000 iterations and 10 time steps*

After 1 000 time steps, the monitor plots for Cx, Cz and Pressure-wake demonstrate we obtain finally an unsteady flow.



*Residuals after 2 000 iterations and 1 000 time steps*



*Cz Monitor Plot after 1 000 time steps*

## 8. Wall Y+ and Courant number

For the mesh generation, we calculated the thickness of the first prism layer to aim values around 1 for the wall  $Y^+$  on the car wall. We also defined a time step to obtain values around 1 for convective Courant number.

We check if we obtain finally these values.

### A) Wall +

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  $Y^+$  »

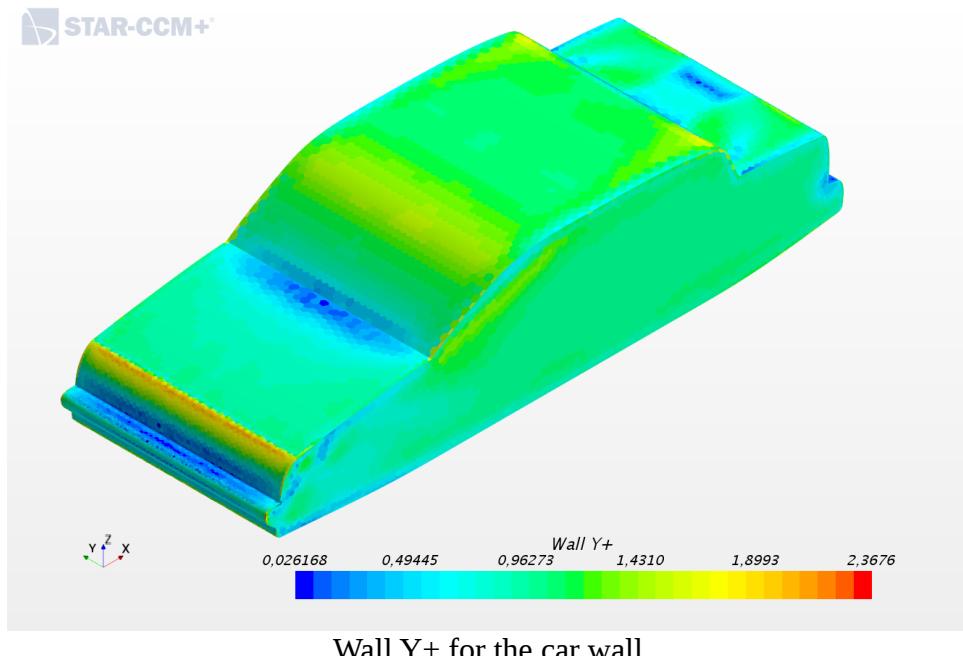
In that scene, in « Displayers », double click on « Outline 1 » to hide it

Go to « Scalar 1 » -> Parts

In Regions -> Region -> Boundaries and Select « Domain.car »

The car wall is displayed without function field.

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



Wall  $Y^+$  for the car wall

The mean value of wall  $Y^+$  on the car 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 = 2.4 in our case).

The thickness for the first prism layer was correctly calculated.

### B) Convective Courant number

We create a scalar scene of Convective Courant Number :

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 Courant Number»

