

# Ongoing Race Project

Secondary education project for learning CATIA V5



# About the course

## Course objectives

This course provides the fundamental know-how and skills to work with CATIA V5 on the creation of parts, assembly of components and creation of CNC manufacturing data to participate in the "Race in Progress" competition of mini F1 .

## Design philosophy

This course has been designed in a modular format to allow its application over consecutive or non-consecutive periods. Each module includes a demonstration, explanations and practice exercises, with particular attention to the ability to produce a concrete result.

## concerned public

Educator in secondary education.

## Prerequisites

No prerequisites.



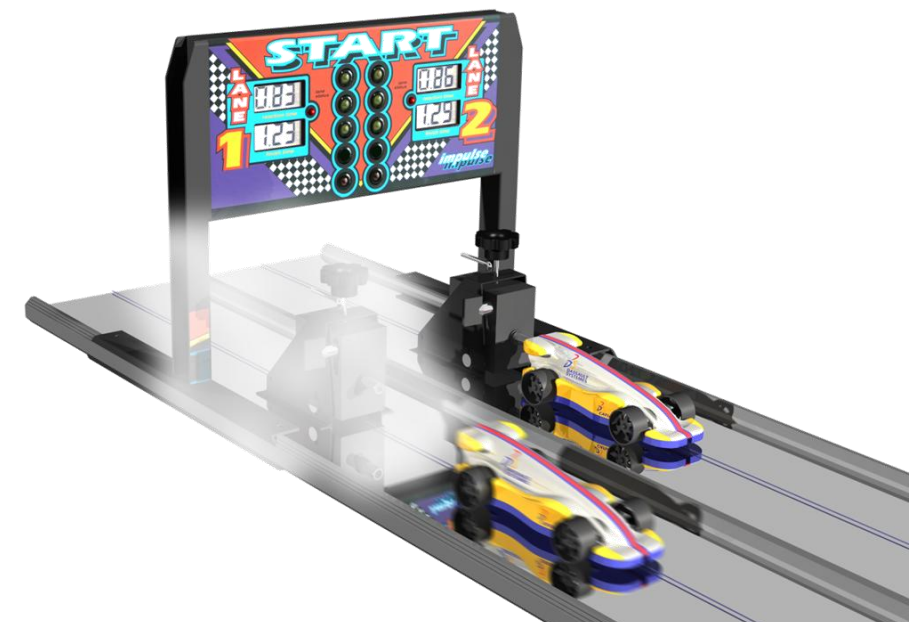
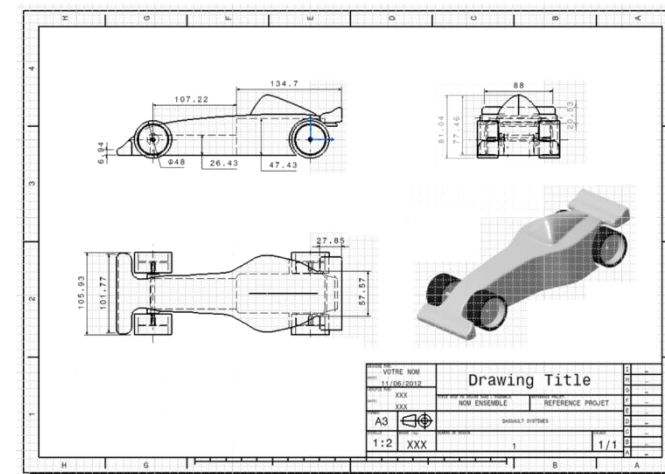
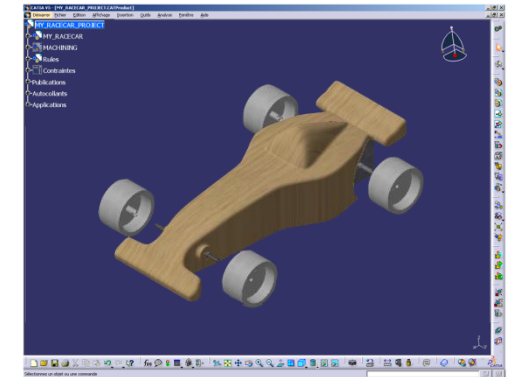
Update of the training document produced by: Jusuf BALTIC, Loïc ETTIGHOFFER, Rémi MULLER, Ghislain ROUZIER and Maxime SCHEFFLER, 3rd year students at ESSTIN under the direction of Mr Emmanuel JACQUOT.

# Foreword

- ▶ That training is based on a project secondary education for learning CATIA V5 design software.
- ▶ The objective is to design a digital model of a mini-racing car and to manufacture it according to specifications.
- ▶ This course introduces the essential principles of modeling in CATIA V5 and the techniques to create a mini-racing car.

At the end of this training, you will be able to:

- Understand the CATIA V5 interface,
- Open a project and design your own 3D racing car,
- Assemble the different elements of your car and make a detailed plan,
- Control the different characteristics to meet the requirements,
- Define a machining process,
- Use realistic rendering calculation tools.



# Training philosophy

- ▶ This manual illustrates a complete PLM\* project covering all disciplines from preliminary design to manufacturing.
- ▶ The purpose of this training is to teach you how to manage an advanced design project starting from its specifications.
- ▶ The training provides you with the key concepts, the methodologies, and an application of the basic techniques, as well as method sheets, to reinforce your knowledge of the CATIA V5 design software.
- ▶ The file that accompanies this manual contains:
  - ▶ an electronic version of the manual;
  - ▶ files to start computer-aided design;
- ▶ Presentation videos are available on the internet, on the Youtube channel [3DS Academy](#) .



\* PLM: Product Lifecycle Management. In French, this corresponds to “product life cycle management”.



# How to use this training

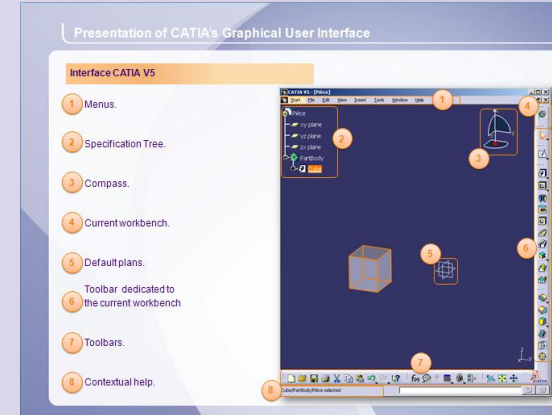
This training uses the graphic convention below:



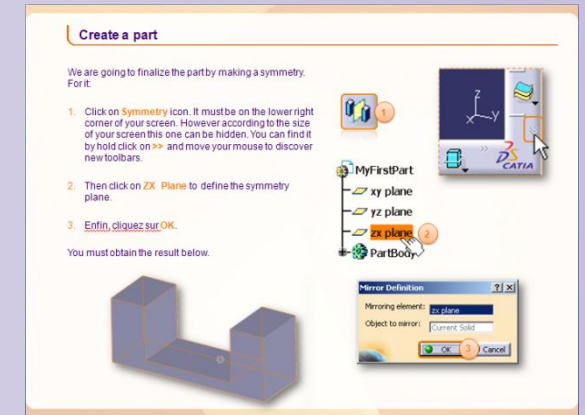
*lesson introduction*



*Theory*



*Main tools*



*Exercises*



*Single click*



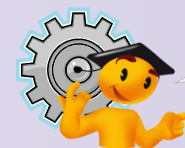
*Double click*



*button hold*



Advice, information  
additional, links to  
a method sheet.



Reference to program points  
school.

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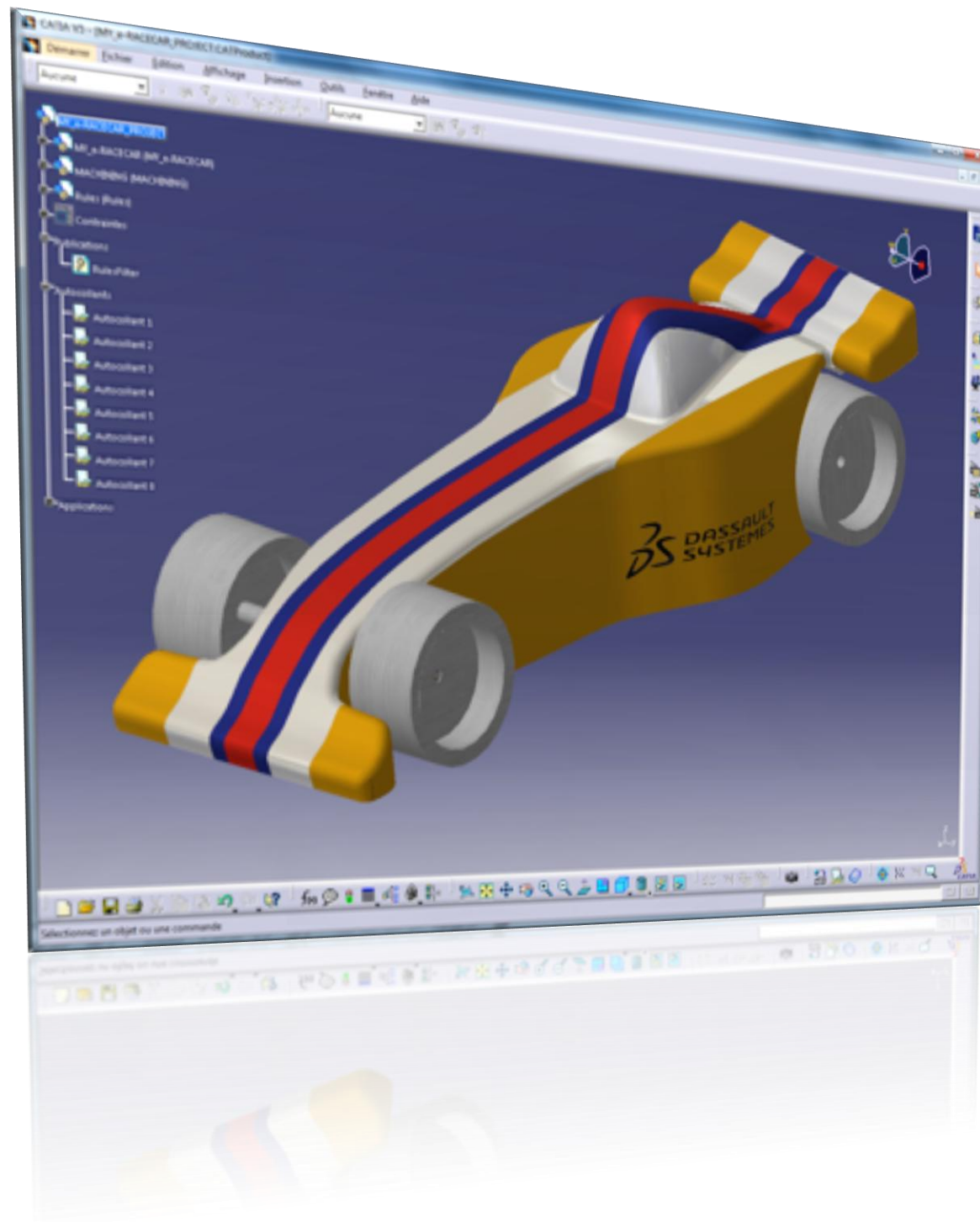
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# Introduction

In this lesson you will learn more about the project.



Here are the steps to follow:

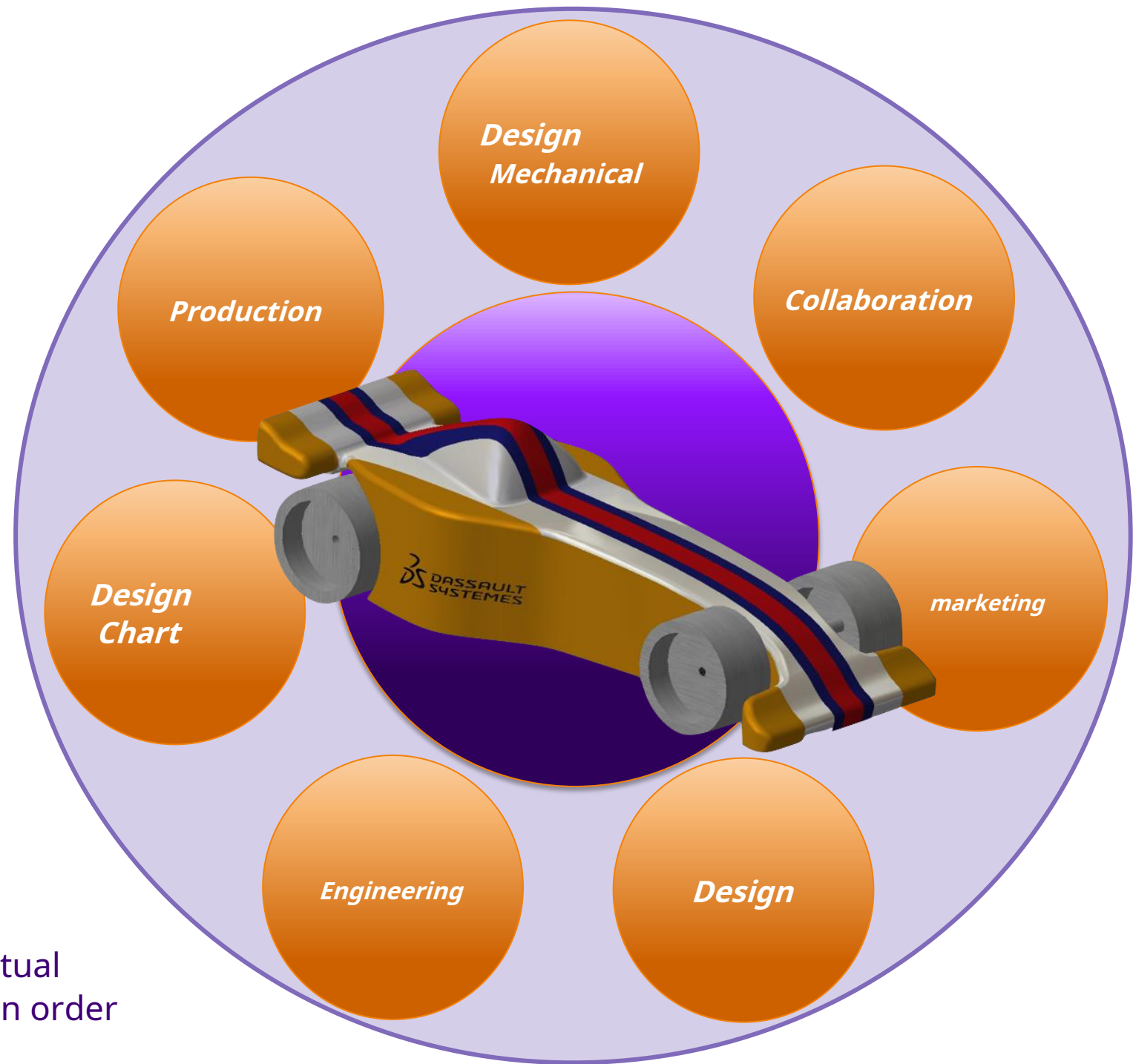
1. *Introduction.*
2. CATIA V5 QuickStart.
3. Your racing car.
4. Method sheets.

## One project, one team

- ▶ The goal of this project is to design a racing car and manufacture it while respecting the design constraints defined in the specifications.
- ▶ A team is made up of four to six students. They must manage the project on the basis of specifications and establish a marketing file to find sponsors and thus finance all the stages of the project. An oral presentation of the project is then made before a jury.
- ▶ This training will show you the effectiveness of CATIA V5 in responding to this type of challenge. Not just for design or manufacturing, but for all areas covered by this project.
- ▶ Each phase of the project requires specific skills:
  - Project management & collaboration,
  - Engineering,
  - Mechanical concept,
  - graphic arts,
  - Design,
  - Marketing,
  - Manufacturing.

# Skill mix

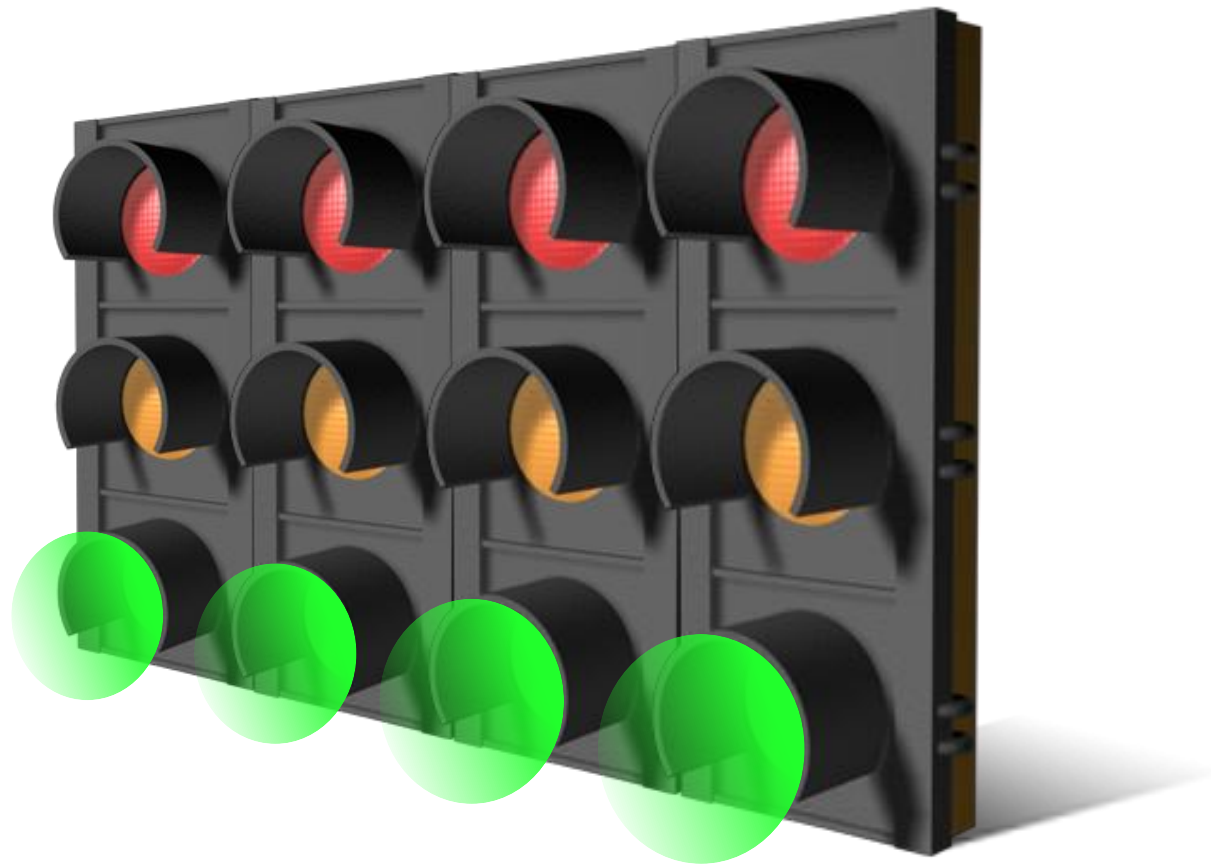
- ▶ Project Management & Collaboration
  - Four to six students per team.
  - Introduction to concurrent
- ▶ engineering. Engineering
  - Finding innovative solutions to improve performance.
- ▶ Mechanical concept
  - Model a virtual 3D model for production.
- ▶ Production
  - Create an NC\* program.
  - Simulate machining.
- ▶ Graphic Arts
  - Define the graphic identity of your team.
- ▶ Design
  - Create stylish shapes in line with good aerodynamic performance.
- ▶ Marketing
  - Establish a portfolio and use the virtual model as communication material in order to canvass sponsors.





# Welcome to CATIA V5

Quick familiarization with CATIA V5 software.



Here are the steps to follow:

1. Introduction.

**2. CATIA V5 QuickStart.**

has. Welcome to CATIA V5.

b. Presentation of the CATIA Interface.

vs. Create a part.

d. Mouse manipulation.

e. The different views.

f. Viewing modes.

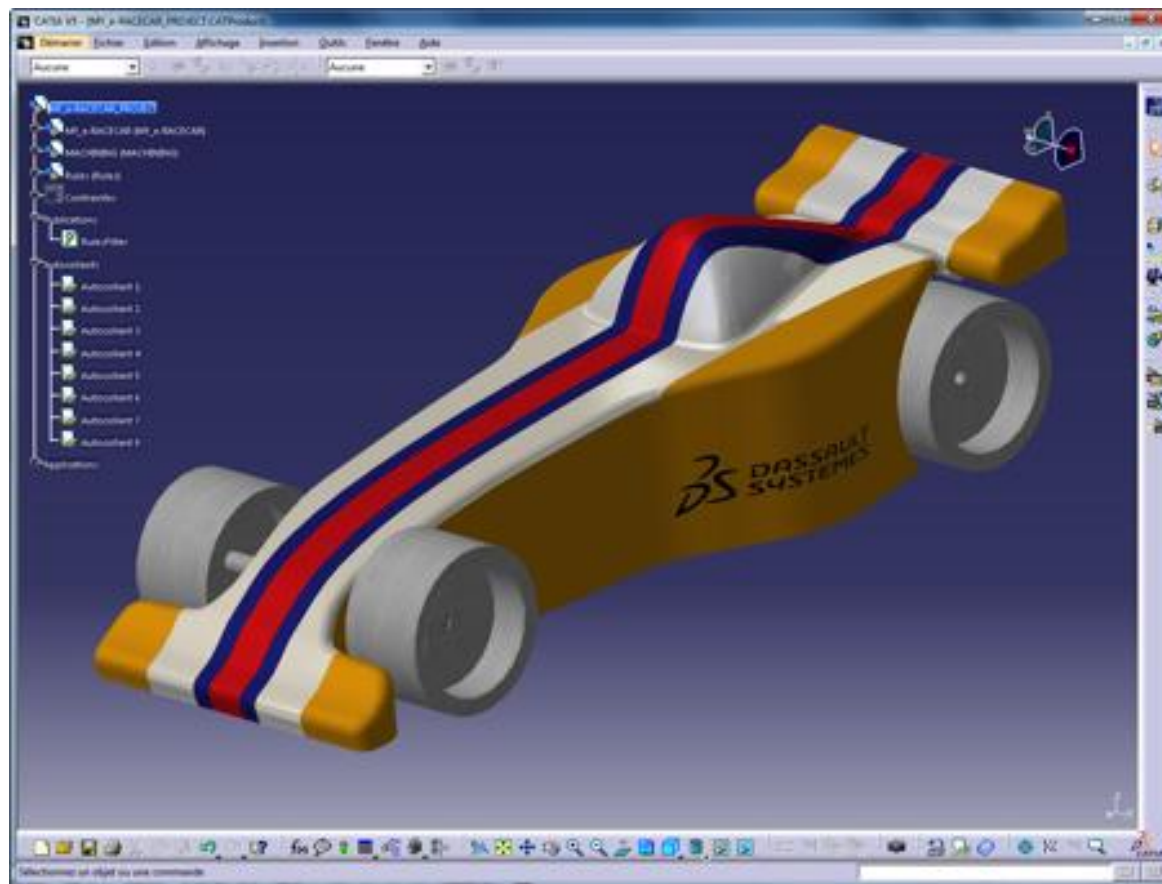
g. Tips and tricks.

3. Your racing car.

4. Method sheets.

# Welcome to CATIA V5

In this step you will discover the possibilities of CATIA V5.



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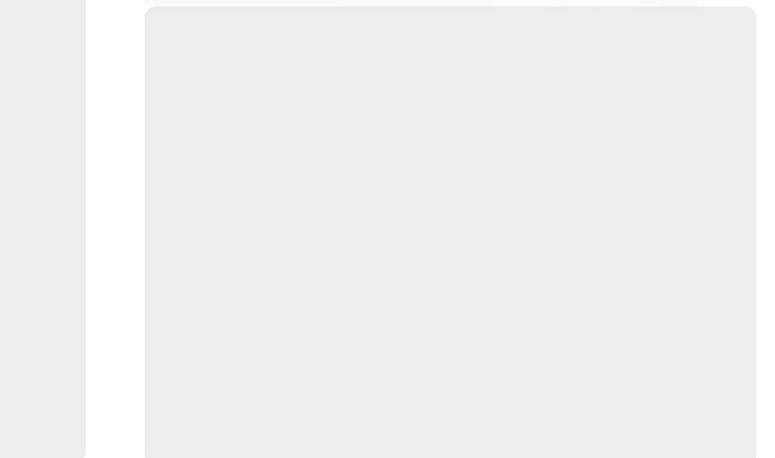
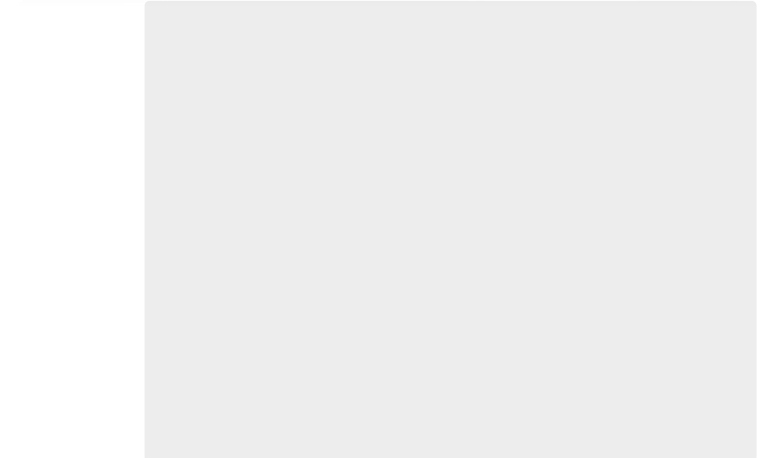
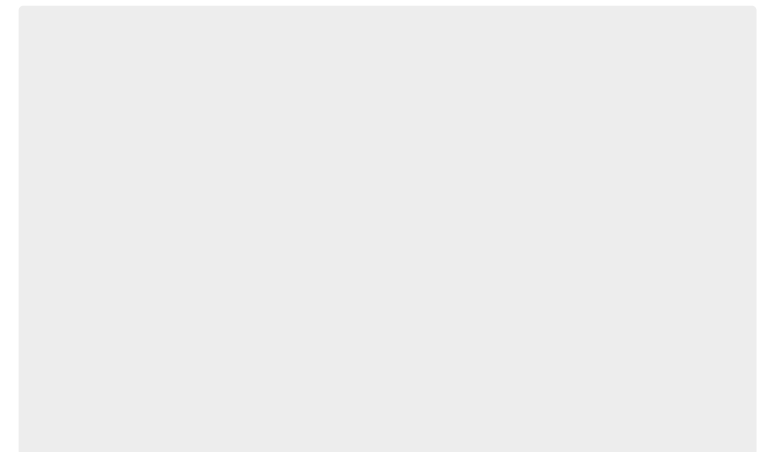
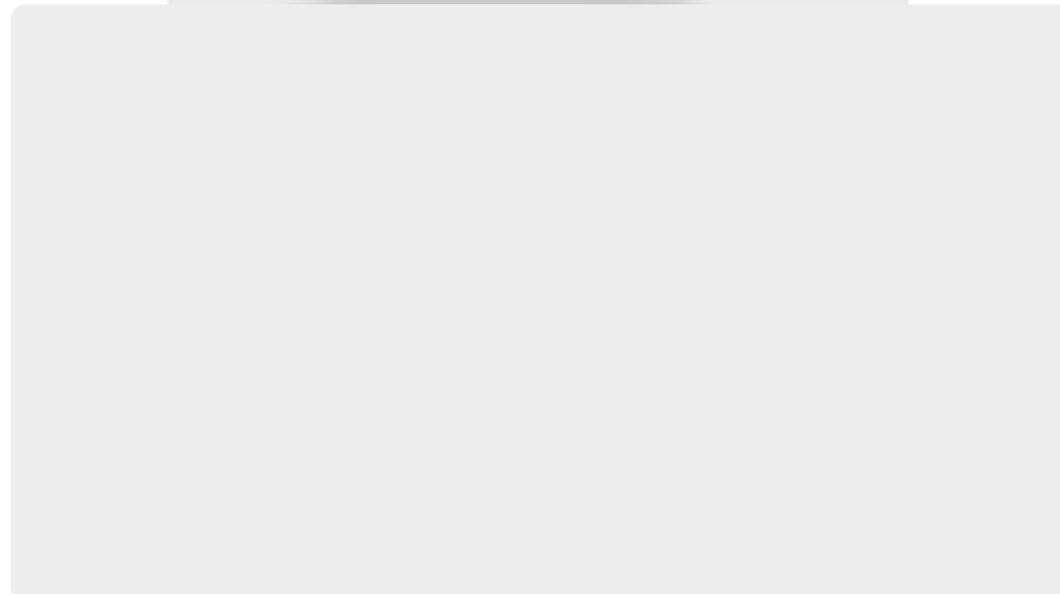
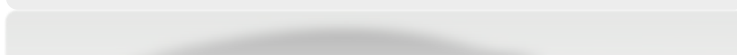
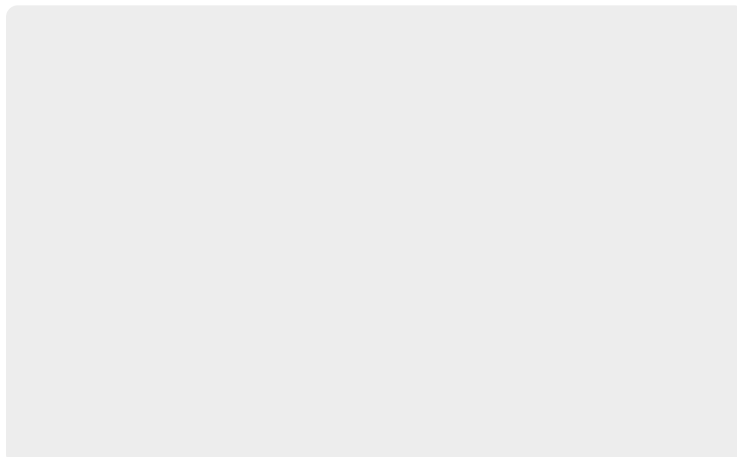
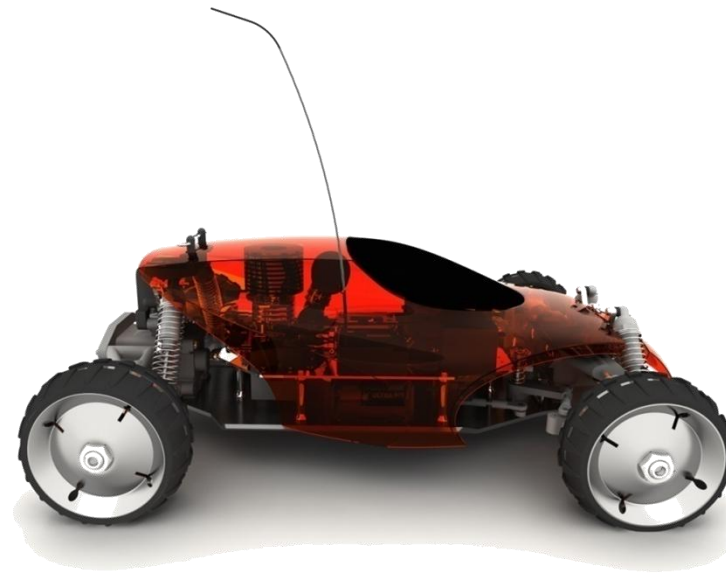
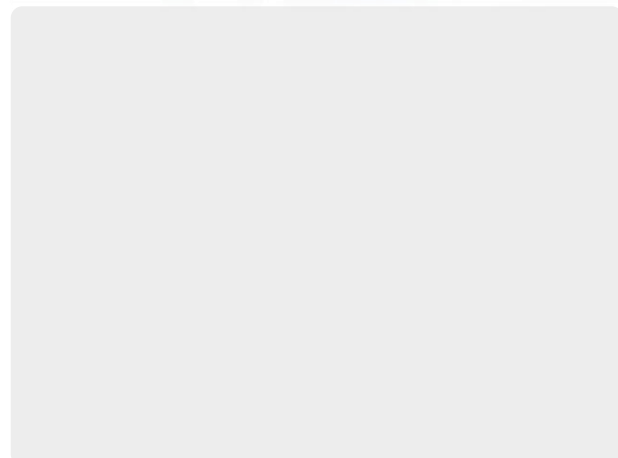
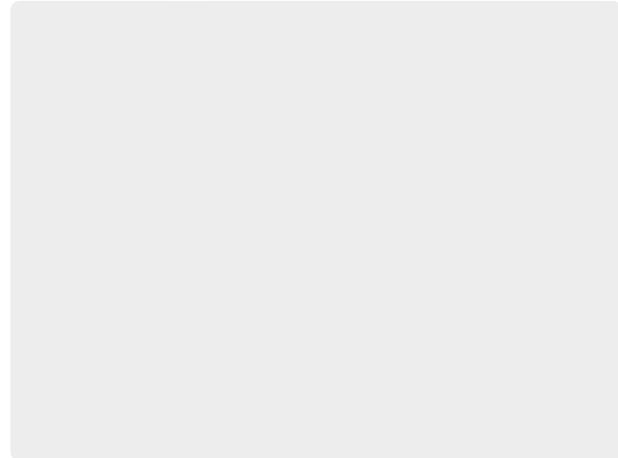
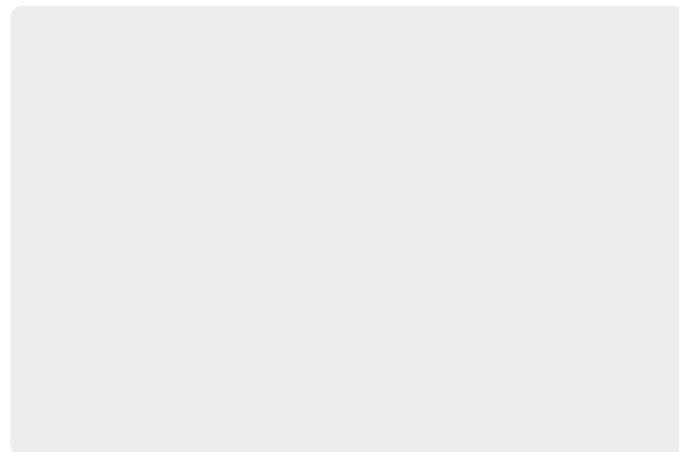
f. Viewing modes.

g. Tips and tricks.

3. Your racing car.

4. Method sheets.

What do these objects have in common?

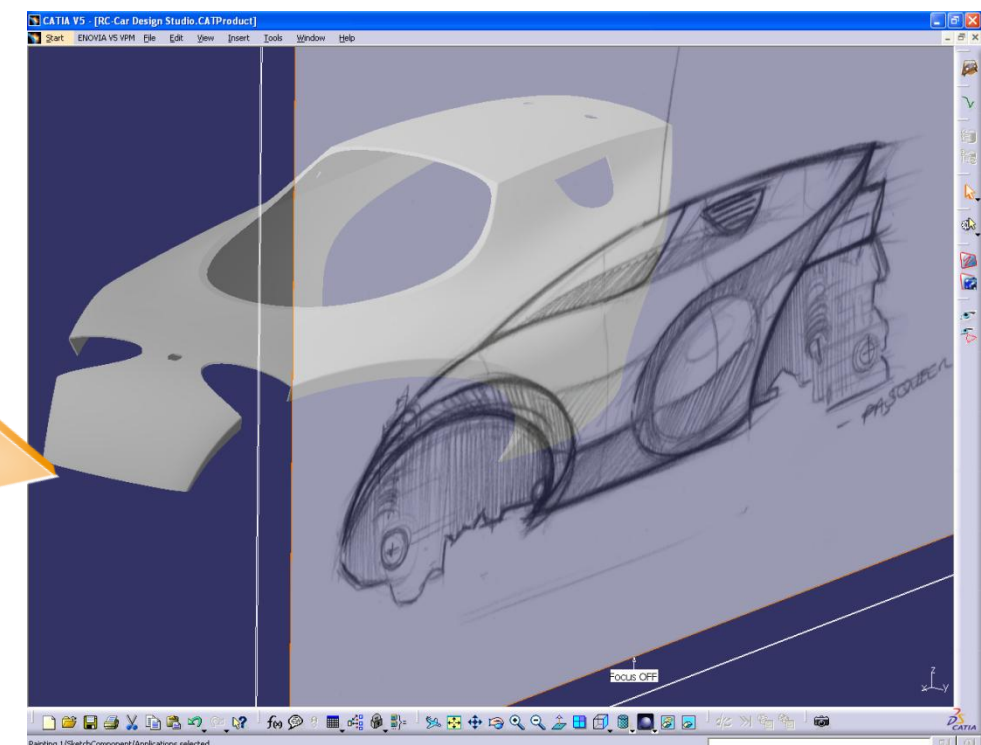
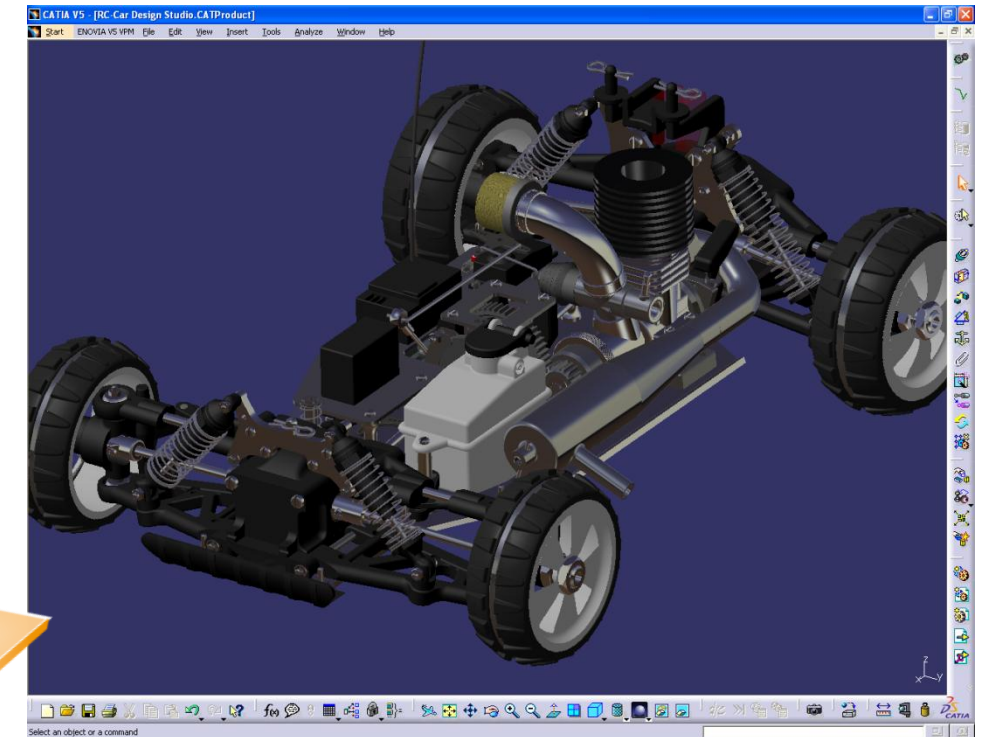
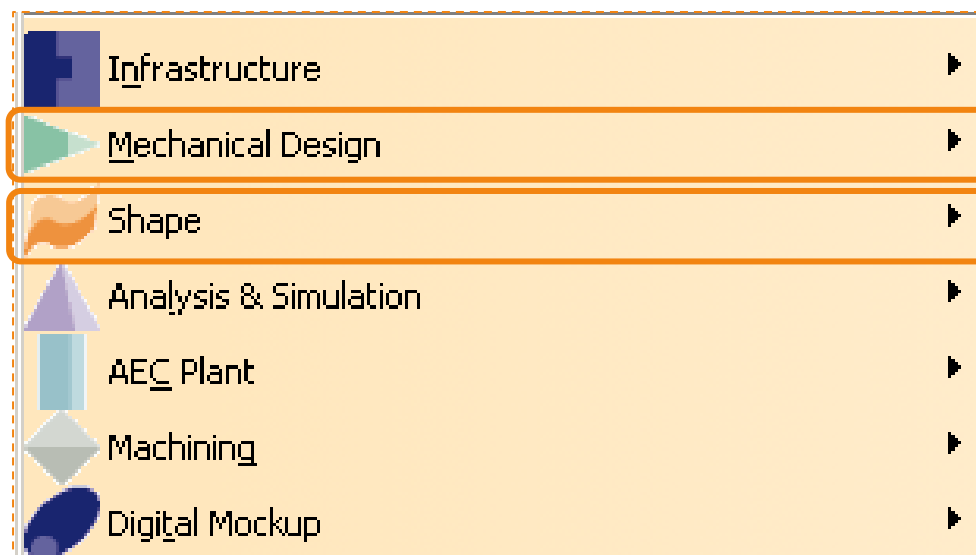


# CATIA V5

- ▶ All of these objects were imagined, designed, modeled and manufactured using 3D CATIA software!
- ▶ CATIA (Applied Interactive Three-Dimensional Aided Design) is a design software computer-aided (CAD) which can model all sorts of objects in 3D.
- ▶ CATIA alone integrates more than 130 workshop Each of these modules corresponds to a precise u in the industry.

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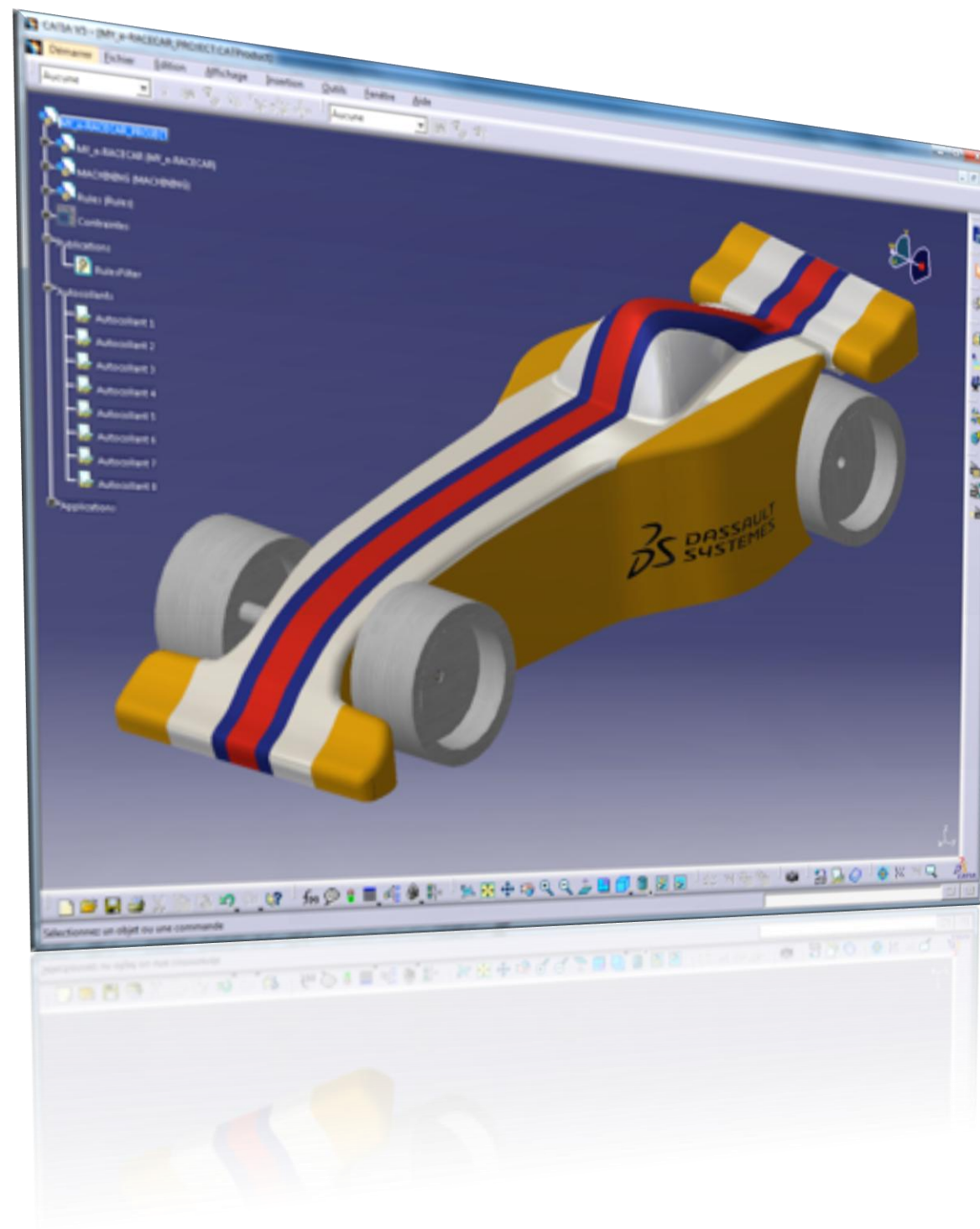
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# CATIA Interface Overview

In this step you will discover the interface of CATIA V5.



Here are the steps to follow:

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## CATIA V5 interface

1 Toolbar.

2 Specification tree.

3 Compass.

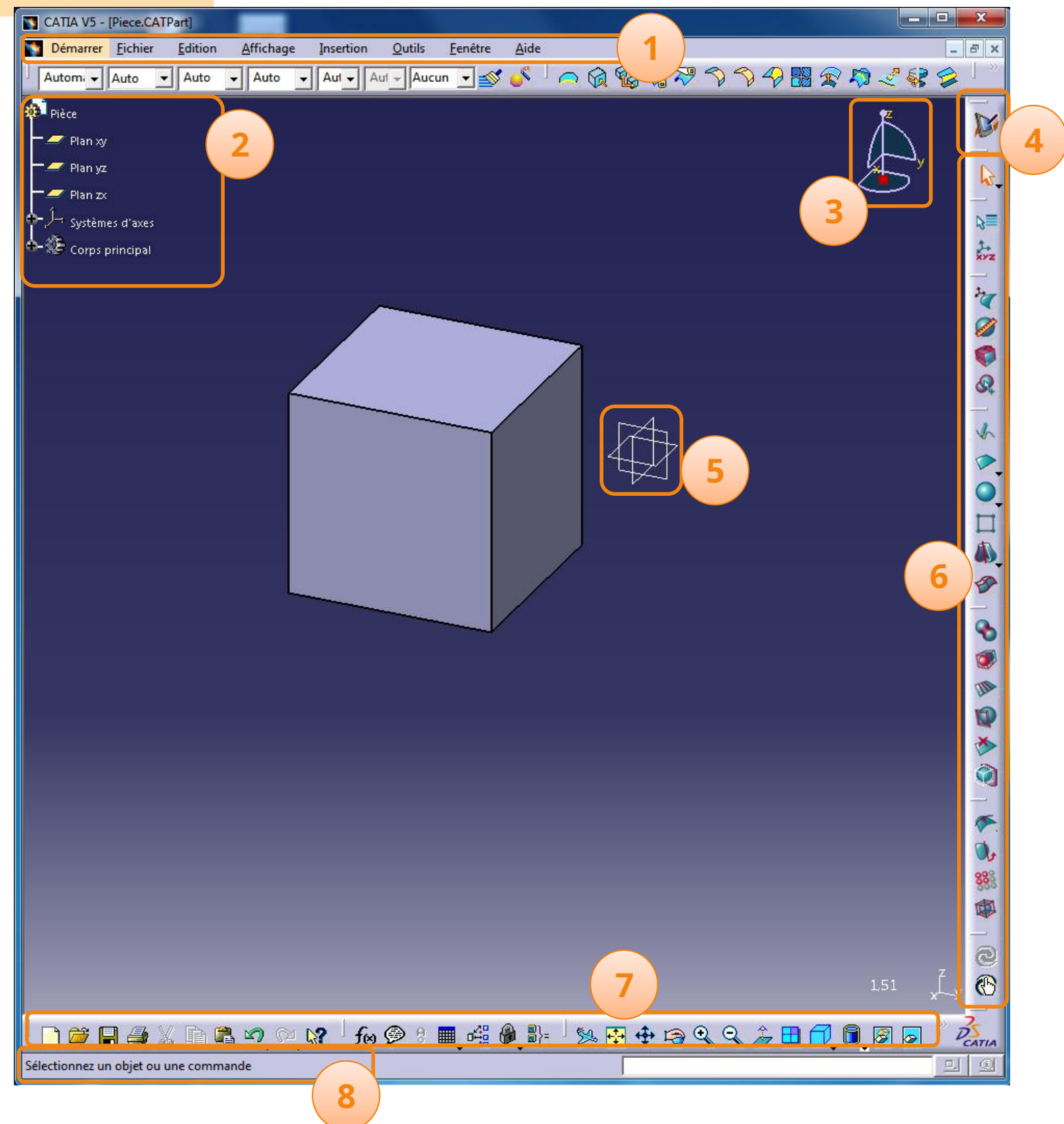
4 Workshop in progress icon.