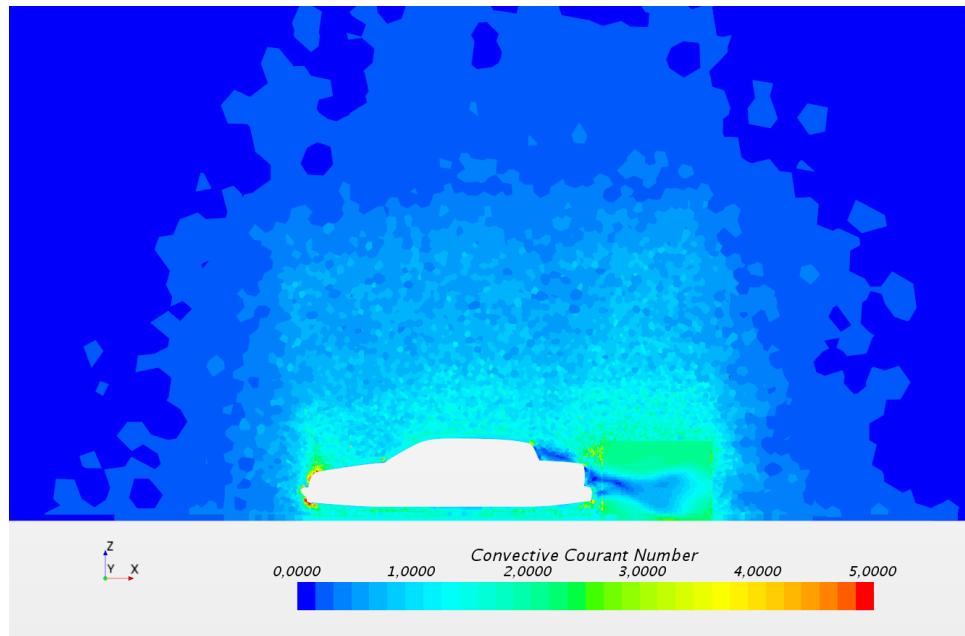
In that scene, in « Displayers », double click on « Outline 1 » to hide it

Go to « Scalar 1 » -> Parts

In Derived Parts -> Select « Plane Section Y=0.0m »

The plane section is displayed without function field.

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



*Convective Courant Number in Plane  $Y = 0$*

We obtain values close to 1 for the Convective Courant Number around the car. They are very low far from the car, because of the coarser mesh. We obtained the highest values where we have flow acceleration and a fine mesh nearby the car. However the maximal value around 15 is acceptable.

The time step was correctly defined.

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

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

Then we decrease the time step until the results does not change any more.

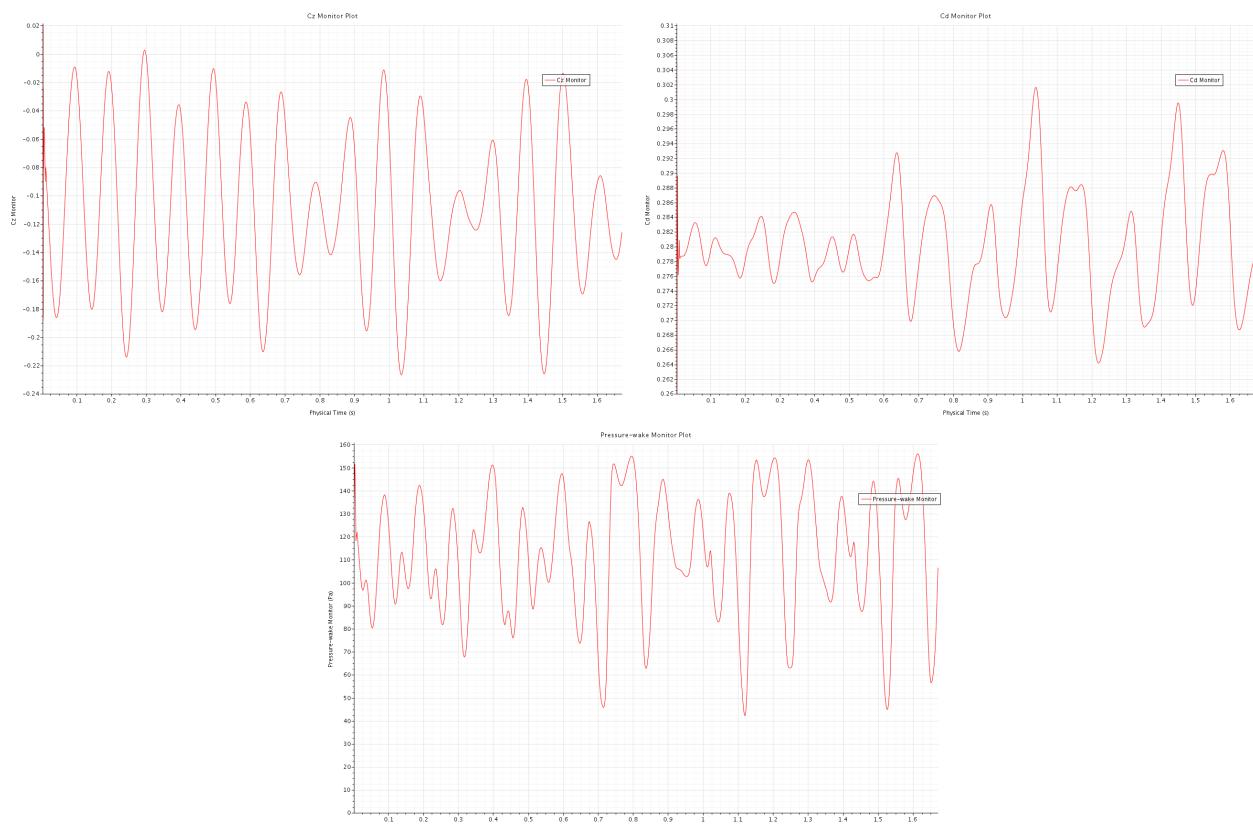
Do not forget that, for unsteady simulations, the mesh refinement will change the Convective Courant Number, and the time step must be decreased !

## 9. Post-processing

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

### A) Signal analysis

First, we can analyze the evolutions of the monitors plots presented below.



*Cz, Cd and Pressure-wake Monitor Plots after 1 000 time steps*

We can guess from these three monitor plots that several frequencies appear. To find all frequencies, we create a FFT for each monitor :

Go to « Tools » -> right click on « Data Set Functions » -> New -> Point Time Fourier Transform

G(p) 1 is created. Rename it « FFT » :

Right click on « FFT » -> « Rename » and rename « FFT »

The Cd Monitor plot shows the flow can be considered established after 0.7s. It will be the start time for the calculation of the FFT.

Go on « FFT » and change the following parameters :

Start Time = 0.25s

Cut-off Time = 1.67s

Window Function = Hann

In FFT, right click on « Monitor » -> New derived data from monitor

Rename (right click -> « Rename ») the created Monitor « Cz »

In Cz, change Input Data 1 to « Cz Monitor »

Do the same three last operations for Cd and Pressure-wake

Select Cd, Cz and Pressure-wake, right click and select « Update Output »

FFT were generated for each monitor.

We want now to plot those FFT :

Go on « Plots », right click -> New Plot -> Monitor Plot

Monitor Plot 1 was created.

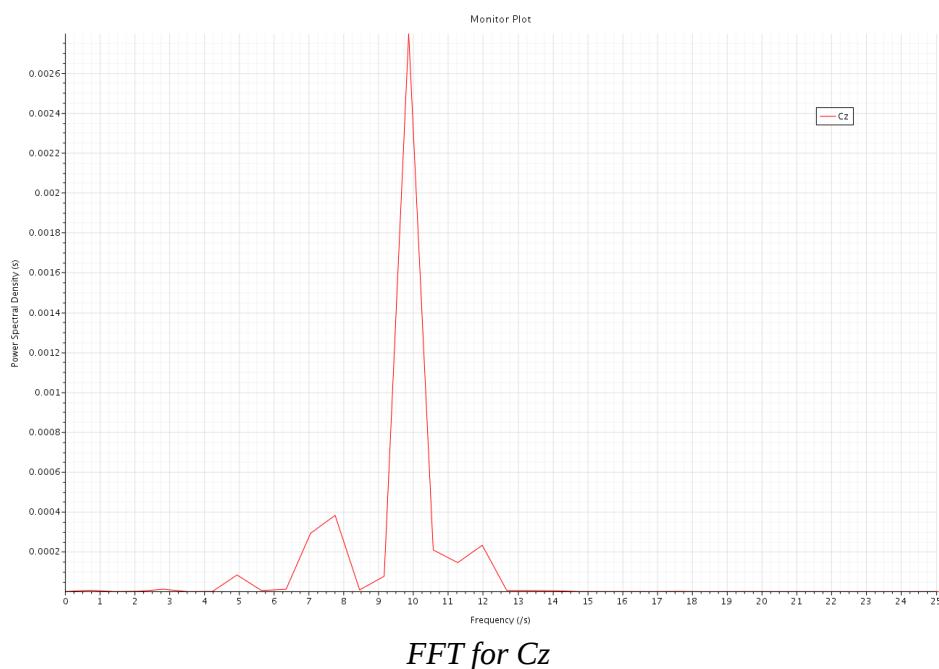
Rename it (right click -> Rename) « Cz FFT »