5 Default planes.

6 Workshop functions.

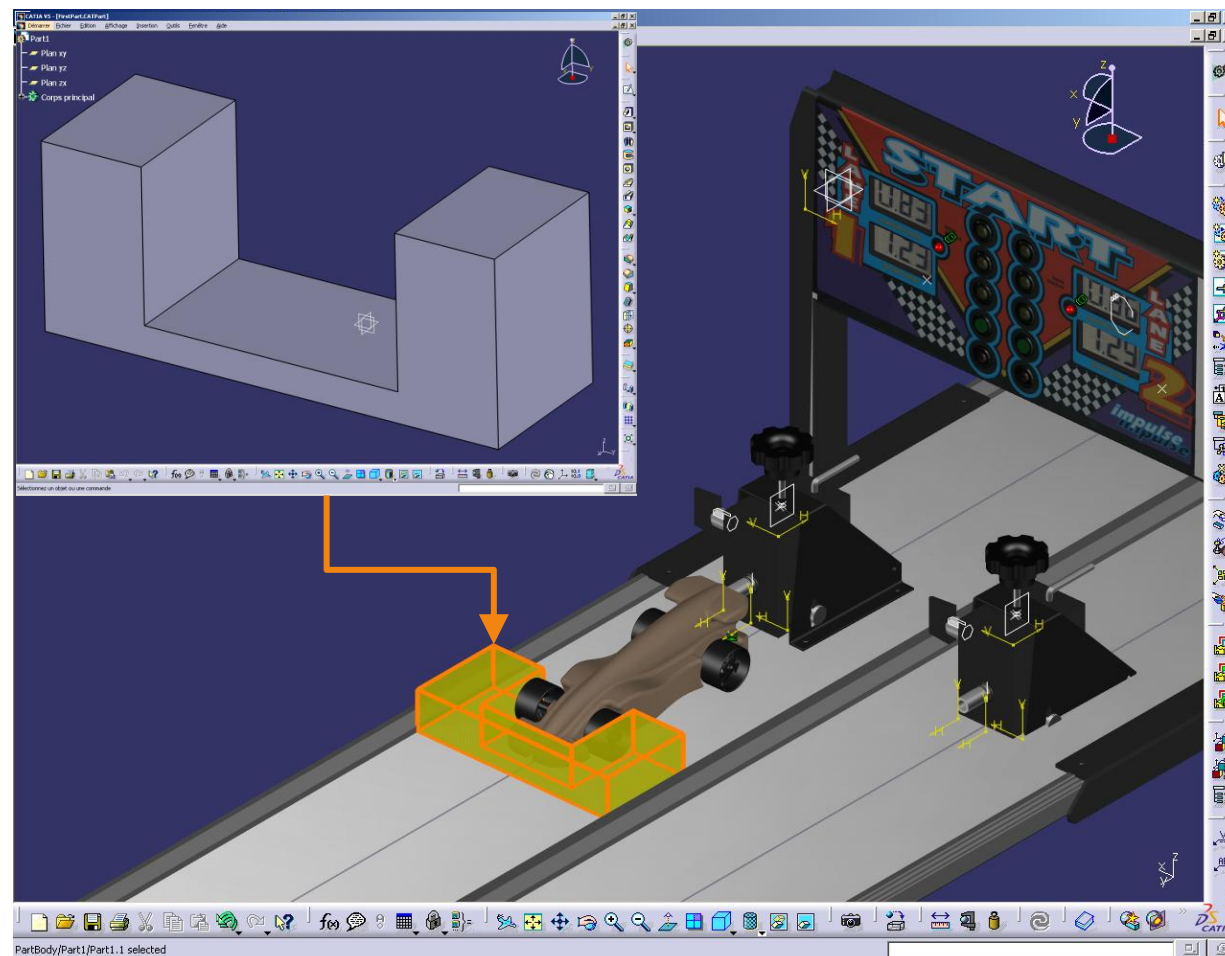
7 Toolbar.

8 Contextual help.



# Create a room

In this step you will create your first part with CATIA V5. We want to model a wedge in order to position a model car on a track.



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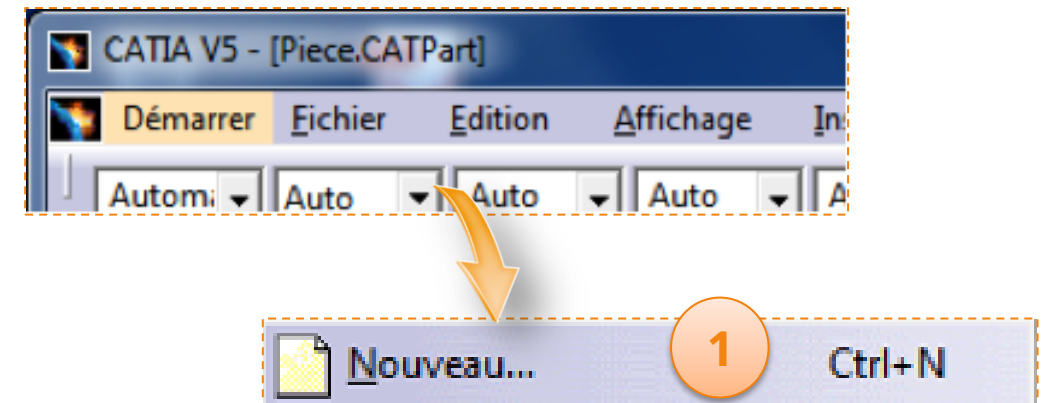
4. Method sheets.



# Create a room

We are going to create a new room. For that :

1. Click on **File** > **New...**

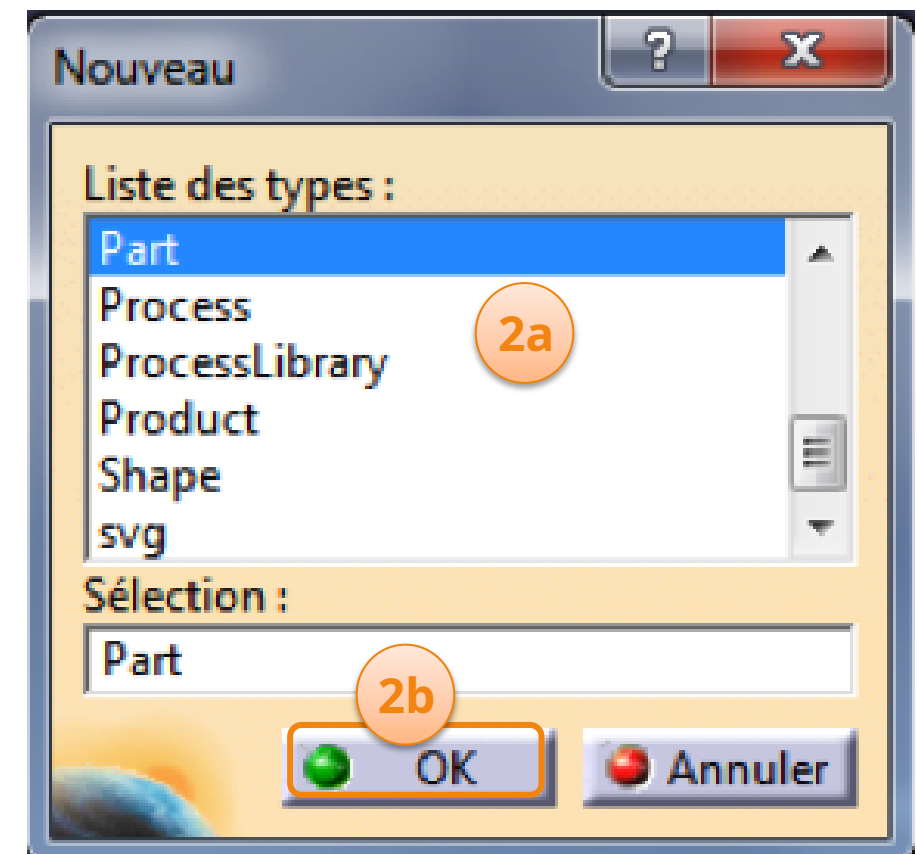
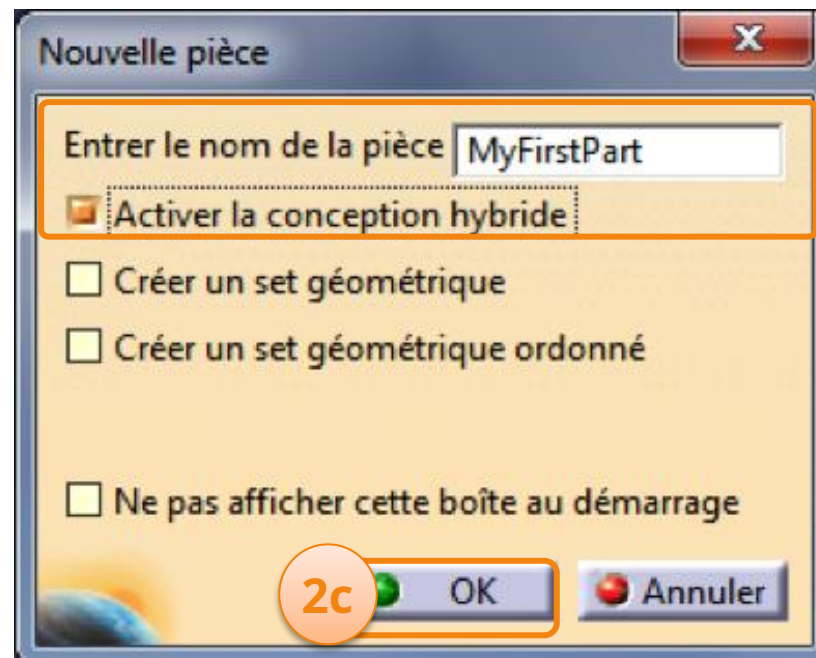


2. To create a new part:

has. Select **Go**.

b. Click on **okay**.

vs. Use the settings below and click **okay**.

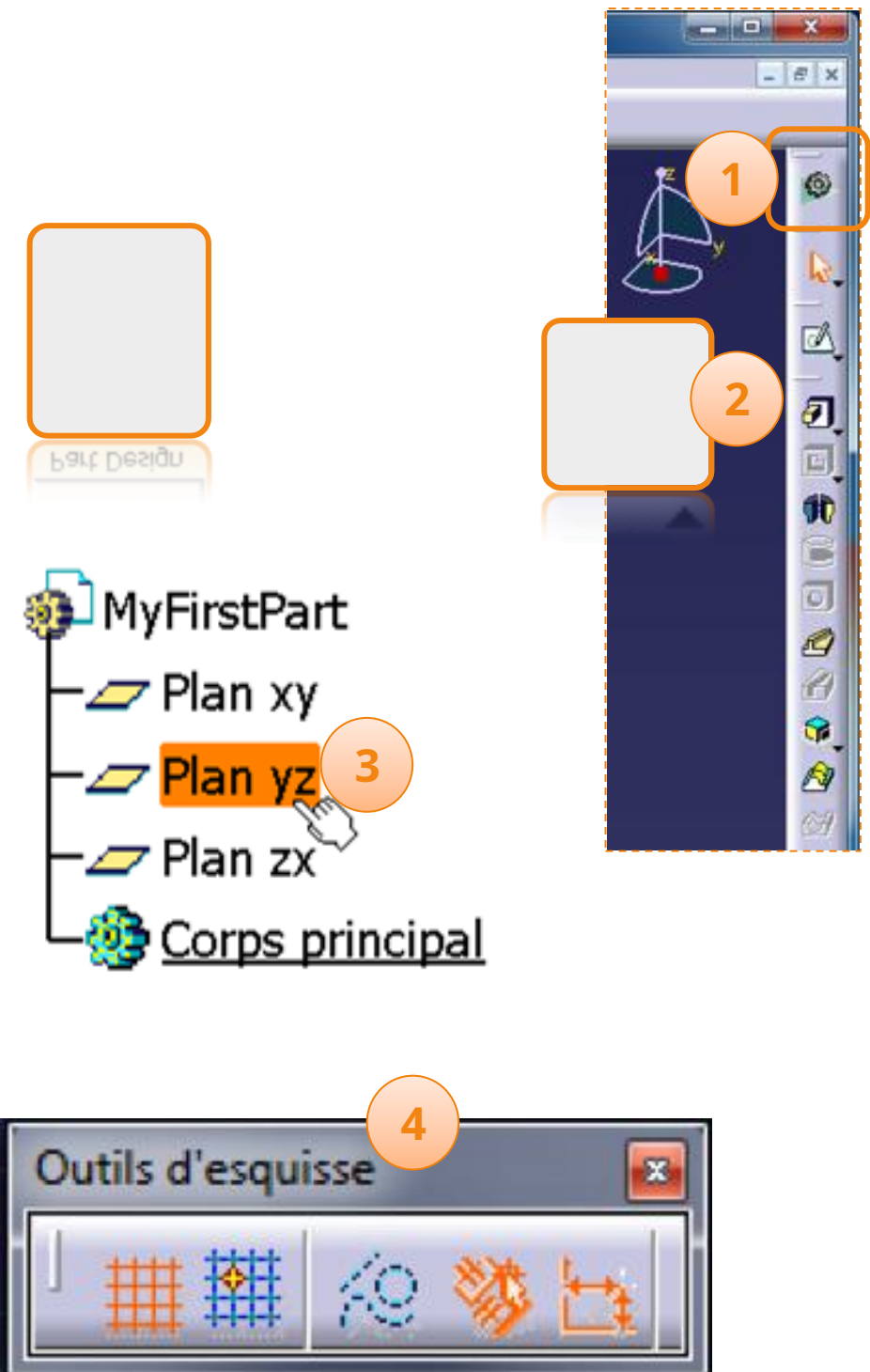


# Create a room

First, we will draw the outline of the part in a sketch.  
Then, this sketch will be used to generate a 3D volume.  
For that :

1. Thanks to the active workbench icon, check that you are in the Part Design workbench. If not, click **Start > Mechanical Design > Part Design**.
2. Click the icon **Sketch**, which is located at the top right of the screen.
3. Click on the **YZ Plan** to define the sketch plane.

4. In the Sketch Tools window that appeared, make sure to use the same settings as shown here. If an icon is a different color, click it.



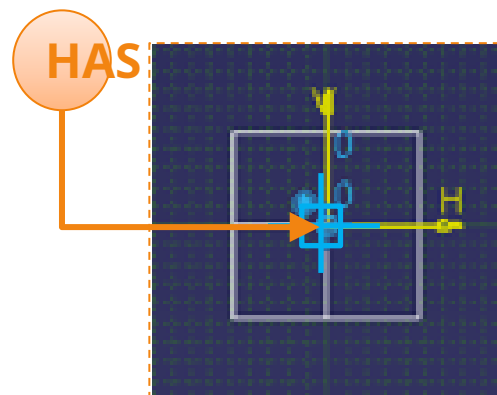
# Create a room

To trace the profile, we will use the contour tool. For that :

1. Click the icon **Outline** which is on the right side of your screen.

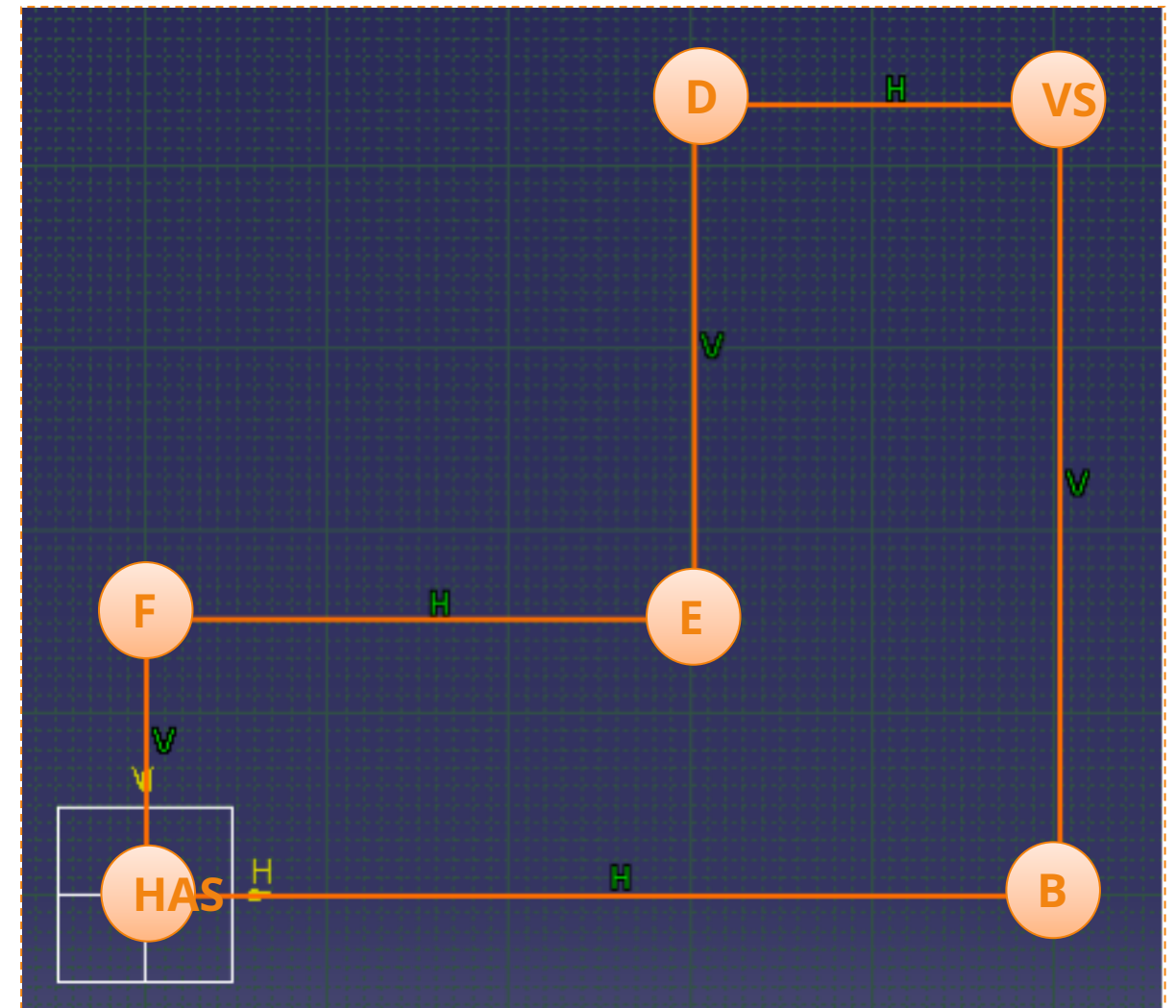
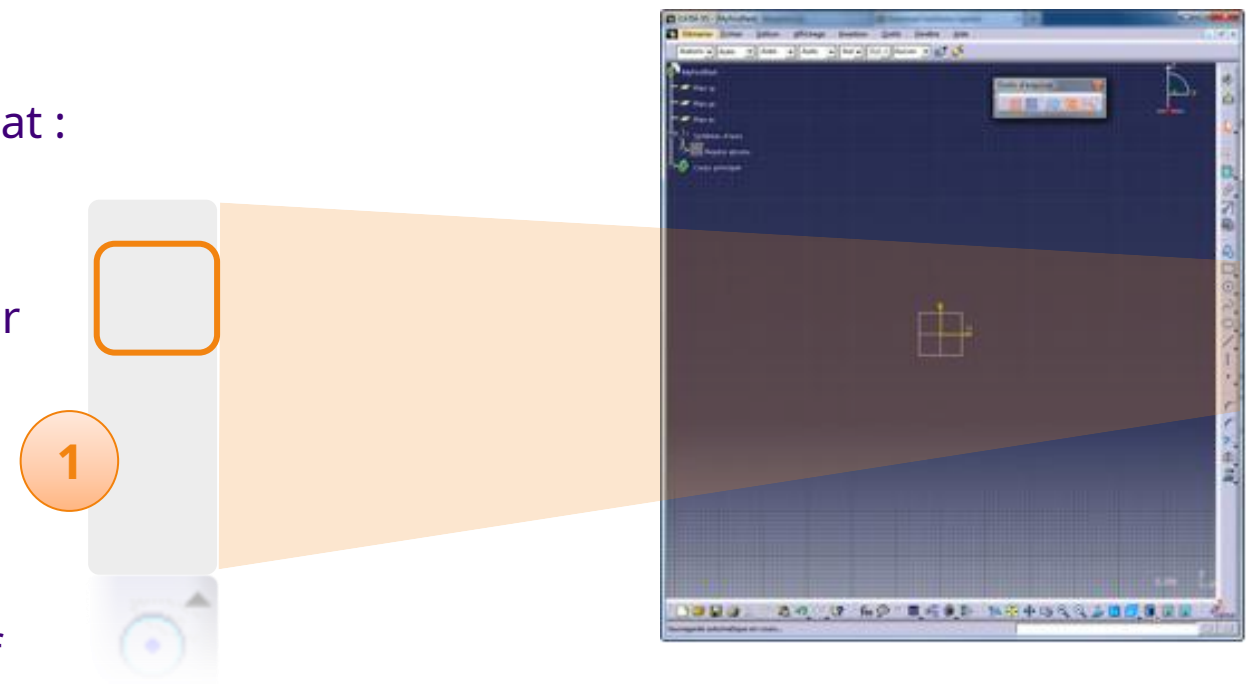
We will draw the outline opposite.

2. Single click on the point **HAS**, which is at the center of the frame.



3. Then draw **approximately** the rest by clicking on the points of **B to F**. Warning ! Between each point, arrange so that the segment is blue.

4. Click again on the point **HAS**, to close the outline.



# Create a room

We are now going to dimension some segments of the sketch. For that :

1. Click the icon **Constraint**.
2. Click on the **G-segment**.
3. Move away from the segment then click to place the dimension.
4. Double-click the dimension value to edit it.
5. Enter the value [56.5mm].
6. Click on **okay**.

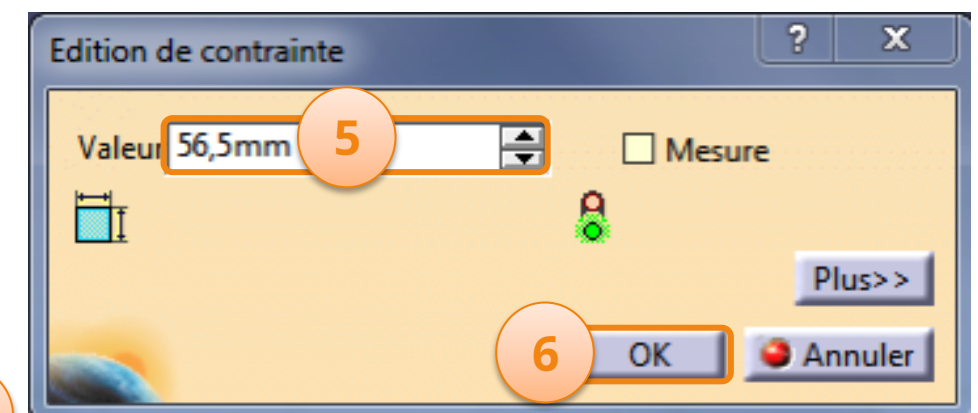
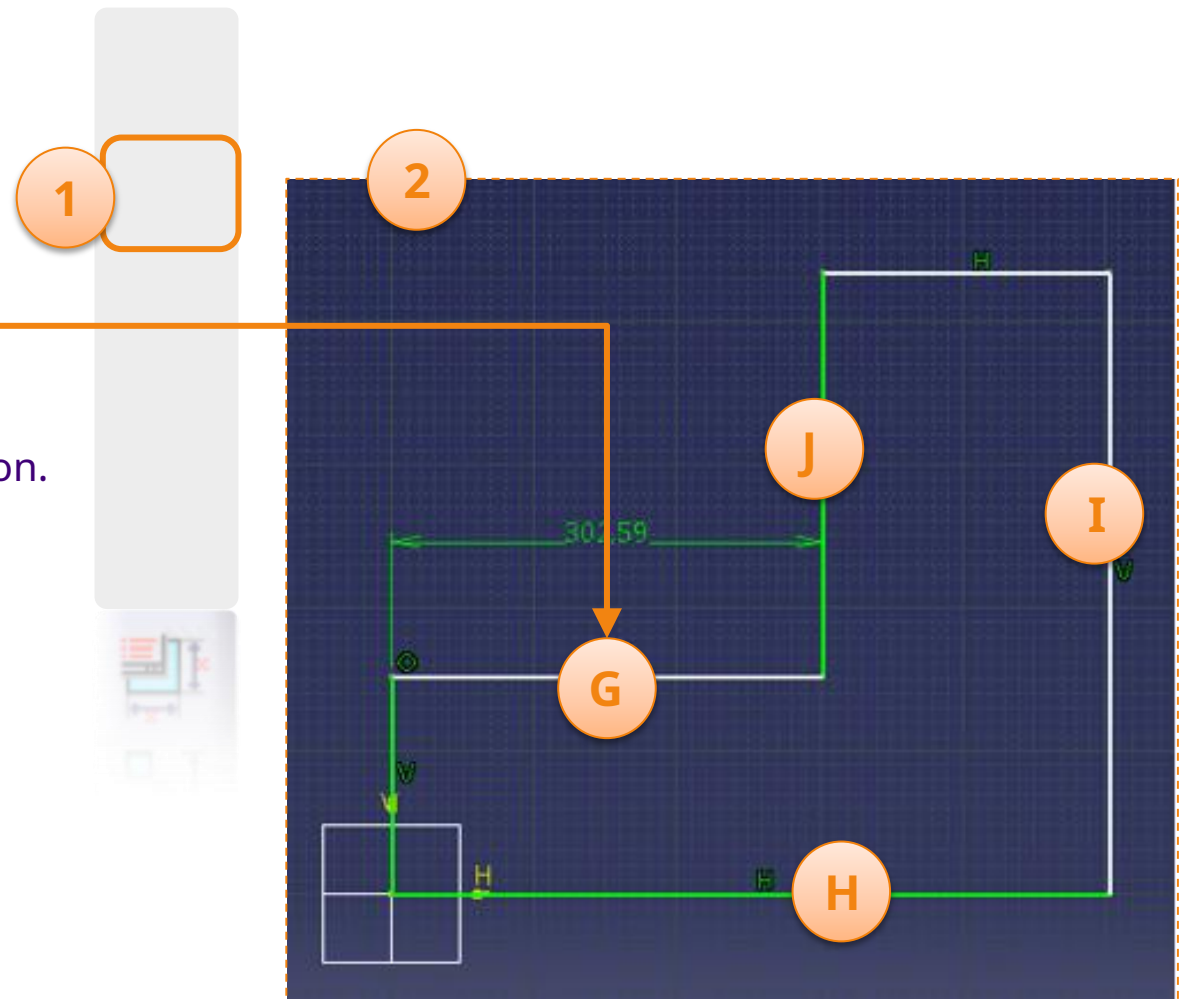
7. Constrain in the same way (from step 1) **the segments from H to J**, using these values:

has.H: [76mm].

b.I: [60mm].

vs.Y: [50mm].

8. Click the icon **Leaving the workshop** located at the top right of the screen.

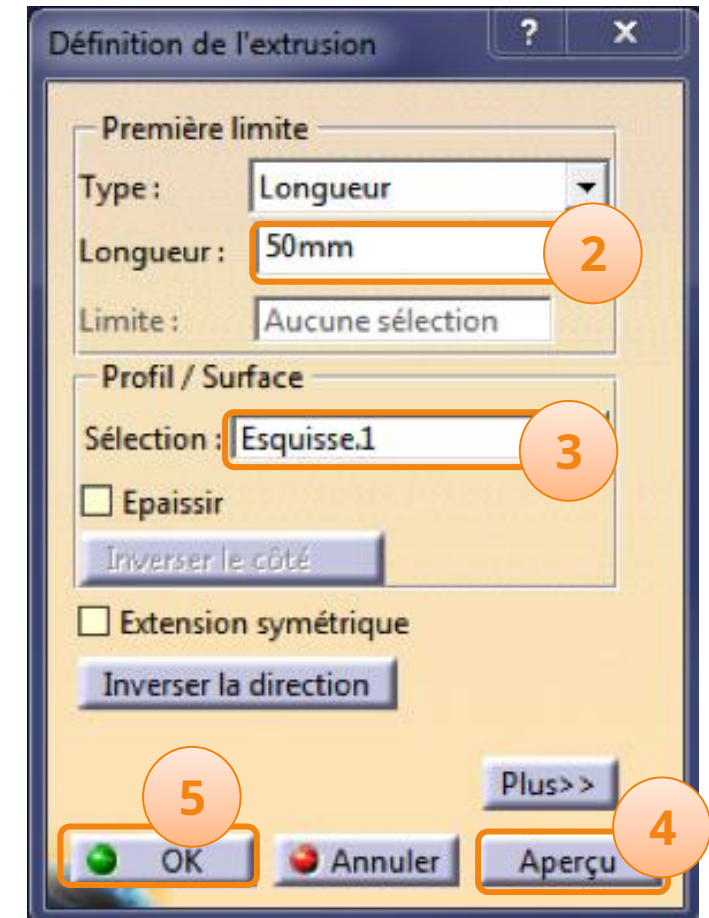




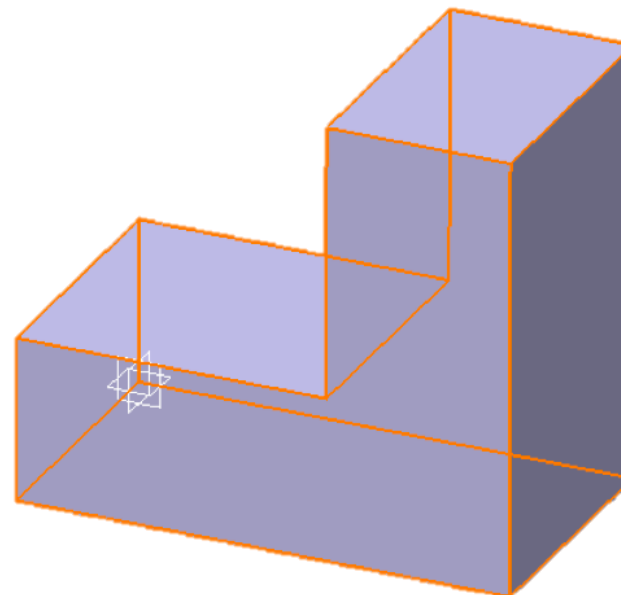
## Create a room

We are now going to create a volume from the sketch created previously. For that :

1. Click the icon **Extrusion**, to open the extrusion definition window.
2. Enter the value [50mm] in the Length field.
3. Verify that Sketch.1 is selected as the profile.
4. Click on **Insight**: this allows you to verify your operation.
5. Click on **okay** to validate.



Here is the result you should get:

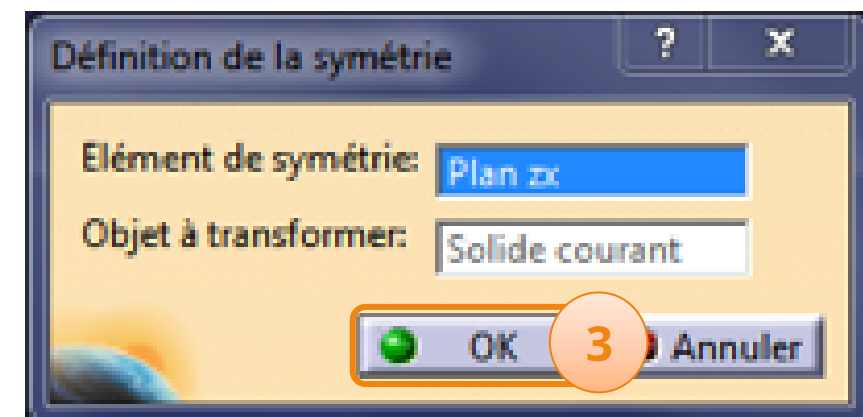
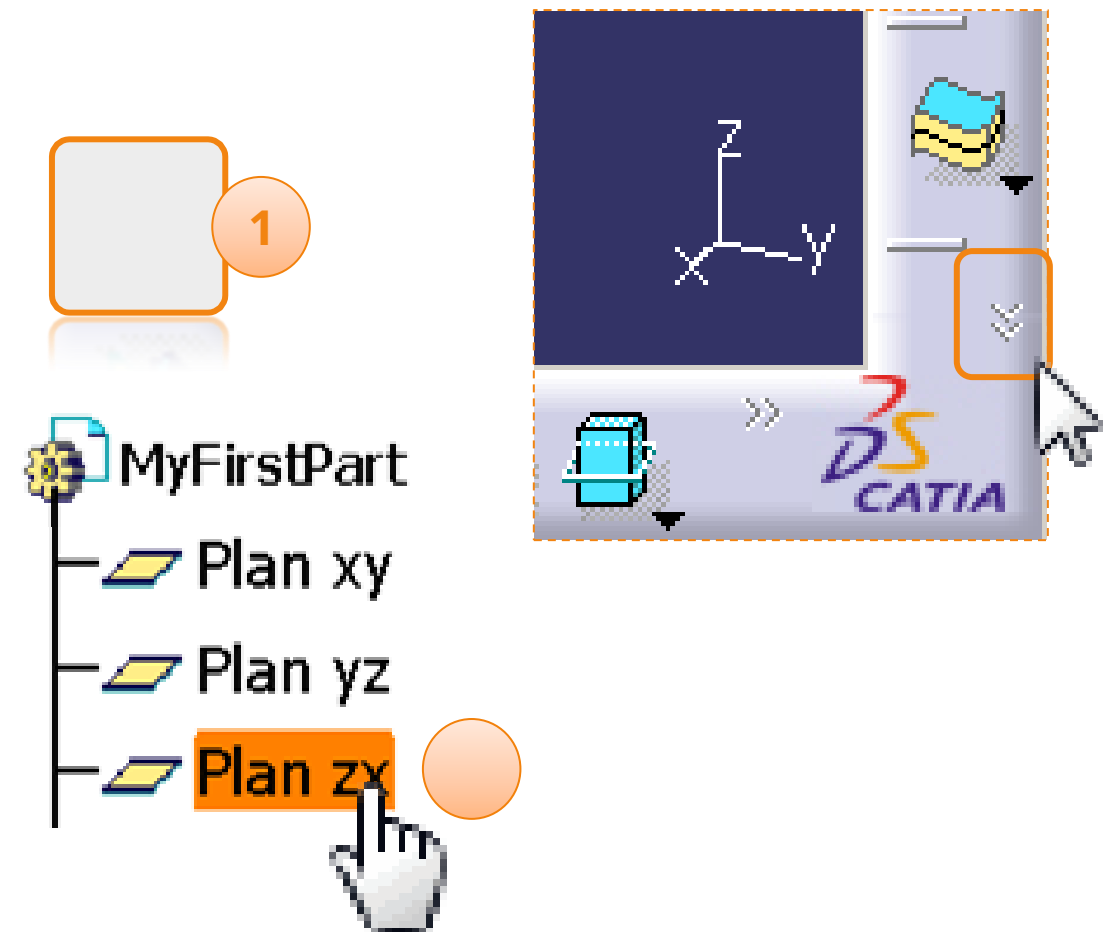
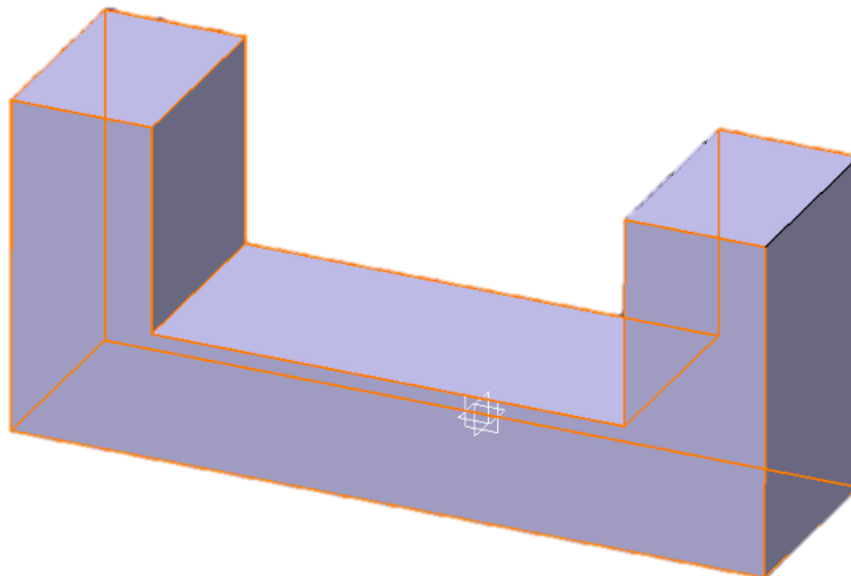


# Create a room

We are going to finalize the piece by performing a symmetry.  
For that :

1. Click the icon **Symmetry** located at the bottom right of your screen. Depending on the size of the latter, the icon may be hidden. To make it appear hold the click on the symbol **>>** as opposite to move the hidden toolbars.
2. Click on the **ZX Map** to define the plane of symmetry.
3. Click on **okay**.

You should get the result below:



# Create a room

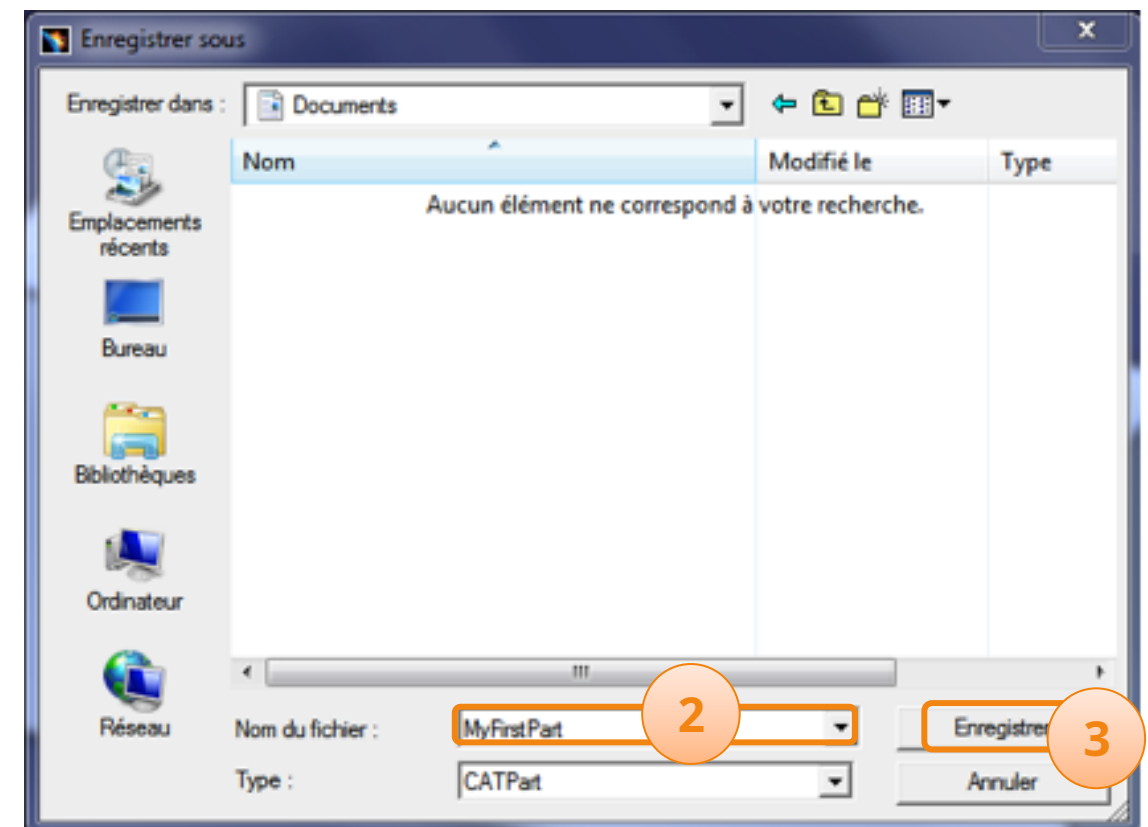
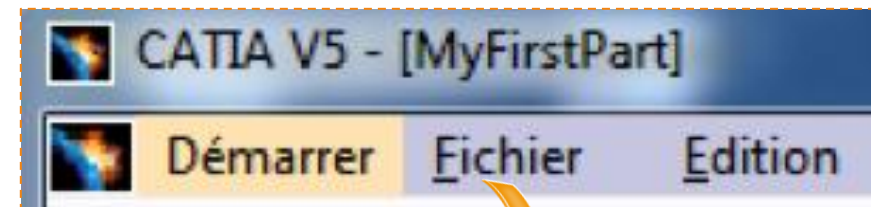
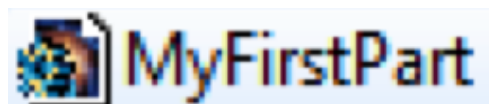
All that remains is to save your work. For that :

1. Click on **File > Save**.

2. Select the directory of your choice and enter the file name [MyFirstPart].

3. Click on the button **SAVE**.

Cheer ! You have just created your first CATPart file.





# Mouse manipulation

In this step you will discover how to manipulate the 3D environment of CATIA V5.

DÉPLACER

TOURNER

ZOOMER



Here are the steps to follow:

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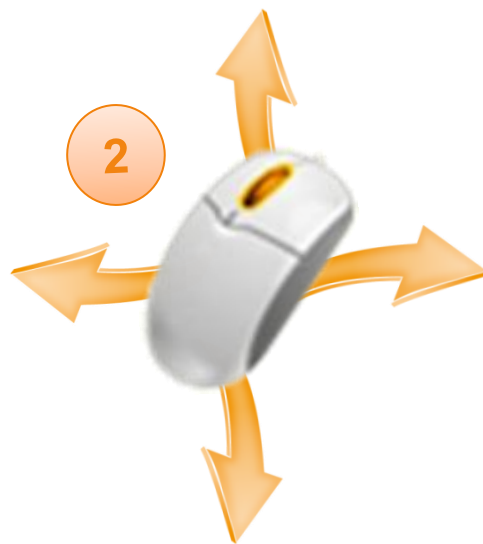
# Mouse manipulation

► To move the view you must:

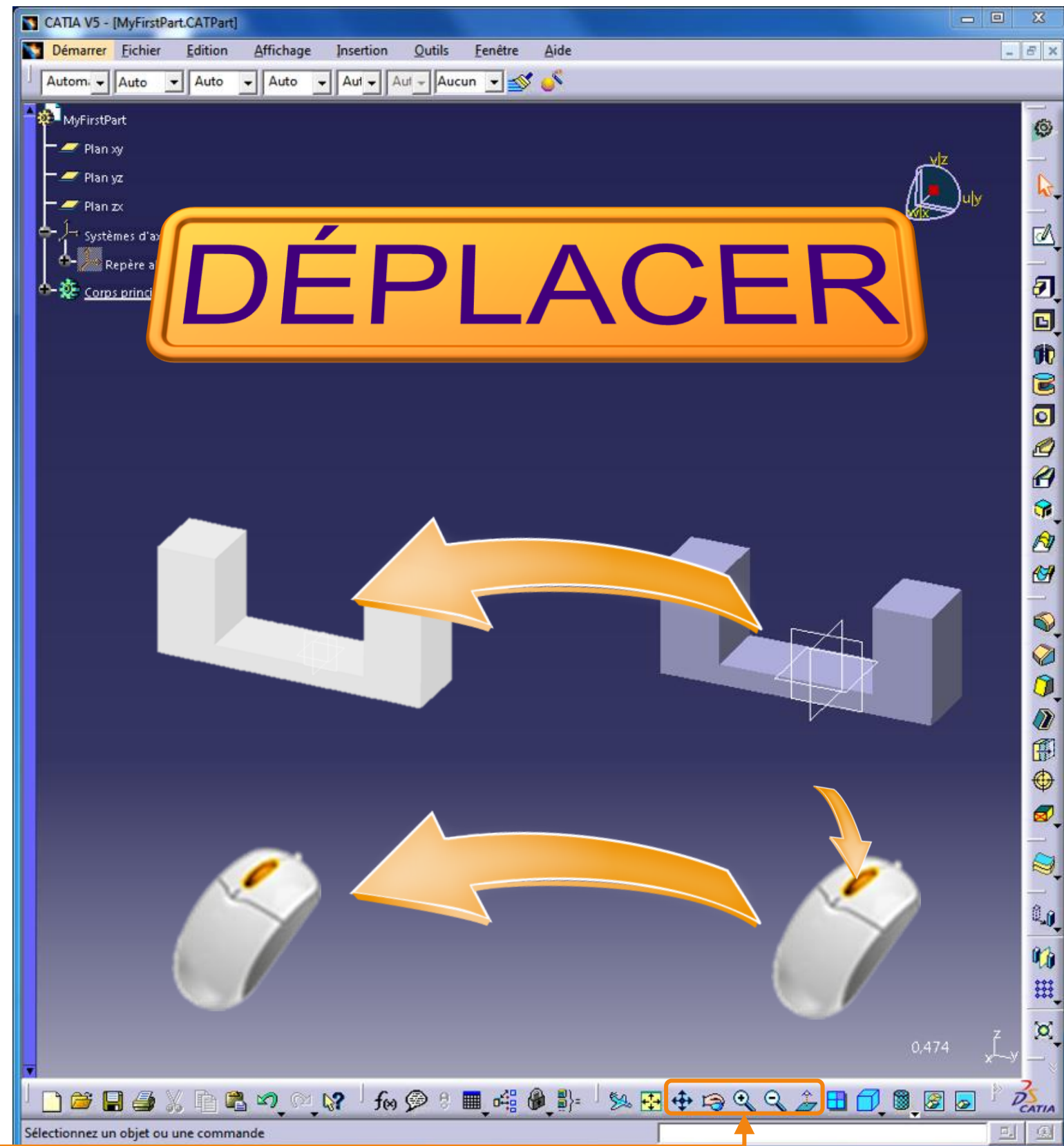
1. Hold down the wheel click.



2. Then, while holding down the scroll wheel, move your mouse to move the view.



► You can also use the icon **To move**.