In « CZ FFT », right click on « Data Series » -> Add Data

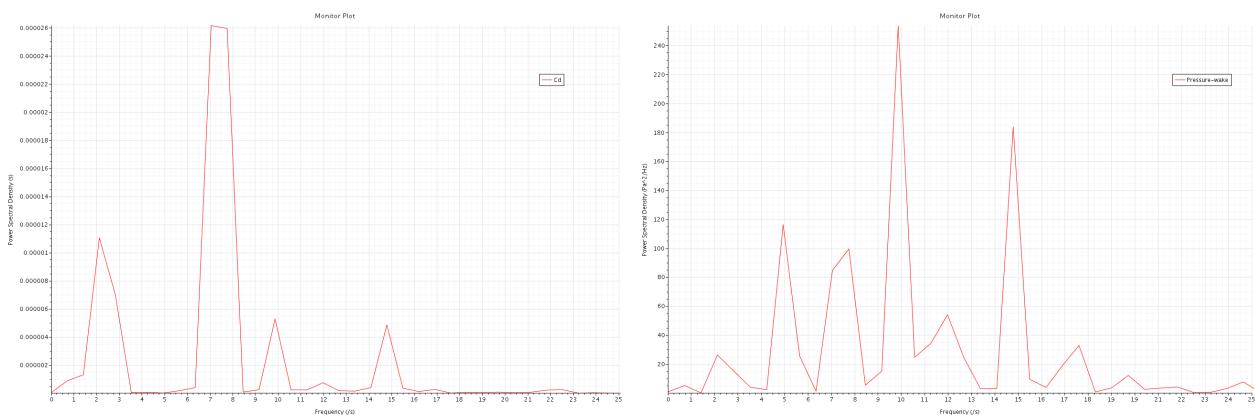
A « Add Data Providers to Plot » window is opening.

In « Derived Data », select « Cz » (it is the FFT of Cz we generated before)

We have now the plot of the FFT for Cz. We obtained a frequency of 9.9 Hz



Create similar plots for Cd and Pressure-wake.



We found for  $C_d$  : 2.1 Hz and 7.4 Hz.

We found for Pressure-wake : 4.9 Hz, 7.7 Hz, 9.9 Hz and 14.8 Hz.

However these FFT were generated from monitors for a short period (around 1.4 second). We should normally run the simulation for a longer period in order to calculate more accurate FFT, especially for the lowest frequency.