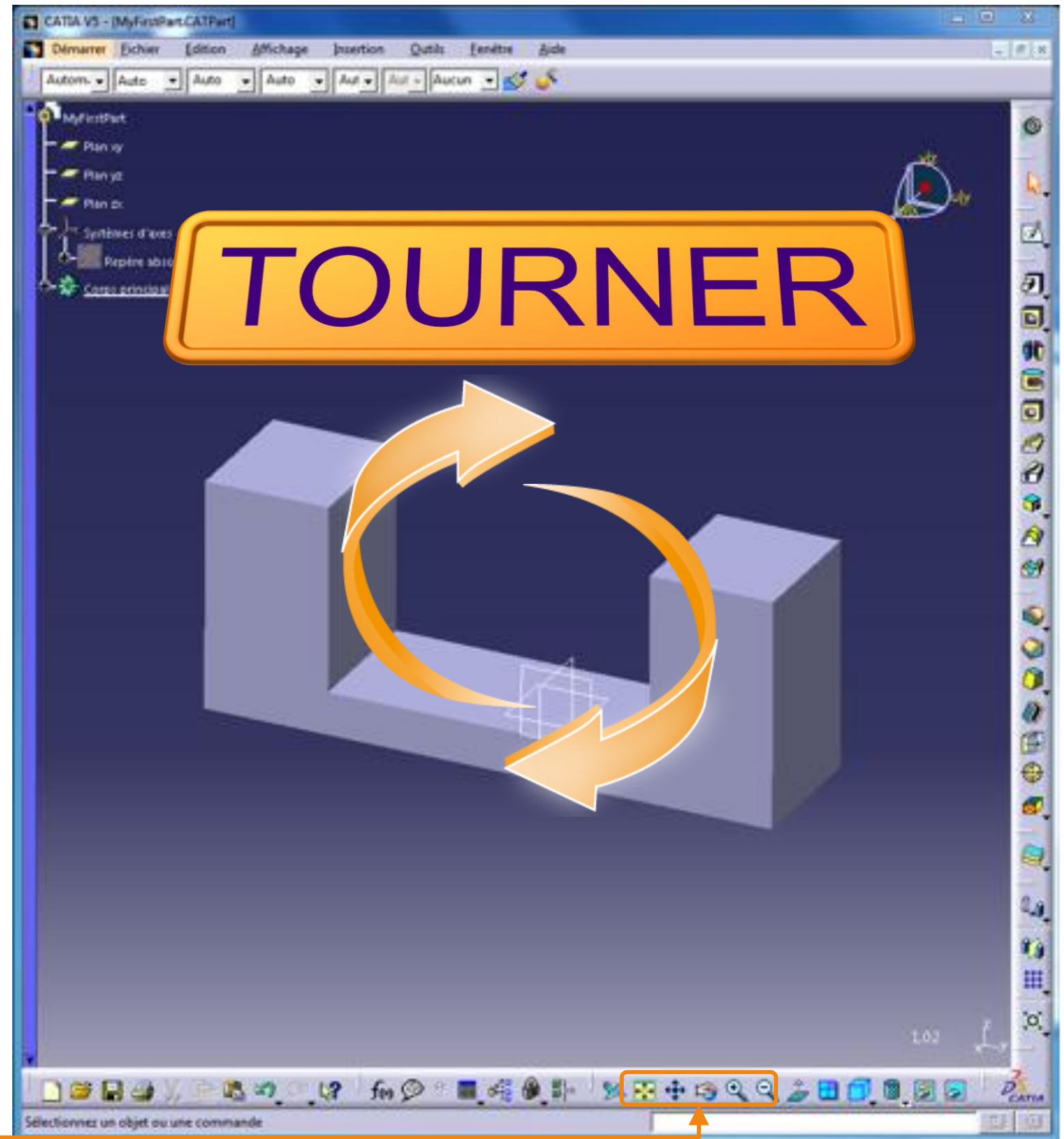
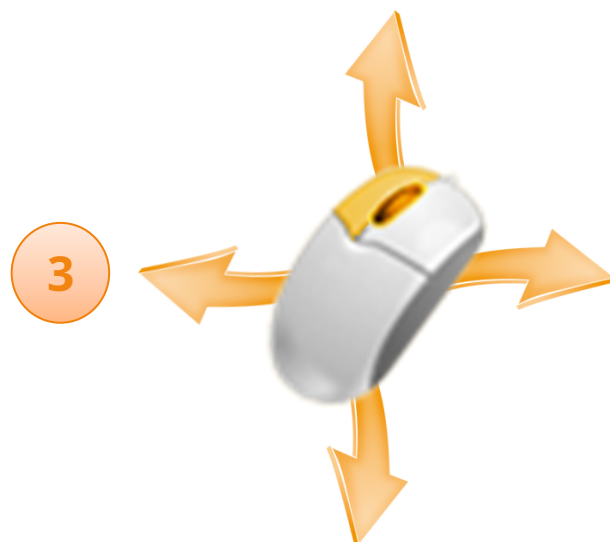
# Mouse manipulation

► To perform a rotation you must:

1. Hold down the wheel click.

2. Then, while holding down the scroll wheel, hold down the left mouse button.

3. With both buttons still pressed, move the mouse to rotate the view.

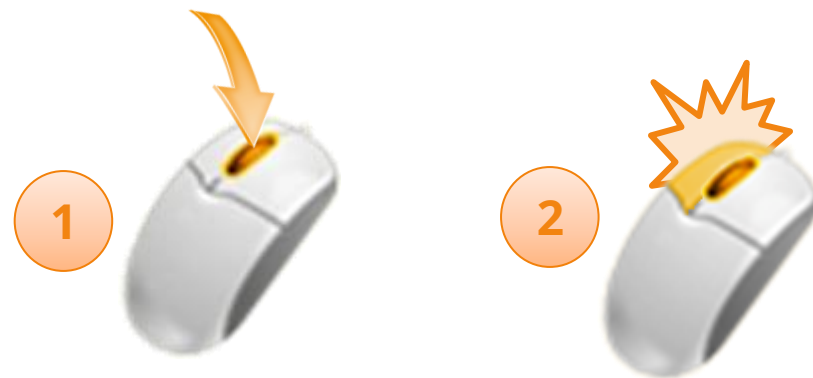


# Mouse manipulation

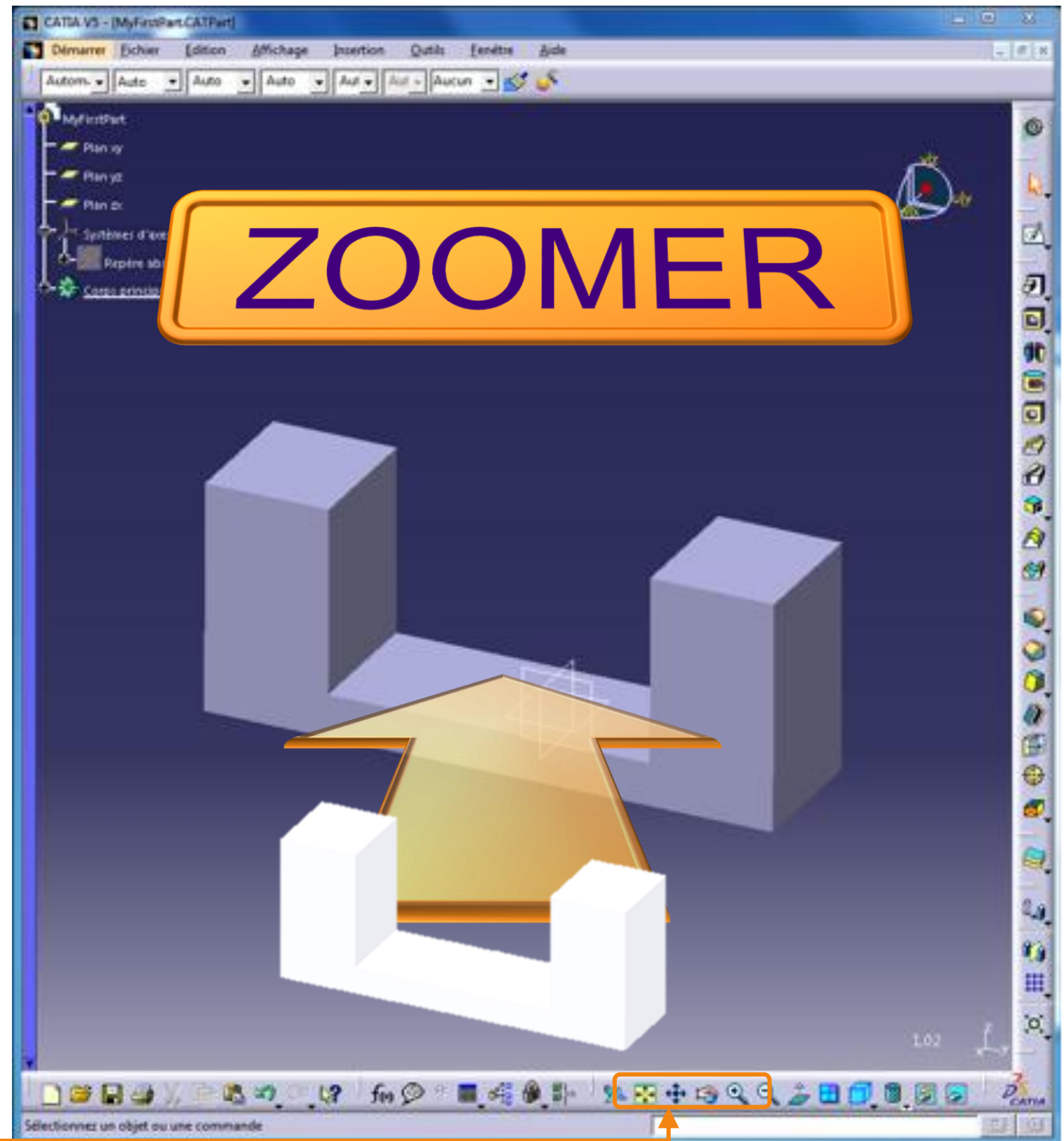
► To zoom you must:

1. Hold down the wheel click.

2. While holding down the scroll wheel, click (without holding down) the left mouse button.



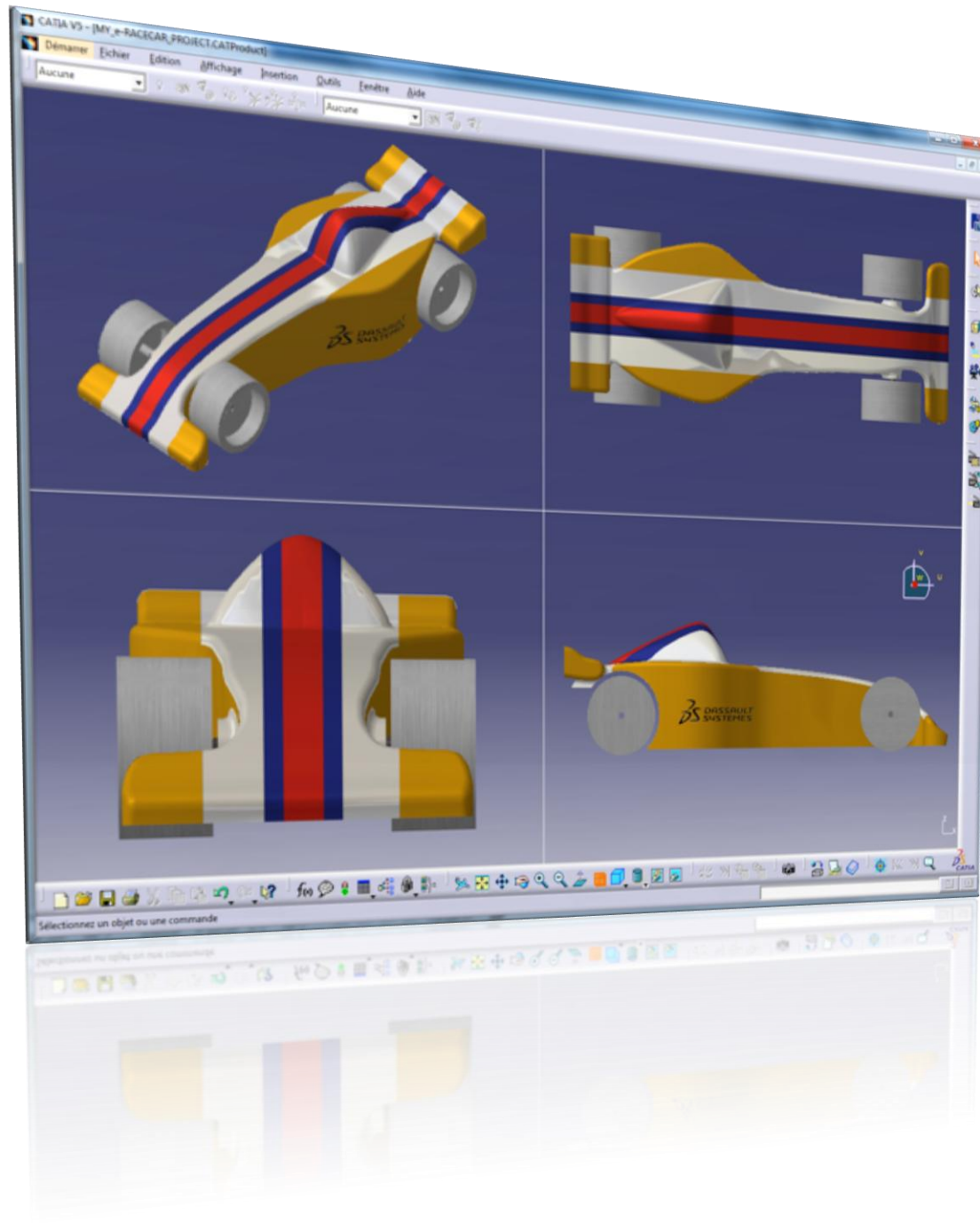
3. Without releasing the scroll wheel, move the mouse forward or backward to zoom in or out.





# The different views

In this step you will learn how to use the predefined views.



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f. Viewing modes.

g. Tips and tricks.

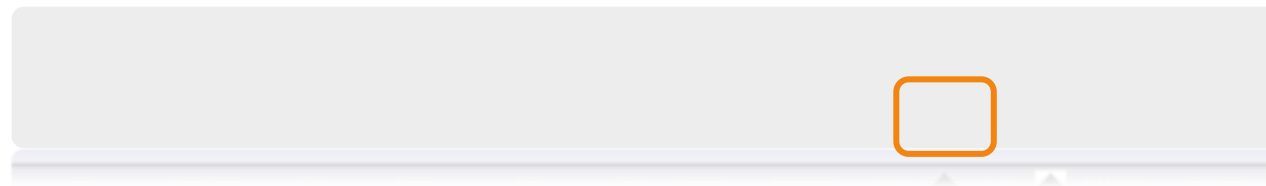
3. Your racing car.

4. Method sheets.

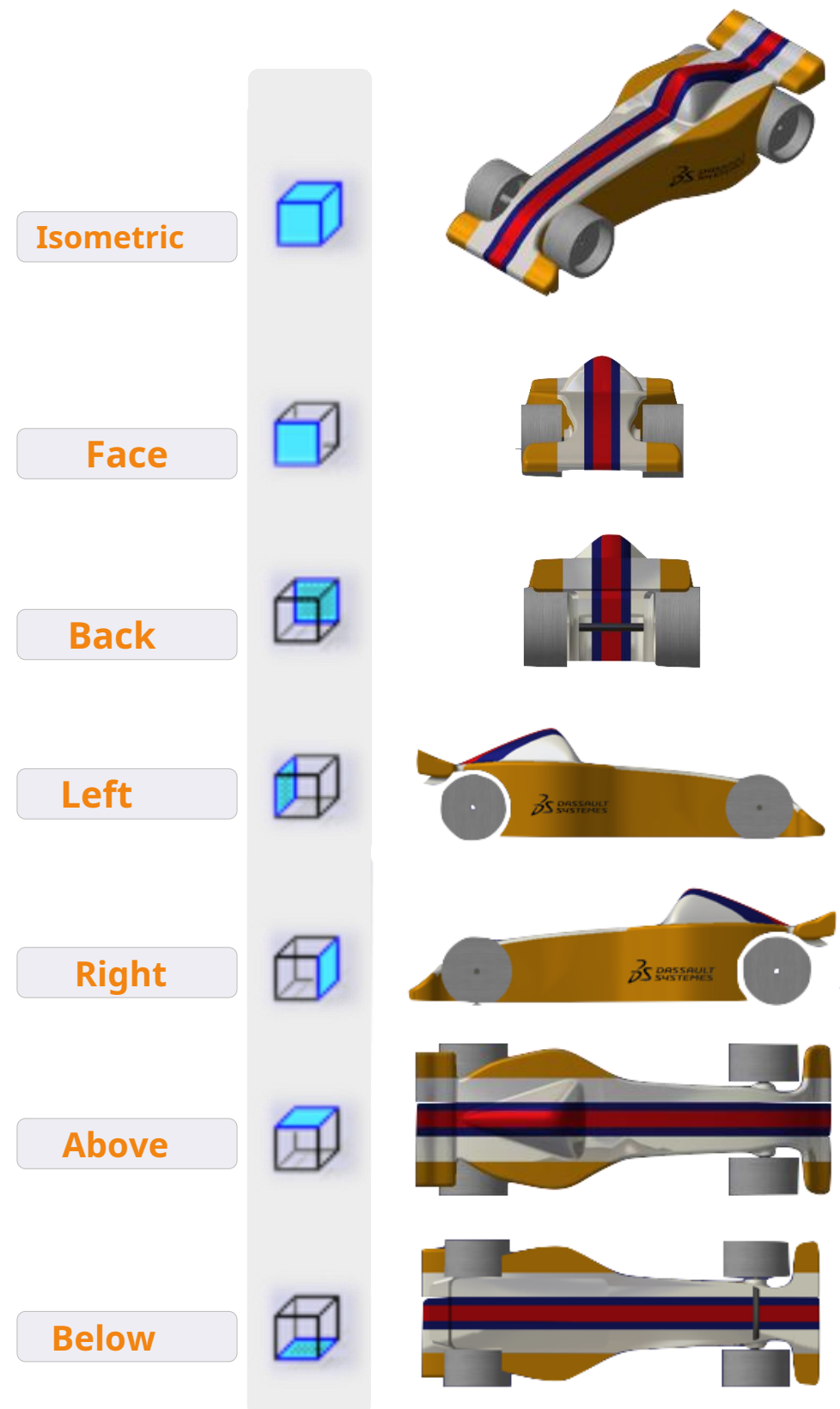
# The different views

You can navigate between the different default views of CATIA using the following icons:

- ▶ To access it, click on the small black arrow below.

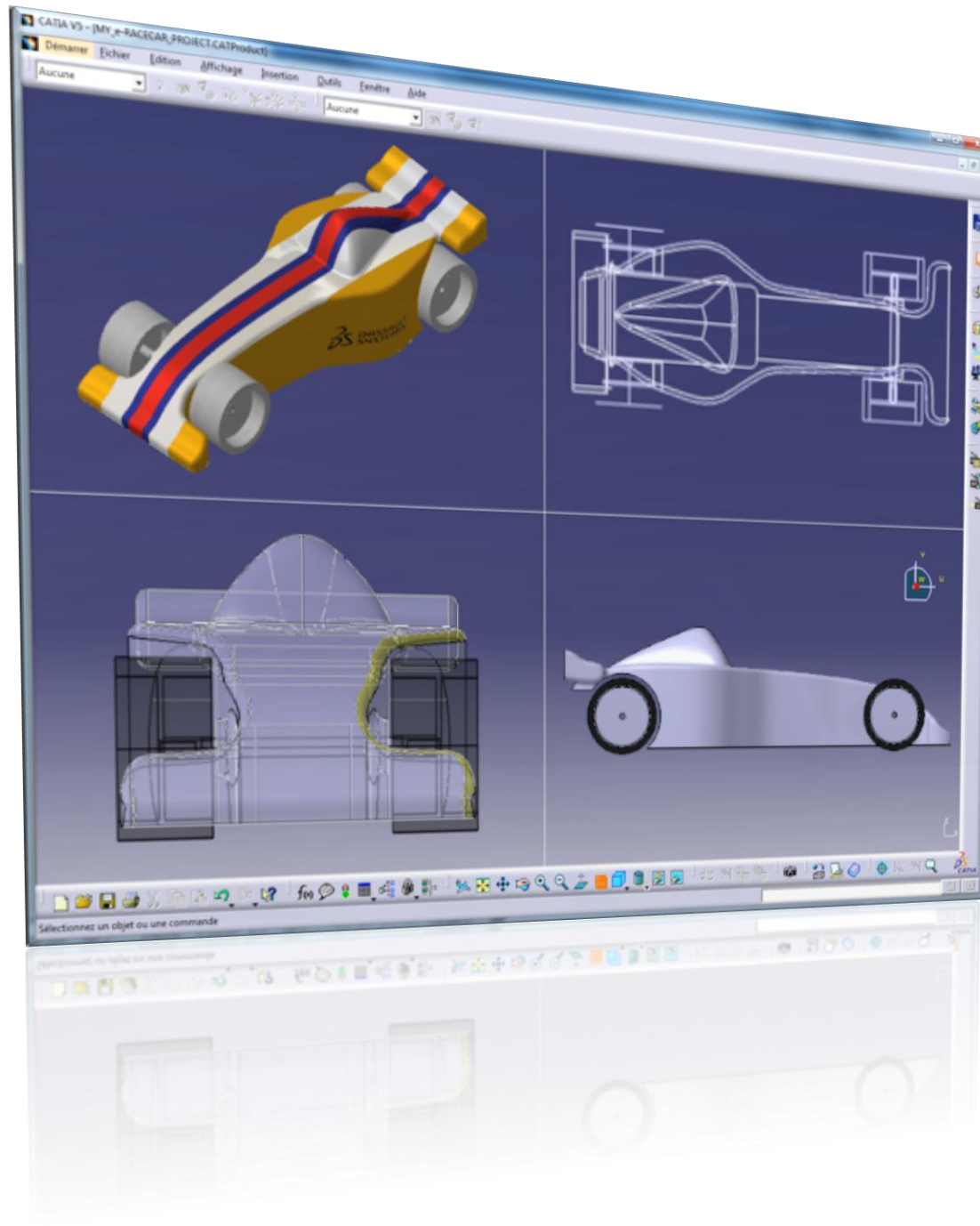


- ▶ giving access to the various default views.



# Viewing modes

In this step you will discover the different display modes offered in CATIA V5.



Here are the steps to follow:

1. Introduction.

**2. CATIA V5 QuickStart.**

has. Welcome to CATIA V5.

b. Presentation of the CATIA interface.

vs. Create a part.

d. Mouse manipulation.

e. The different views.

**f. Viewing Modes.**

g. Tips and tricks.

3. Your racing car.

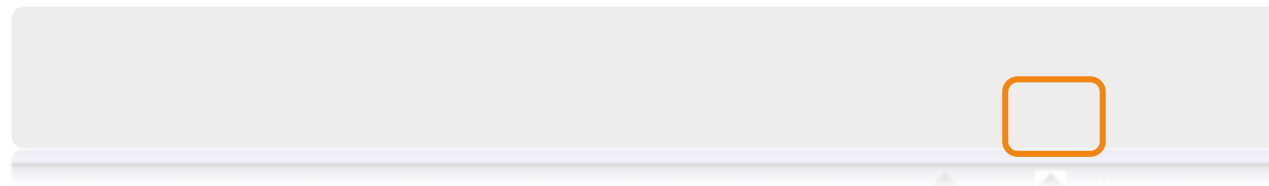
4. Method sheets.



# Viewing modes

You can navigate between CATIA display modes using the following icons.

- ▶ To access it, click on the small black arrow below.



- ▶ giving access to the different display modes.



# Tips and tricks

In that stage you will find a few useful information for using CATIA V5.



Here are the steps to follow:

1. Introduction.

**2. CATIA V5 QuickStart.**

has. Welcome to CATIA V5.

b. Presentation of the CATIA interface.

vs. Create a part.

d. Mouse manipulation.

e. The different views.

f. Viewing modes.

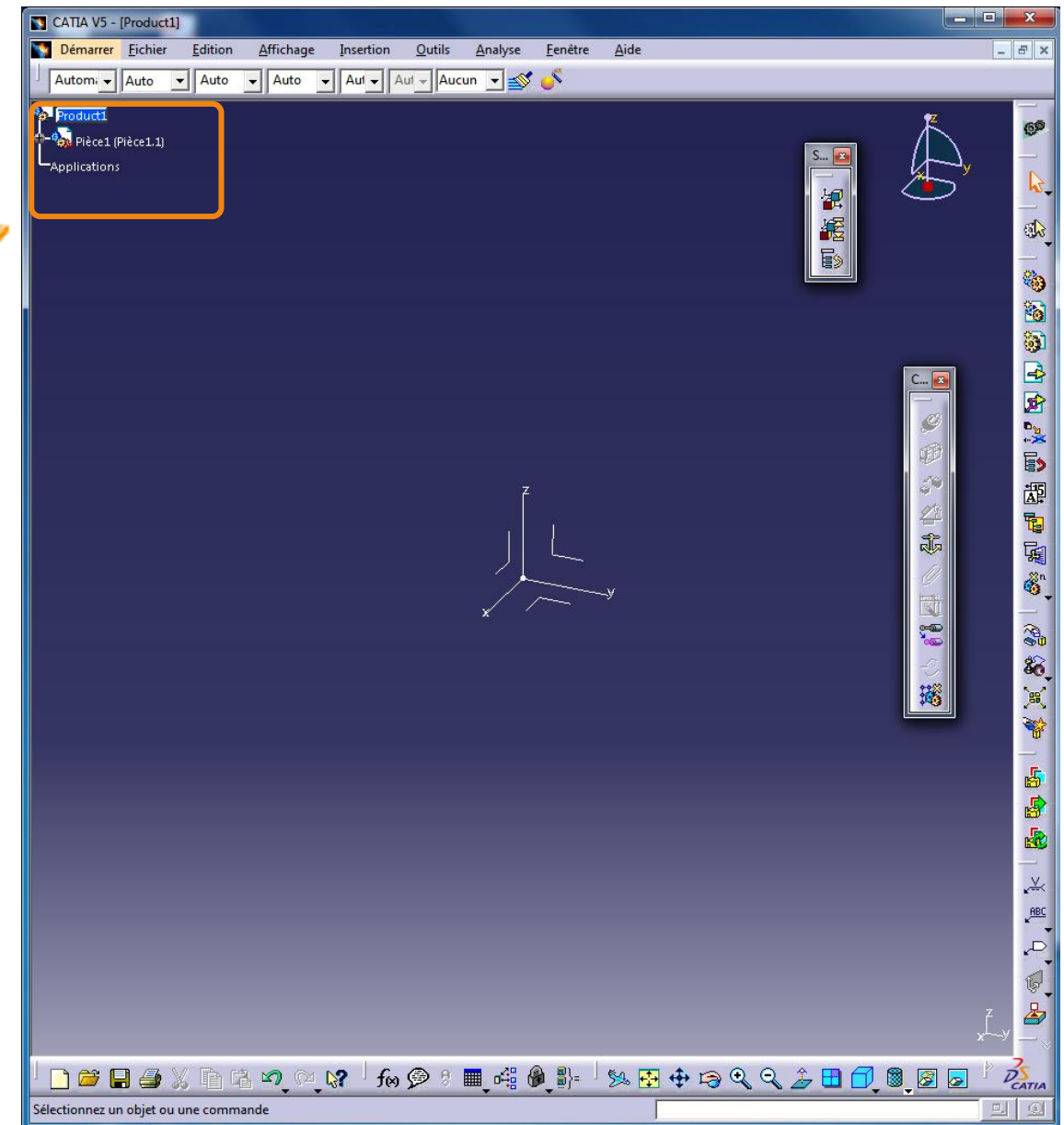
**g. Tips and tricks.**

3. Your racing car.

4. Method sheets.

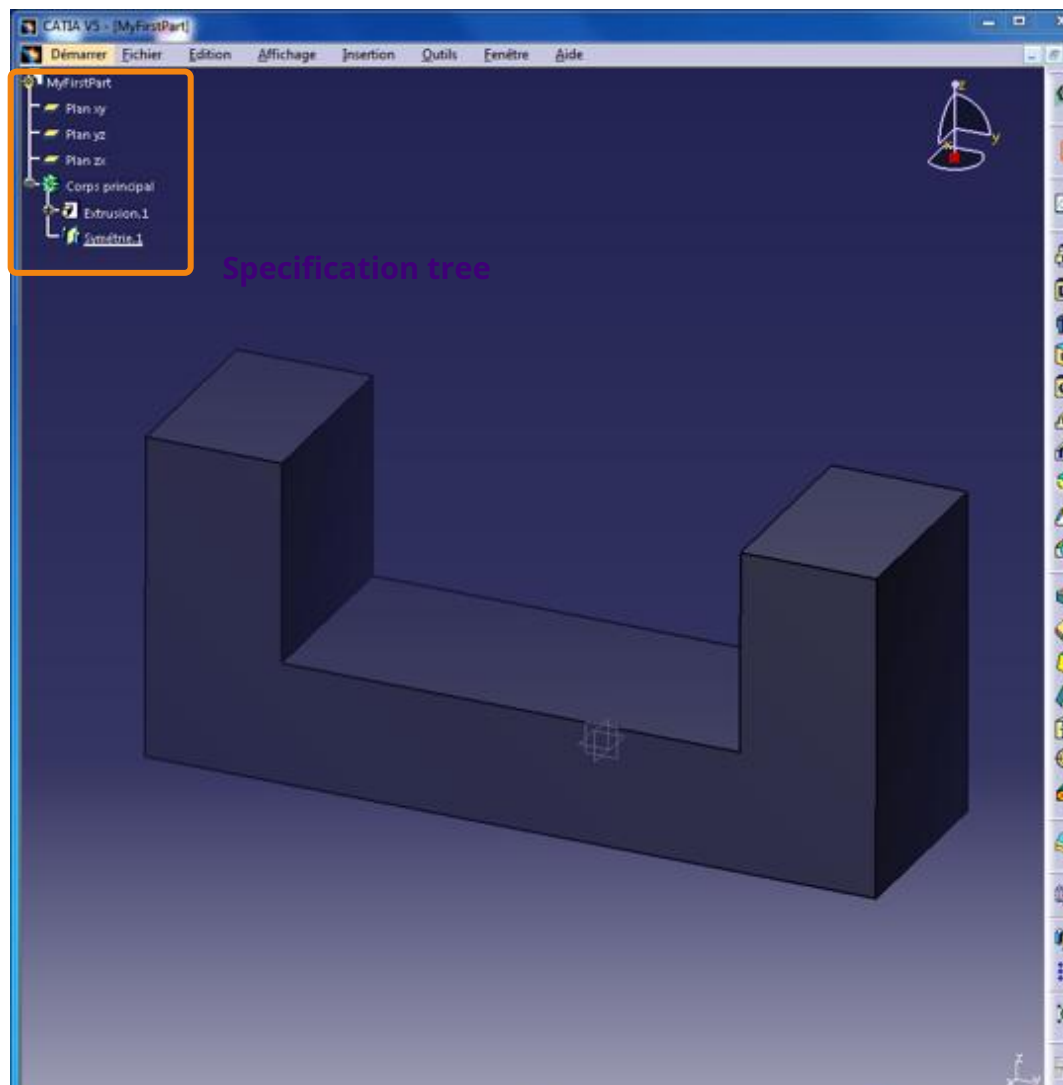
# Tips and tricks

- ▶ For each function of CATIA, there is a contextual help which is displayed in the lower left corner of the screen. This will tell you the steps to follow.
- ▶ You can hide or show the specification tree by pressing the F3 key on your keyboard.

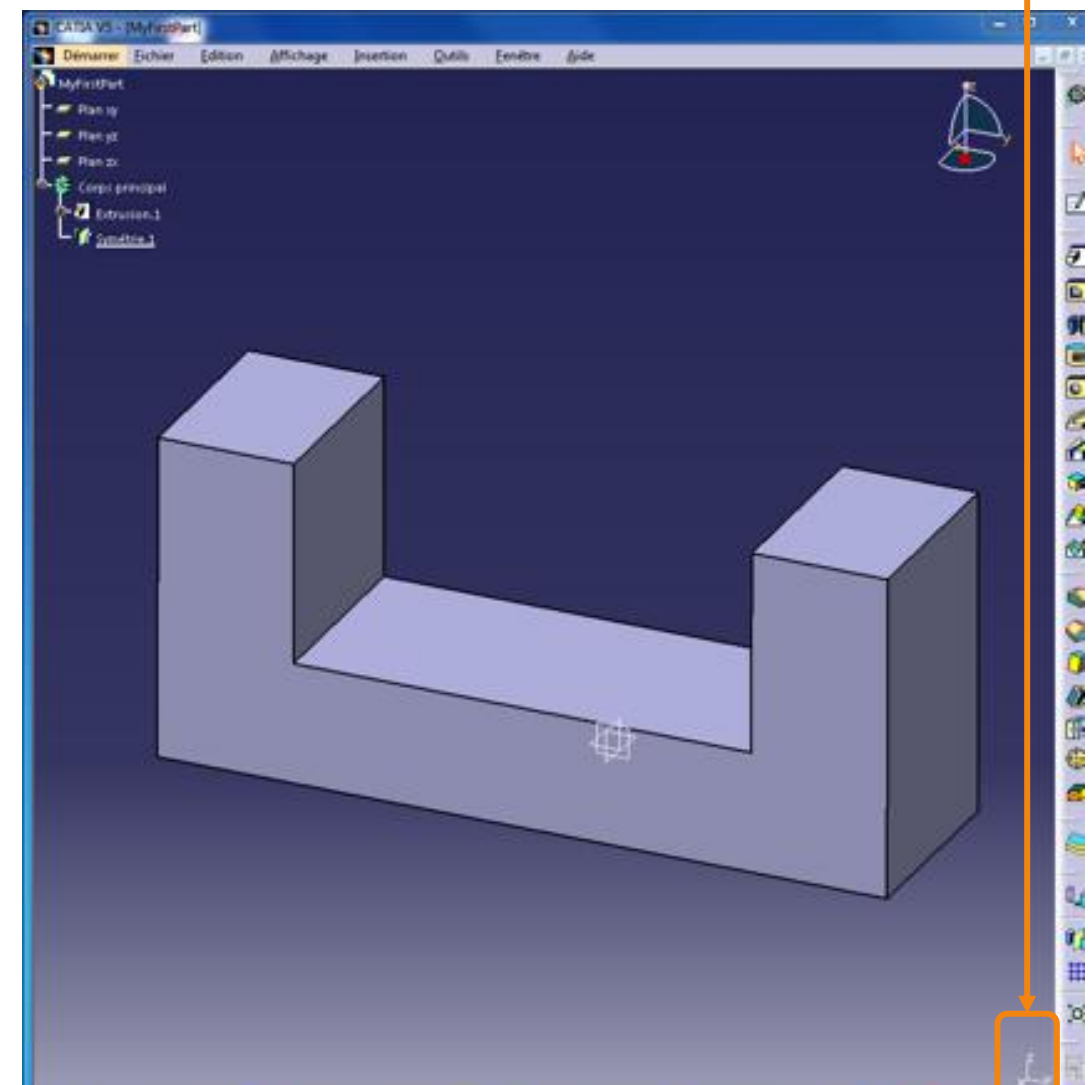


## Tips and tricks

- ▶ Sometimes, when trying to manipulate in the 3D view, the view freezes, and the specification tree moves. This is because you have enabled tree manipulation. To enter and exit this mode, click on the small marker located in the lower right corner of the screen.
- ▶ You will notice that when it is the tree that moves, the 3D elements are darkened.



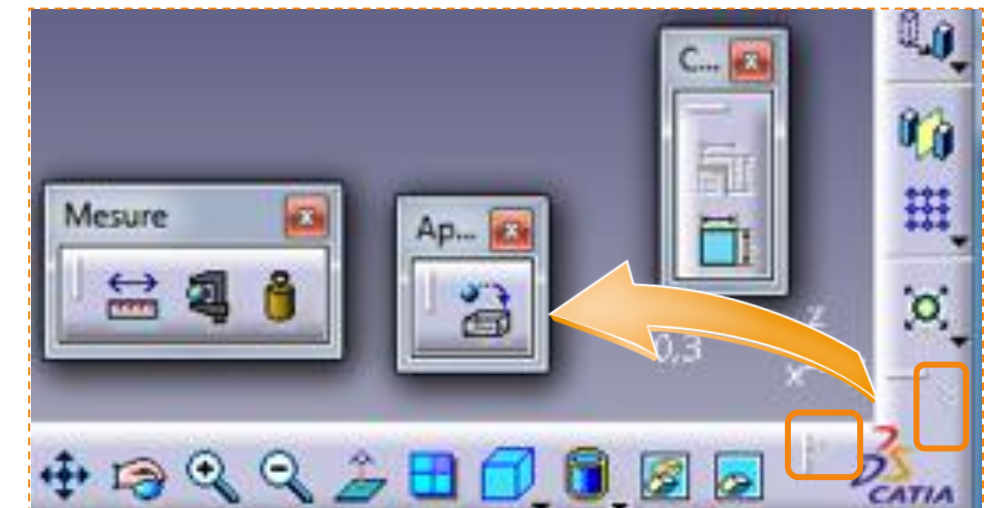
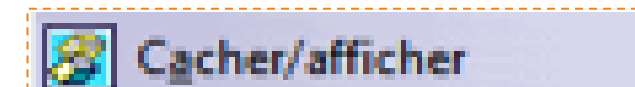
Manipulation de l'arbre



Manipulation normale

# Tips and tricks

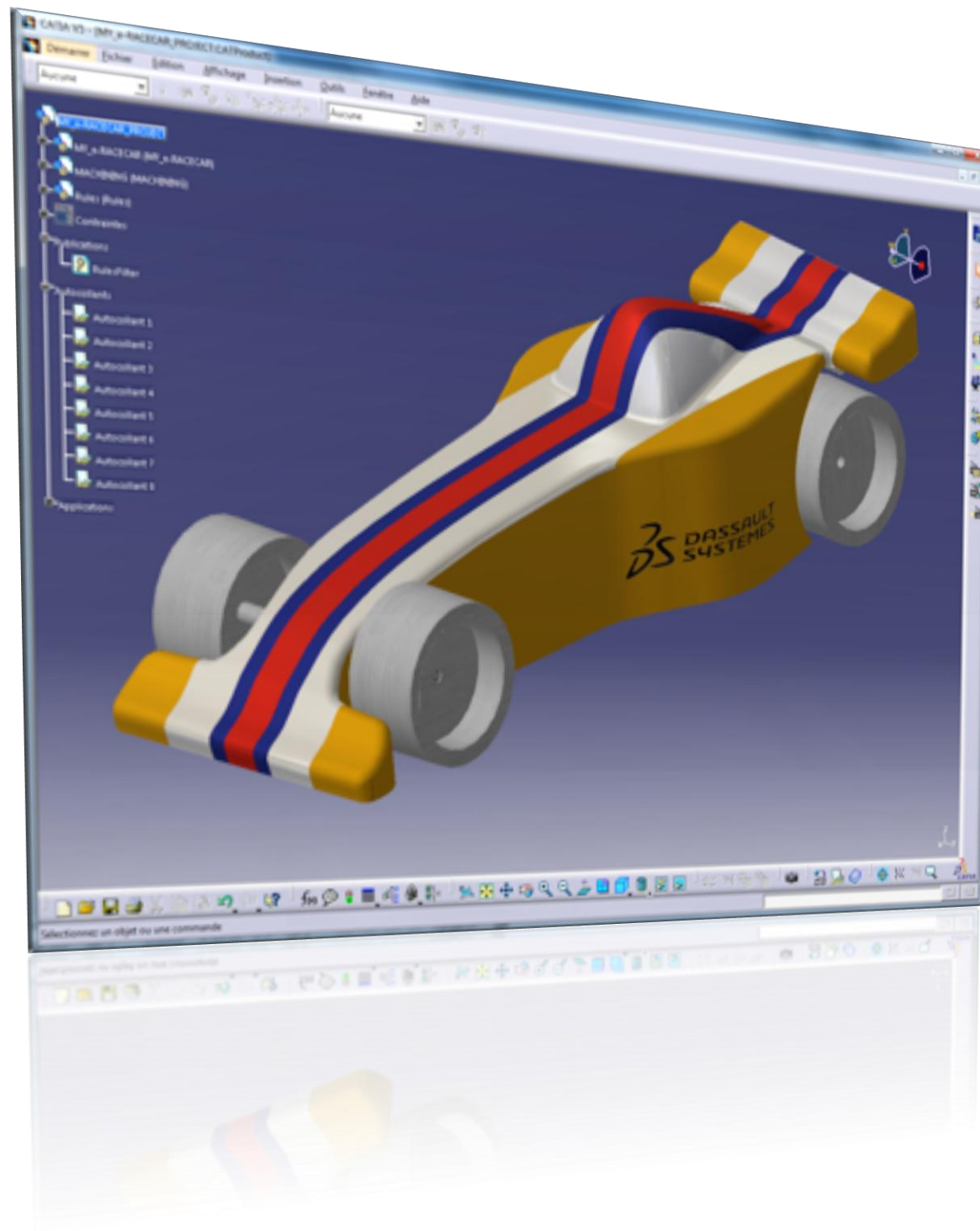
- ▶ As you work, the display may become cluttered, or your vision may be obstructed by a large number of graphical elements. To avoid this, you can hide 3D elements. To do this, right-click on an element in the specification tree or in the 3D environment and select **Hide/show**.
- ▶ Warning ! Hiding an element does not deactivate it: to find out, click on the icon **Show hidden objects**. It is an alternate 3D space where all hidden elements are grouped together. Click the icon again to return to the normal display.
- ▶ You can reframe the view on all the elements displayed using the icon **Center All**.
- ▶ Depending on the number of tools, the resolution of your screen, you can see the symbol >> on the corner lower right of the screen. This means that other toolbars are available. You can make them appear by holding the click on them, then moving the mouse. This will display a new toolbar. You can return a toolbar to its original position by double-clicking it.





# Your racing car

In this step you will build your car using CATIA V5.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. *Your racing car.*
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a plan.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.



# Before you start

## ► Training content :

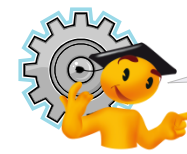
- A DVD containing all the CATIA digital models so that you can use them during the training. You must copy these files to your computer before starting your project.
- During the training, you will be invited to use specific files. You can find all the data related to the project on the Dassault Systèmes academic website: <http://campus.3ds.com> .
- Extract the contents of the Zip file all at once to preserve the directory structure.
- You must have a directory named **Courseware\_Data** on your hard drive. You will find for each lesson a dedicated sub-directory containing:
  - **Start\_Data**: gathers the data needed to start the lesson. **End\_Data**: gathers
  - the data corresponding to the expected results.
- The training has been designed so that it can be followed in parts or in full. You can start from any lesson of your choice by loading the results (**End\_Data**) from the previous lesson.



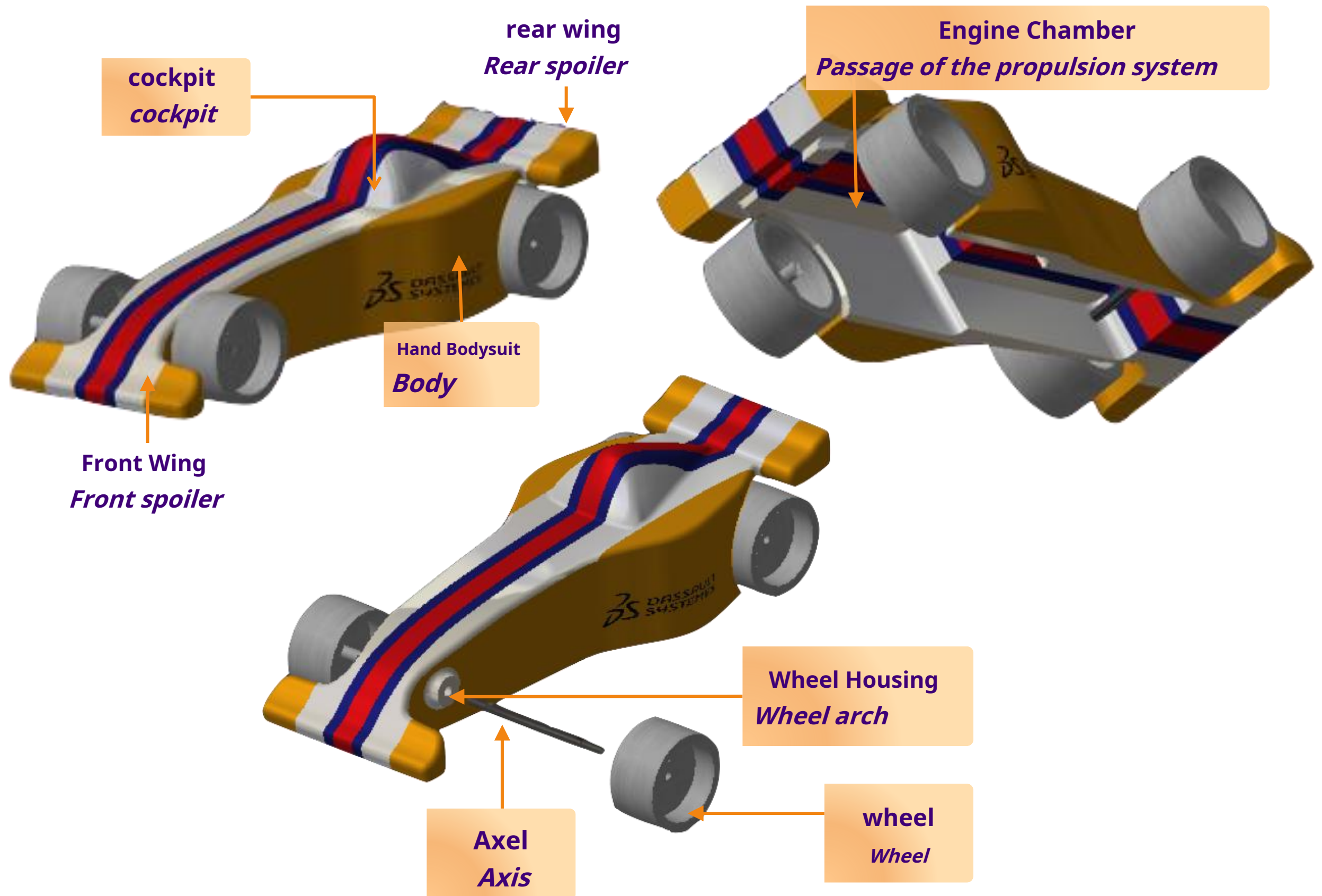
### Before you start!

Personalize your CATIA V5: take 5 minutes to configure CATIA and define your favorite workshops. Consult the method sheet "Configure CATIA" located in part 4 of the training.

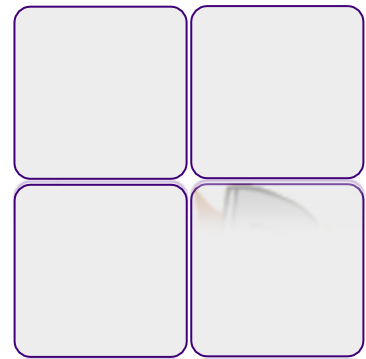




# Vocabulary

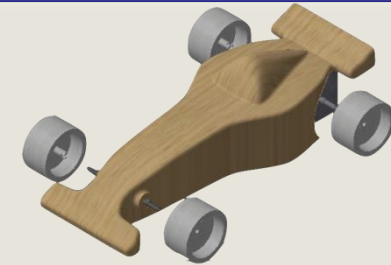


# The key stages of the project



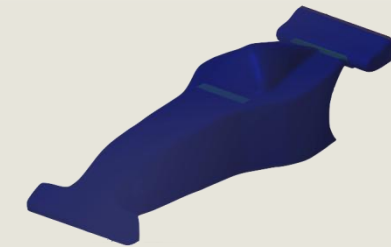
## Modelization

- Model your car and assemble the different components.



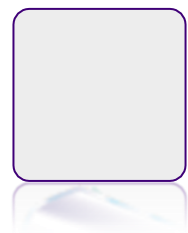
## Control Dimensional

- Check the dimensions of your car by drawing up a detailed plan and using the analysis tools.



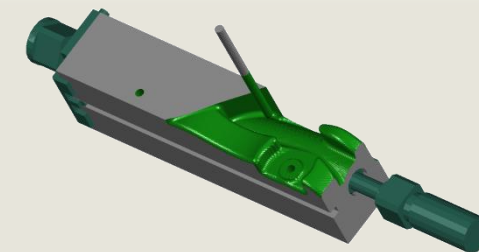
## Design Chart

- Personalize your car with stickers then create photo-realistic images.



## Manufacturing Assisted by Computer

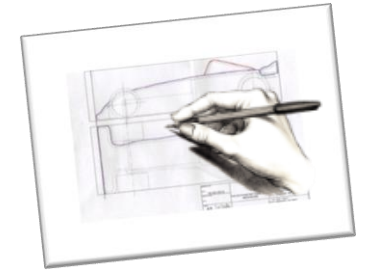
- Carry out a machining simulation of his car and generate a file to drive a numerically controlled machine.



# The Body Modeling Process

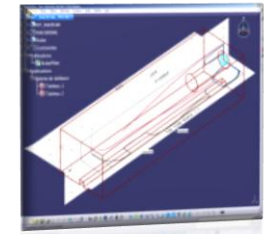
Work  
Manual

- Print and draw his car on a rough plan of balsa block.



Sketch Tracer

- Create immersive sketches using the Sketch Tracer workbench to allow easy tracing of the model.



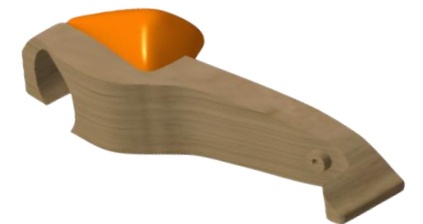
Shape Design

- Reproduce the guiding curves of the car to model the bodywork.



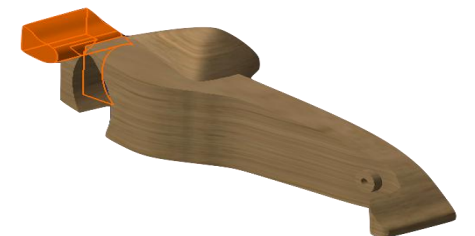
Mechanical  
Design

- Design the various mechanical elements of the car using the Part Design mechanical design workshop.



Shape Design

- Model the cockpit and the rear wing using the Imagine & Shape workshop allowing unlimited freedom of shapes.



Finalization

- Use various essential features in a design, such as fillets, and symmetry.





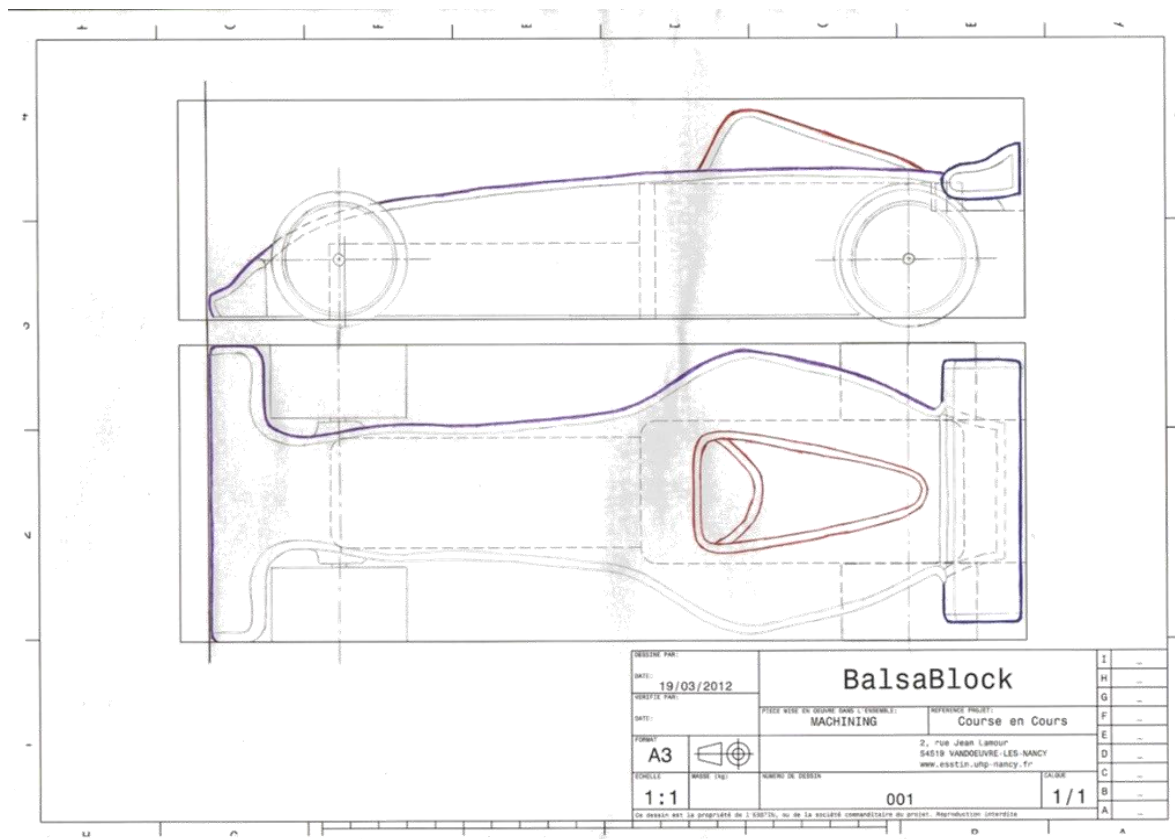
# Draw your car

In this part we will see how to draw your car in a template in order to respect the main dimensions.



Here are the steps to follow:

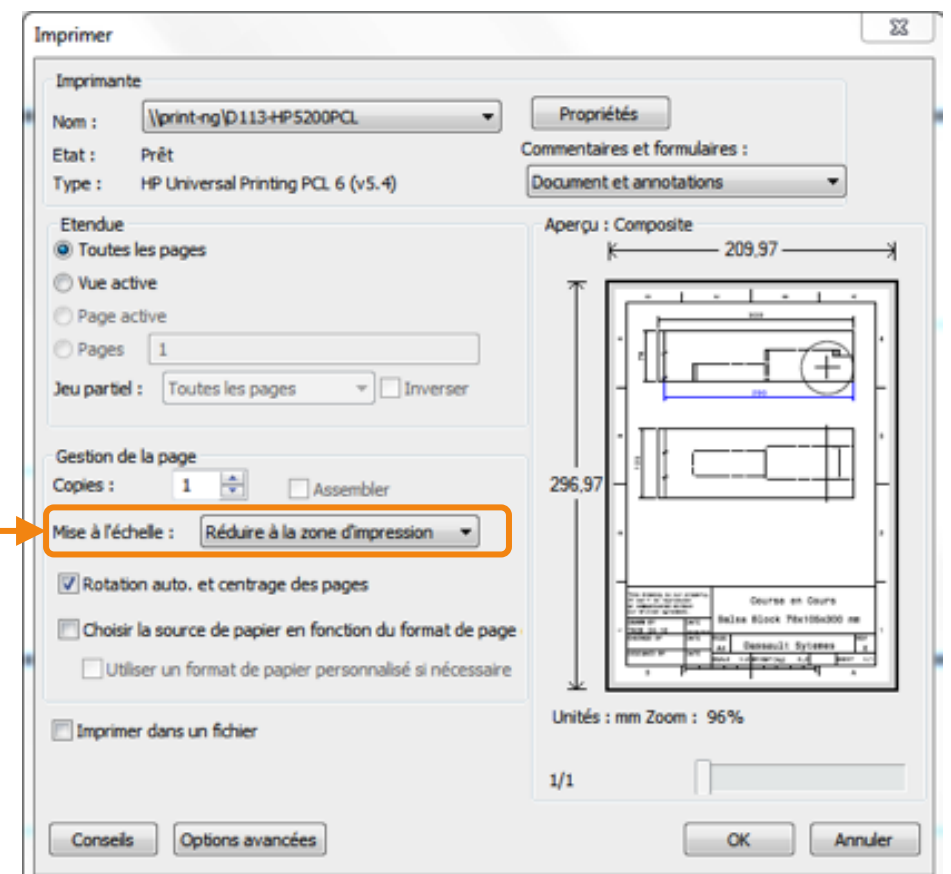
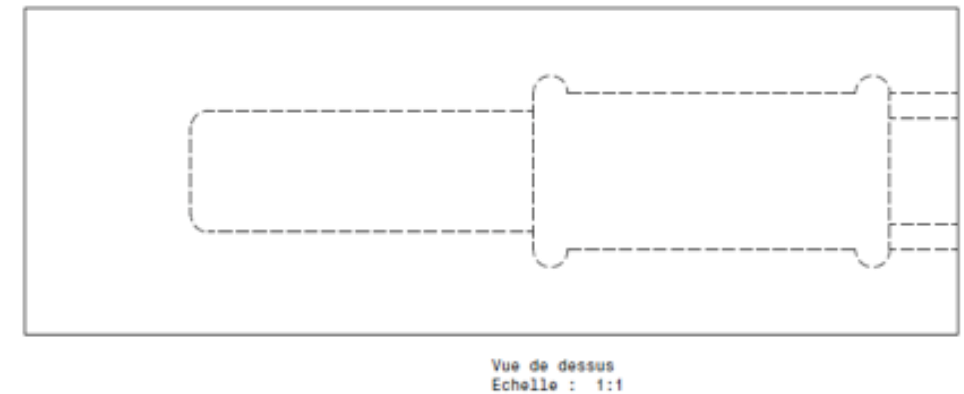
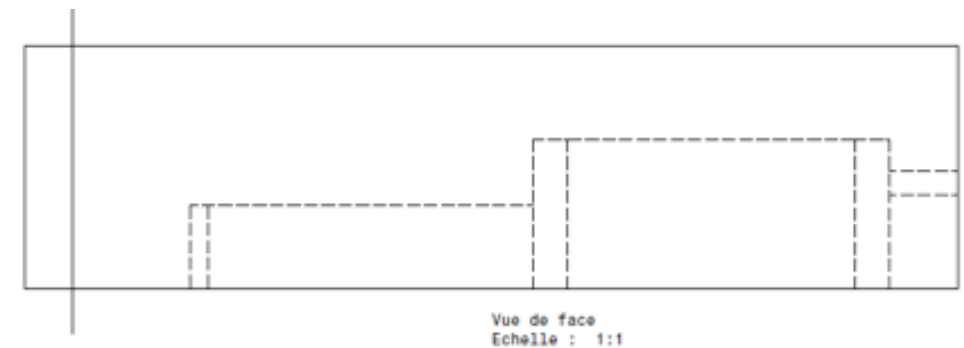
1. Introduction.
2. CATIA V5 QuickStart.
3. *Your racing car.*
  - has. *Draw your car.*
    - i. Print the plan.
    - ii. Draw two views.
    - iii. Scan your sketch.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a plan.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.



# Print the map

You will use an A3 plan (297 x 420mm) representing the balsa block to draw the car in the correct proportions.

Print the plan of the balsa block in which the car will be machined **DrawingA4\_BalsaBlock.pdf** making sure to **do not scale**.





# Draw the two views

Sketch the profile and top of the car, taking care to:

**has.** Be consistent between the two views.

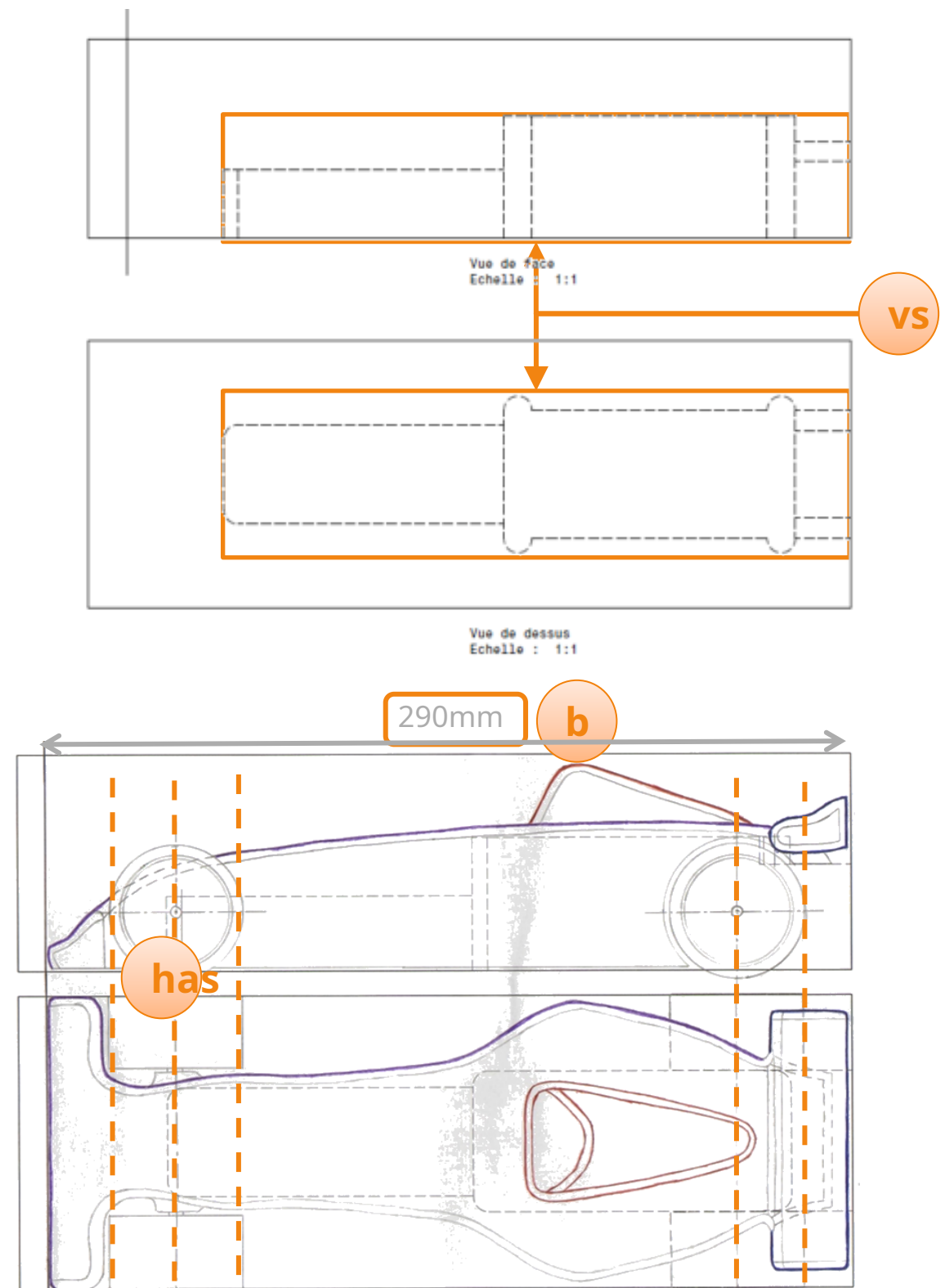
**b.** Respect the dimensions.

**vs.** Consider motor and battery housings

It is advisable to differentiate each part of the car with colors.

Here we have:

- in red the cockpit,
- in purple the frame,
- in blue the rear wing.



# Scan your sketch

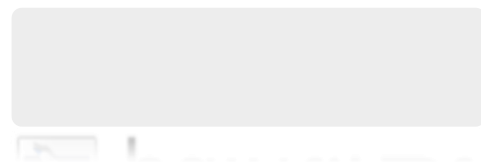
We will now scan these drawings so that they are suitable for import into CATIA.

Scan the drawings making sure you are:

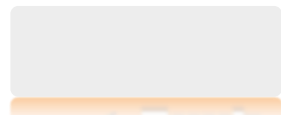
1. In landscape mode. (\*)
2. Best oriented.
3. In the following resolution: **200dpi** Where **200dpi**.
4. Save image as JPEG with name '**My\_e-Racecar\_Handwriting**'.

(\*) If you cannot directly scan in landscape, Windows XP and above is able to reorient your image. To do this, open the image and click on one of the icons at the bottom.

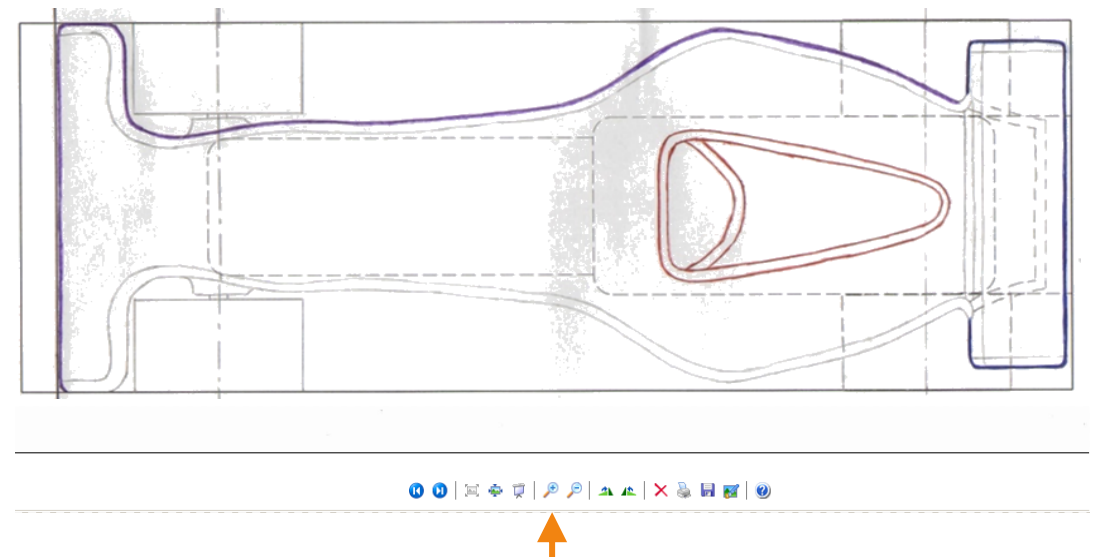
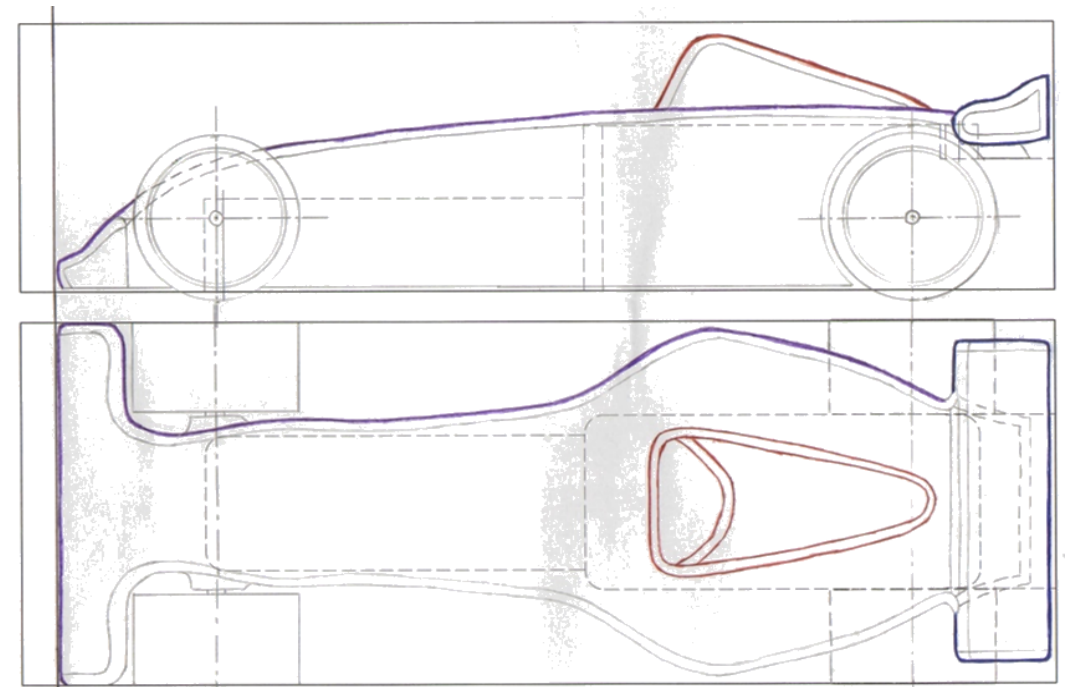
Free image editing software:



<http://www.getpaint.net/>



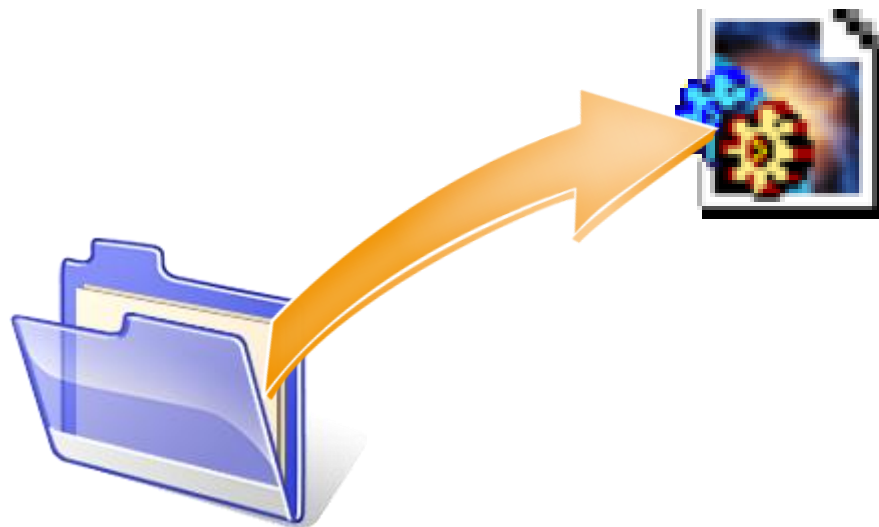
<http://www.gimp.org/>



# Open your project

We will work in a 3D environment in which we will import the 2D sketches of your car. We will start by opening a file in which some elements will already be present. We will complete this file throughout the const

of your car.



Here are the steps to follow:

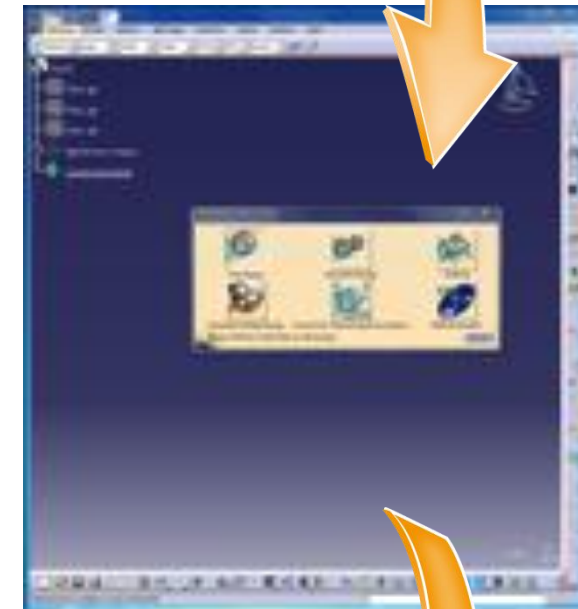
1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.**
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a plan.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

# Open your project

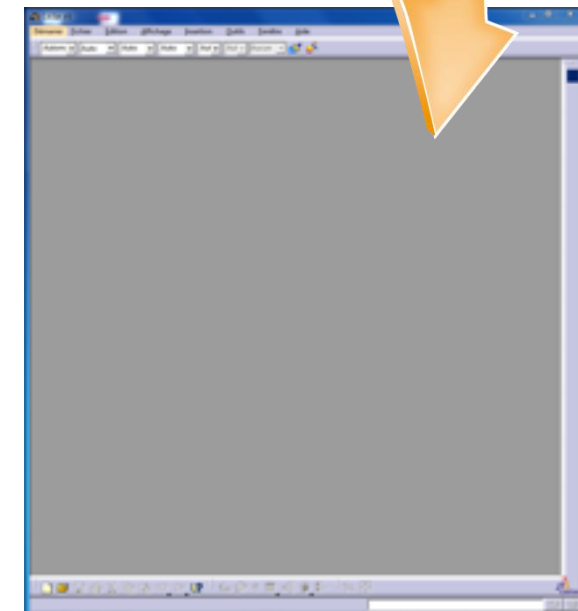
1. Start by launching CATIA V5.



2. After launching it is possible that your screen will look like the image opposite.



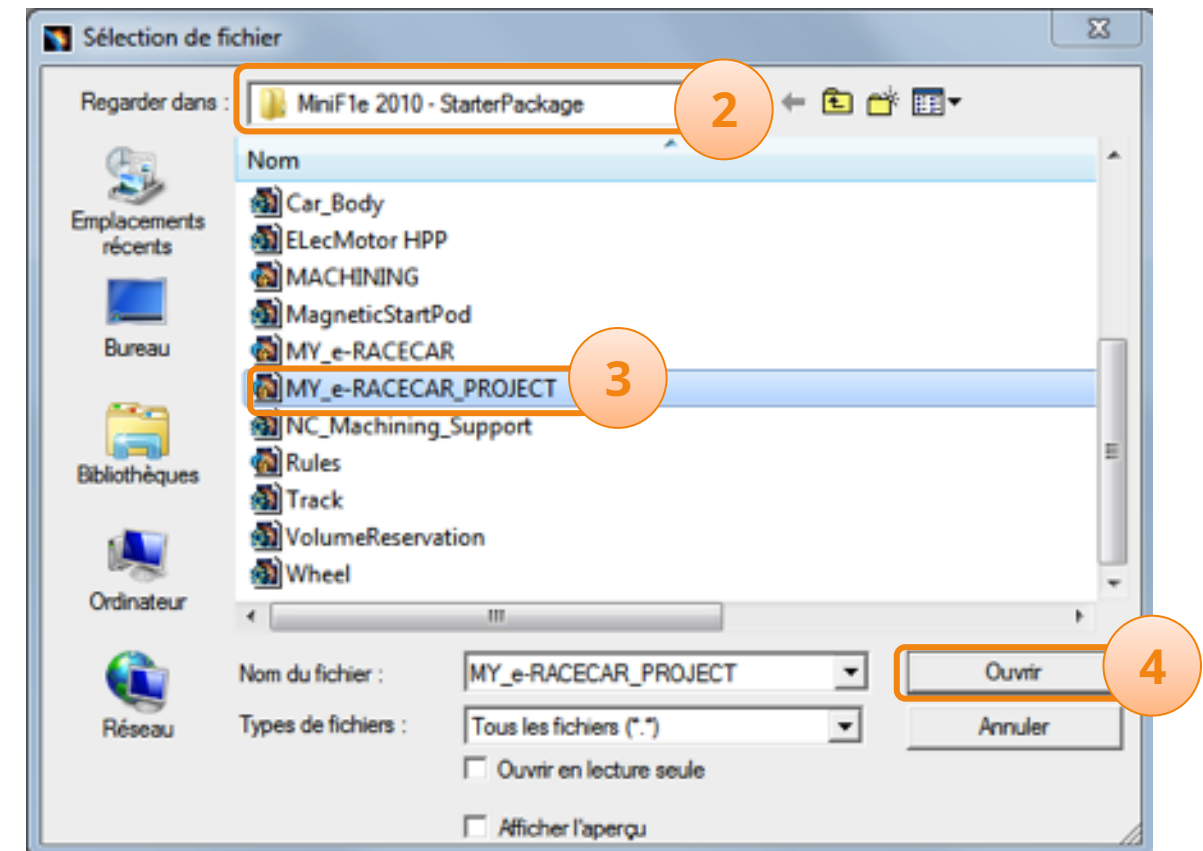
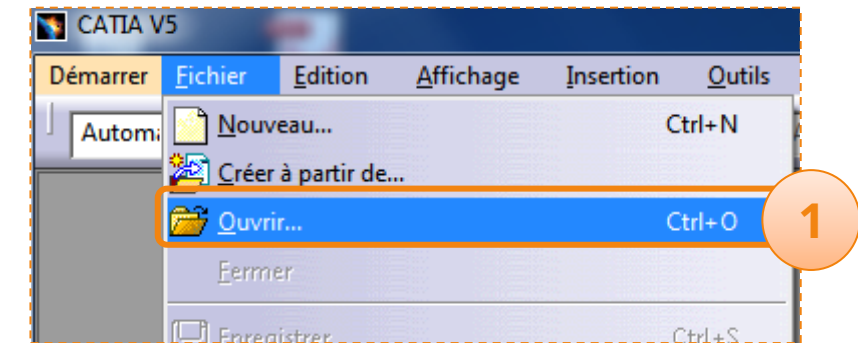
3. Close the visible windows until you get the blank screen opposite.



# Open your project

We will start by opening the project, for this:

1. Click on **File > Open...**
2. Go to the directory **Mini F1e 2010 - StarterPackage**.
3. Select the file named **MY\_e-RACECAR\_PROJECT**.
4. Click on **Open**.



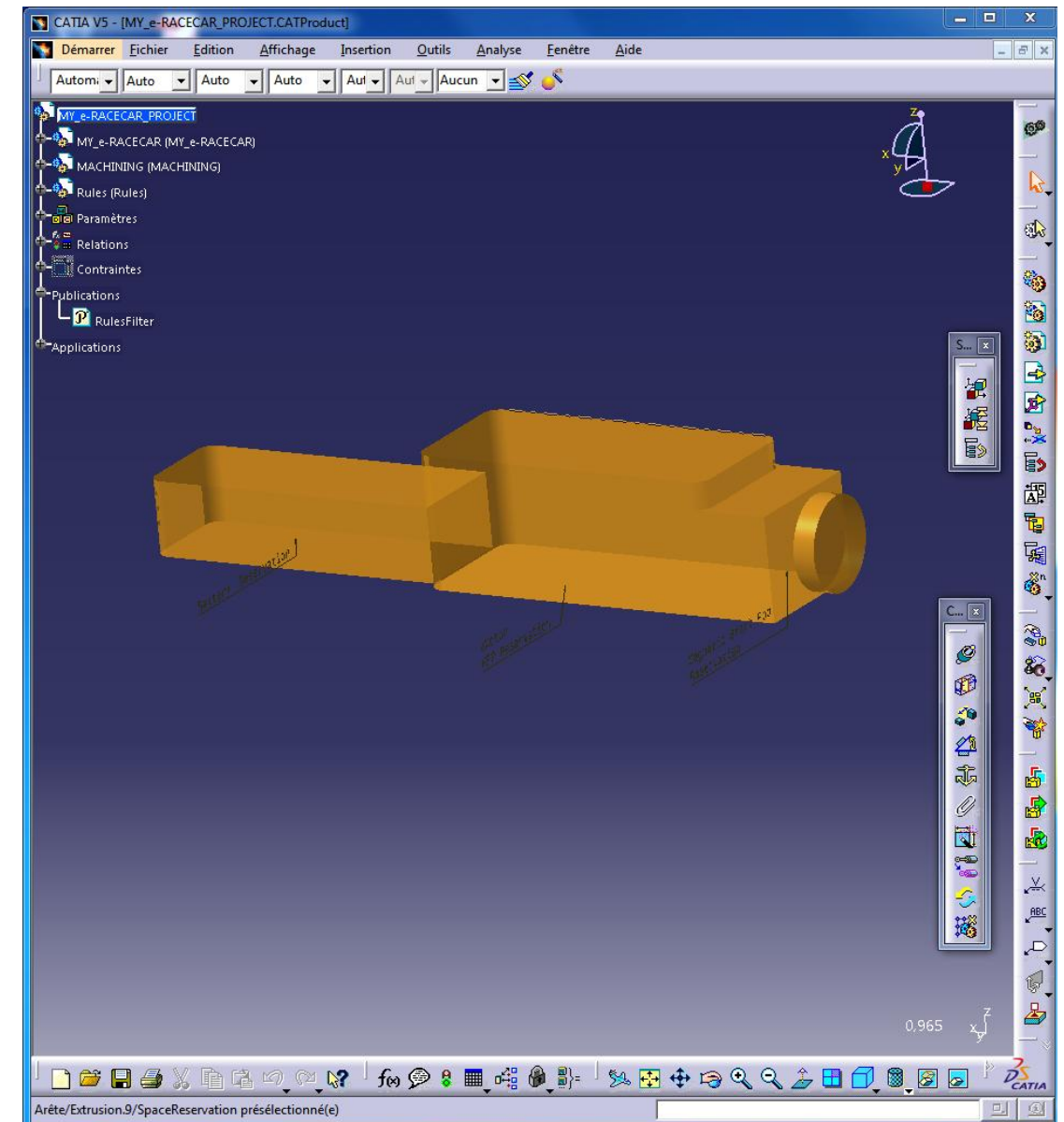


# Open your project

This is what your screen should look like now.



For more information on the different elements that make up the CATIA screen, see the Quick Start.



# Open your project

Here is the structure of your project:

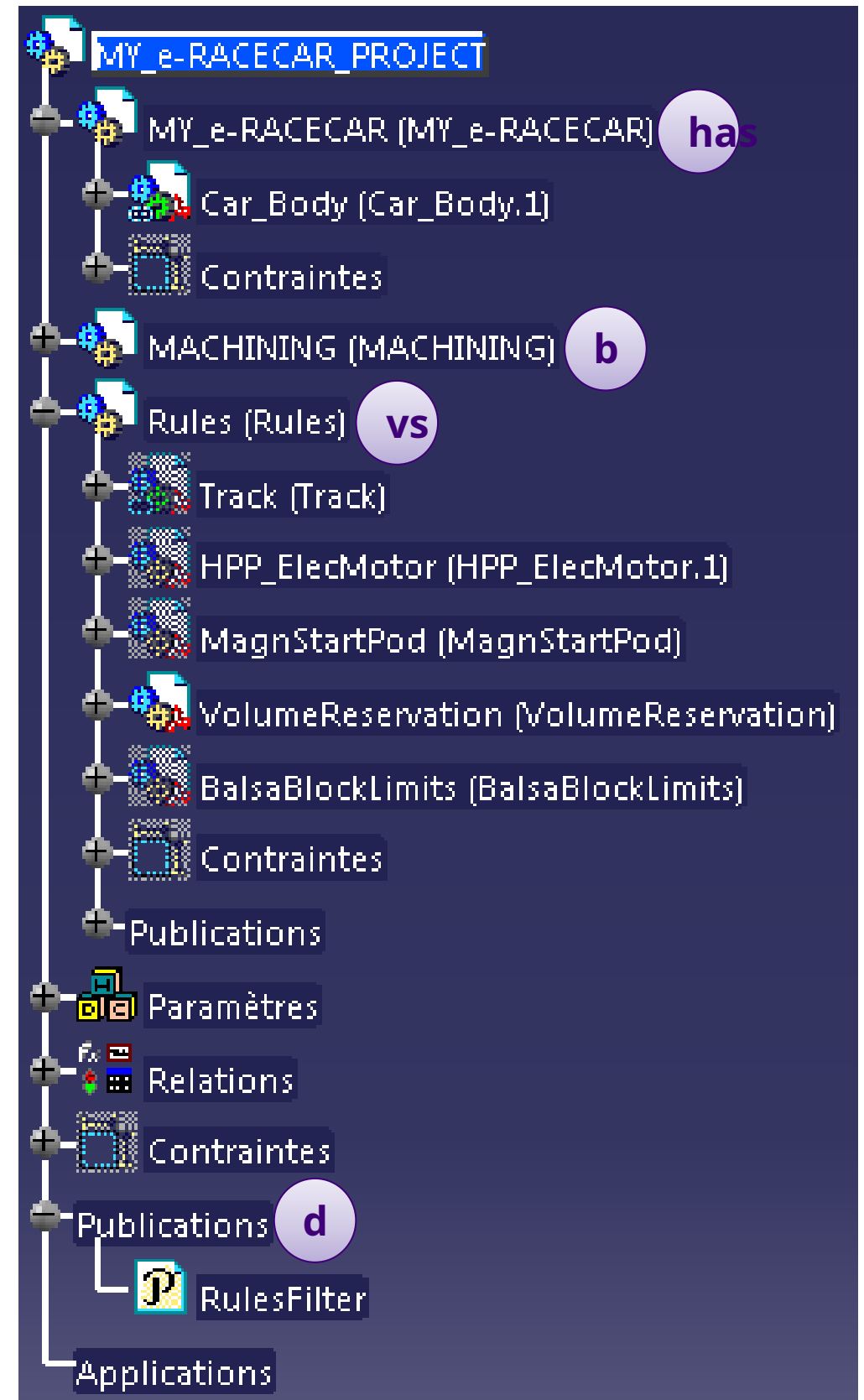
has. The product **MY\_e-RACECAR** will contain the car assembly, for example, 'Car\_Body' is the body.

b. The product **MACHINERY** contains the elements useful for Computer Aided Manufacturing.

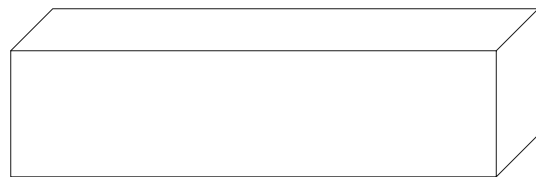
vs. The product **Rules** makes it possible to check the dimensional requirements of the specifications.

d. A publication, in which there is a display filter.

We will come back in more detail on each of the parts that make up the project.



# Initial structure of your project

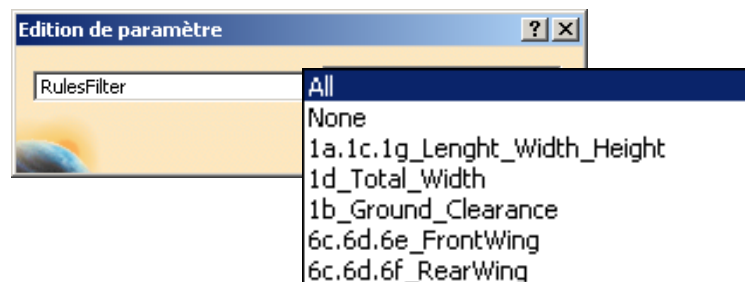
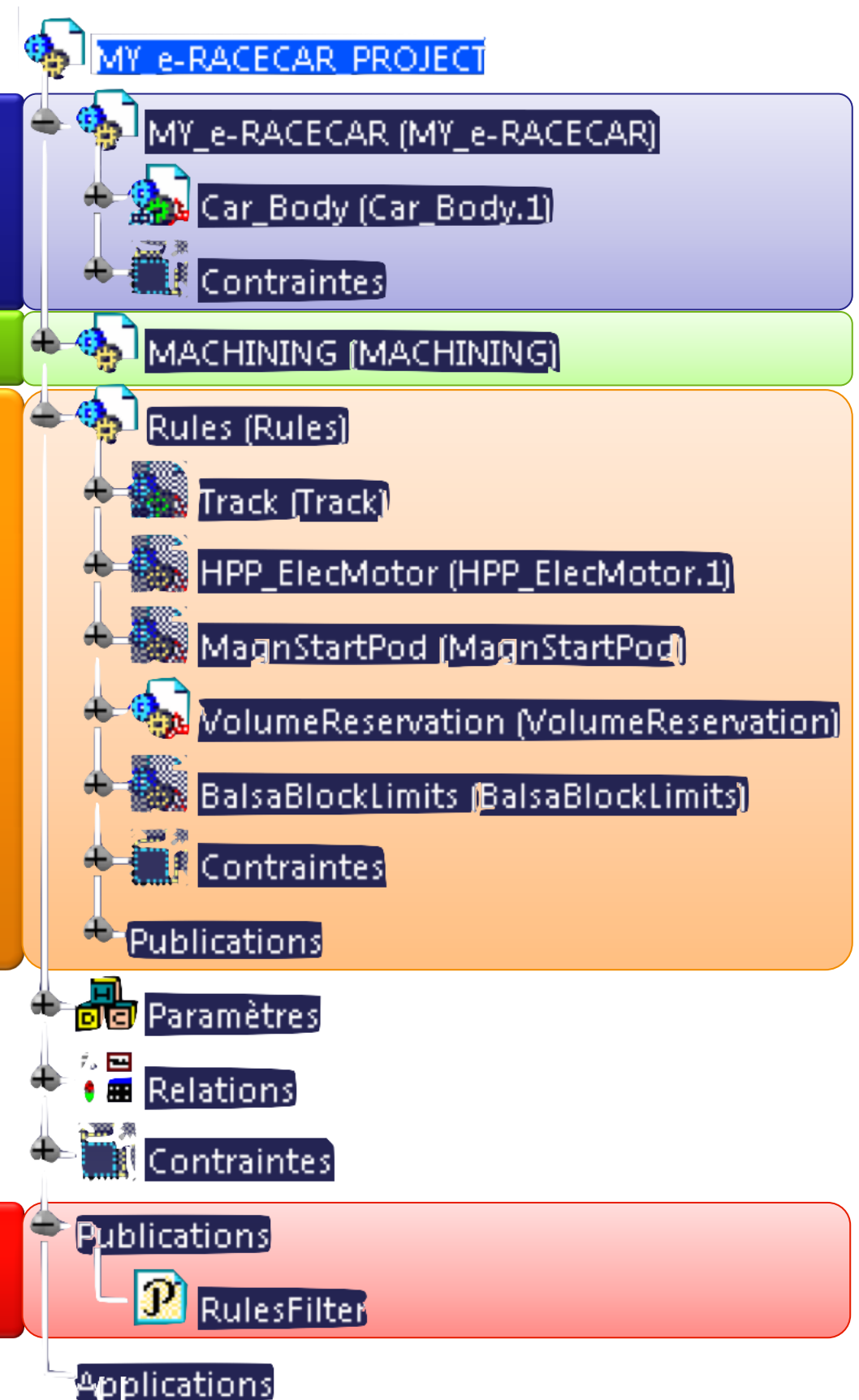


**car assembly**

**Components for machining**

**Components for the control**

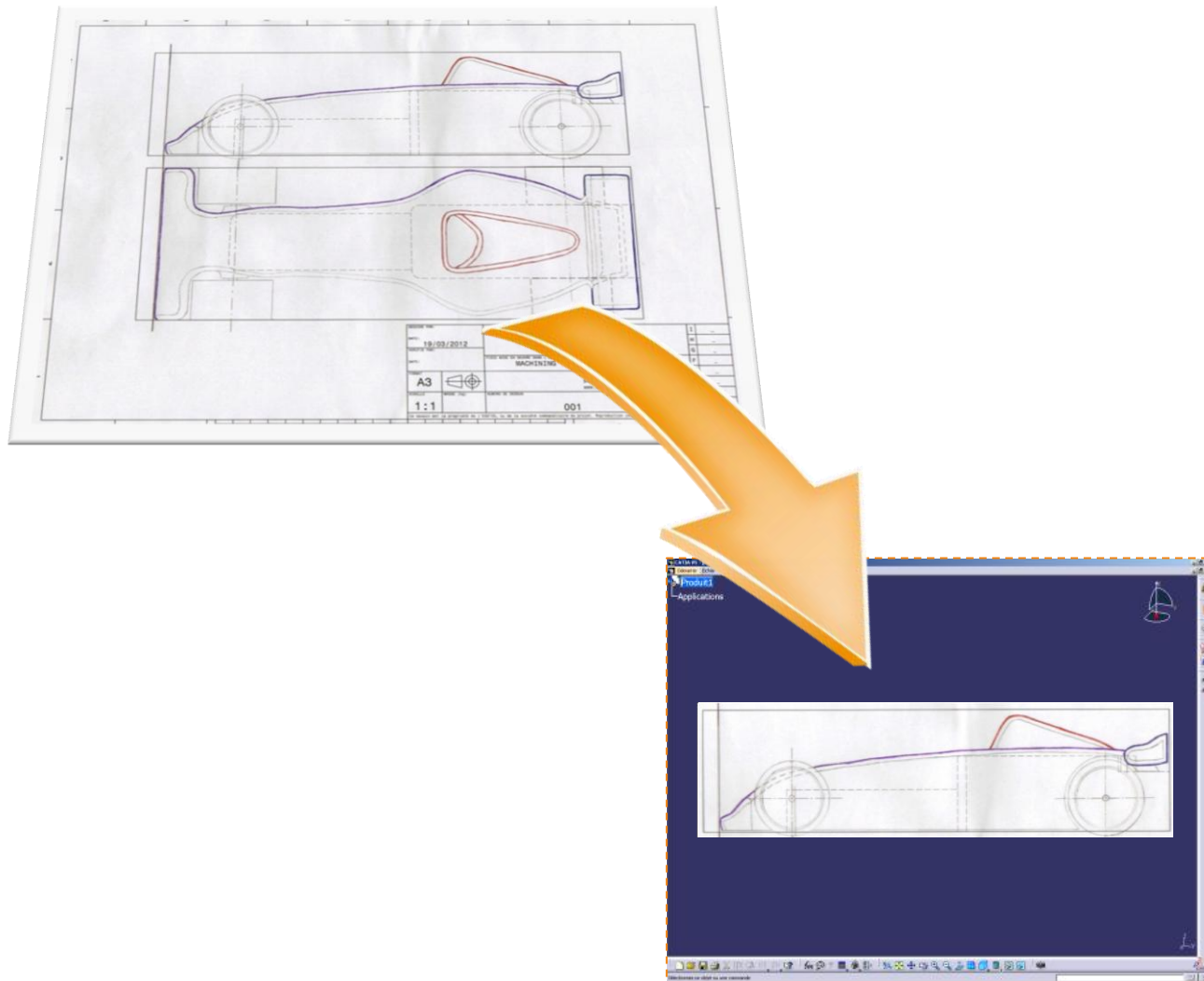
**Control management**



# Create an immersive sketch

We will import scanned to place it This will serve as a support for the characteristic curves

previously 3D purr. you will trace vehicle.



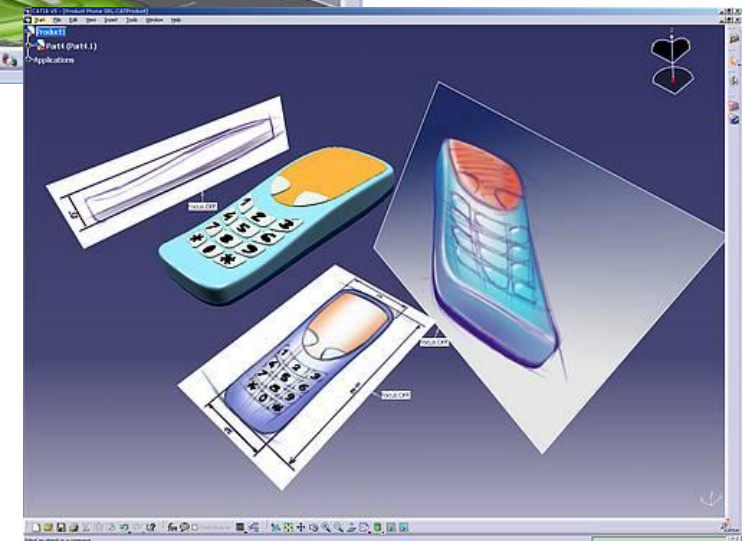
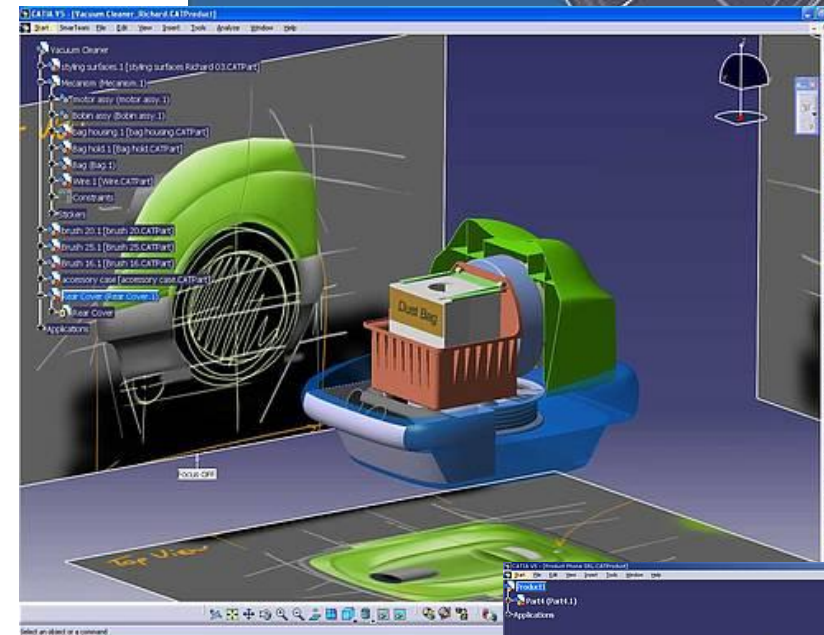
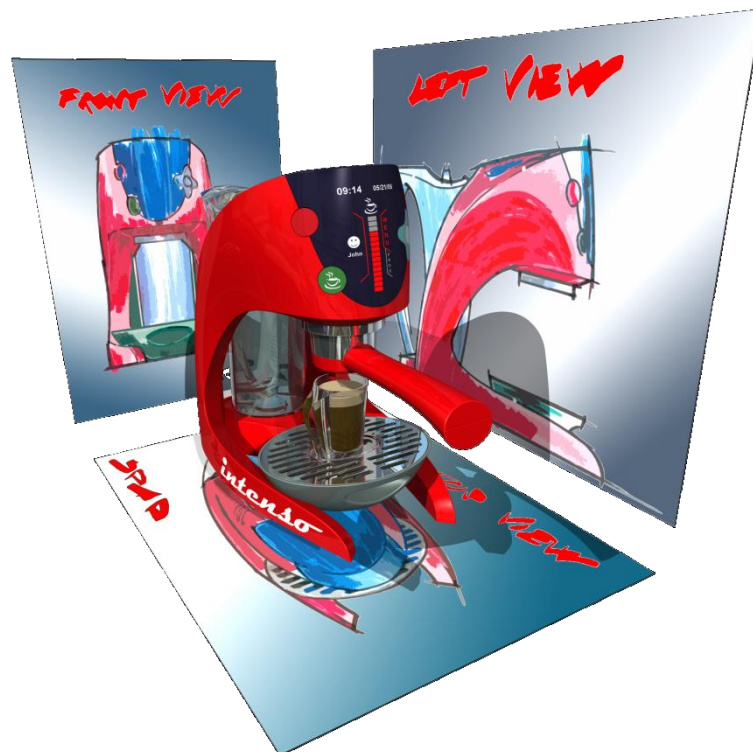
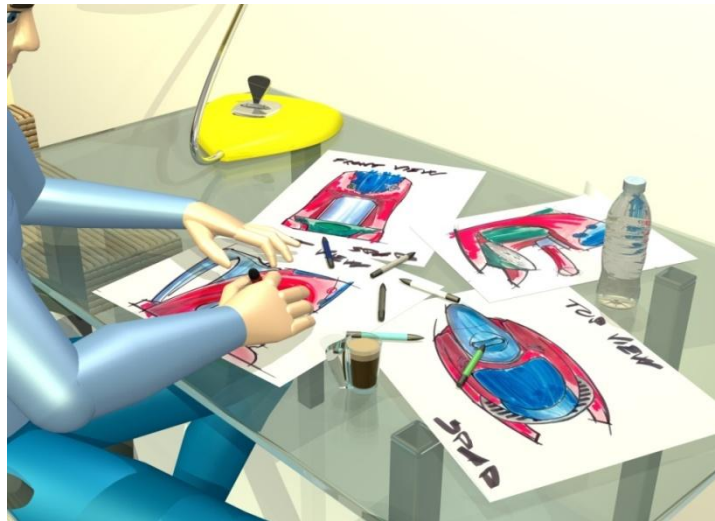
Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. *Your racing car.*
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.*
    - i. Introduction.
    - ii. Side view.
    - iii. Top view.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a plan.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.



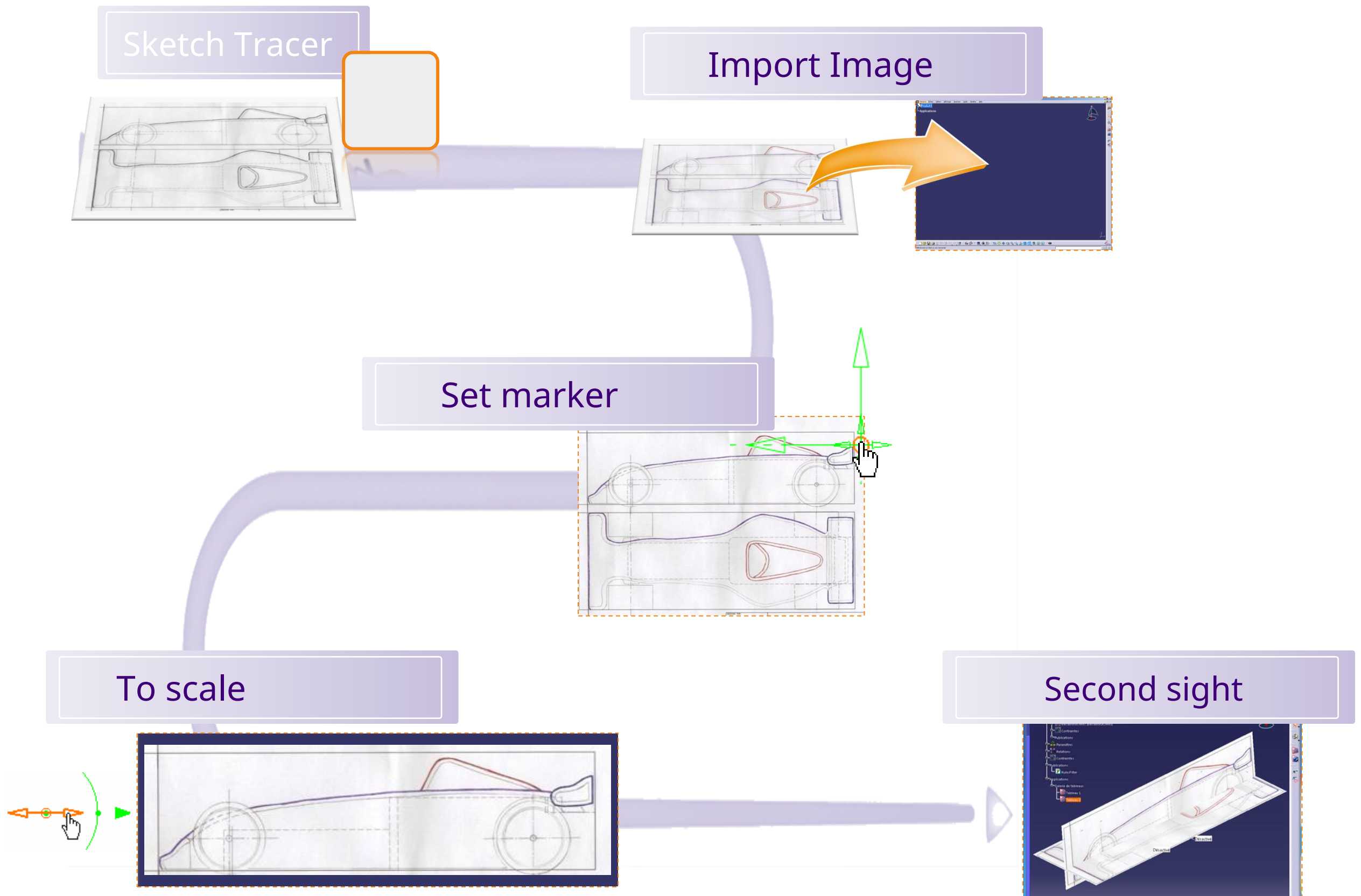
# Introduction

The Sketch Tracer workbench allows images to be imported into a 3D environment. Thanks to this, you will be able to create three-dimensional geometries from your drawings, by "tracing" your characteristic curves.