### B) Contours

During the simulation, the setup for the scalar scene was detailed. We displayed contours of velocity magnitude or Courant number in the plane  $Y = 0$ , or contours of wall  $Y^+$  on the car wall. Many other 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 car walls :

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

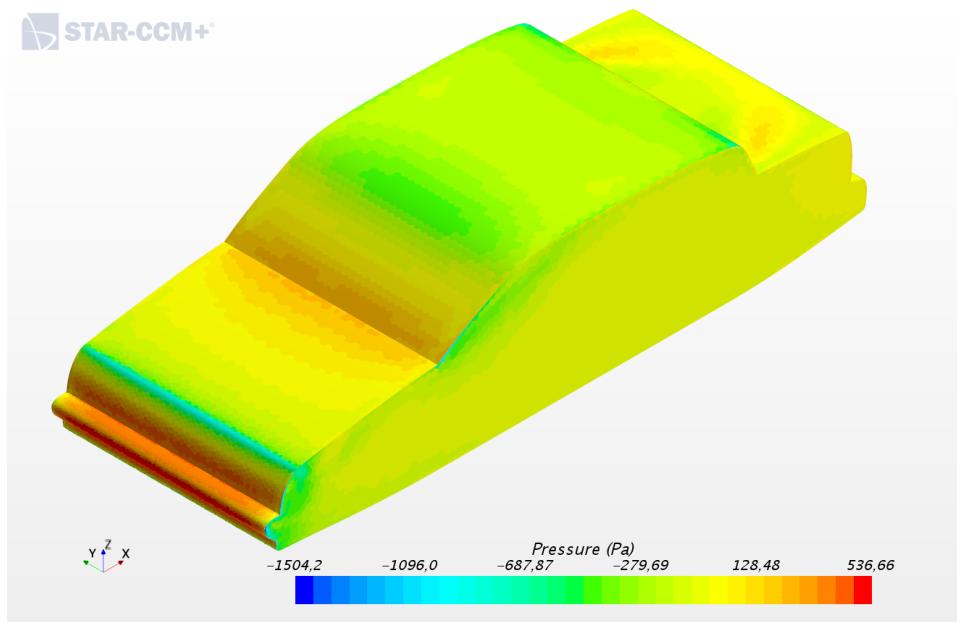
Go to « Scalar 1 » -> Parts

In Regions -> Region -> Boundaries and Select « Domain.car »

The car wall is displayed without function field.

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

We obtain the following pressure field on the car.



*Pressure on the car walls*

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

### C) Plots

For a more detailed analysis of the pressure distribution, we can plot a pressure coefficient distribution. We create first the car profile, which is the intersection of the car 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, Domain.car (to specify the plane splits only the car wall, and not the whole « Region »)

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

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 ») « Car profile »

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

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

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

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

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

The pressure coefficient on the car profile is now plotted. However, the values are very high, since a maximum value of 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

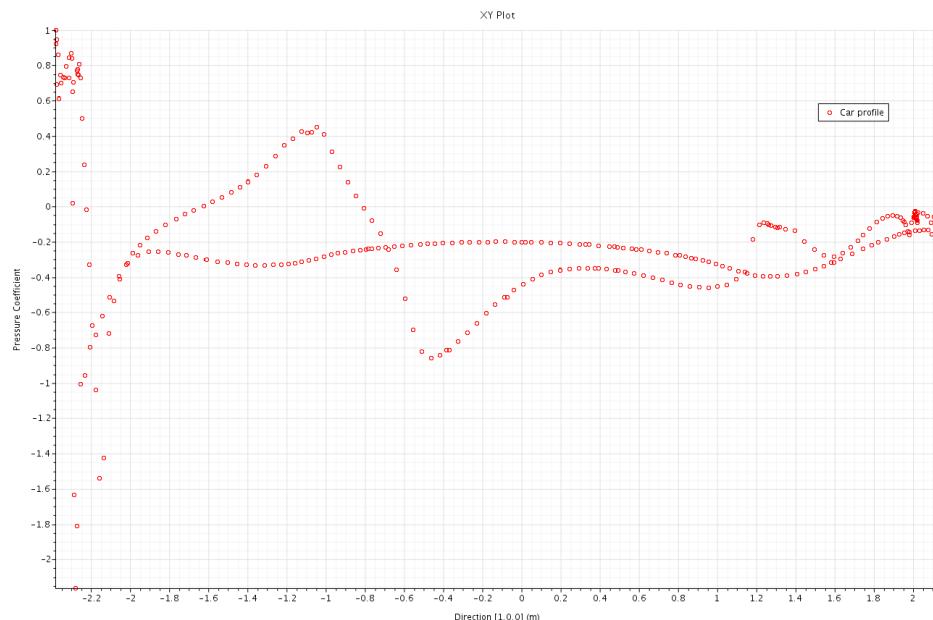
In « Pressure Coefficient - Properties » window, change the following parameters :