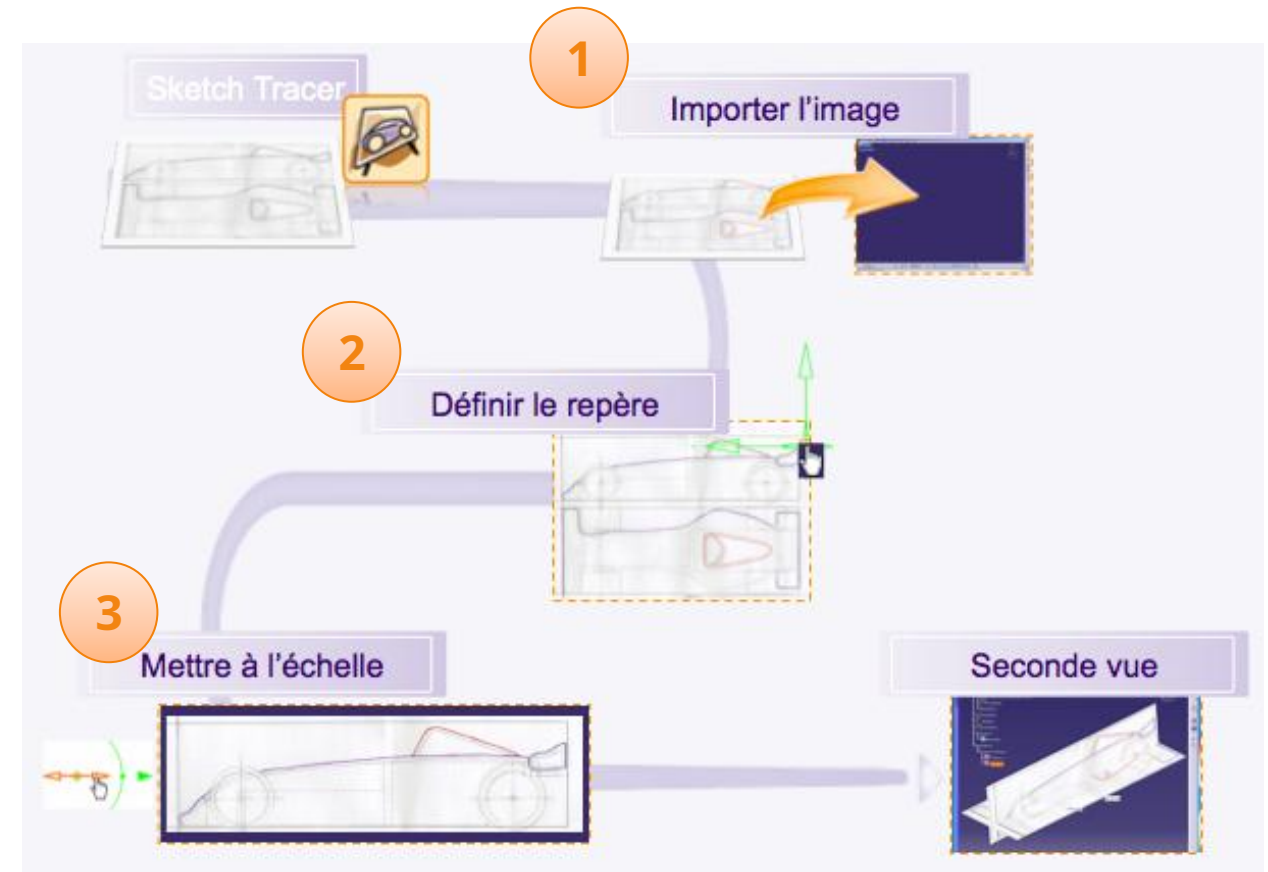
## Introduction – The procedure to follow (1/2)



## Introduction - The procedure to follow (2/2)

The configuration of the 3D space is carried out by following the procedure below:

1. Import the image having previously selected the appropriate view.
2. Set marker position.
3. Size the image in the 3D environment using the dimensions present on the plan, then crop the image.
4. Perform operations 1, 2 and 3 for the second view.

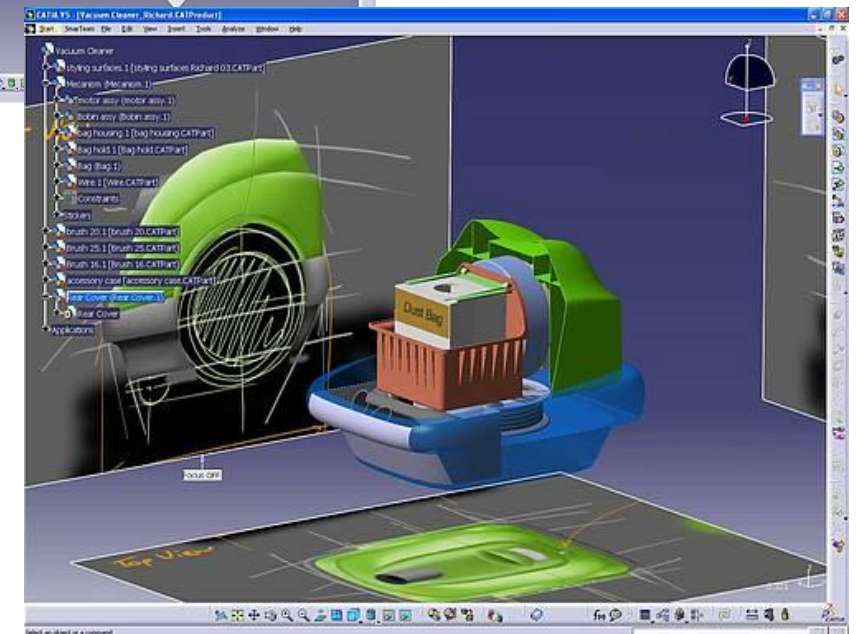
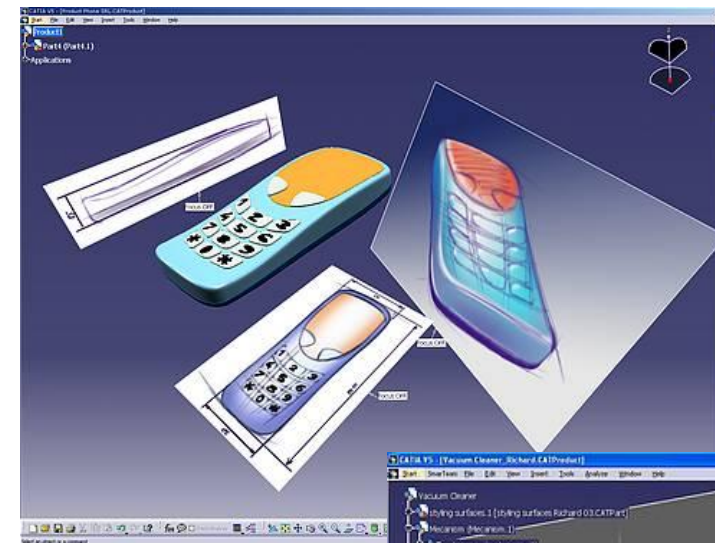
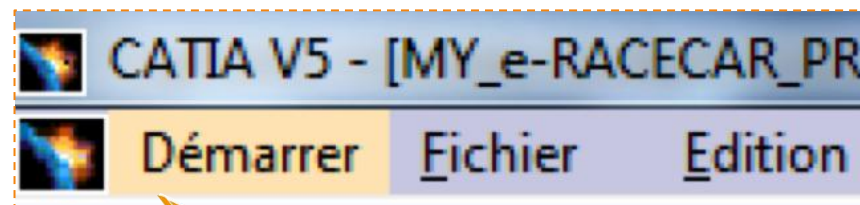
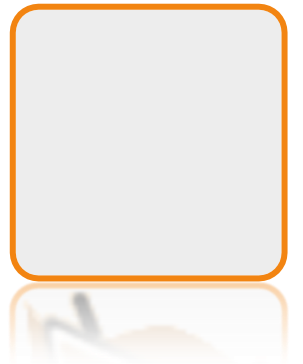


Attention it is the image which is resized and not the 3D space.  
It is a question of perspective.

# Side view

We are going to open the Sketch Tracer workbench which will allow us to import images.

1.To do this click on: **Start > Shape > Sketch Tracer**



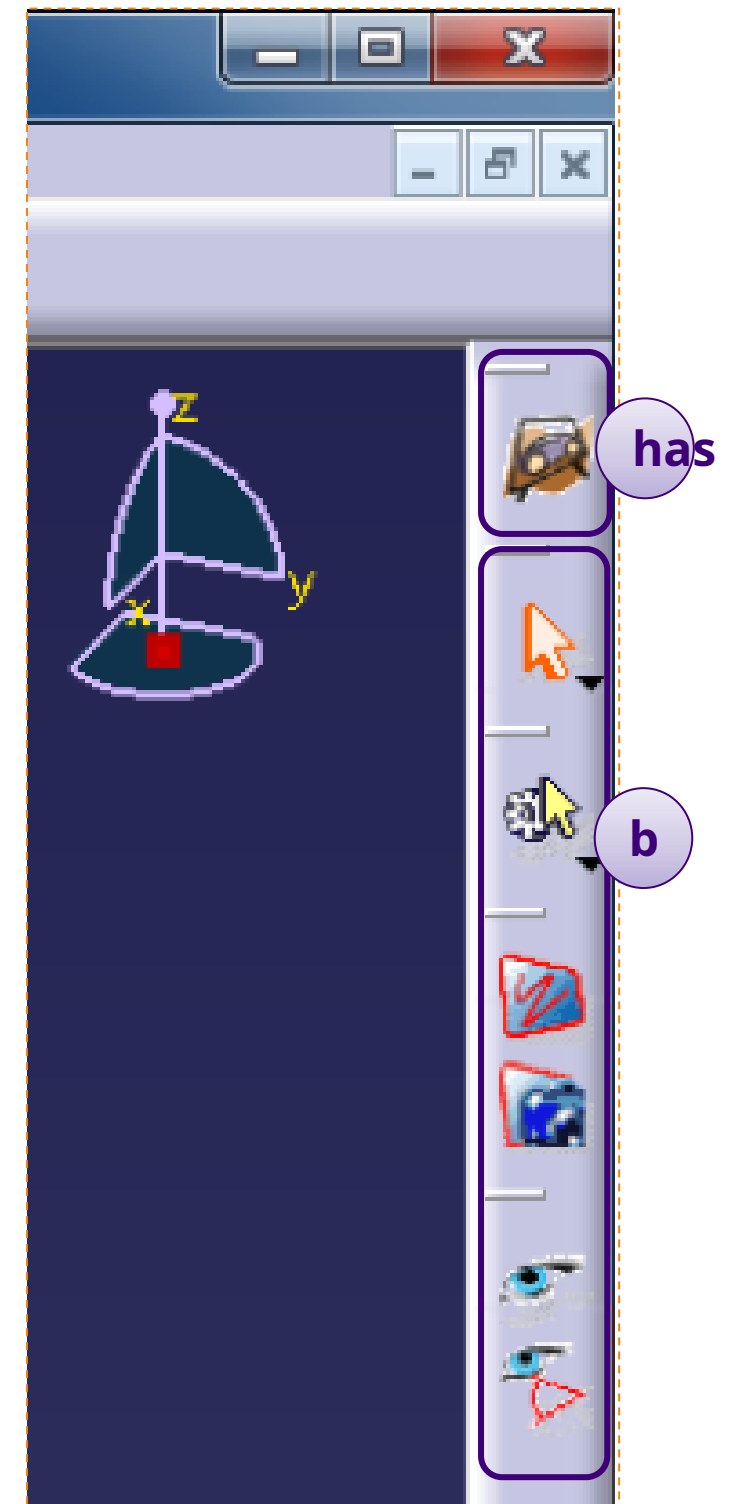
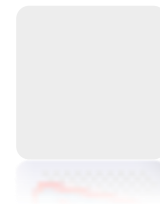
## Side view

This is what the right side of your screen should look like.  
This groups together the various tools of the Sketch Tracer workbench.

**has.Active workshop icon:**allows you to make sure that you are in the workshop in question.


**b.Toolbar:**workshop-specific tools.

- ▶ In this project only the **sketch creation tool** immersive will be used.



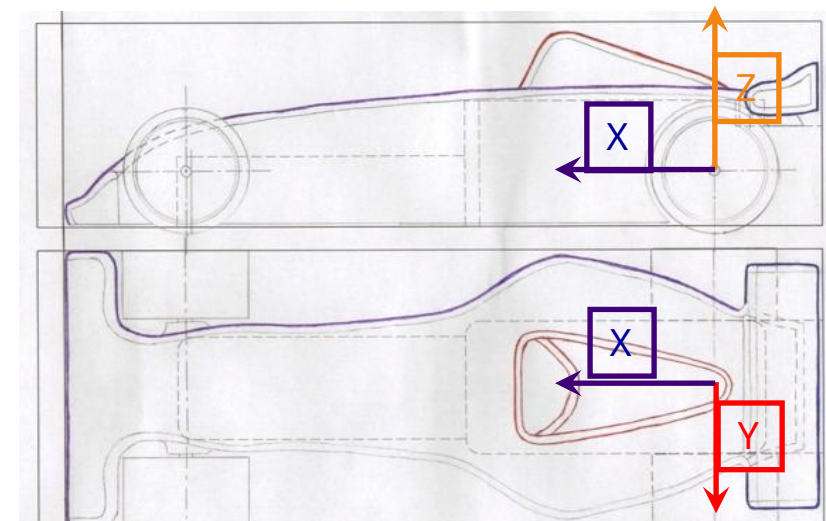
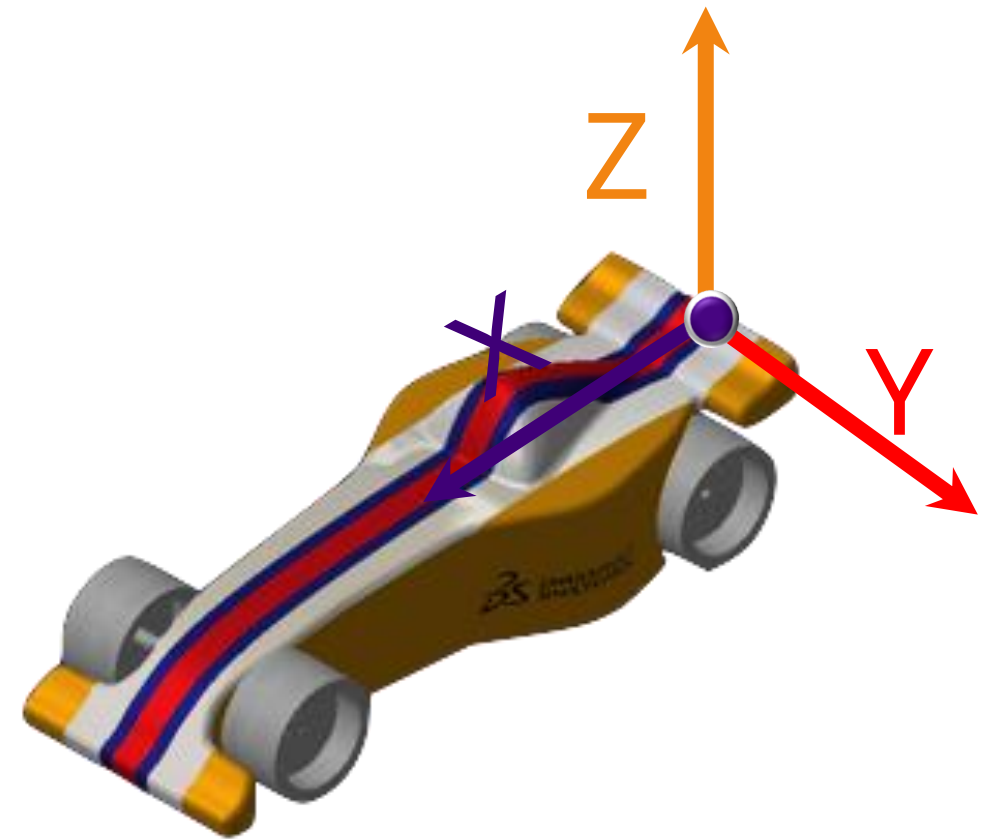
## Side view

What we want is to orient our car in the position opposite:

- ▶ Origin of the mark. 
- ▶ Z axis, will represent the vertical and point to the sky.
- ▶ X axis, will point to the front of the car.
- ▶ Y axis, will point to the left of the car.

This position is that of the isometric view of CATIA, accessible with the icon **Isometric view** opposite.

In relation to the plan of the car, this is how the mark must be placed (see opposite):





## Side view

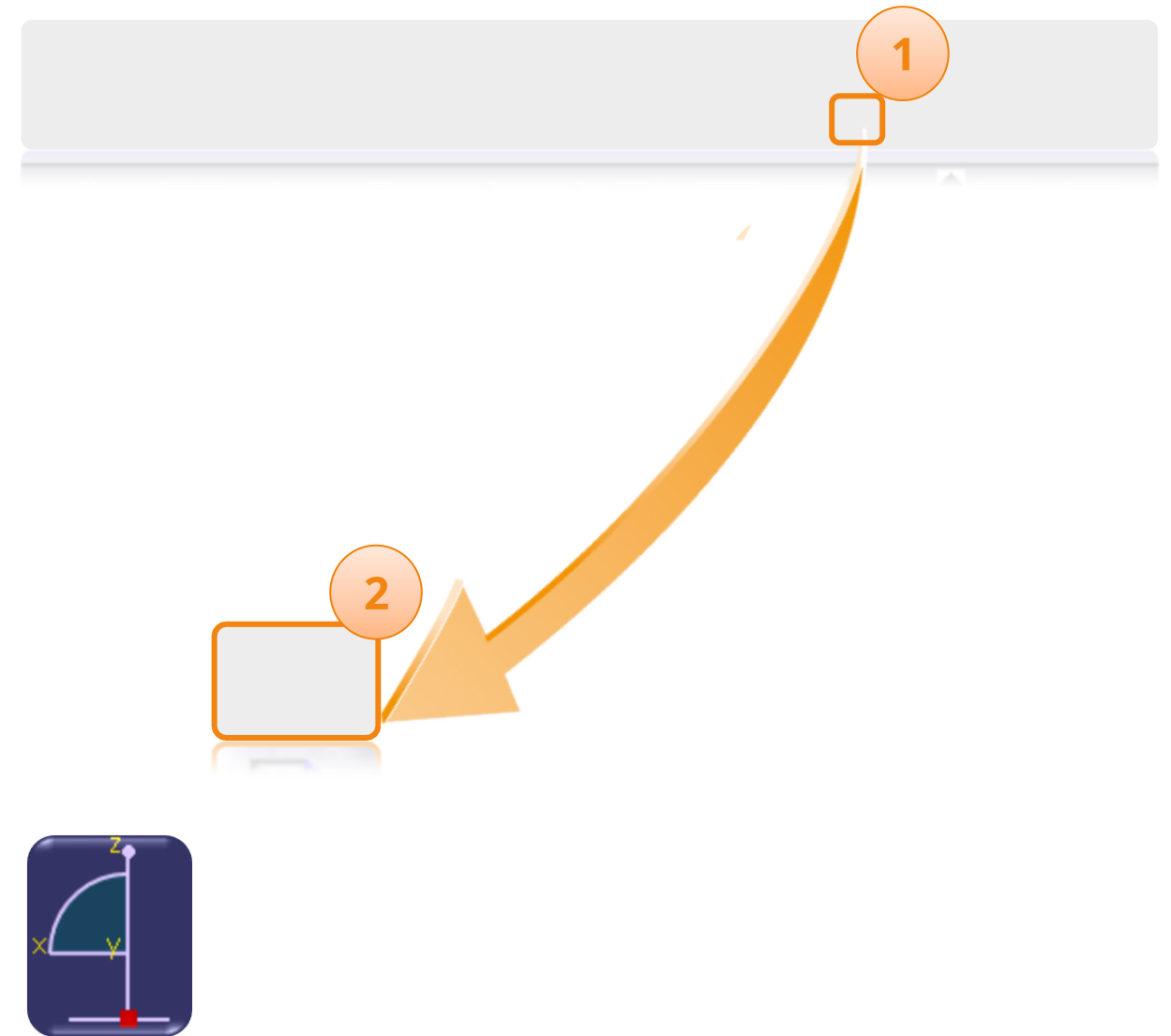
We want to import the side view first. So you have to orient the 3D view properly.

At the bottom of your screen should be the toolbar opposite.

1. Click on the **small black arrow** of this icon to display the icons of the different views by default.

2. Click on the icon of the **right view**.

You will notice that the compass orients in this way.

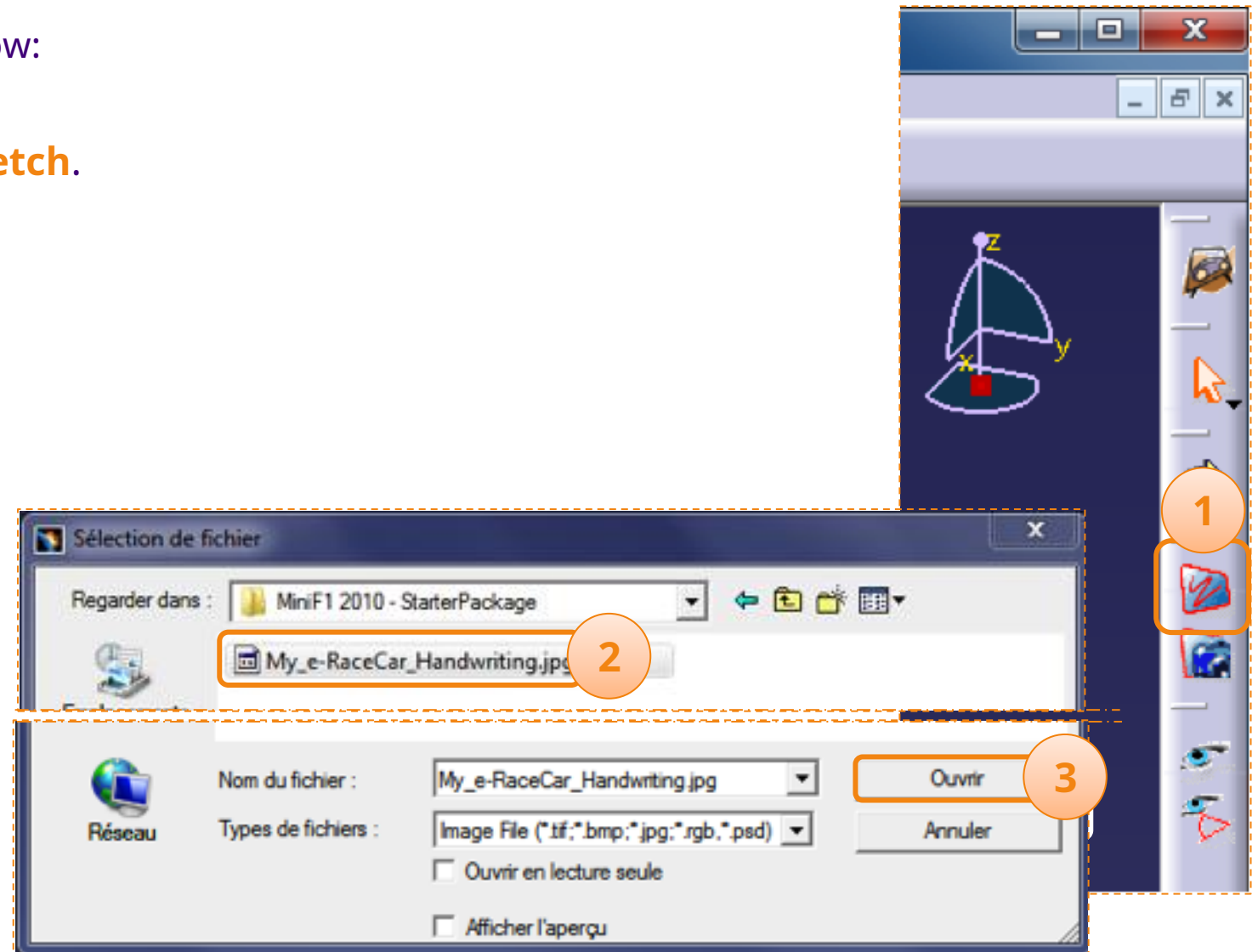


For more information on the different views,  
see the Quick Start.

## Side view

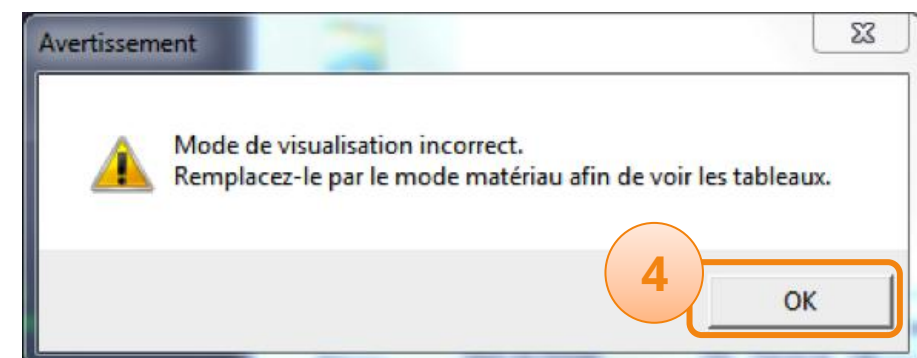
To import the image, here are the steps to follow:

1. Click the icon **Create a new immersive sketch**.
2. Select from directory **MiniF1e 2010 – StarterPackage** the image **My\_e-RaceCar\_Handwriting**.
3. Click on **Open**.



This warning may appear. This is the display mode used which is not suitable for displaying the image.

4. Click on **okay**.



# Side view

The gray rectangle in your work area represents the location of your image.

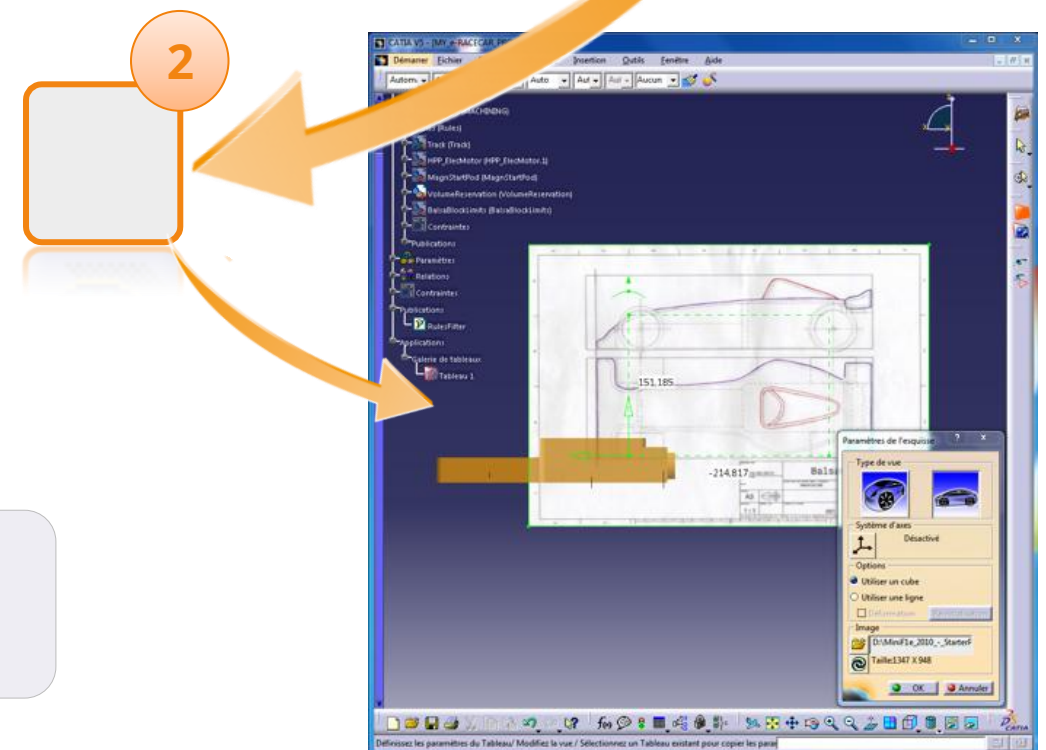
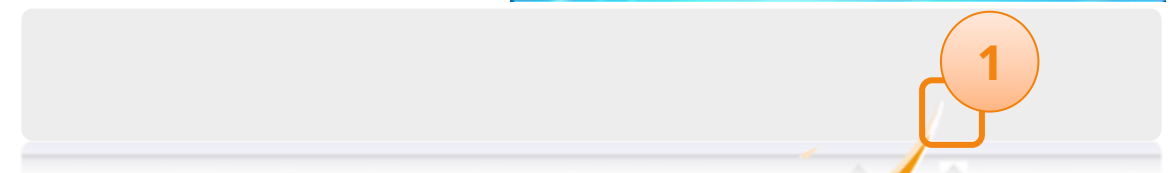
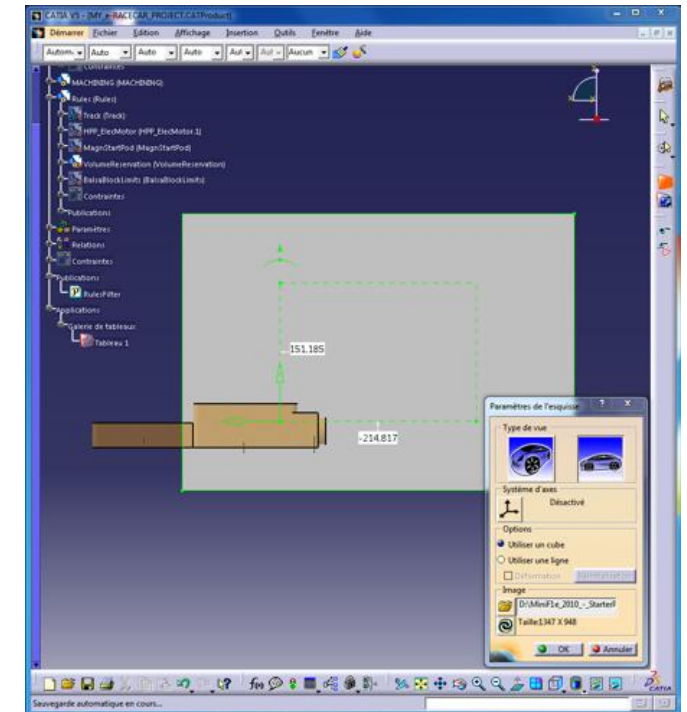
We are going to change the viewing mode.  
At the bottom of your screen should be the toolbar opposite.

1. Click on the **small black arrow**.
2. Click the icon to **Display in realistic render mode with textures**.

You should be able to view your image now.



For more information on the different modes of viewing, see the Quick Start section.



## Side view

In the window opposite which is displayed. Verify that the following options are selected:

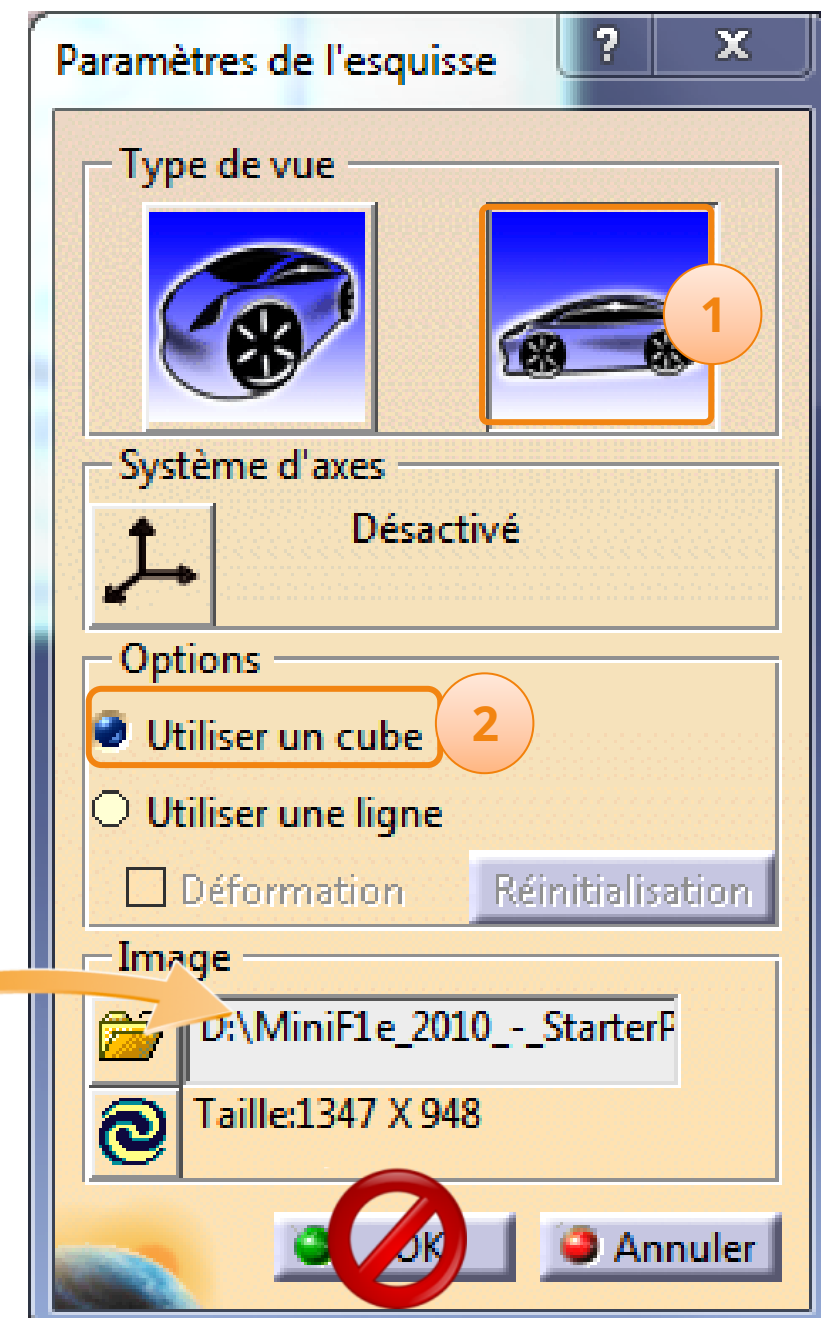
1. Click on **Apply a cylindrical view**.

2. Select option **Use a cube**.

3. **DO NOT CLICK** sure OK.



You can change the previously selected image by clicking on the icon **Open a new image**. Once selected make an update by clicking on the icon just by below to apply the change.



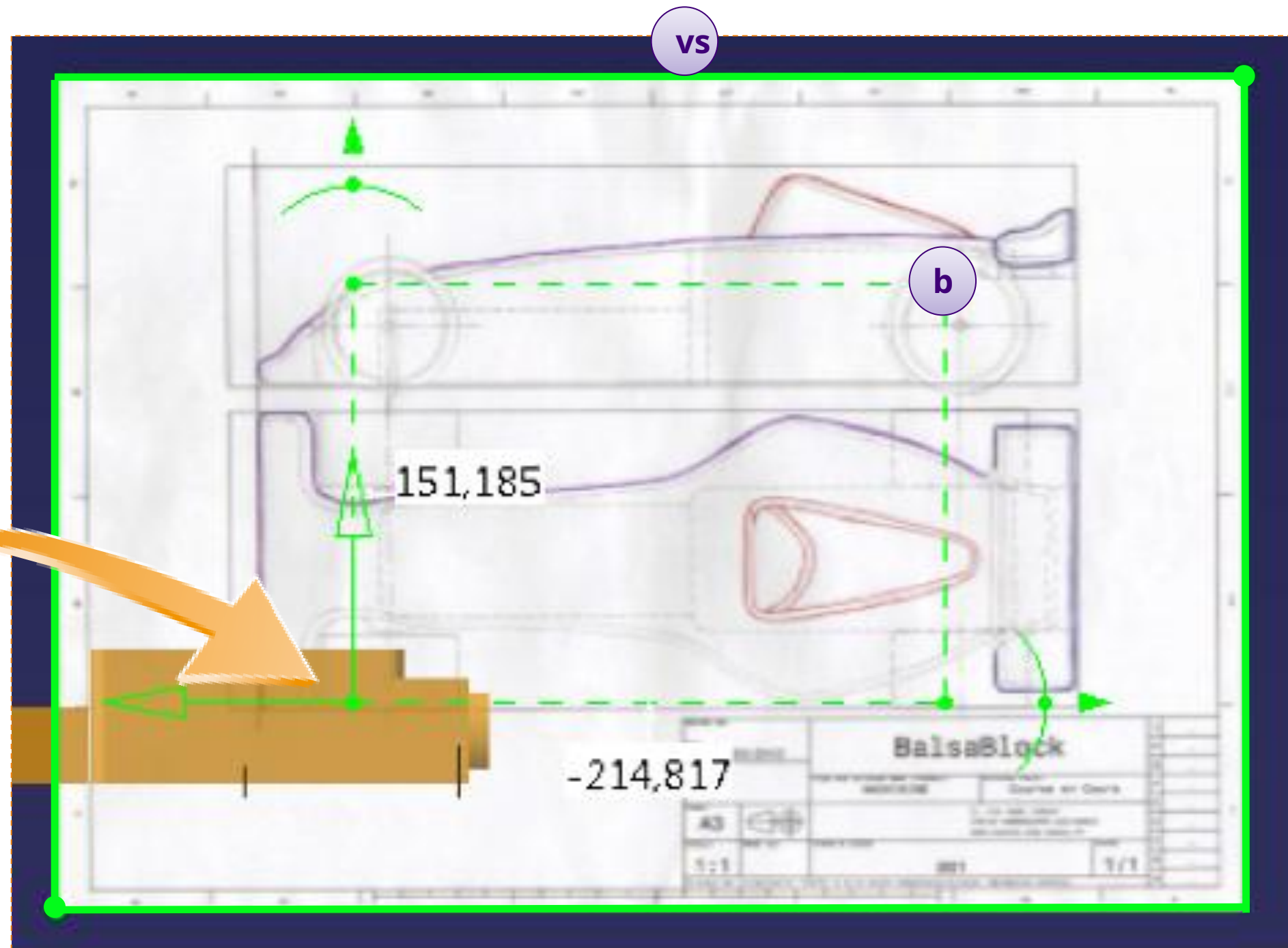
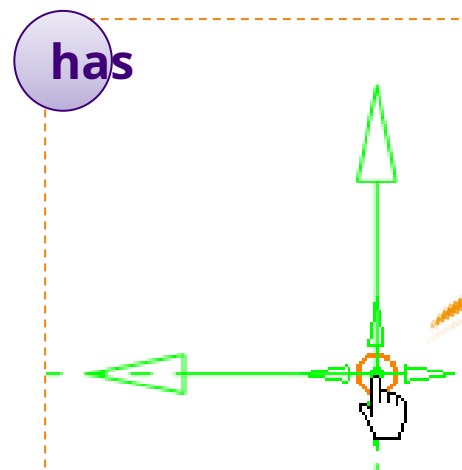
## Side view

Now, learn about the different elements that make up the screen:

has. Current marker position.

b. Scale frame (green dotted lines).

vs. Image frame (solid green lines).





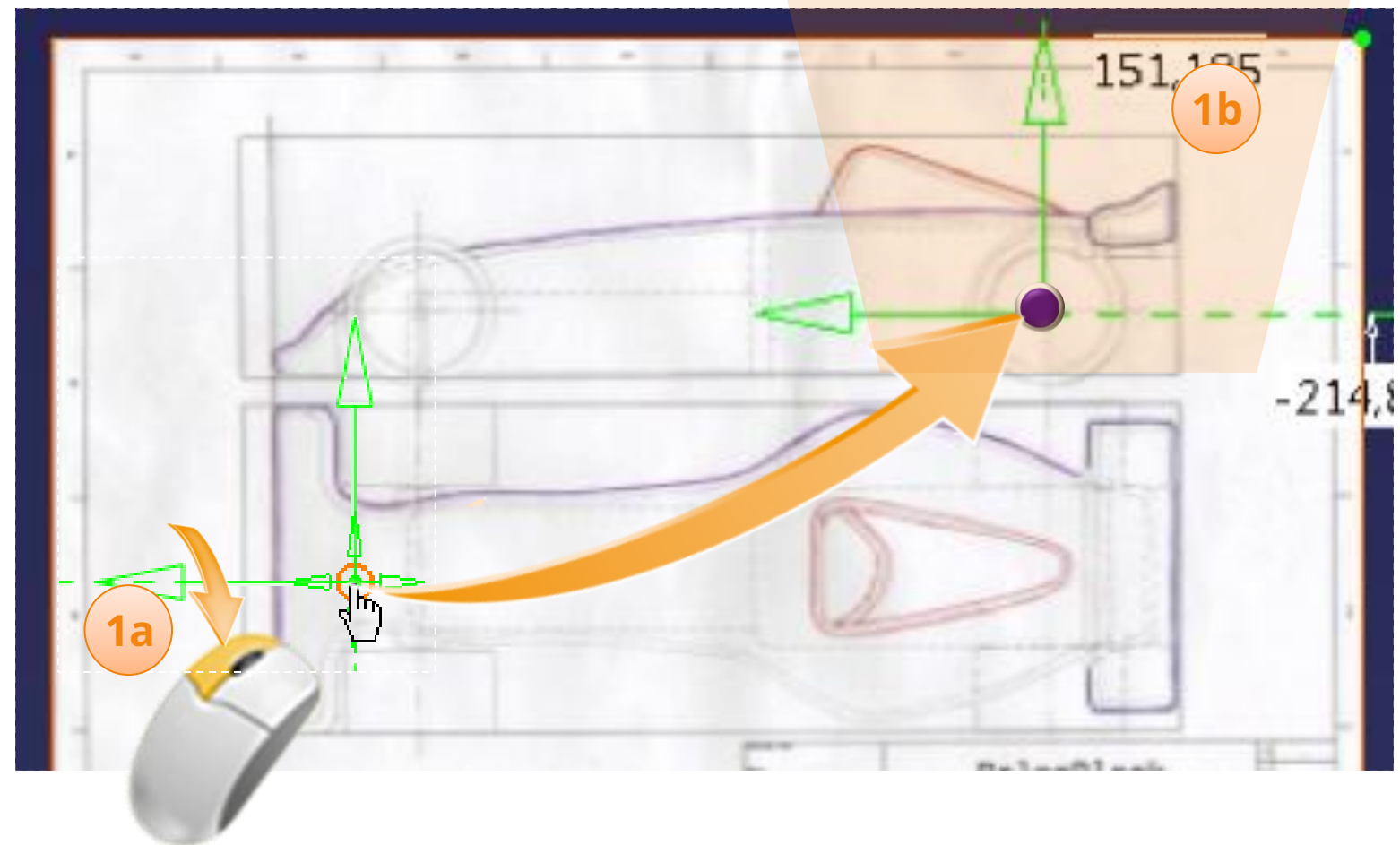
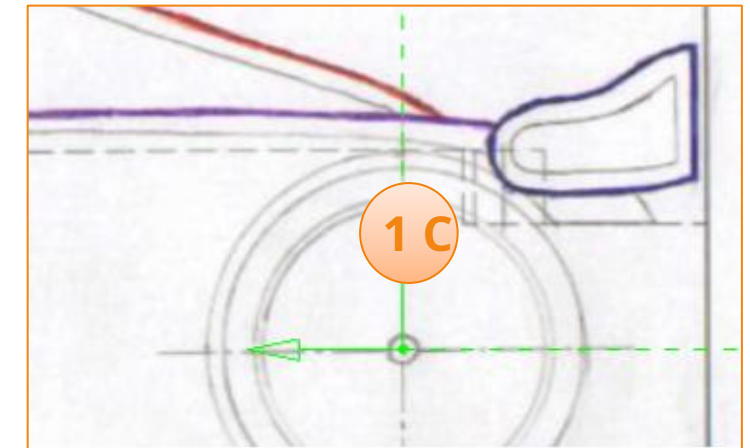
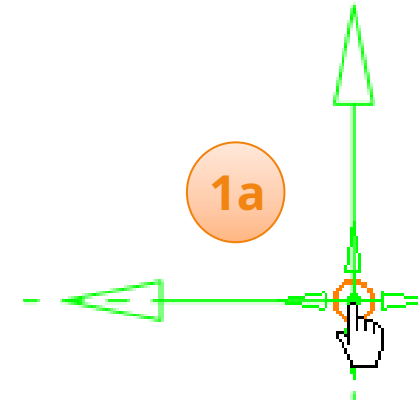
## Side view

1. The second step is to position the marker relative to the side view:

has. Place your mouse at the origin of the marker.  
The orange circle means you will move the marker in two directions at the same time.

b. Click and hold the click to roughly move the marker to the origin shown in the image below.

vs. Zoom in to fine-tune marker placement.



## Scaling tools

The ladder frame has several manipulation tools.

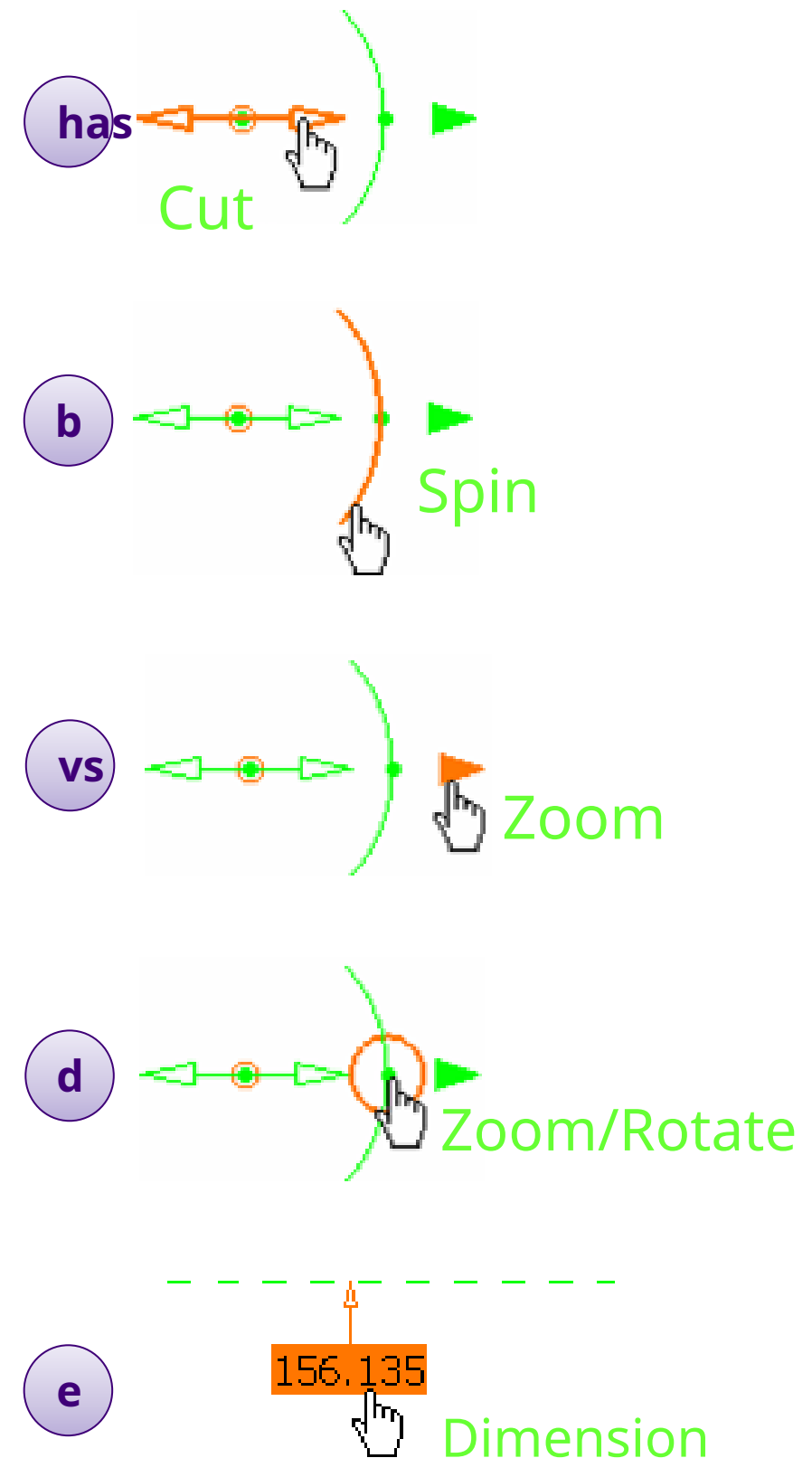
has. The double arrow that only appears when you bring the mouse closer to the corner of the rectangle. It allows you to move the limits of the dotted rectangle.

b. The arc of a circle which makes it possible to make a rotation of the reference.

vs. The solid triangle at the end that allows you to zoom.

d. The point on the arc (b) that combines zoom and rotation.

e. And two rectangles containing the dimensions of the rectangle, which will allow scaling.



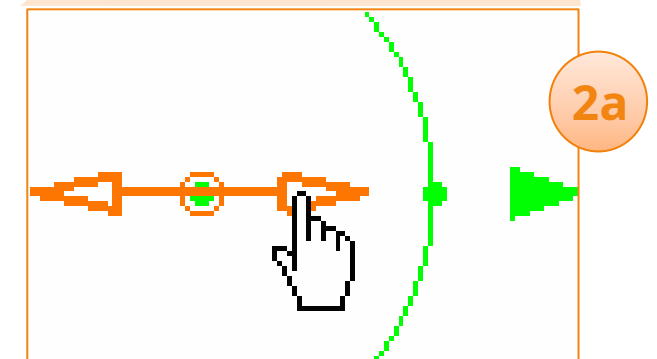
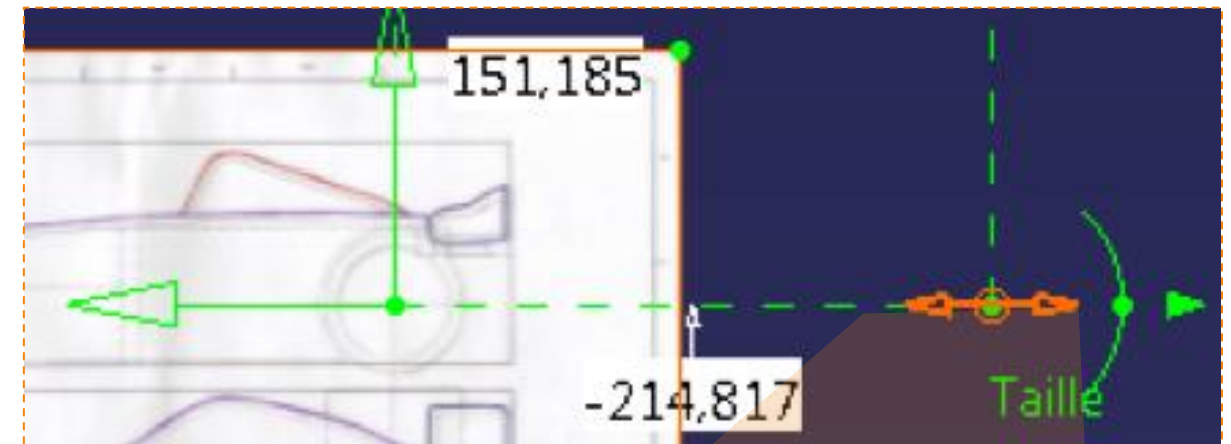
## Side view

2. The third step is to set the scale of the image:

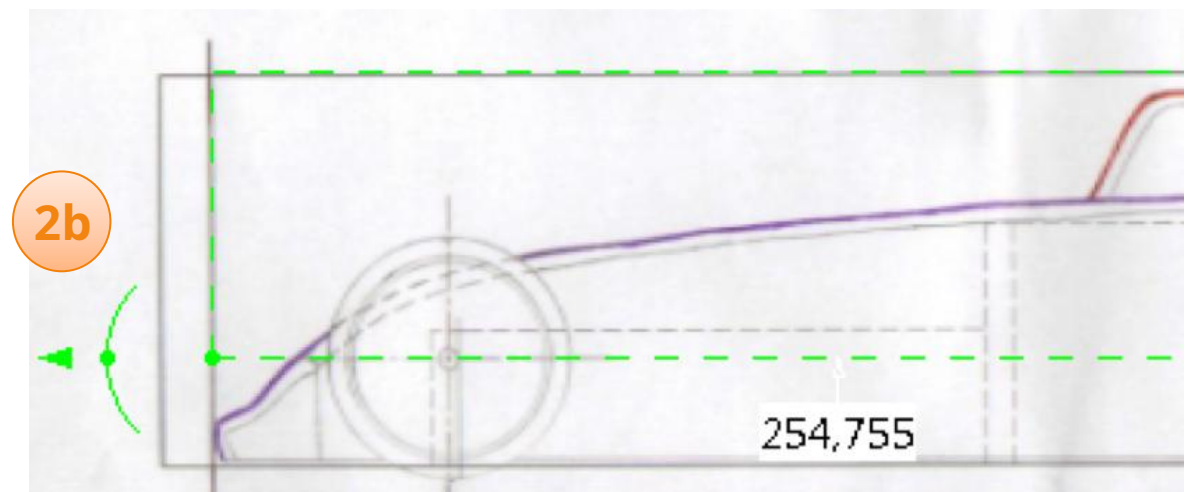
has. Click and hold the bottom right corner of the ladder frame (arrows appear when you bring your mouse near the corner).



*(Do not confuse with the arrow located on the right of the arc of circle which does not have the same utility)*

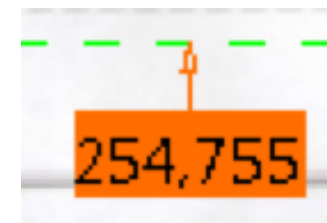


b. While holding the click, move the frame to the end of the balsa block. Do not hesitate to zoom in to be more precise.



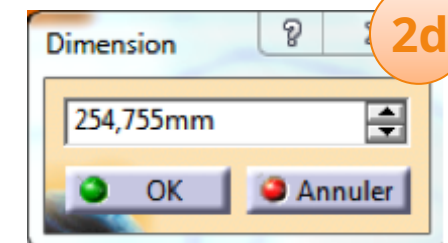
## Side view

vs. Click on the value located in the rectangle that points to the horizontal side of the frame (the value currently entered may be different).



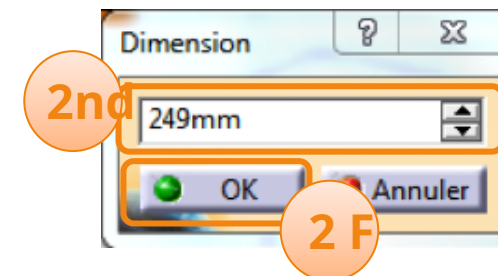
2c

d. The window opposite appears.



2d

e. Enter the value [249mm].



2nd

2 F

f. Click on **okay**.

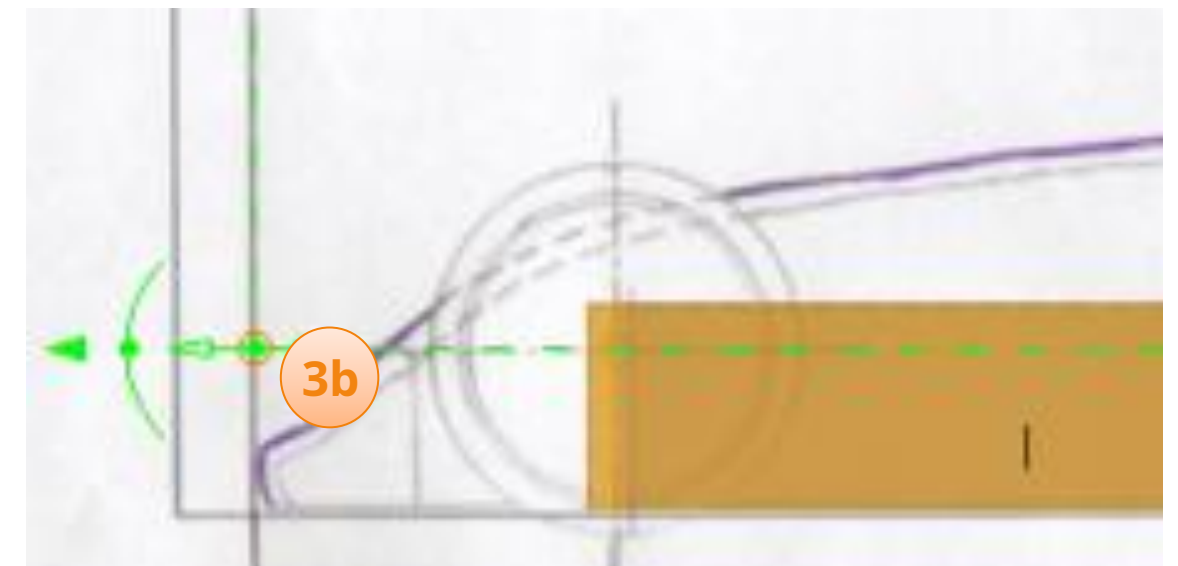
## Side view

3. If the horizontal boundaries of the balsa block and the drawing do not overlap, you can use the rotate tool to correct this alignment issue:



has. Click and hold the click on the arc at the end of the ladder frame.

b. Move the mouse to skew the guide and fix this orientation issue to get the desired result.





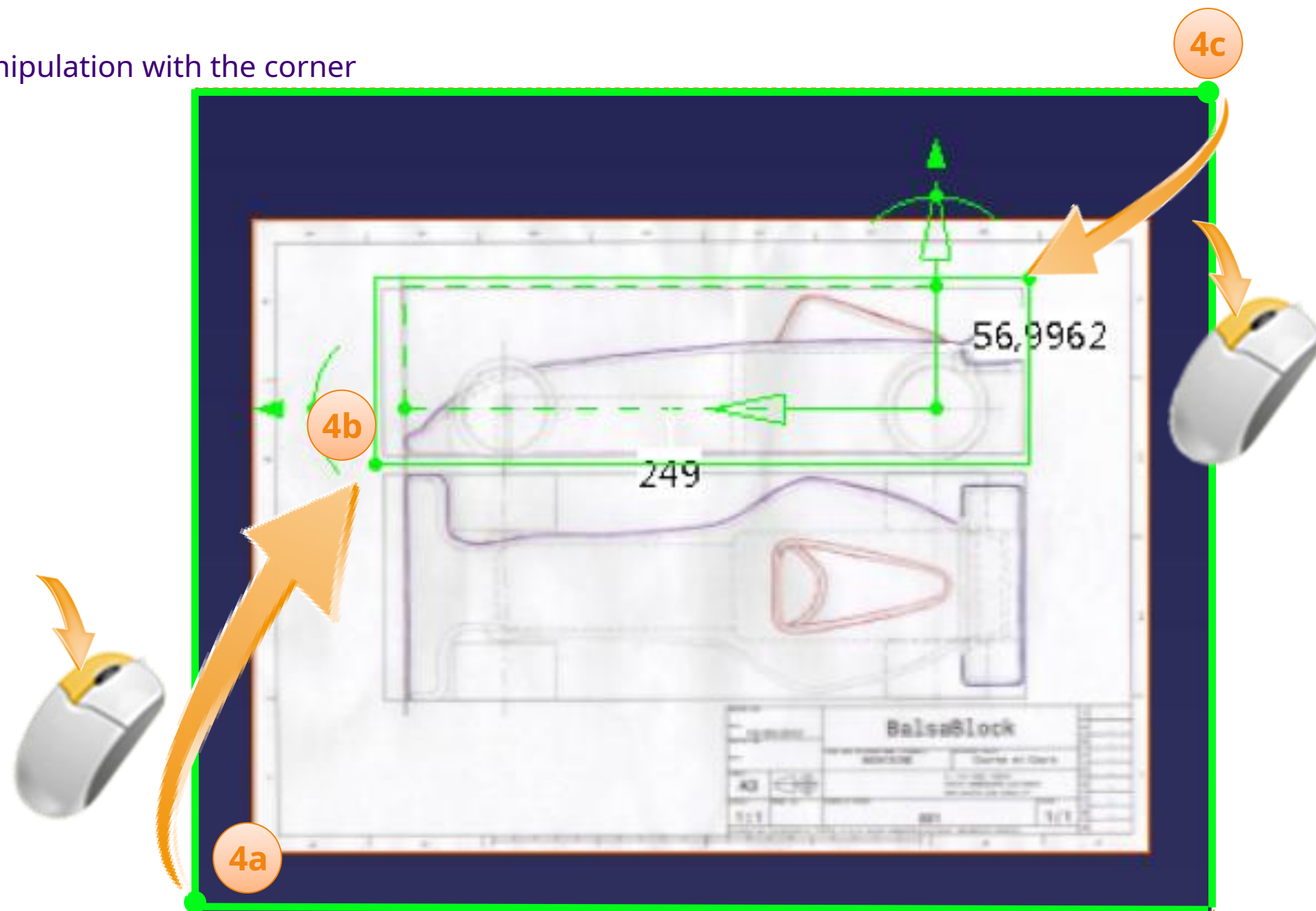
## Side view

4. The last step is to crop the image to keep only the part concerned. In this case only the side view interests us. For that :

has. Click and hold the click on the lower left corner of the image frame.

b. Move this corner so that it only surrounds the side view.

vs. Perform the same manipulation with the corner upper right.



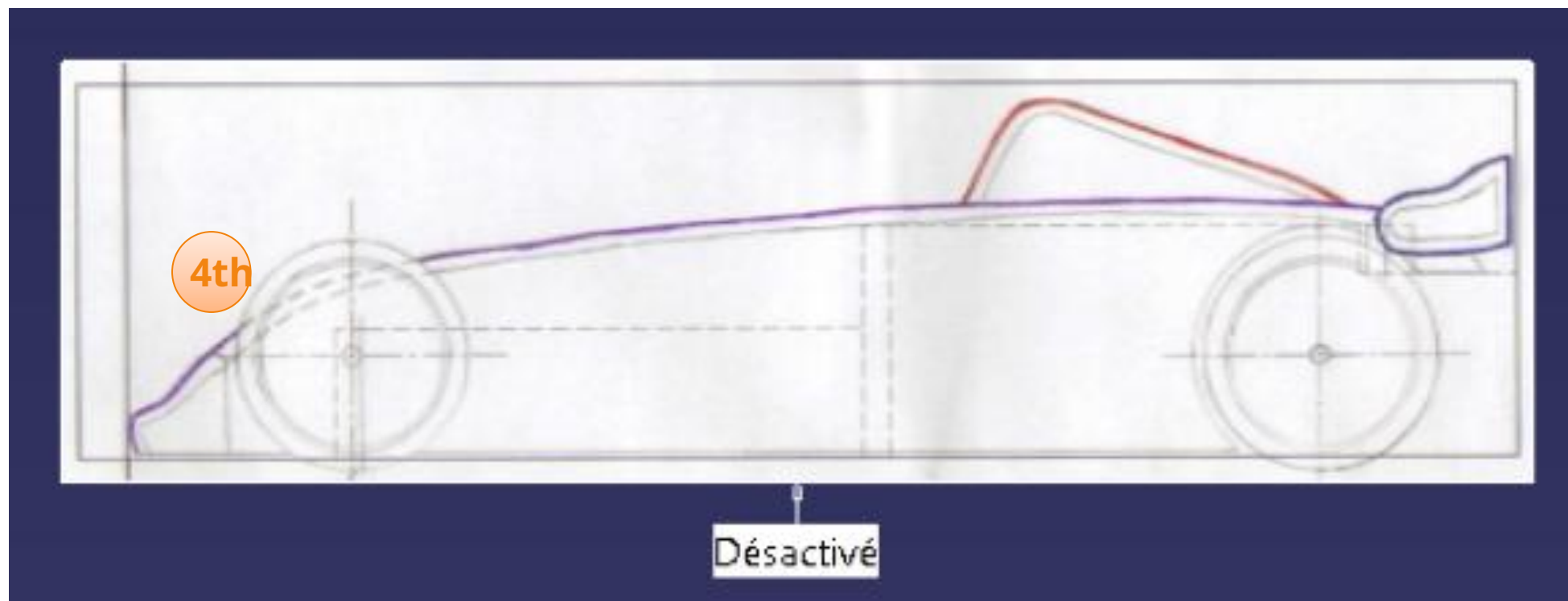
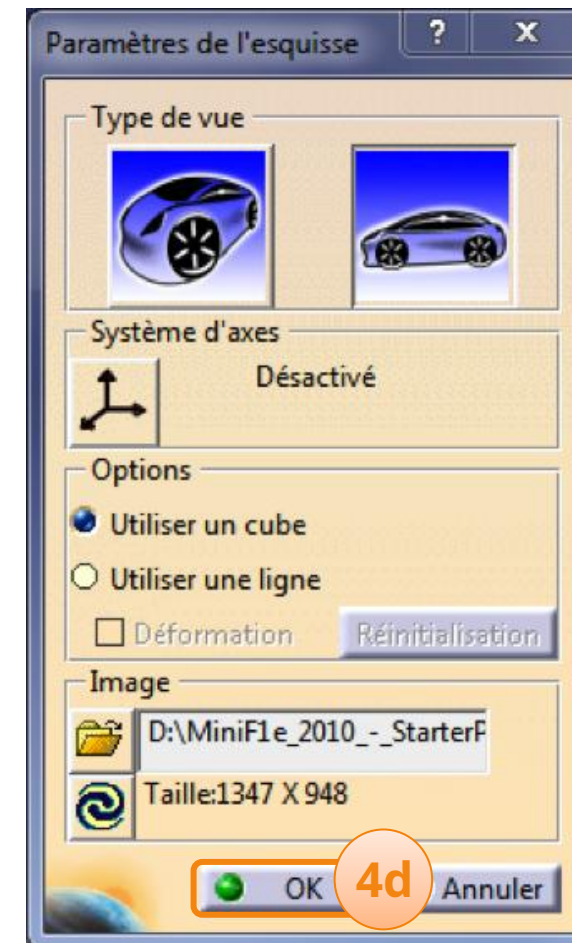
## Side view

d. Validate by clicking on **okay**.

e. You can check that the wireframe version (in brown) of the balsa block matches well with the one in the image as below.



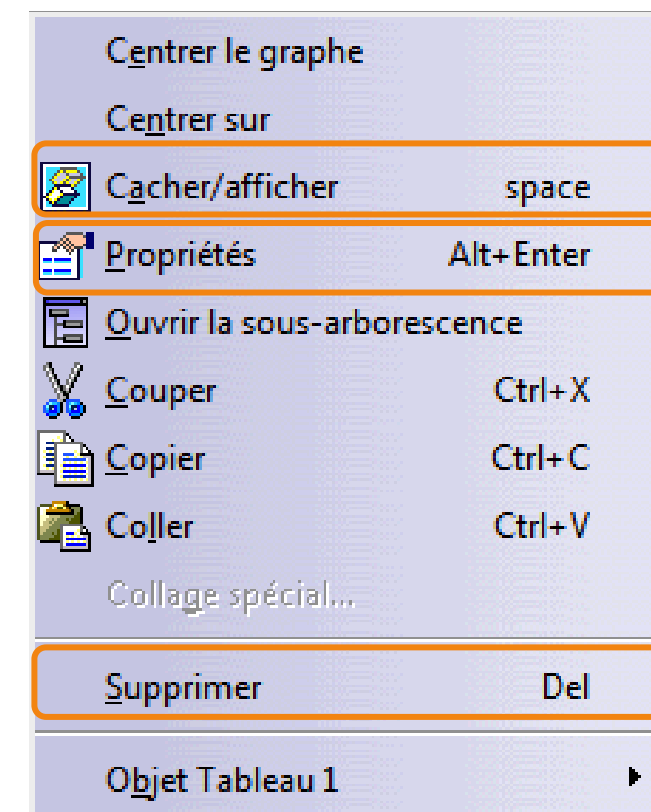
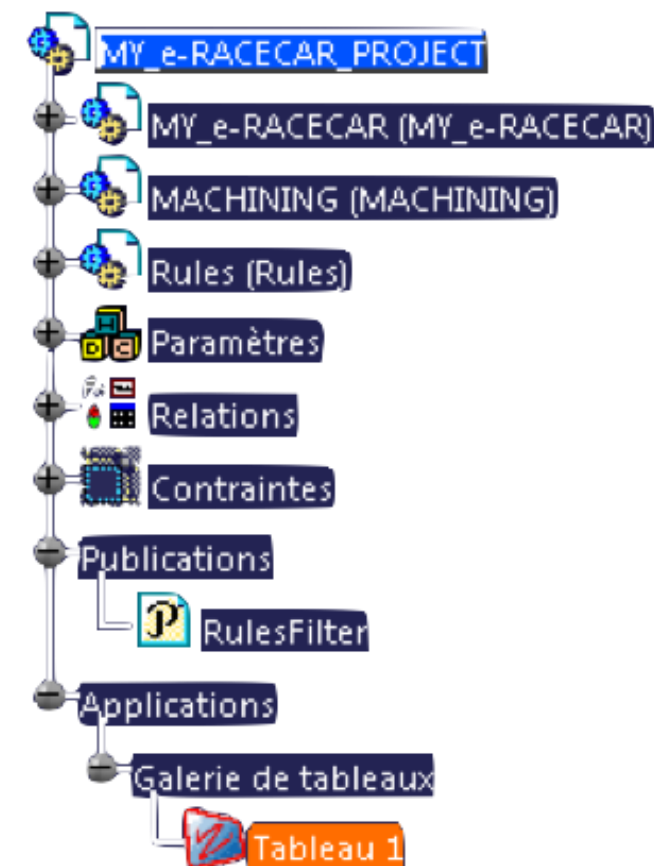
It is essential to have an equivalent result before continuing to build your car.



## Side view

You can now see in the last line of the specification tree which is called **Apps** the presence of "Table 1". You can right-click on it to:

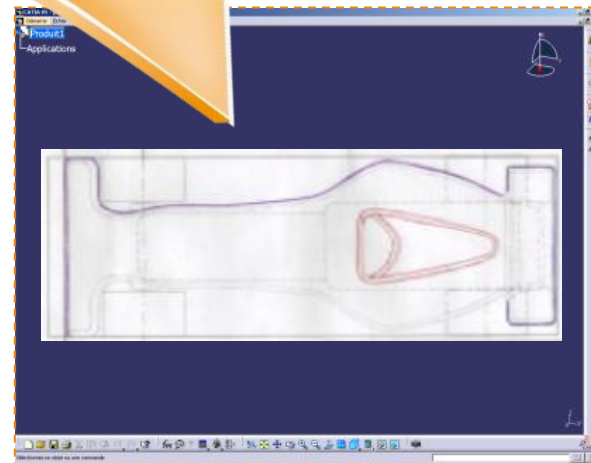
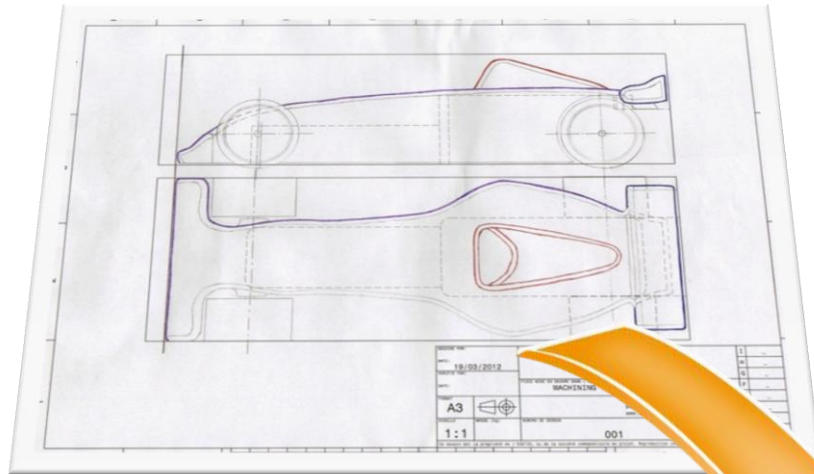
- ▶ the to hide Where display it grace at the option **Hide/Show**.
- ▶ Access its properties.
  - Change its name. We advise you to rename "Table 1" to "SideView".
  - Add a transparency effect.
- ▶ Delete it.



# Top view

We will perform  
previous ones in order to define

at the steps  
above.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
    - i. Introduction.
    - ii. Side view.
    - iii. Top view.**
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a plan.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

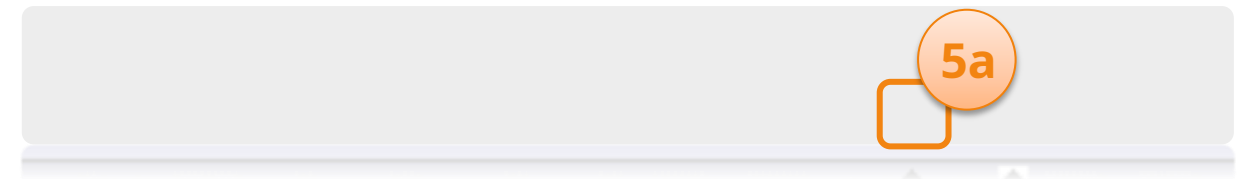
# Top view

We are now going to insert the top view, for this we are going to follow the same method as before.

5. We are going to orient the view in top view:  
has. Click on **the little arrow** to display all predefined views.

b. Click on the icon of the **top view**.

vs. Check compass orientation.

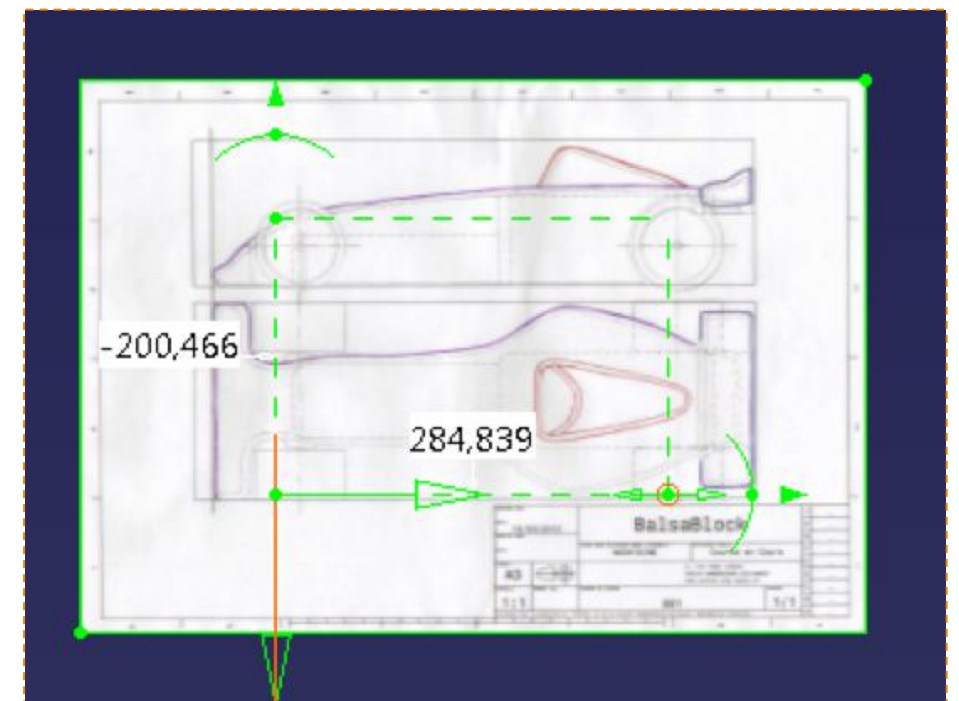


6. We will now import the image. For that:

has. Click the icon **Create a new immersive sketch**.

b. Select from directory **MiniF1e 2010**  
– **StarterPackage** the image being called **My\_e-RaceCar\_Handwriting**.

vs. Click on **Open**.





## Top view

7. We will position the marker relative to the top view:

has. Place your mouse at the origin of the marker.  
The circle means that you will move the marker in two directions at the same time.

b. Click and hold the click to roughly move the marker to the level of the origin indicated on the image below.

vs. Zoom in to fine-tune marker placement.

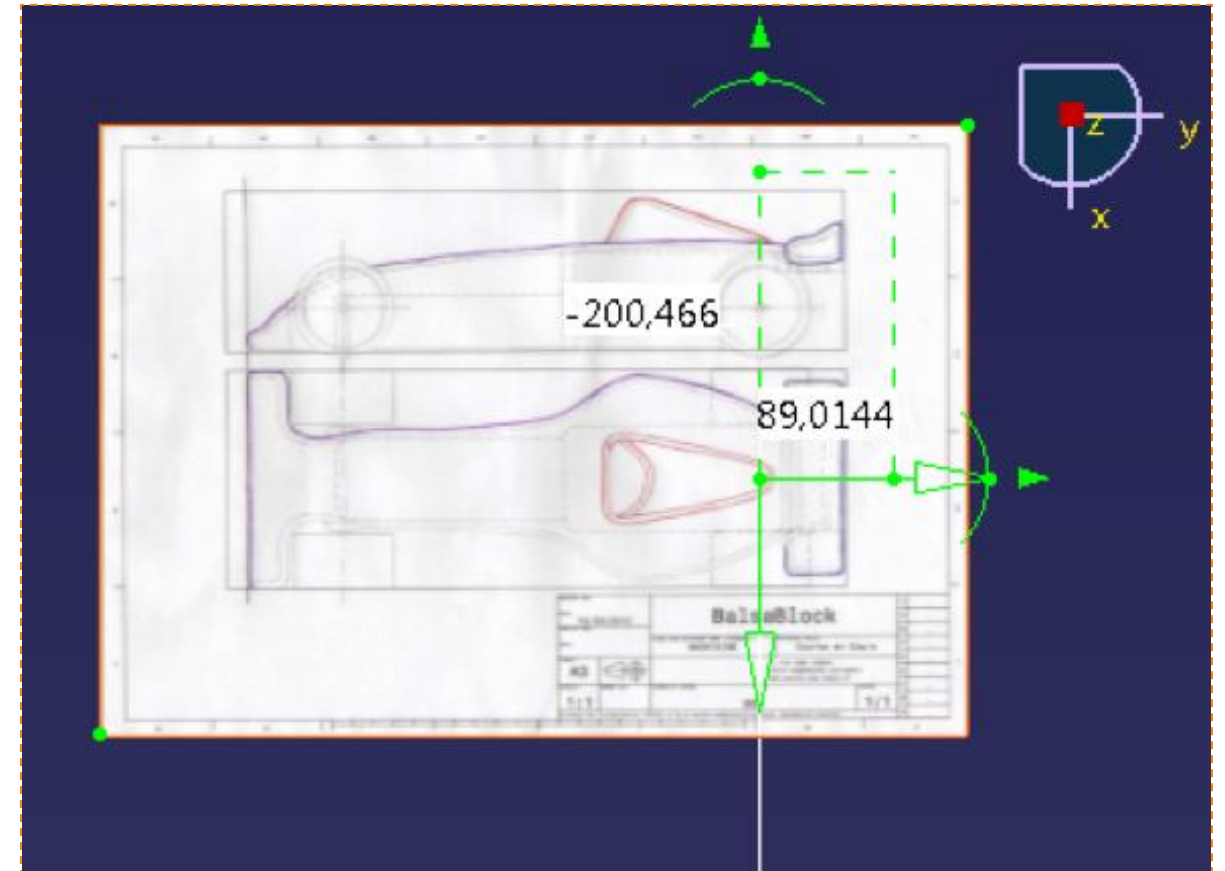
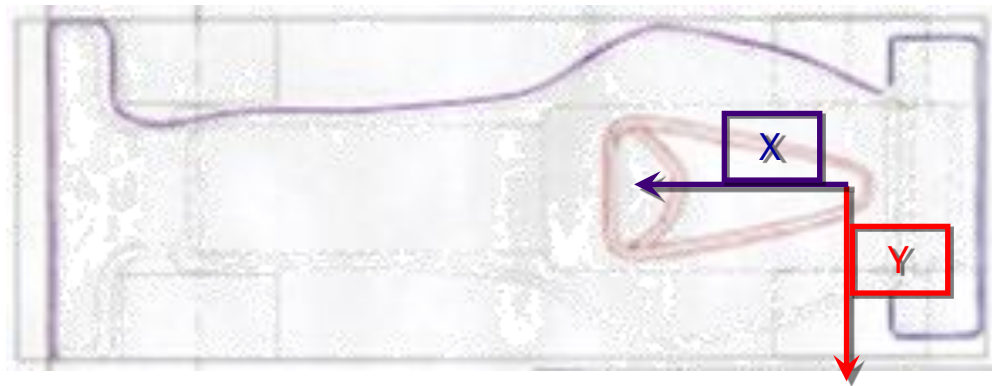


For more information on handling of the view, consult the method sheet corresponding.



## Top view

Thanks to the orientation of the balsa block and comparing with what we want, you can see that the orientation of the marker is not good.



## Top view

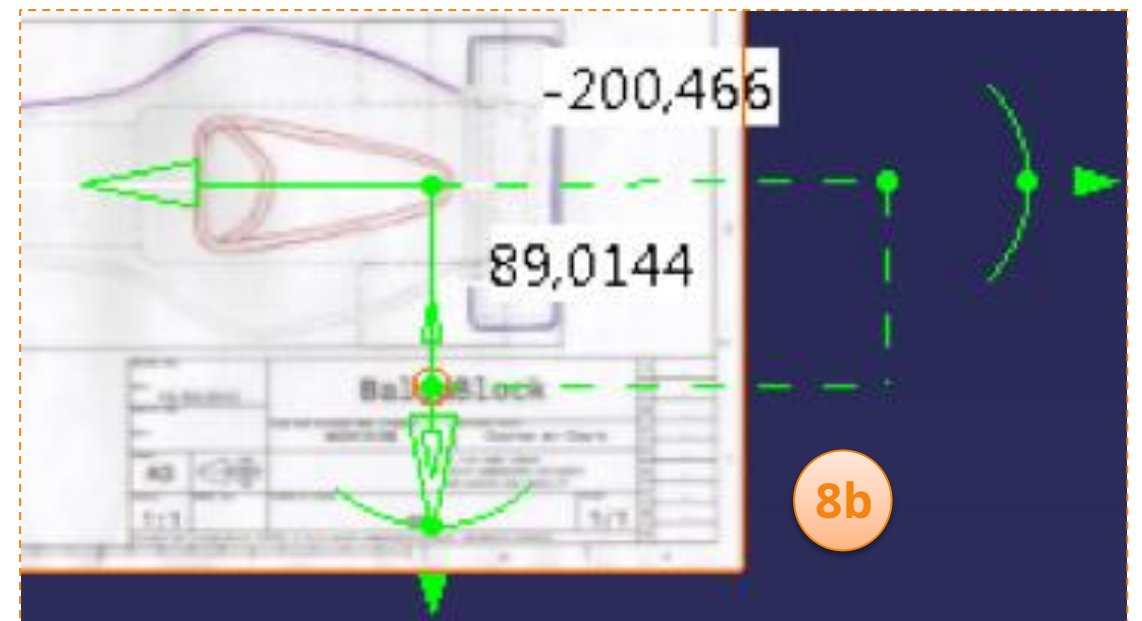
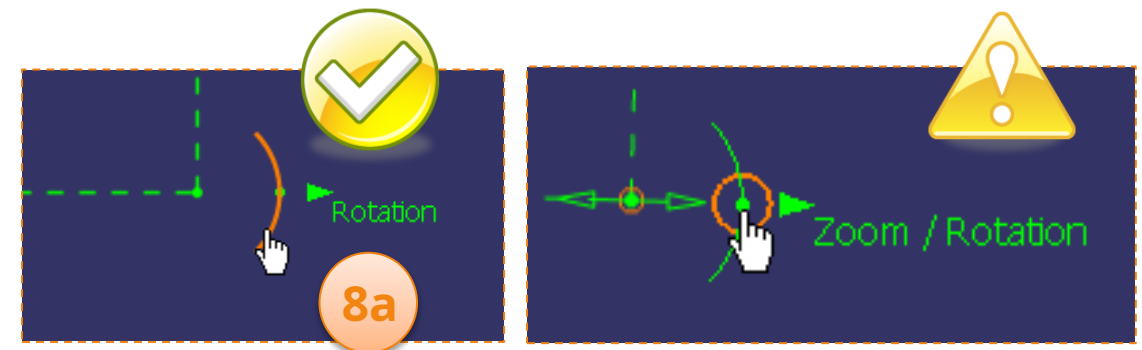
8. We will therefore reorient the marker:

has. Click and hold the click on one of the circular arcs present at the extremities.



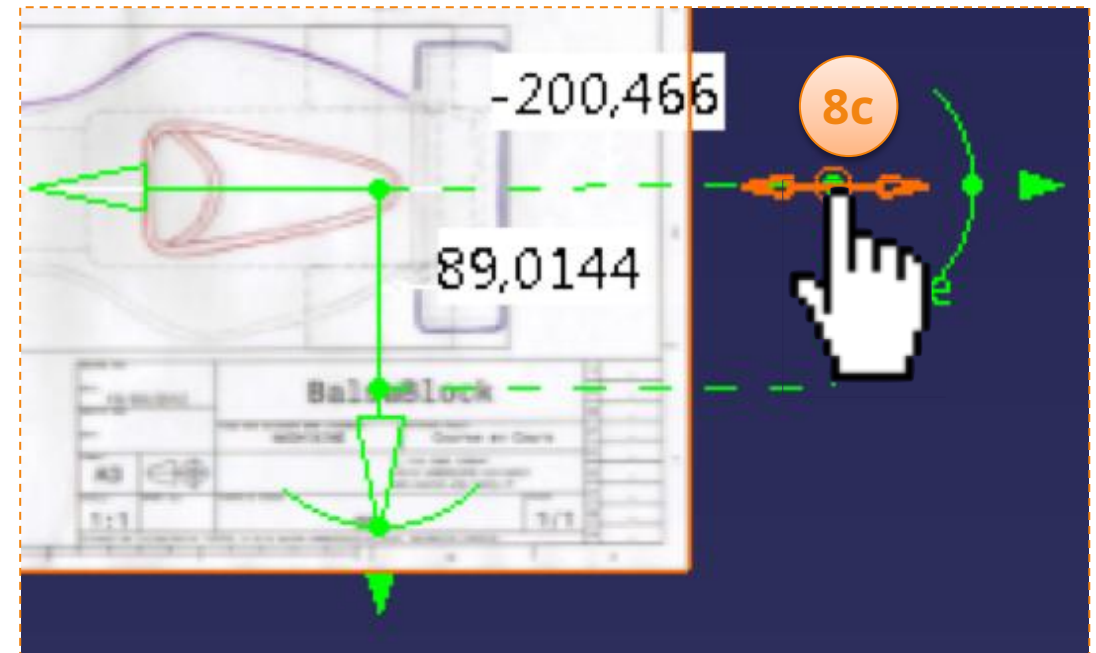
Be careful not to click on the point in the center of the arc of a circle which groups the zoom and rotate!

b. Move the mouse, which will rotate the area representing the engine block. Do it enough so that the block is roughly horizontal as shown.



## Top view

vs. Click and hold the click on the top right corner of the ladder frame. (Arrows appear when you bring your mouse closer to the corner).

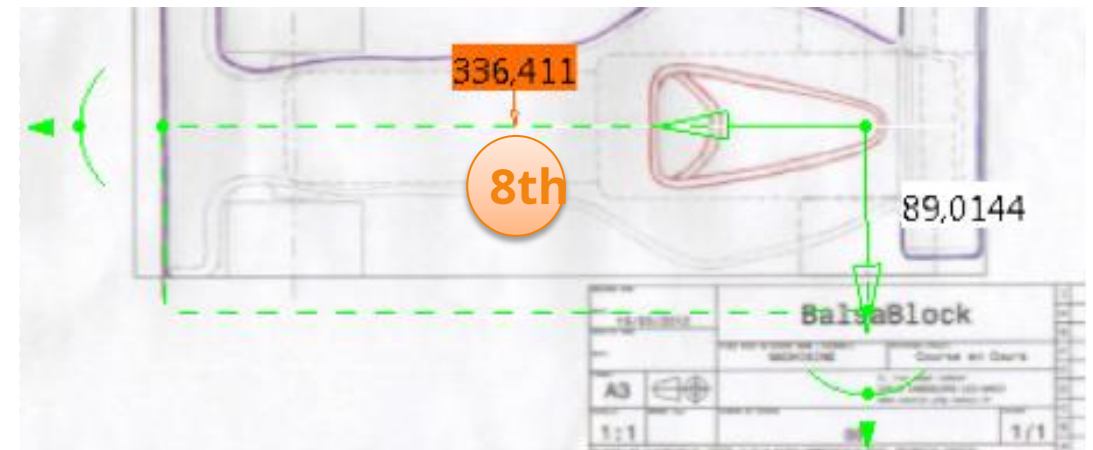


d. While holding the click, move the frame to the end of the balsa block. Do not hesitate to zoom in to be more precise.

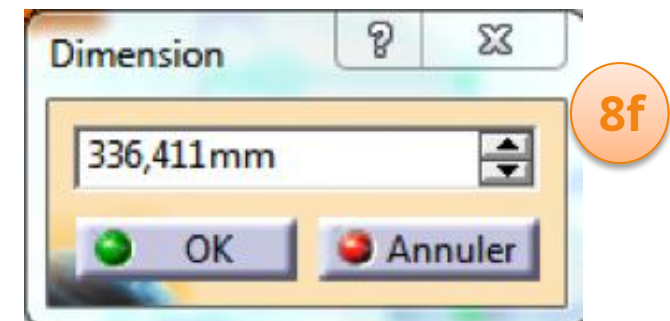


## Top view

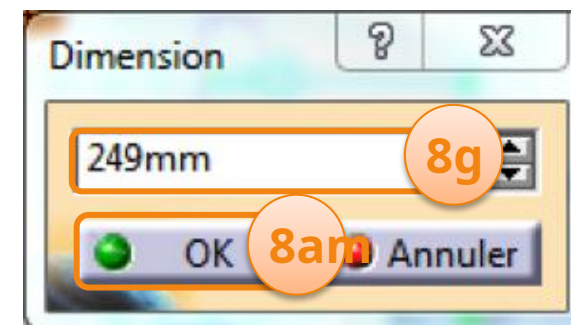
- e. Click on the value located in the rectangle that points to the horizontal side of the frame. (The value currently listed may be different)



- f. The window opposite appears.



- g. Enter the value [249mm].



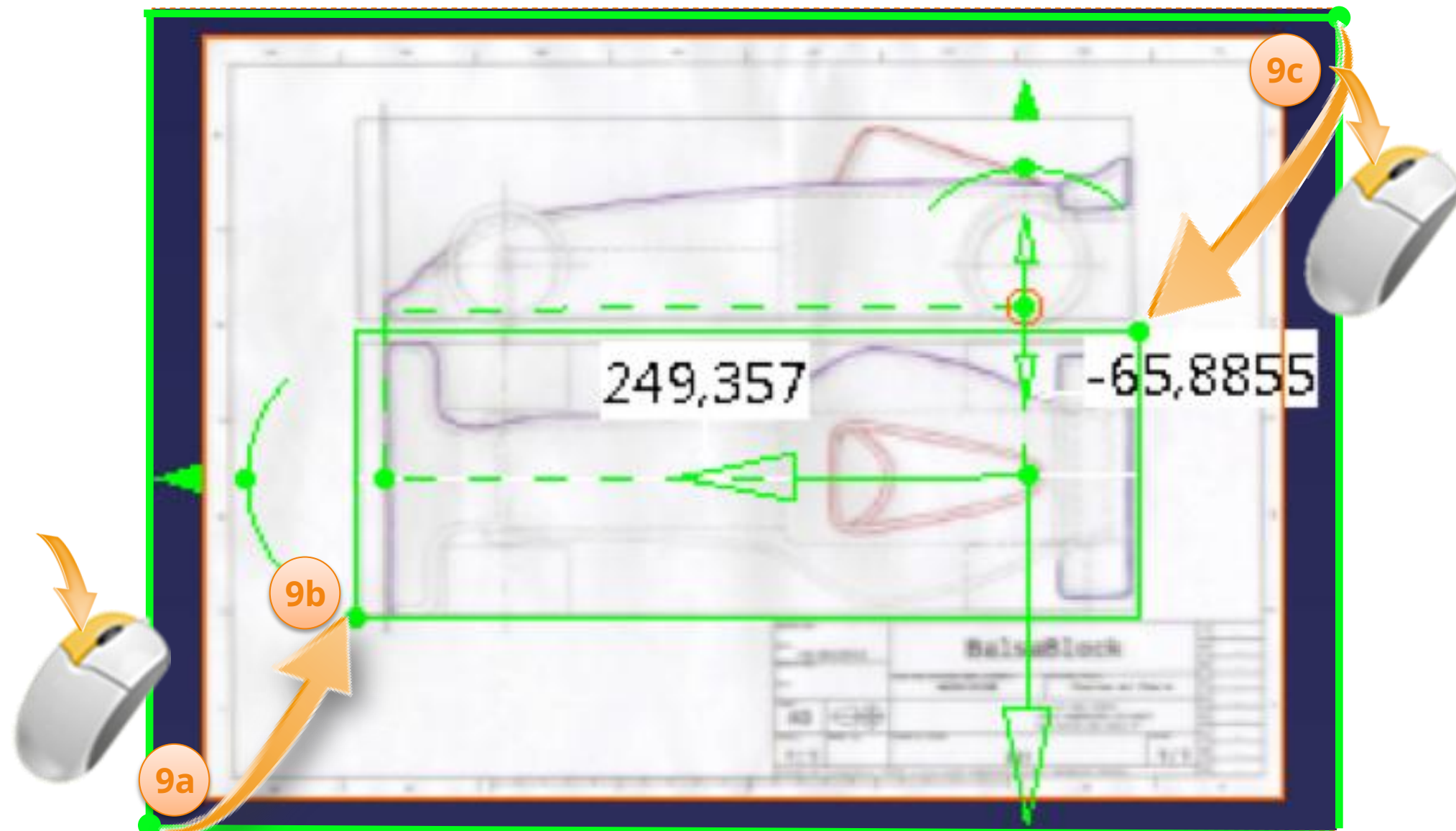
- h. Click on **okay**.



## Top view

9. Finally we will crop the image:

- has. Click and hold the click on the lower left corner of the image frame.
- b. Move this corner so that it only surrounds the top view.
- vs. Perform the same manipulation with the corner upper right.

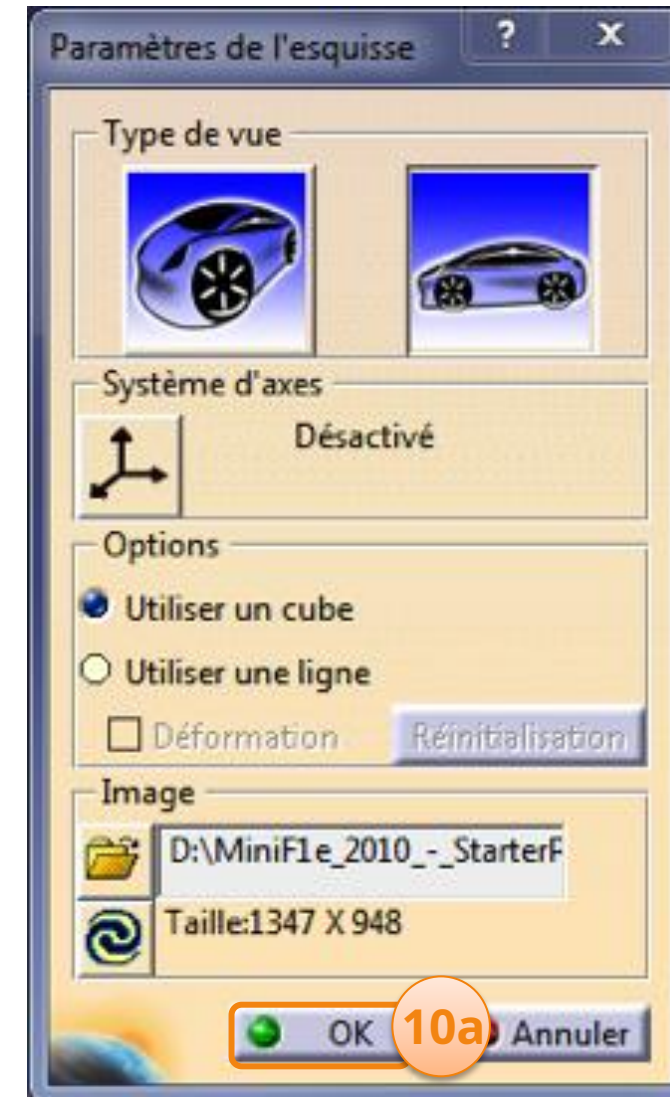
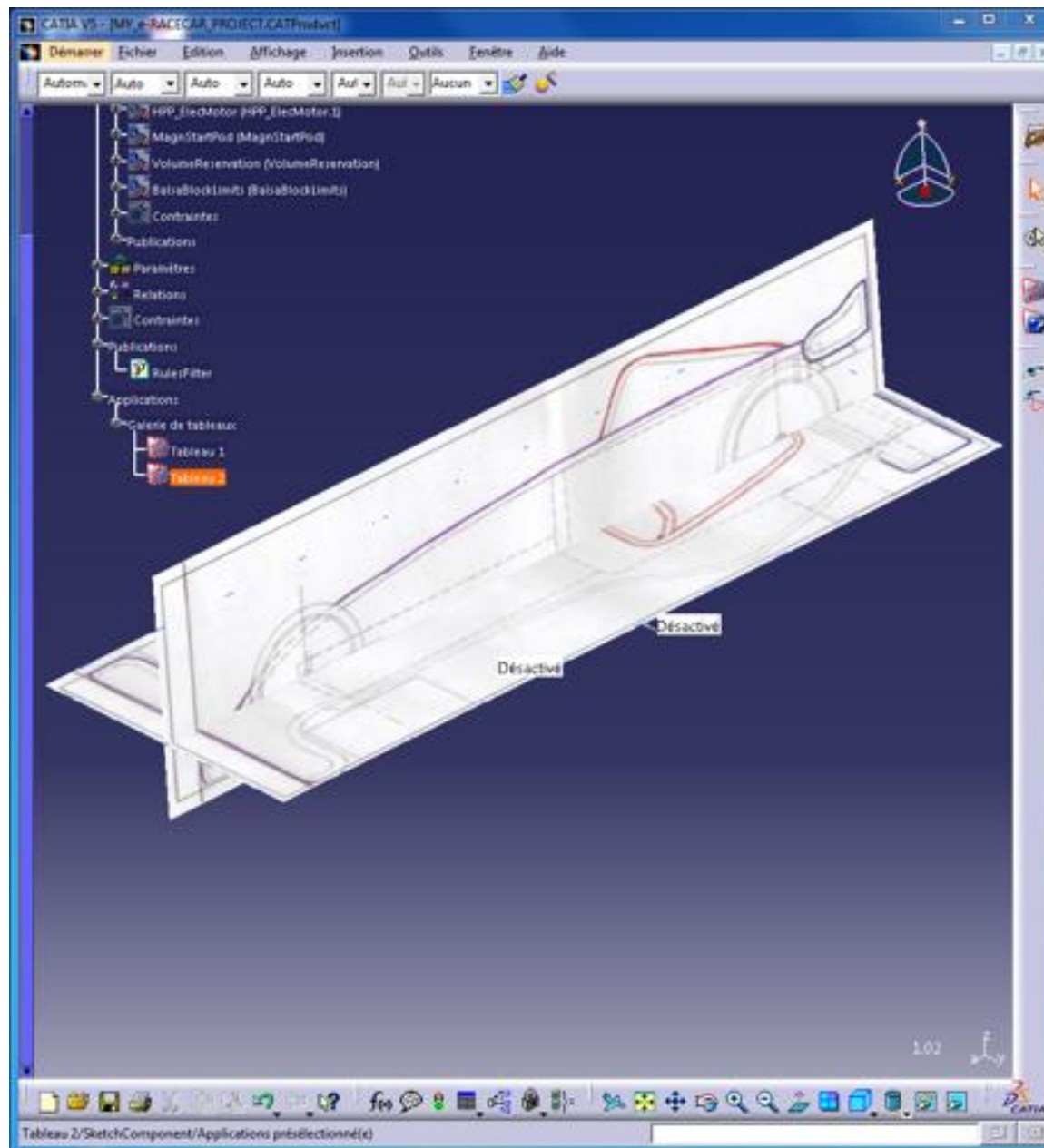


## Top view

10. We are going to validate the configuration of the immersive sketch:

has. Click on **okay** in the window of **Sketch parameters**.