Reference Density = 1.18415 kg/m<sup>3</sup>

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 = 30.0 m/s (Far-field velocity)

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



*Pressure coefficient on the car profile*

#### D) 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 hide the eventual planes (double click on the displayers « Section Geometry »).

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 »)

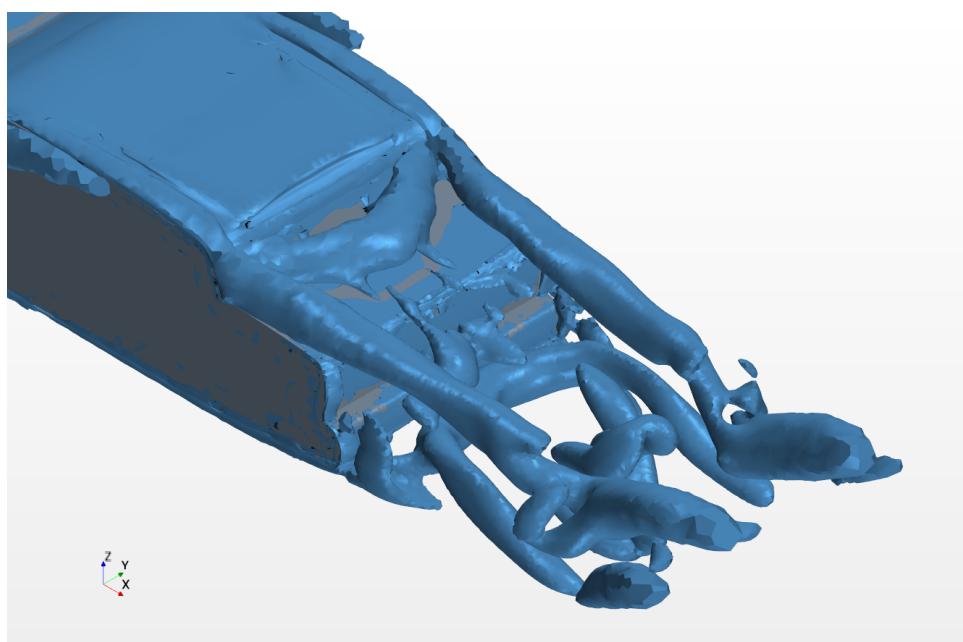
Scalar = Q-Criterion

Isovalue = 1 000 s<sup>-2</sup>

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 car can be visualized with the isosurface of Q-criterion = 1 000 s<sup>-2</sup> shown below.



*Isosurface of Q-criterion = 1 000 s<sup>-2</sup>*

#### E) Line Integral Convolution

To visualize the flow separation, we can display wall shear stress vectors on the car 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 »

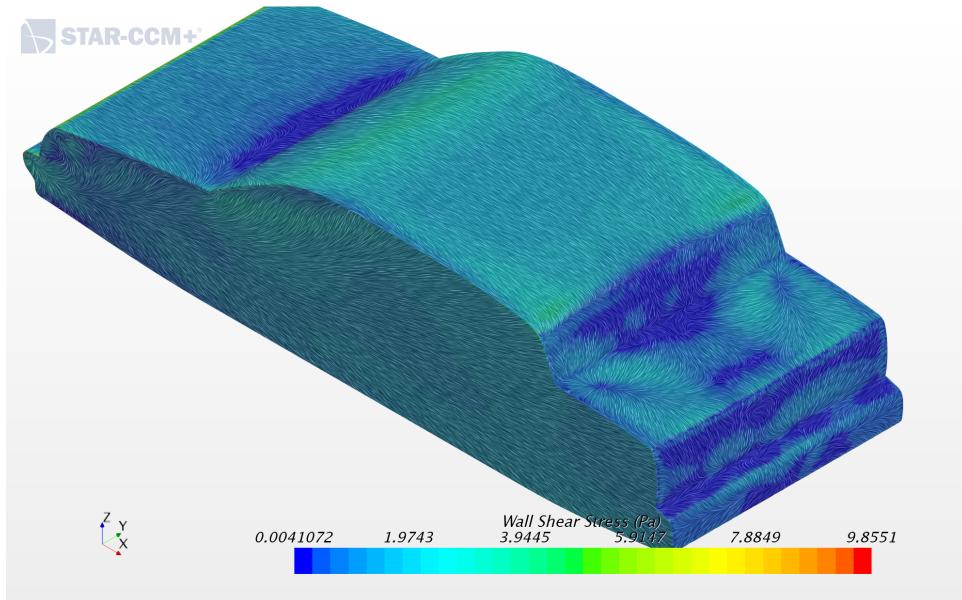
Still in Vector 1 », in « Parts » -> Regions -> Region 1 -> Boundaries, select « Domain.car »

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*

#### F) 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) in front of the car :

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 = [-2.5, -0.9, 0.0] m, m, m

Point 1 = [-2.5, 0.9, 0.0] m, m, m

Point 2 = [-2.5, -0.9, 0.85] m, m, m

The presentation grid is now in front of the car.

If needed, display the « Geometry Scene 1 » and hide (by double click) all displayers, except « Geometry 1 » (the car 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 =10 (to have 10 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)

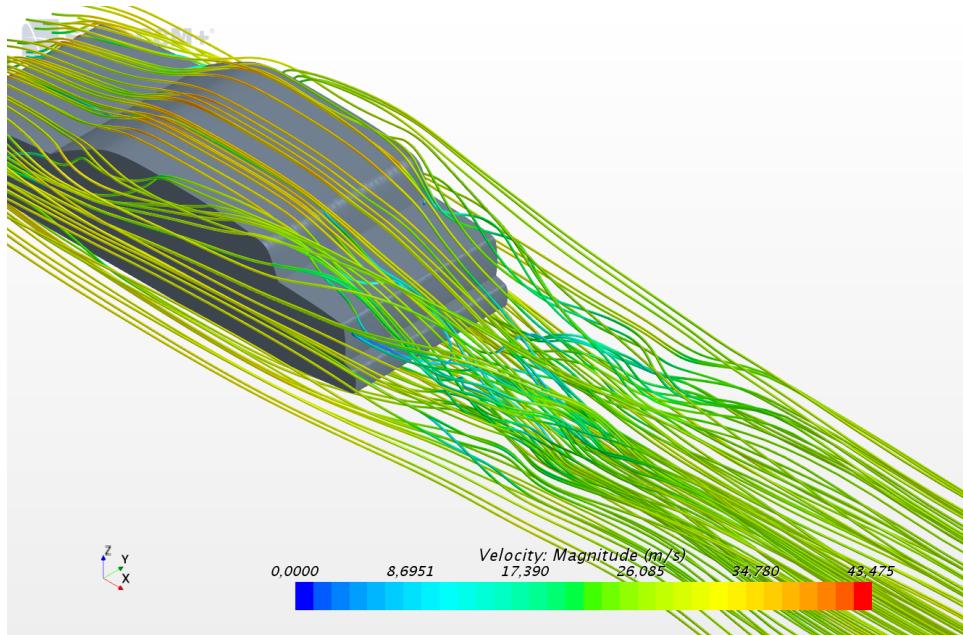
Click « Create » then « Close »

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 « Velocity Magnitude» for Function (to colour the streamlines with velocity magnitude)

We have now streamlines generated from the presentation grid in front of the car.



Streamlines generated from the presentation grid