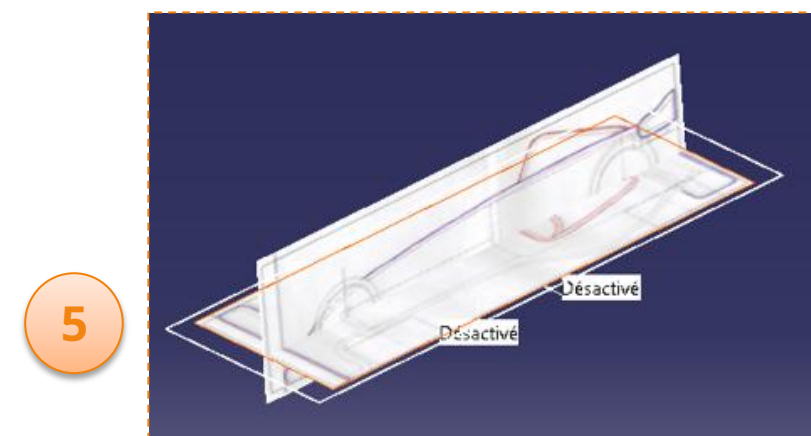
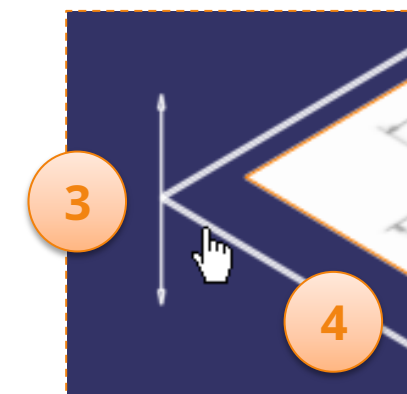
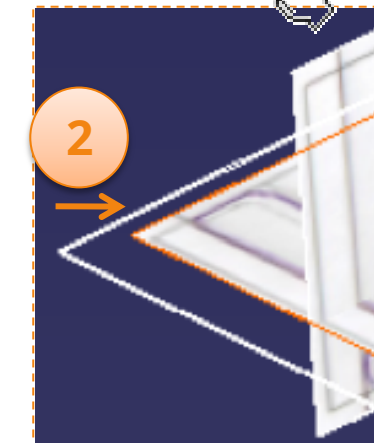
You should get the result below by selecting the isometric view.



# Top view

We will check the positioning and the update the scale of our images in the 3D environment:

1. Select the **Table 2** in the tree by clicking on it.
2. You will notice that a white frame surrounds the image in 3D space, indicating that it is selected.
3. Place your mouse over this frame.
4. Click and hold the click, then move the mouse in the direction of the arrows. The table can move.
5. Thanks to this you can highlight the concordance of the views, but also the good alignment of the two images.



# Save your work

At this point you should save your work. For that :

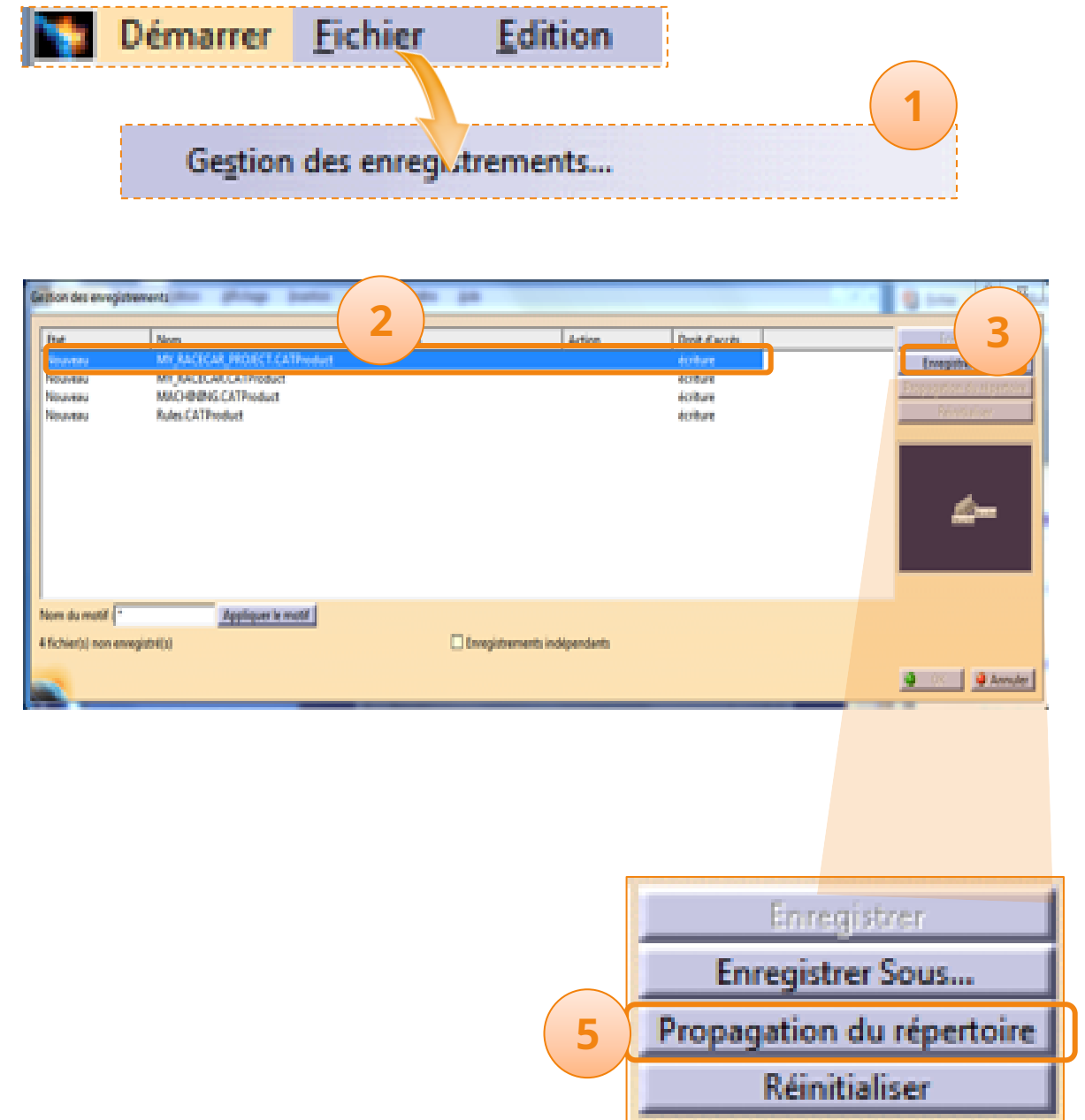
1. Click on **File > Records Management**
2. Select from the list on the left **MY\_e-RACECAR\_PROJECT.CATProduct**.
3. Click on **Save as**.
4. Select the directory of your choice, then click on **To safeguard**.
5. Click on **Directory propagation**.  
This will save all parts related to the product in the same directory.
6. Click on **okay** to validate the recording.



For more information on recording your data, see the method sheet "Manage your data".



Consider backing up your work ! this logo you will call back.



# Model your car

In this introduction we will see:

- ▶ The constraints resulting from the manufacture that you will have to take into account.
- ▶ The presentation of the different steps needed to model your car.
  - ▶ The construction will begin by drawing the characteristic lines of the body.
  - ▶ These will be used to generate the volume of the body in which you will carry out specific operations using the Part Design workbench.
  - ▶ Then you will make the cockpit by manipulating a kind of virtual modeling clay.
  - ▶ Finally to respect the constraint of symmetry, we will model only half of the car. The other part will be generated by a symmetry operation.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.**
    - i. Introduction.**
    - ii. Settings.
    - iii. Model the body.
    - iv. Model the cockpit.
    - v. Model the rear wing.
    - vi. Finalization.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.



# Introduction – Manufacturing constraints

Your car is going to be machined, you must keep in mind these few design rules so as not to have unpleasant surprises during production.

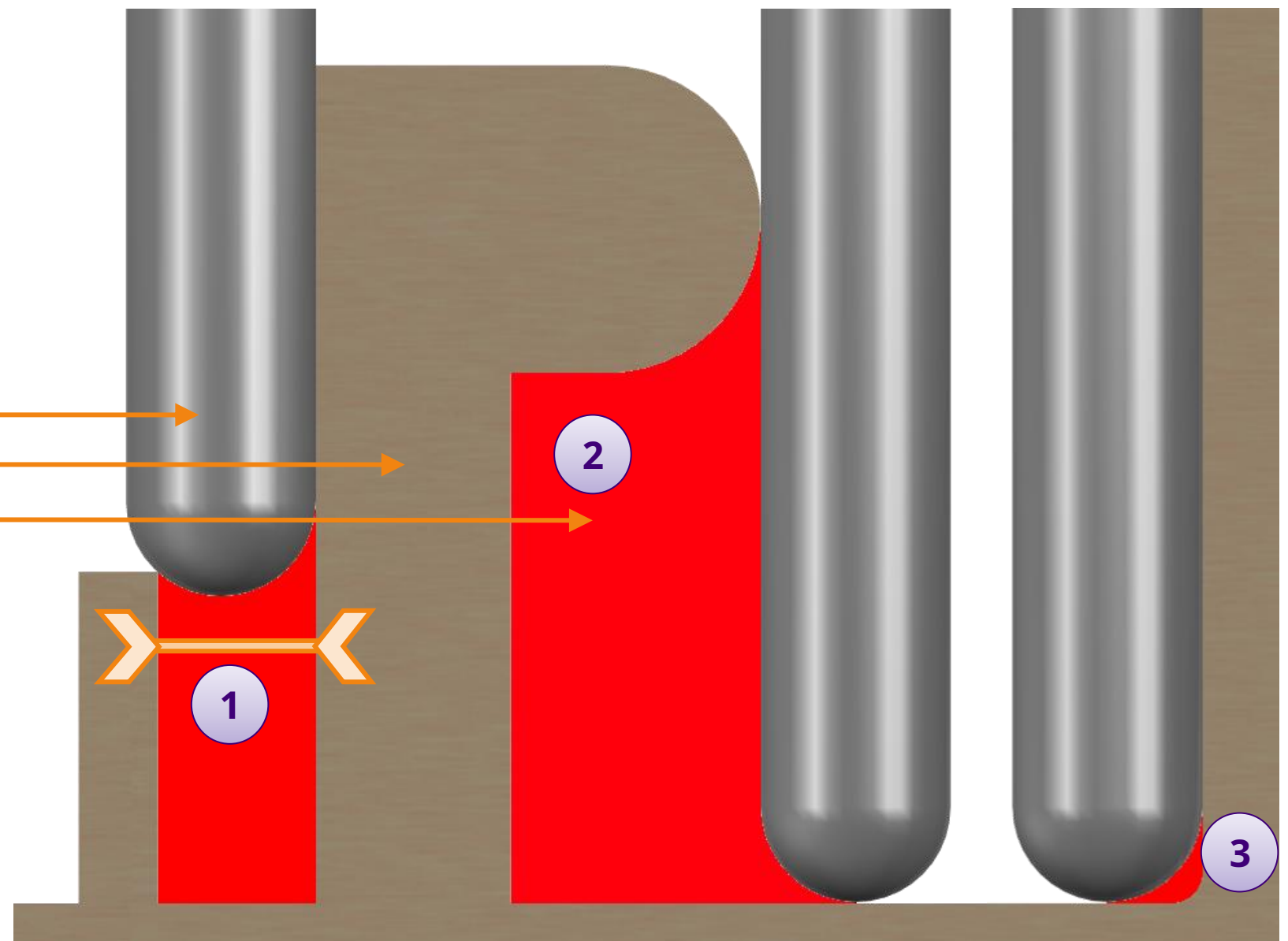
**1** ▶ The distance between two elements must be greater than 6 mm, the diameter of the cutter.

**2** ▶ Pay attention to surfaces that will not be accessible by the tool.

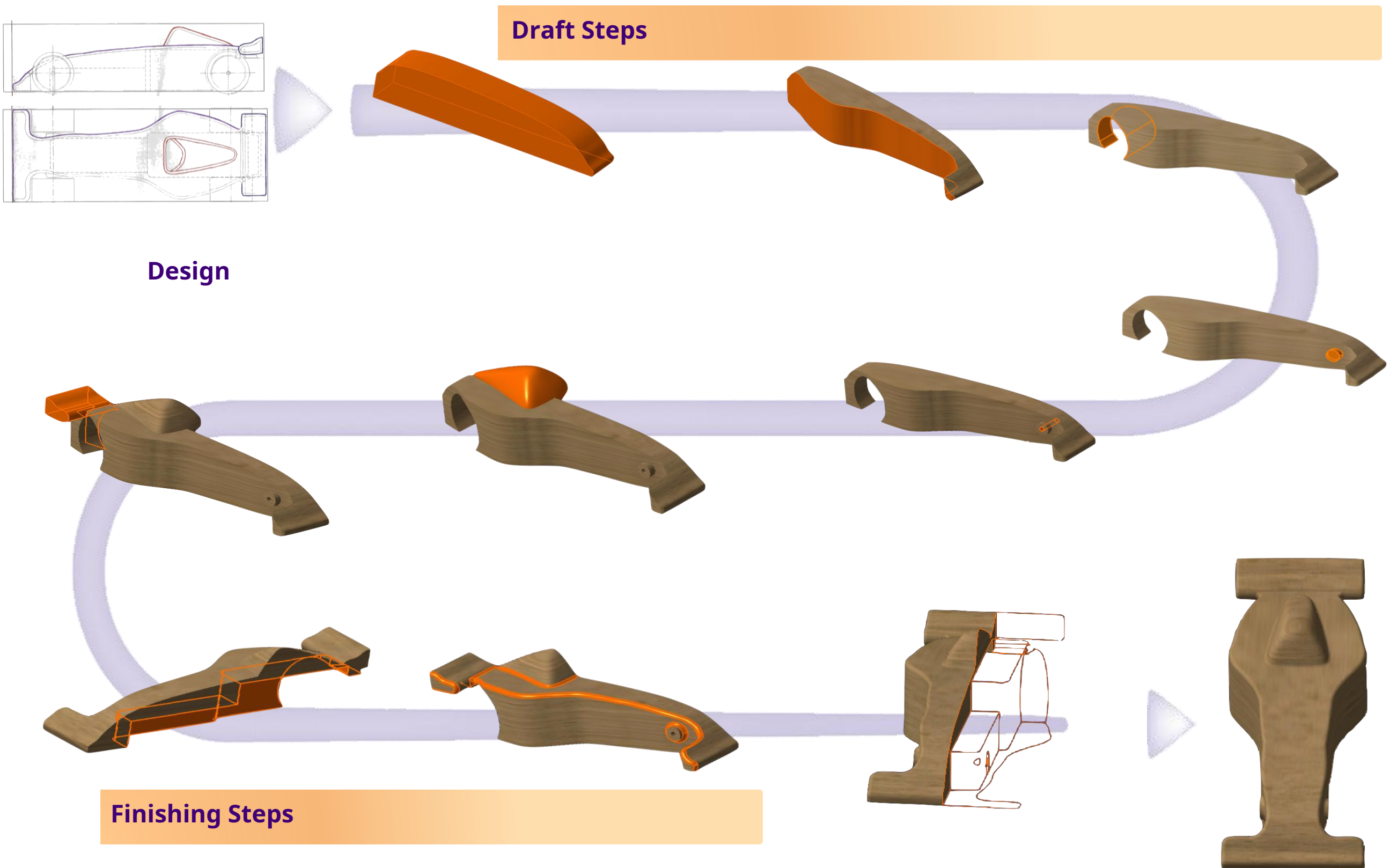
**3** ▶ Concave fillets must have a minimum radius of 3 mm.

▶ In addition, prefer elements with a minimum thickness of 3 mm. To avoid breakage during machining.

- Tool
- Balsa
- Not machined



# Introduction – The modeling process



# Settings

Before starting the modeling and to facilitate the steps that will follow, we will modify certain elements of the environment, such as:

- ▶ **Hide an unnecessary element that clutters the display.**
- ▶ **Modify the appearance of a table to allow better readability of the environment.**



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. *Your racing car.***
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.***
    - i. Introduction.
    - ii. Settings.***
    - iii. Model the body.
    - iv. Model the cockpit.
    - v. Model the rear wing.
    - vi. Finalization.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

# Settings

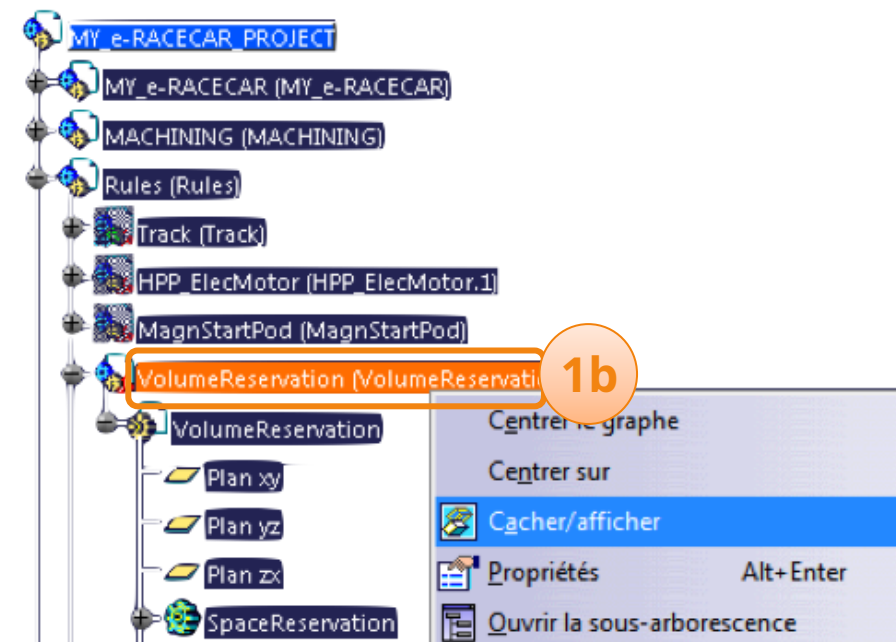
1. We are going to hide the element representing the motor and battery reservation:

has. Click on **+** of the product branch **Rules**.



- b. Right-click on the named part **VolumeReservation**, and click **Hide/show**.

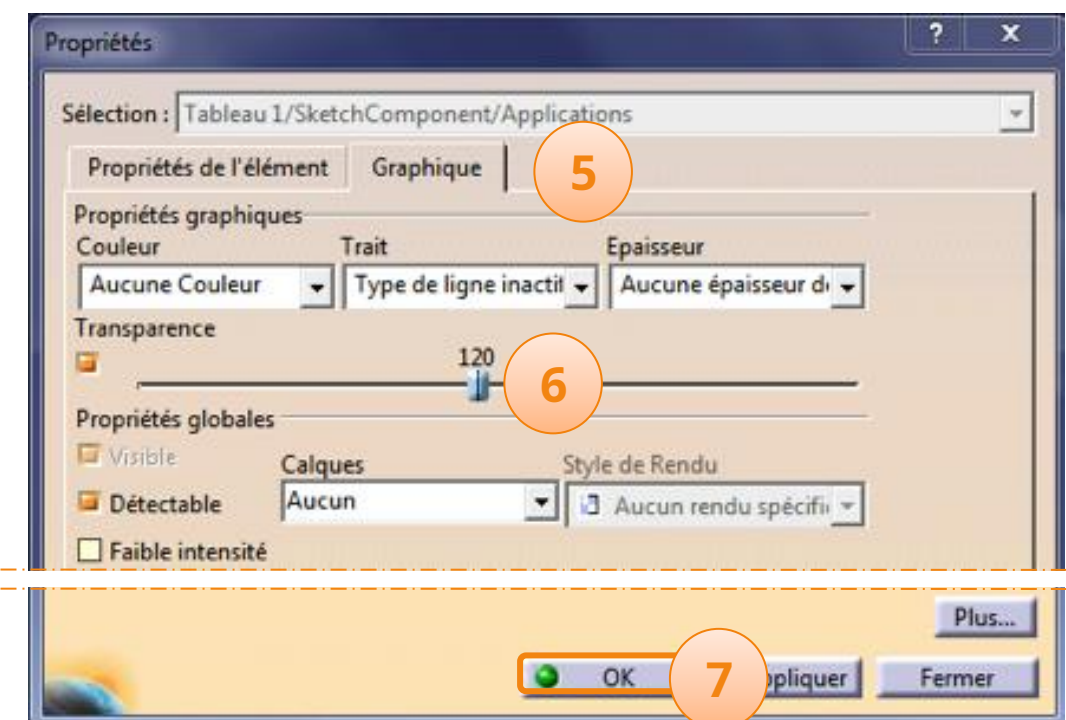
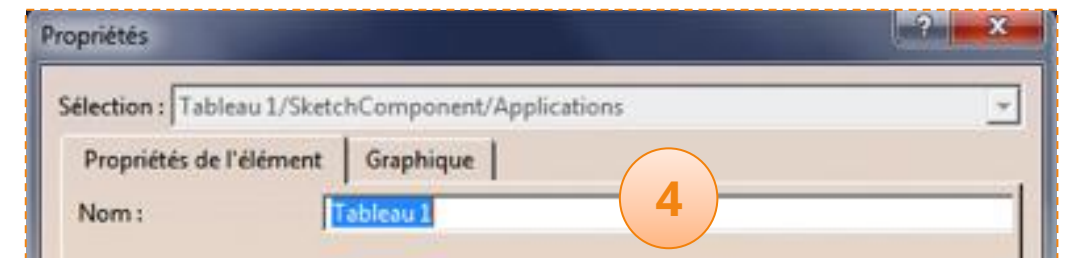
vs. You can close the branch **Rules** clicking on **-**.



# Settings

We are going to modify the appearance of the table to facilitate readability when we trace the curves:

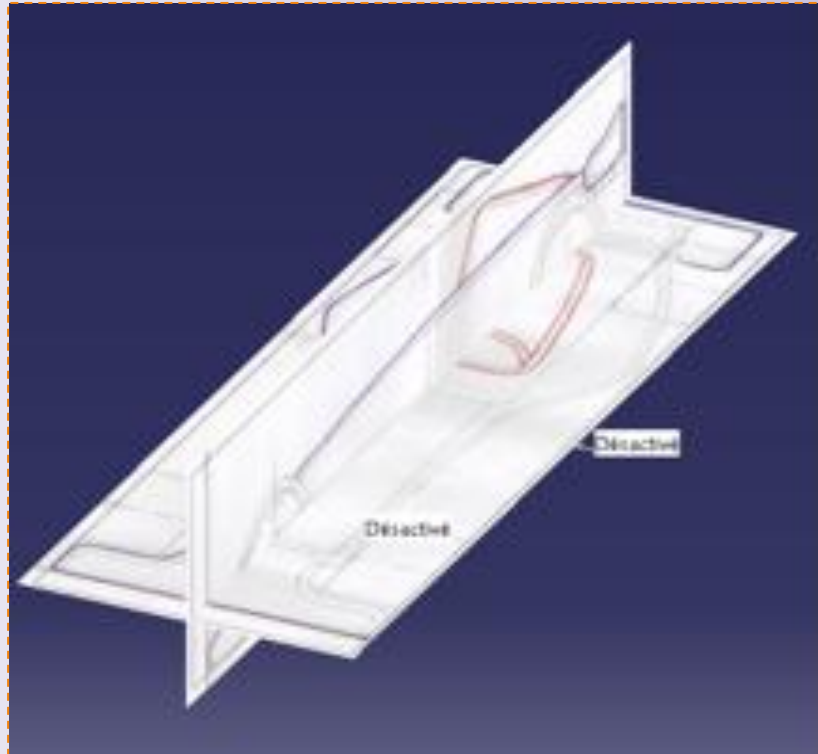
1. Click on the **+** of **Apps** which is at the very bottom of the specification tree.
2. Click on the **+** of **Gallery of paintings**.
3. Right click on **Table 1** and click on **Properties**.
4. In the first tab **Item Properties**, you can change the table name. Rename it [SideView].
5. Click on the second tab **Chart**.
6. **Activate** transparency by putting a coefficient of 120.
7. Click on **okay**.





# Settings

Without transparency



Transparency



0



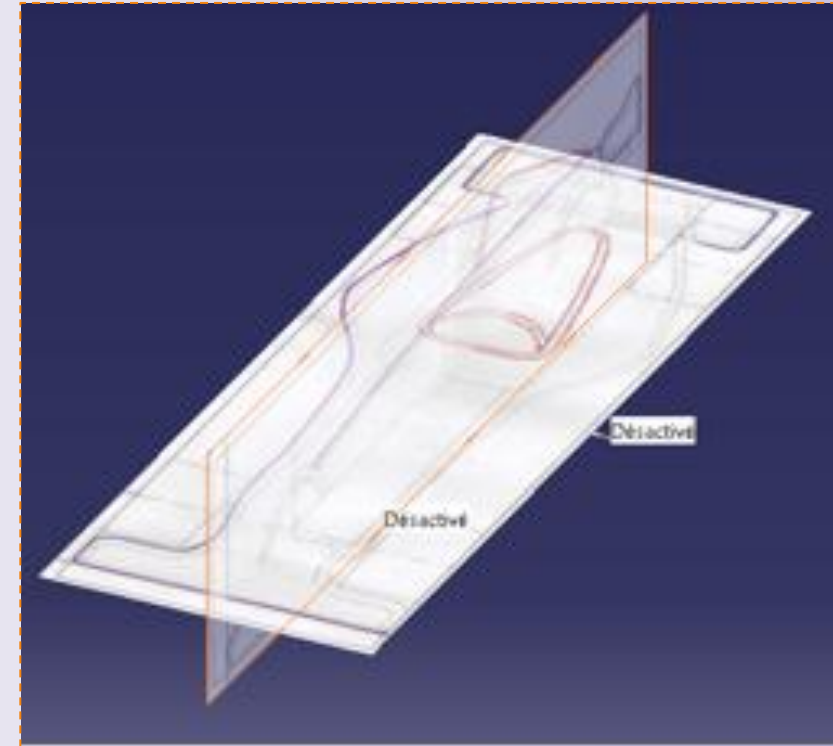
Transparency



120



With transparency



Transparency



120

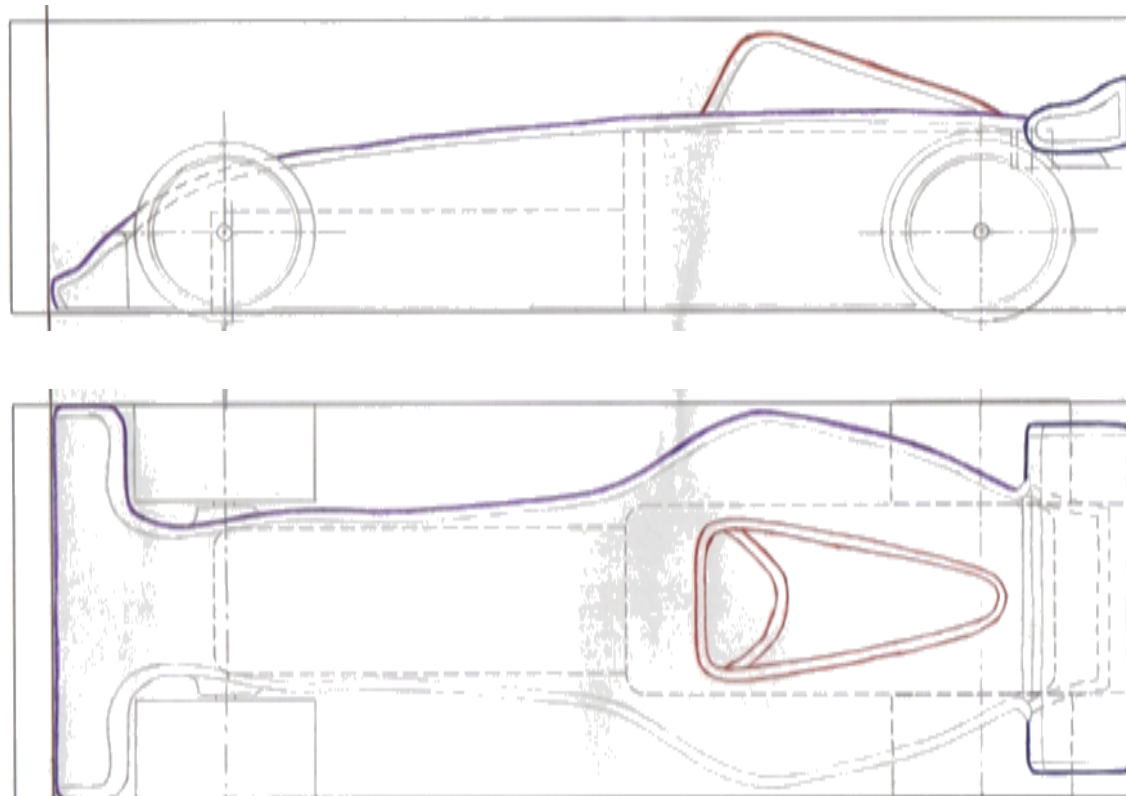


Make sure you have checked the option for it to be enabled.

# Model the body

In this step you will use the Imagine & Shape workbench to:

- ▶ Tracing curves.
- ▶ Edit curves.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.**
    - i. Introduction.
    - ii. Settings.
    - iii. Model the body.**
      - I. Curves features.**
      - II. Mechanical concept.
    - iv. Model the cockpit.
    - v. Model the rear wing.
    - vi. Finalization.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

# Settings

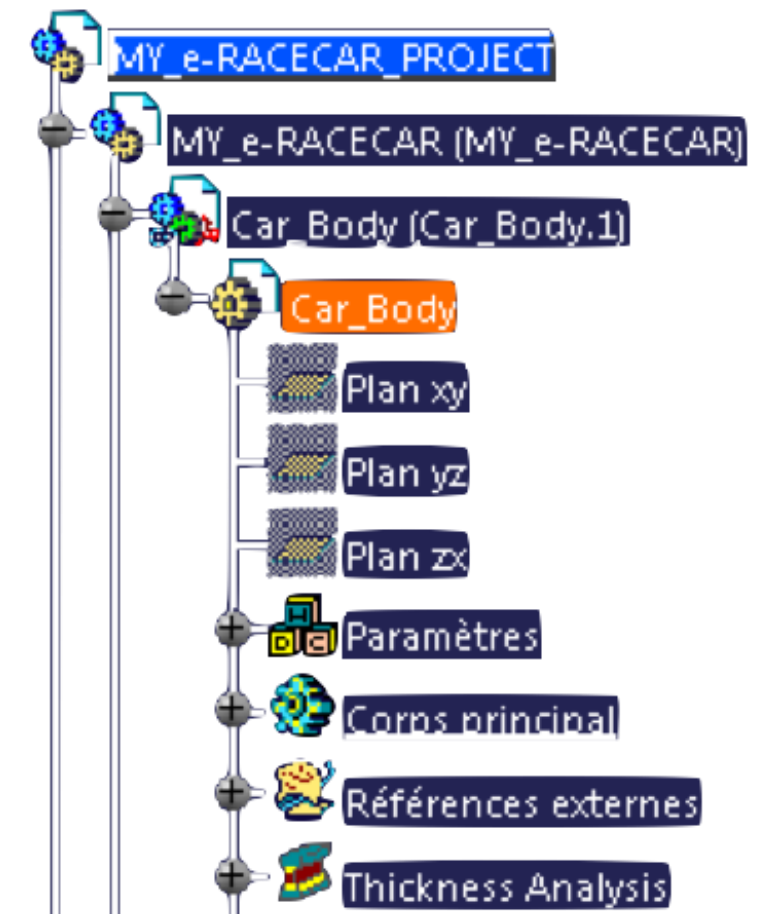
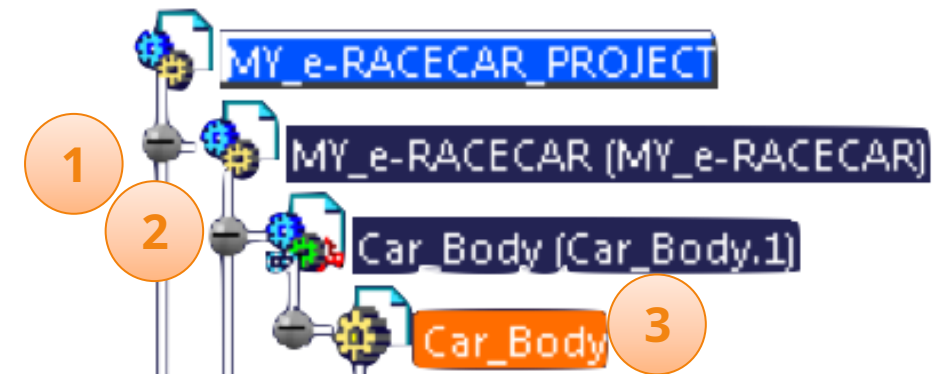
We want to model the body of the car, so we need to indicate at which level of the specification tree we want to work.

For that :

1. Click on the **+** of the branch **MY\_e-RACECAR**.
2. Click on the **+** of the branch **Car\_Body**.
3. Double-click on the part being called **Car\_Body**.

You will see that the active part has changed, and “Main body” is underlined.

This means that this is the new active level and therefore all operations will be performed within it.



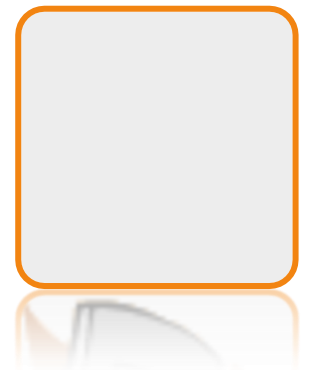
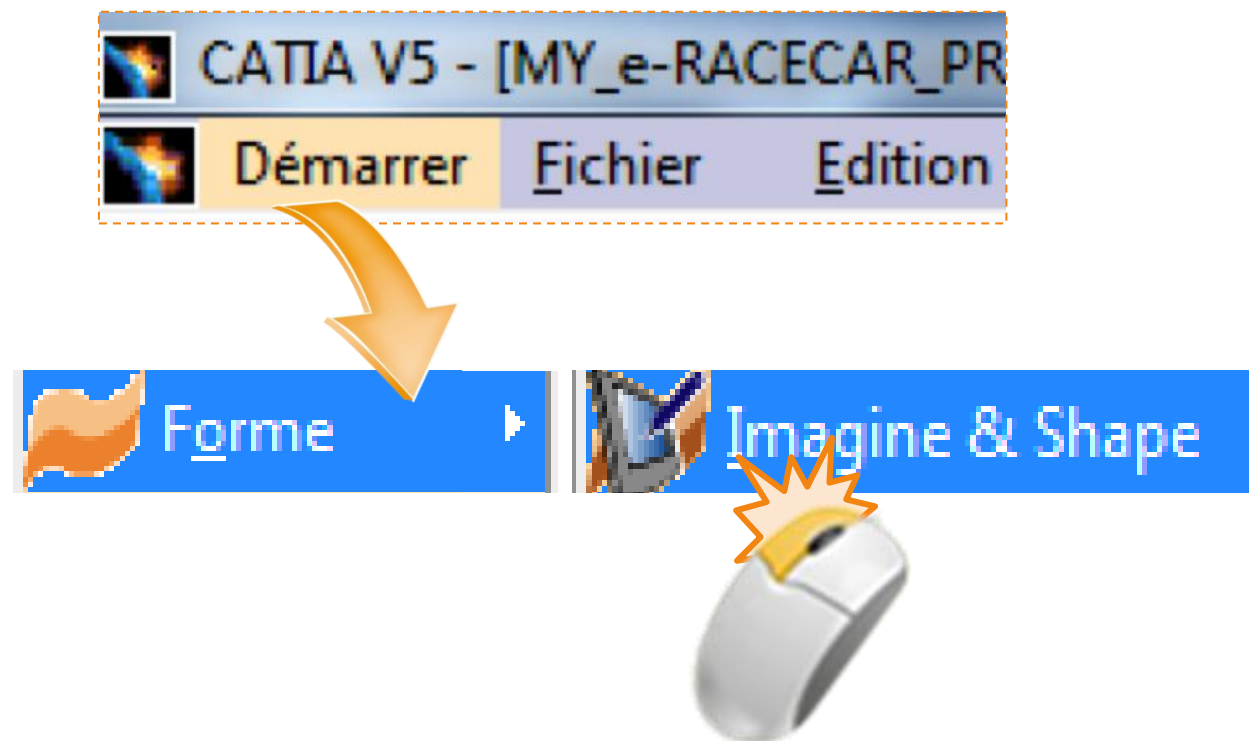
It is important to have a good understanding of what the **active level** in CATIA.

Consult the “Specification tree” method sheet for more information.

# Characteristic curves

Modeling begins with curve drawing. For this we will use the Imagine & Shape workshop.

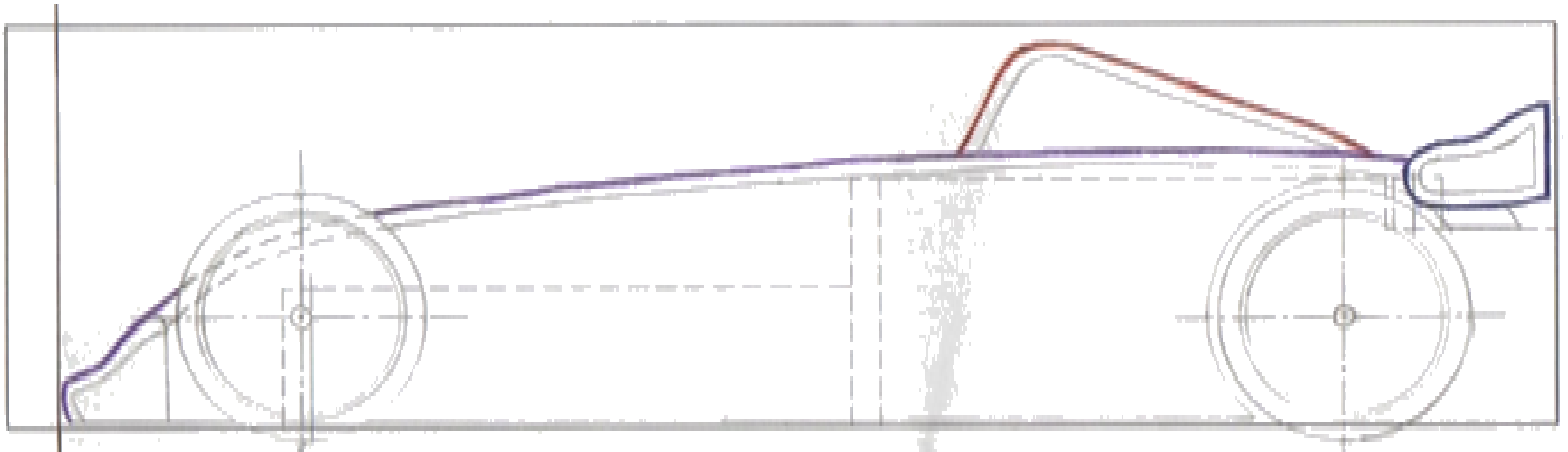
To access it click on: **Start > Shape > Imagine&Shape**



# Characteristic curves

We will start by drawing the curve of the body profile.  
Here are some recommendations to follow:

- ▶ The curve will only represent the top line of the frame (in purple).
- ▶ The curve will not take into account the cockpit (in red).
- ▶ It is advisable to overflow a little on the ends. Excess material will then be automatically re-limited by the software.





# Characteristic curves

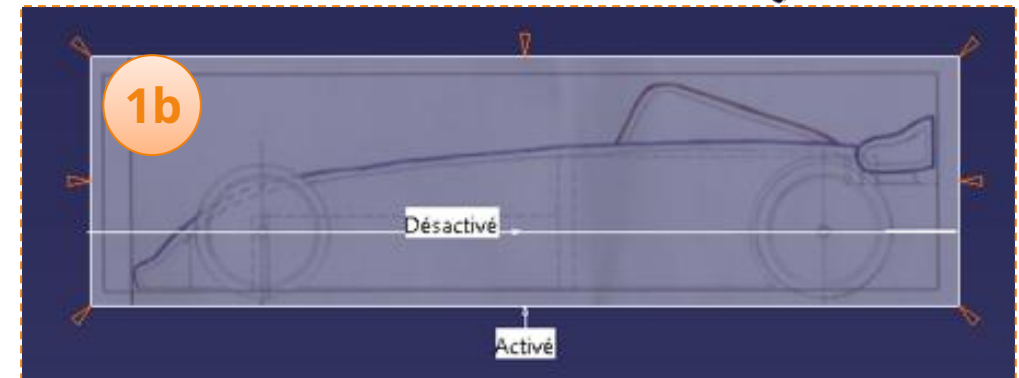
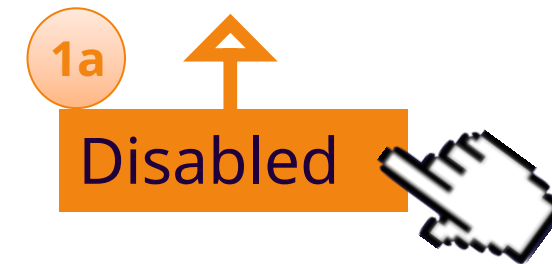
1. We are now going to activate the side view table (this allows to 'lay' the drawing in the plane of the screen):

has. Click on the box **Disabled** which is attached to the side view.

b. The view is locked to the drawing.



Activating a table prevents rotation of the view. Notice that the board displays its Enabled/Disabled state; a click toggles mode.



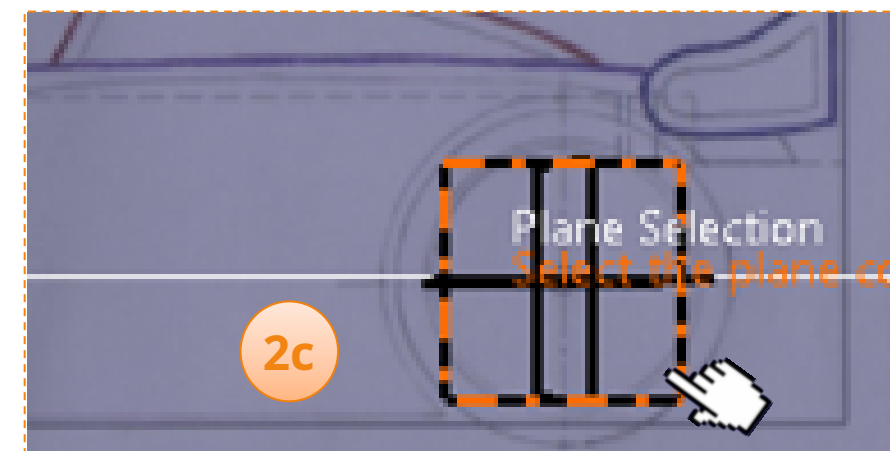
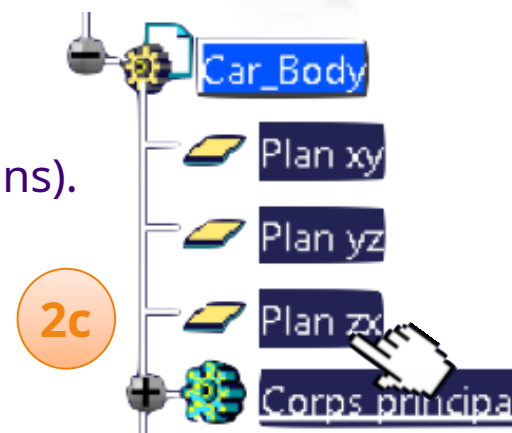
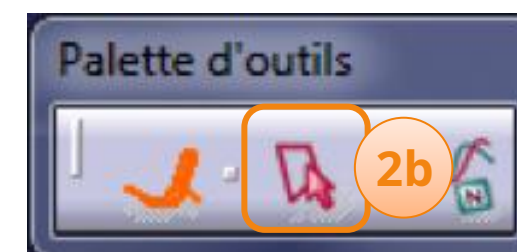
2. Now we have to define in which plane we **2a** let's draw the curve:

has. Click the icon **Sketch Curve**.

b. In the window that appears, click the icon **Plane Selection**.

vs. Click on the **ZX plane** on the geometry, or directly in the tree.

However, the planes may be hidden (grayed out icons). To display them, right-click on them, then **Hide/Show**.



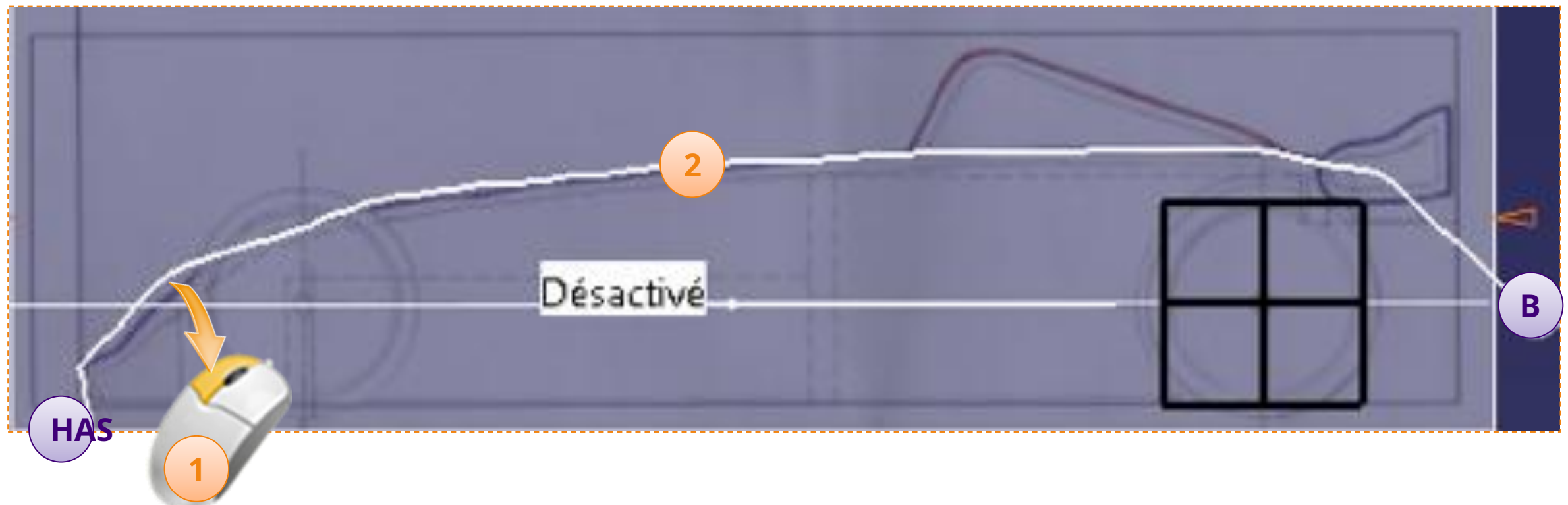
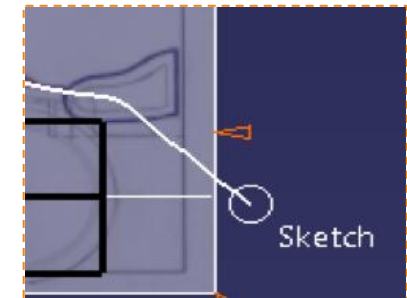
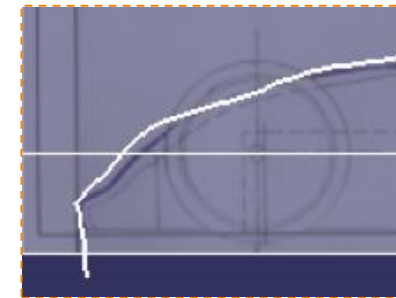
# Characteristic curves

Taking into account the information opposite, draw the curve, for this:



Take care to start and finish later at the extremities ! As below.

1. Click and hold the click **point A**.
2. While holding the click, trace the guideline. Go to **B-spot**, then release the click.



You don't have to iron perfectly the first time. CATIA smoothes your plot and you can modify it later. It is preferable to draw the curve in one go to obtain the result opposite.

If the plot does not suit you, press the key. **<Delete>** and start the operation again.



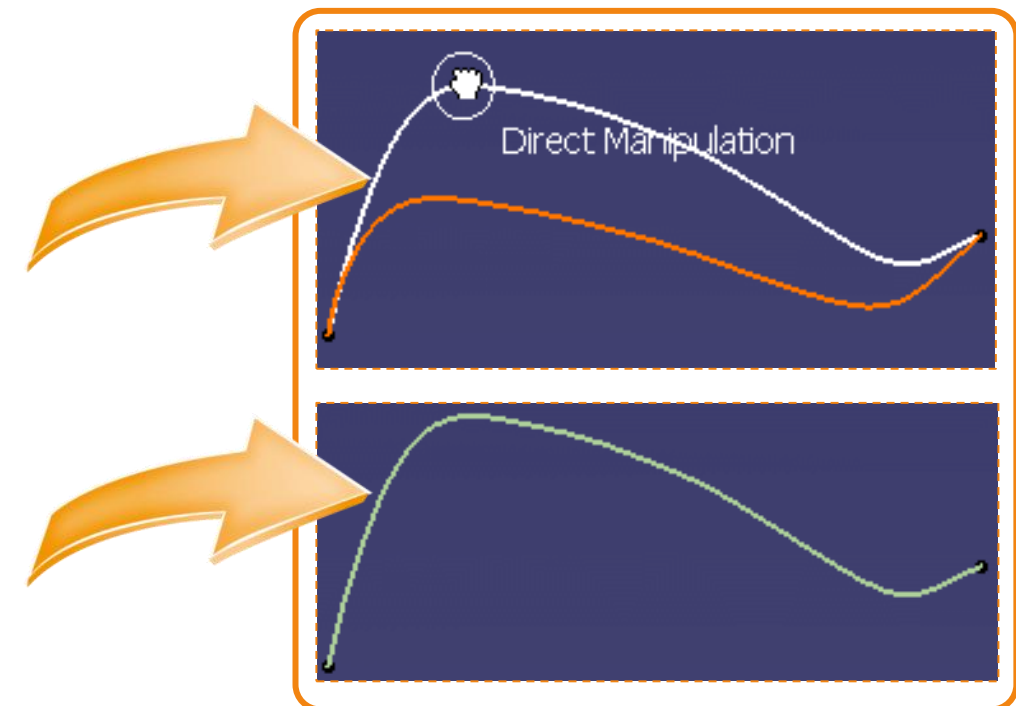
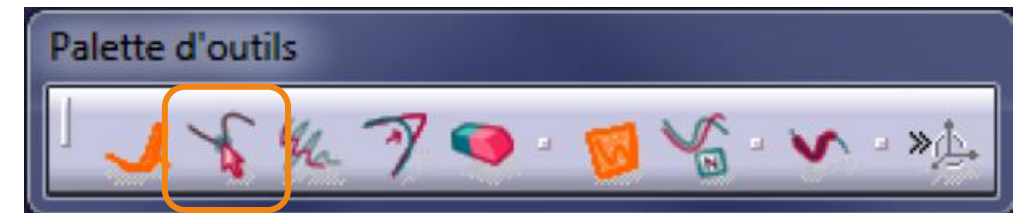
## Tool palette

1. When you have drawn the curve, the tool palette opposite appears. Click on the tool **Handling**.
2. Click in the area of the curve you wish to modify and move the mouse to deform the curve.
3. Release the click, when you are satisfied with the modification. The curve now looks like this (see opposite).



You can define a selection area before modify the curve by pressing the Ctrl key.

Note the help that appears to the right of the mouse for each tool.



# Characteristic curves

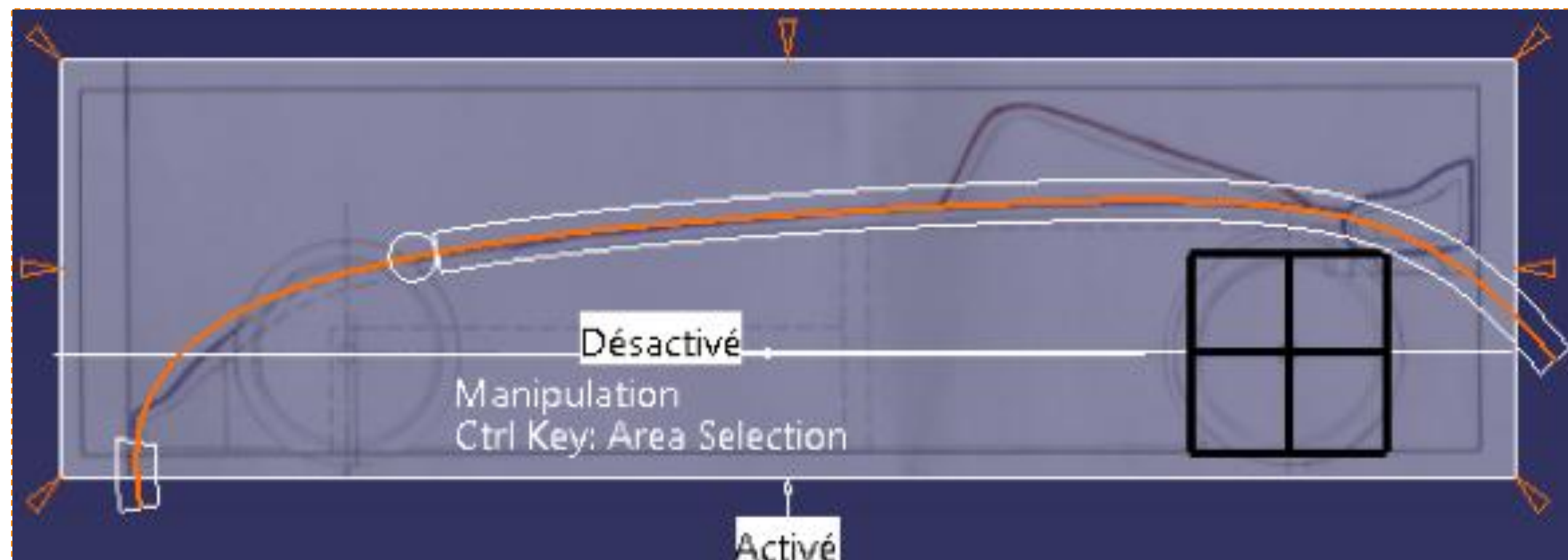
We will create a selection area to manipulate the curve.

This allows:

- ▶ To manipulate the curve more easily.
- ▶ Modify only the parts of the curve that require correction.



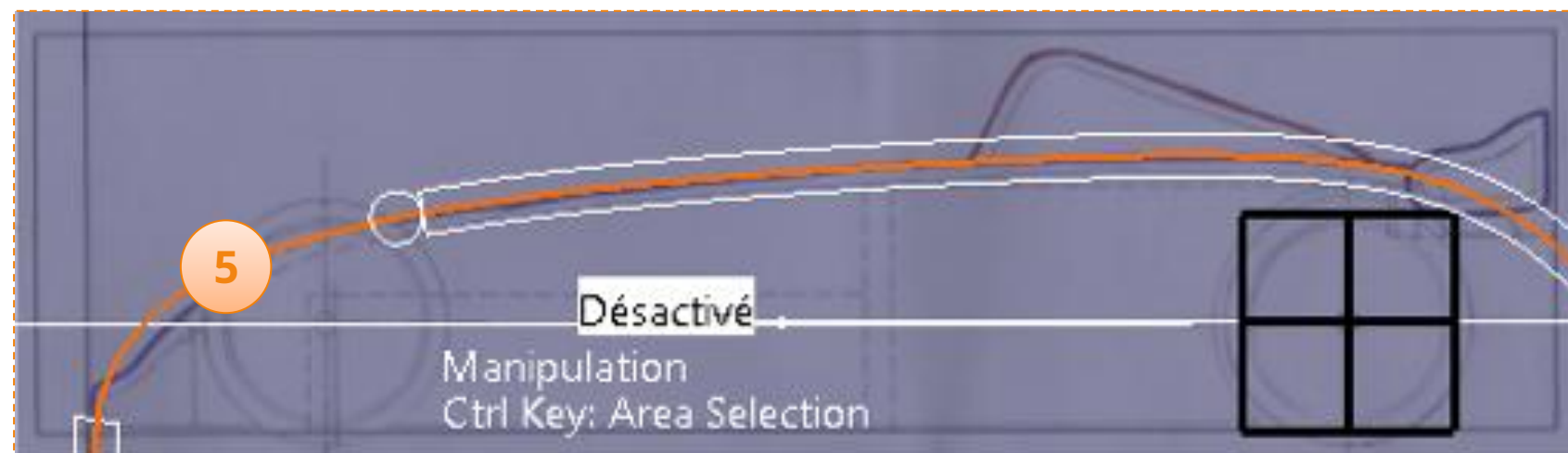
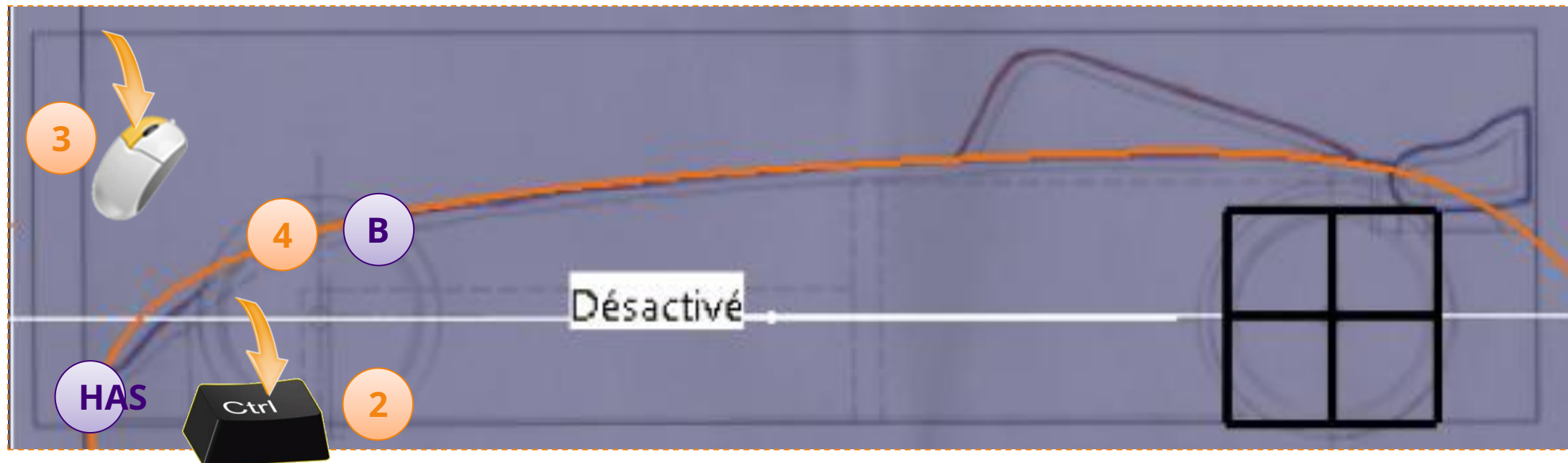
Tip: To choose which areas to select, create selections between two points that are well positioned (see below).



# Characteristic curves

We are going to create a selection area. For that :

1. Click on the tool **Handling**.
2. Hold down **<Ctrl>** depressed.
3. Click and hold the click **point A**.
4. Move the mouse along the curve to define the area that needs to be modified.
5. Release the click at **B-spot**, the area is set.

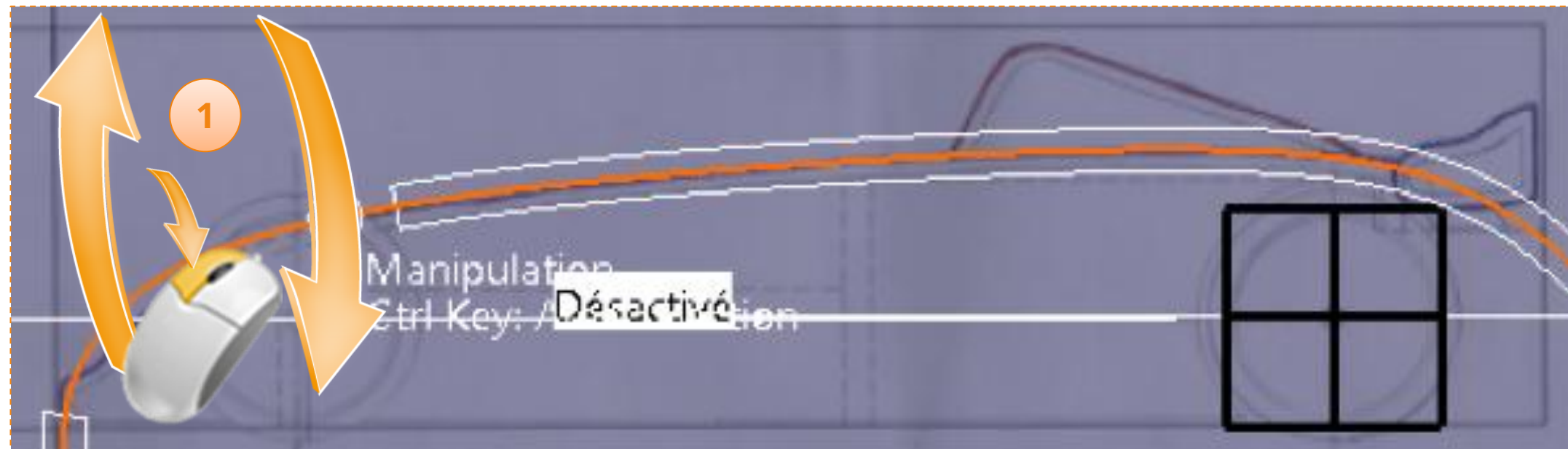




# Characteristic curves

We are going to modify the curve of the area defined with the manipulation tool:

1. Click and hold the click on the curve to move the part you are interested in.



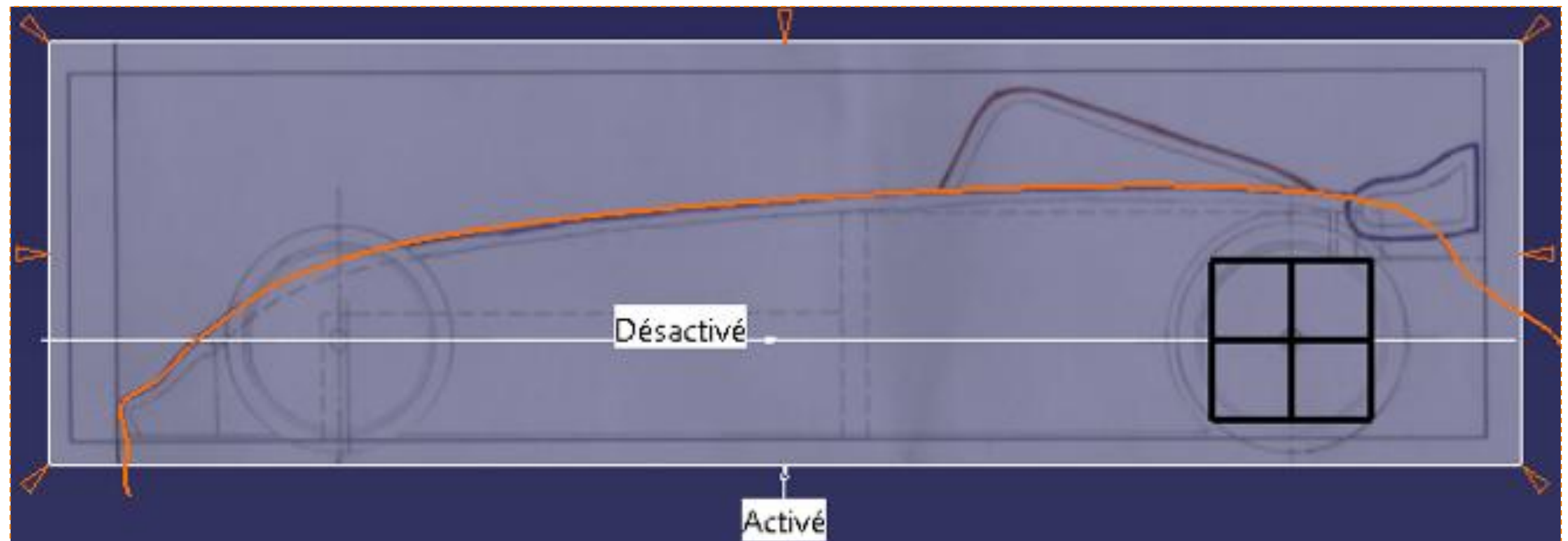
2. You can outline other selection areas to refine the manipulation.



# Characteristic curves

When you are satisfied with your curve, and making sure that it protrudes slightly at the ends, exit the curve editor by:

3. Pressing **two times** on the key <Esc>.

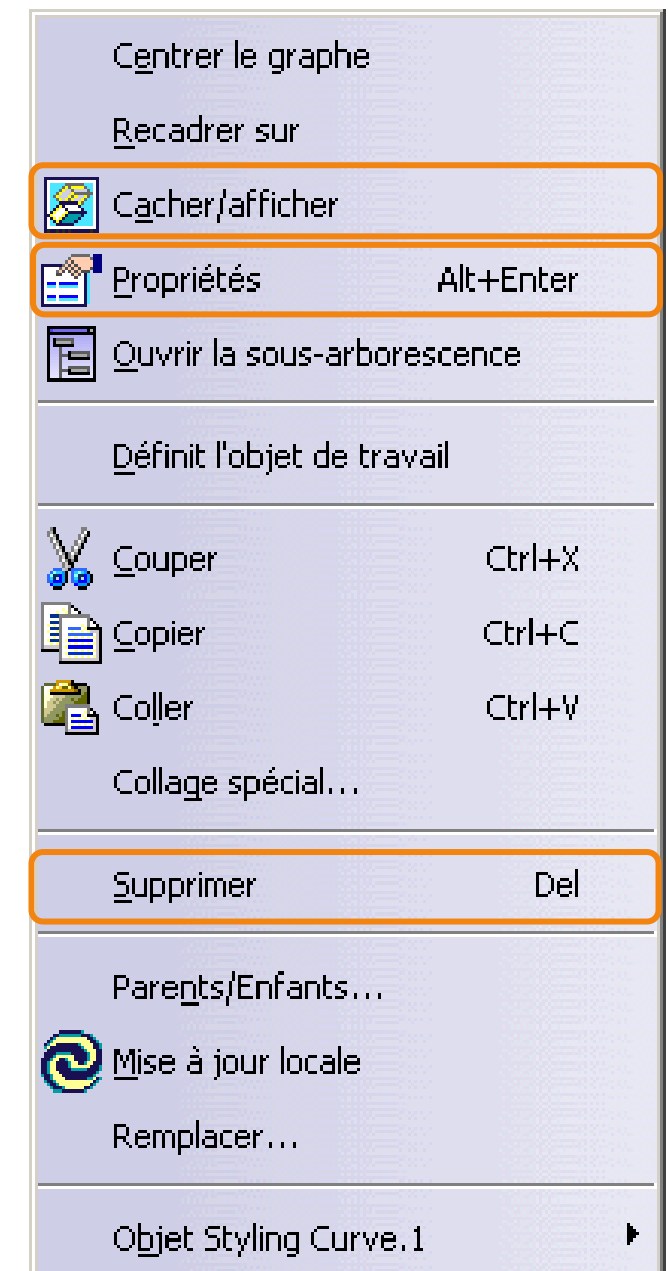
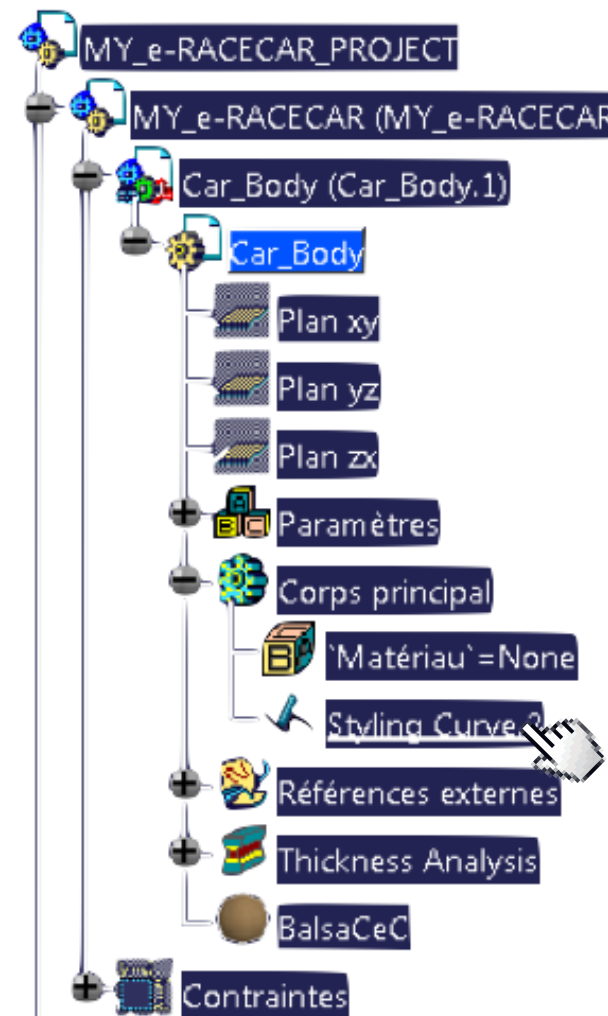


# Characteristic curves

You can now notice the presence of **Styling Curve.1** in the main body.

A right-click on Styling Curve allows you to:

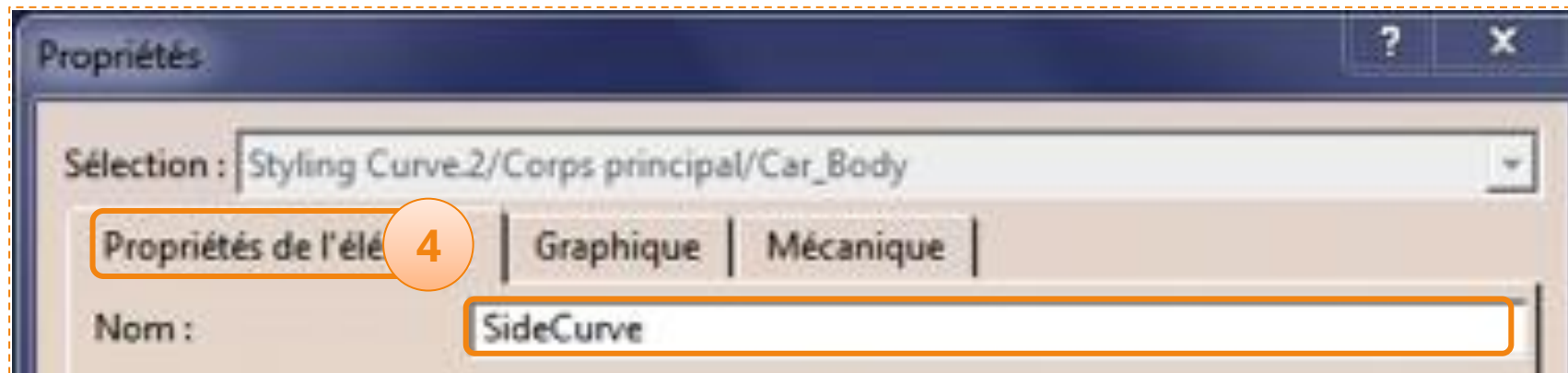
- ▶ Hide/Show .
- ▶ Access its properties.
  - Change its name.
  - Change its color, its thickness. Add
  - a transparency effect.
- ▶ Delete it.



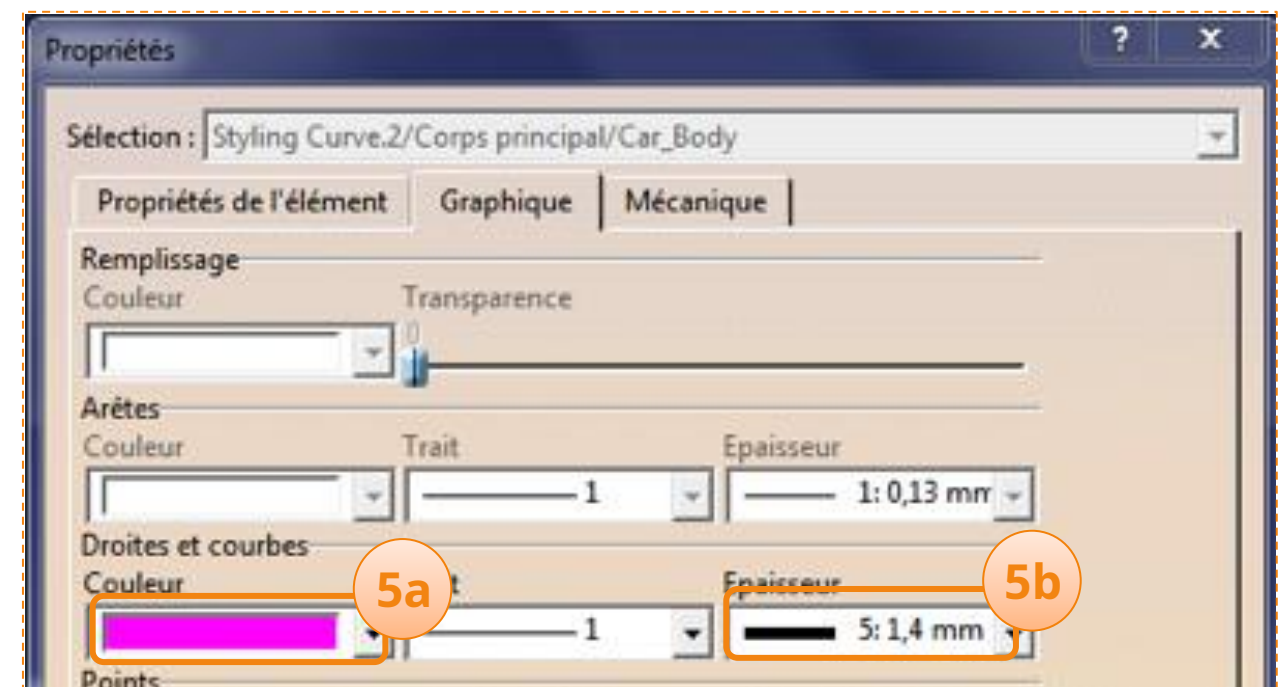
# Characteristic curves

To make the display more pleasant, we are going to modify the properties of this curve:

1. Click on the **+** of the branch **main body**.
2. Right click on **Styling Curve.1**.
3. Click on **Properties**.
4. In the **element properties** tab, rename the curve [SideCurve].



5. In the **Chart** tab,
  - a. Select the color of the curve by **pink**.
  - b. Select thickness **5:1.4mm**.



# Characteristic curves

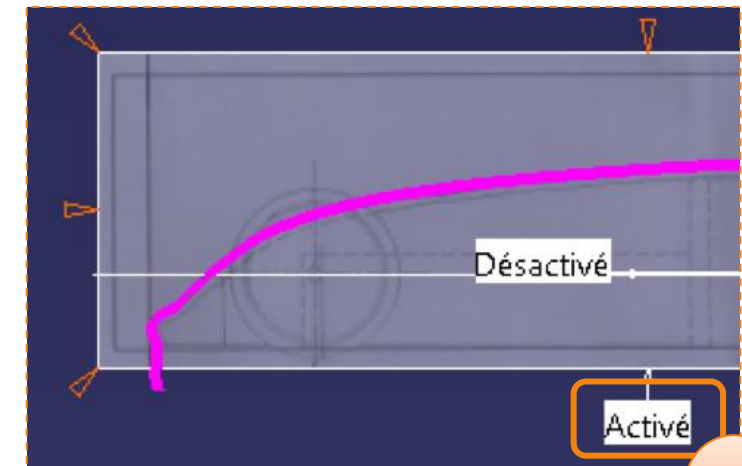
1. To disable the side view table, click on the "enabled" box that is attached to the side view.

The view is now free.

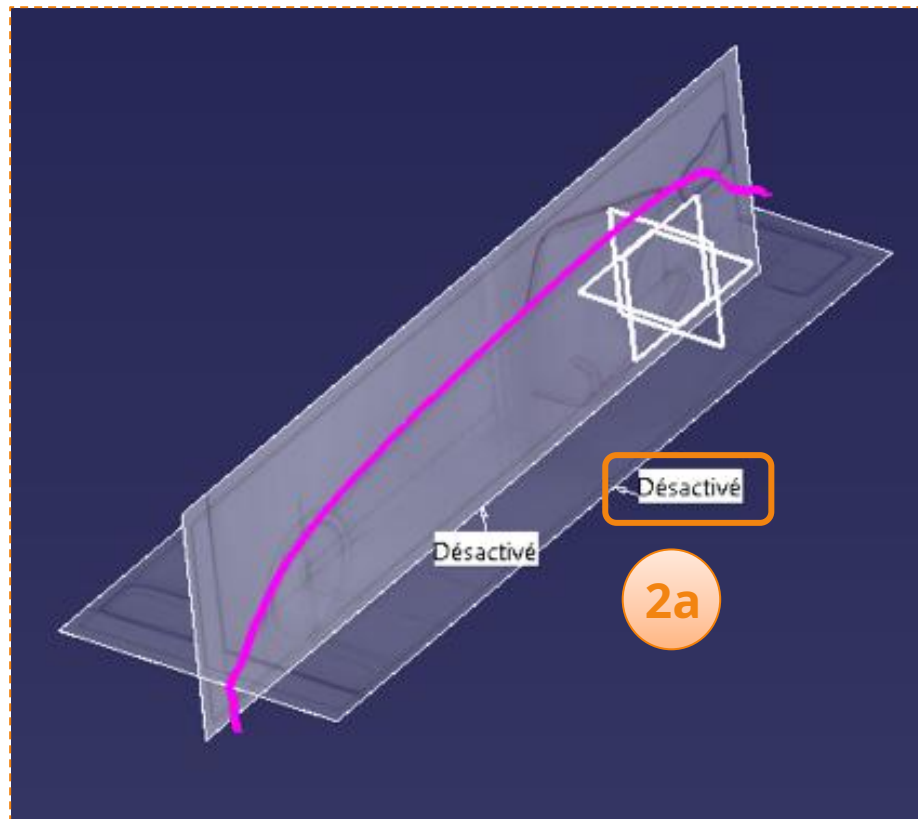
2. To activate the top view table:

has. Click on the "disabled" box that is attached to the top view.

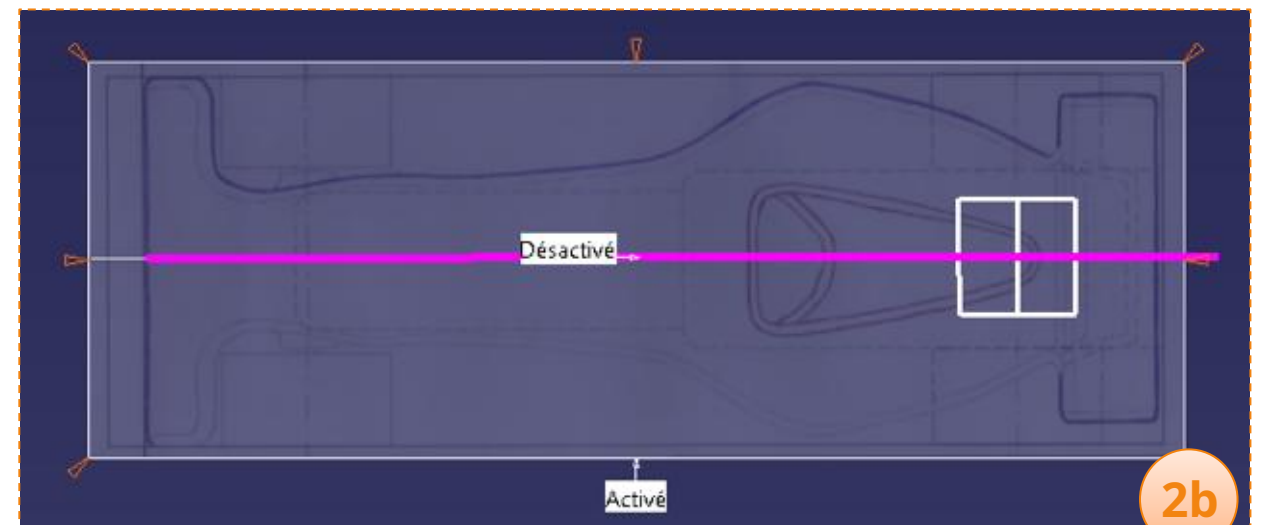
b. The view comes to 'lay' on the screen.



1



2a



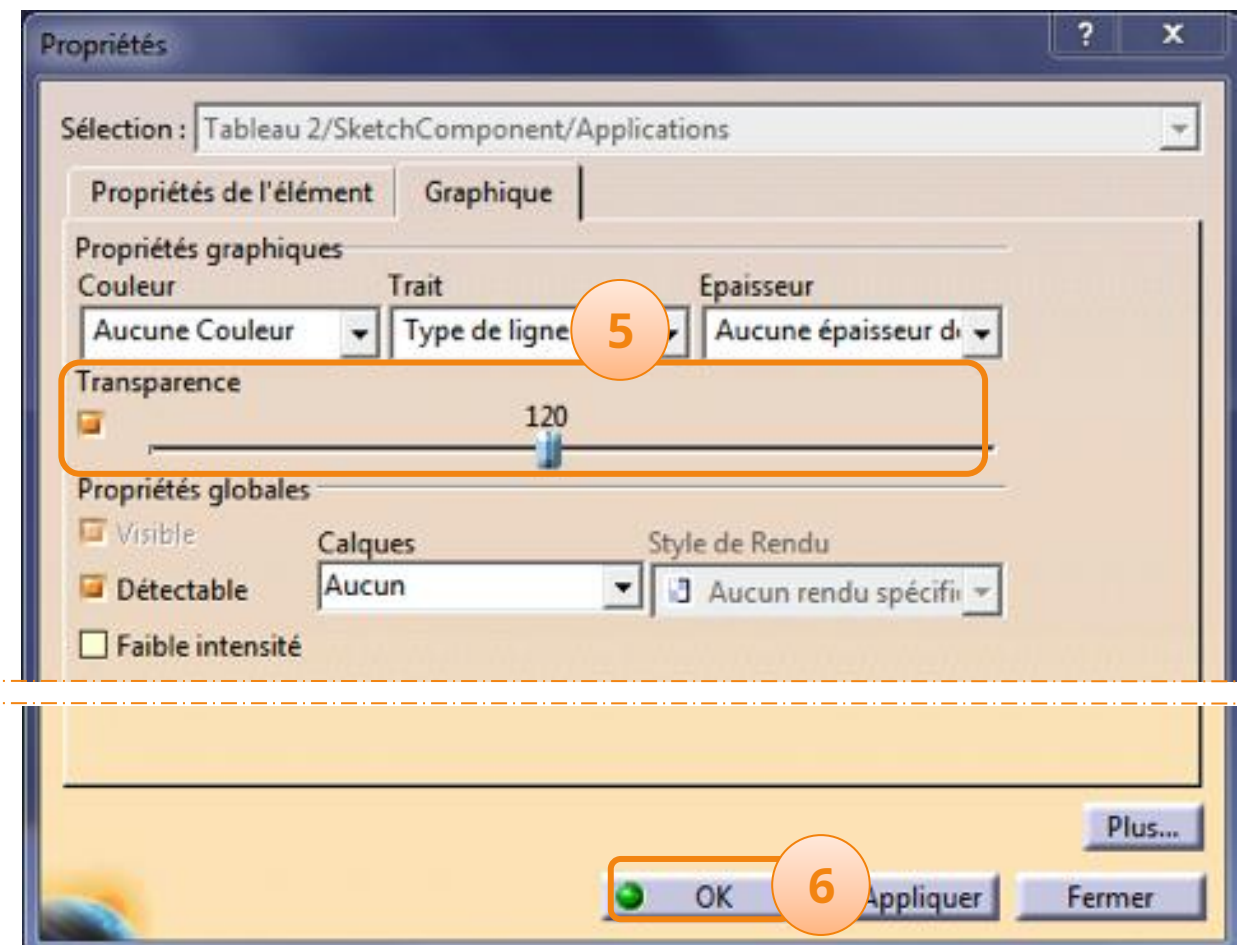
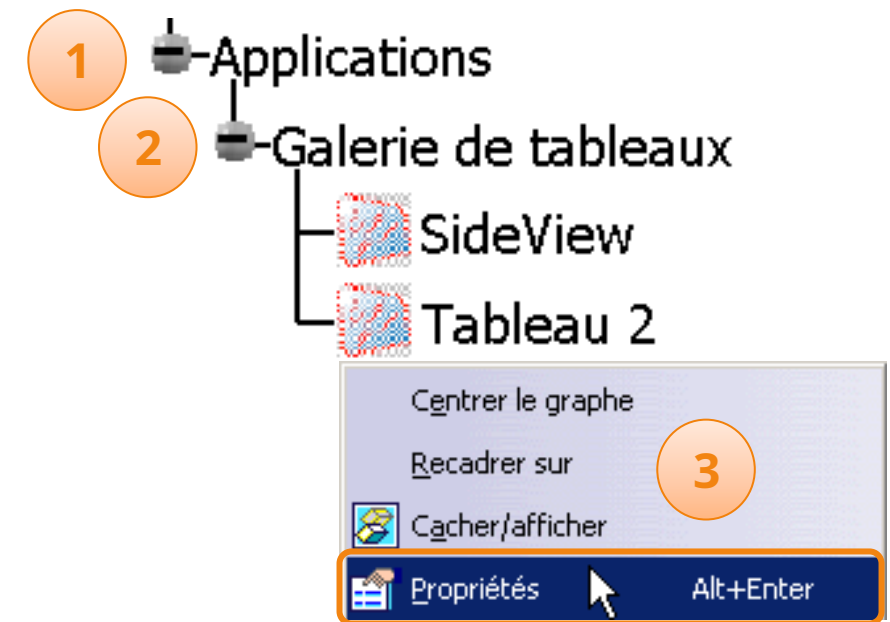
2b



# Characteristic curves

We are going to modify the appearance of the table:

1. Click on the **+** of **Apps** which is at the very bottom of the specification tree.
2. Click on the **+** of **Gallery of paintings**.
3. Right click on **Table 2**, and click **Properties**.
4. In the first tab **Item Properties**, rename it [TopView].
5. Click on the second tab **Chart** and enable transparency by setting the coefficient to **120**.
6. Click on **okay**.

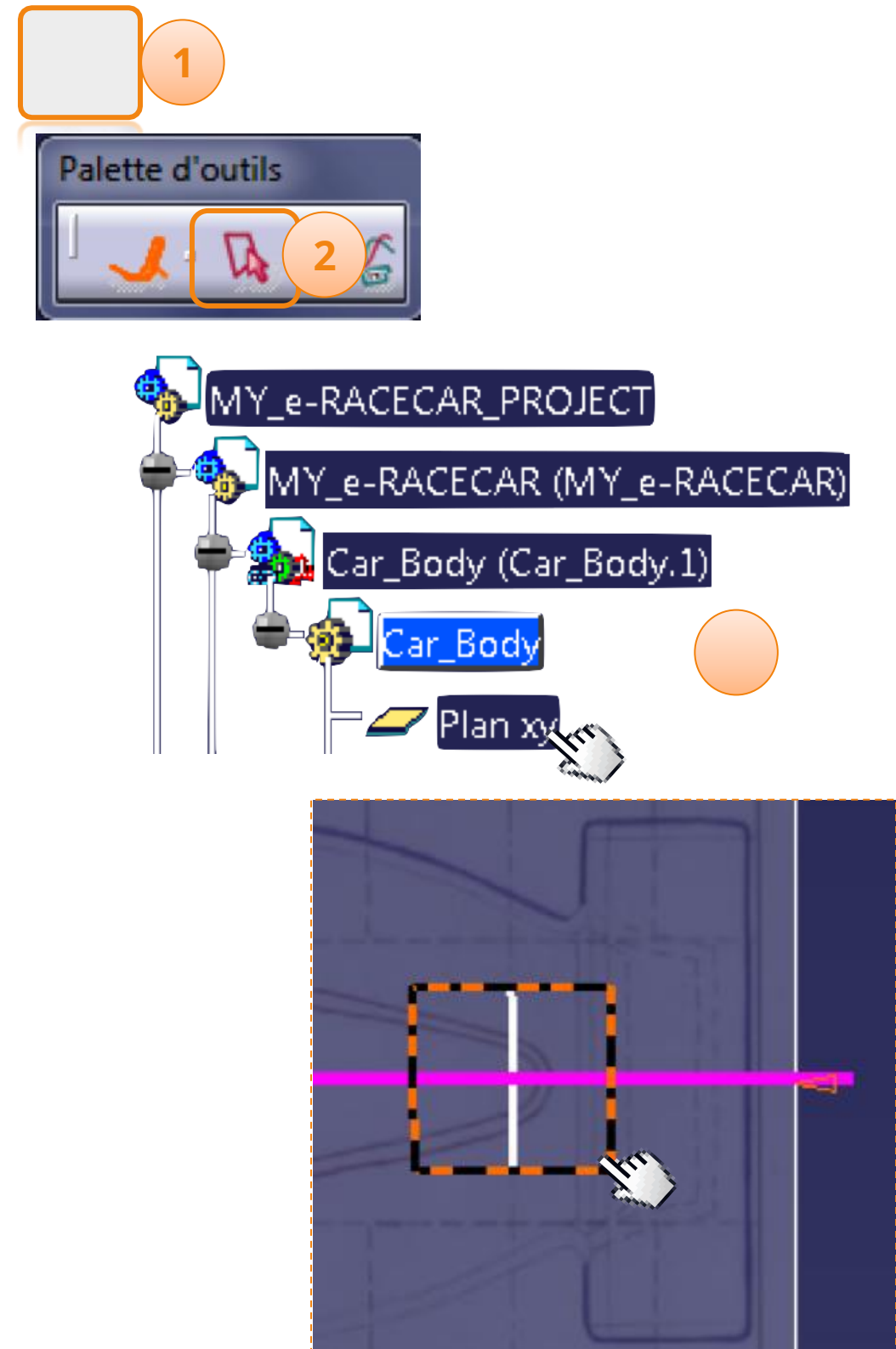


# Characteristic curves

We will now draw the top curve:

1. Click the icon **Sketch Curve**, in the Imagine & Shape studio.
2. In the next window that appears, click the icon **Plane Selection**.
3. Select the **XY-plane** directly in the 3D environment or in the specification tree.

This has the effect of defining the plane in which the curve will be inscribed.

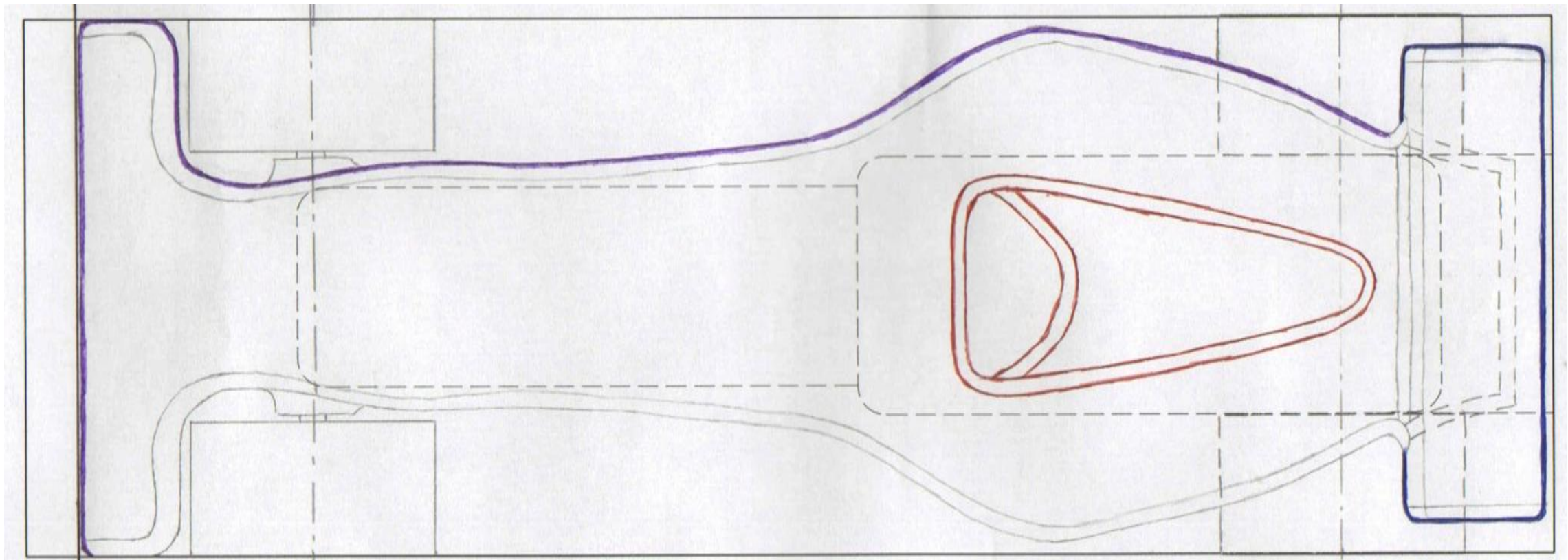


# Characteristic curves

Here are some recommendations for drawing the curve

On top :

- ▶ The curve will only represent one side of the car.
- ▶ The wheel arches will not be taken into account because they will be made later using more suitable tools.
- ▶ The curve should slightly protrude at the ends.



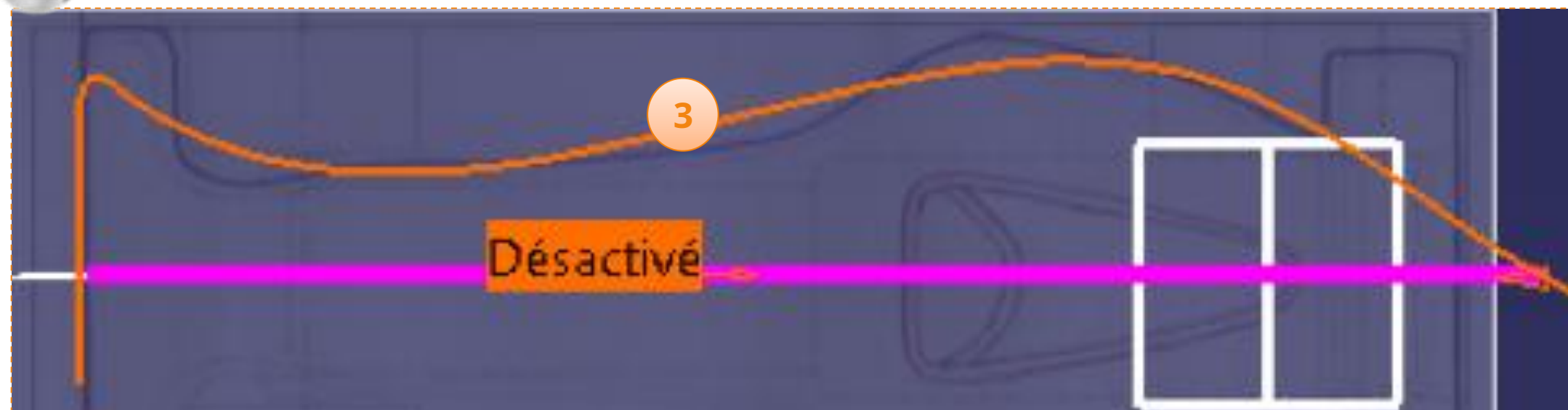
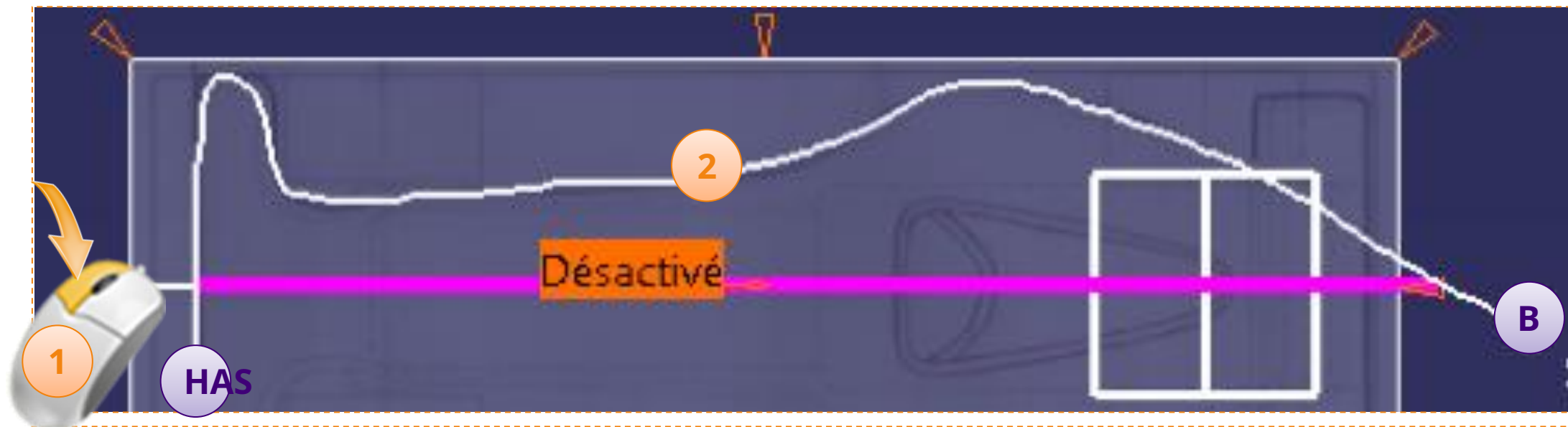
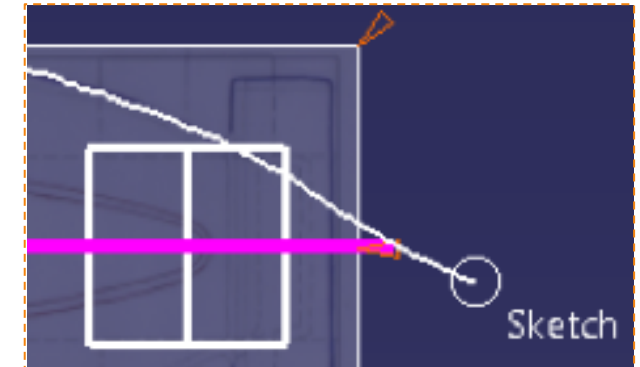
# Characteristic curves

Draw the curve, taking into account the information opposite:

1. Click and hold the click **point A**.
2. While holding the click, trace the guideline. Go to **B-spot**, then release the click.
3. You don't have to iron perfectly the first time. CATIA smoothes your plot and you can modify it later. It is best to draw the curve in one go.



Take care to start and finish later at the extremities ! As below.





# Characteristic curves

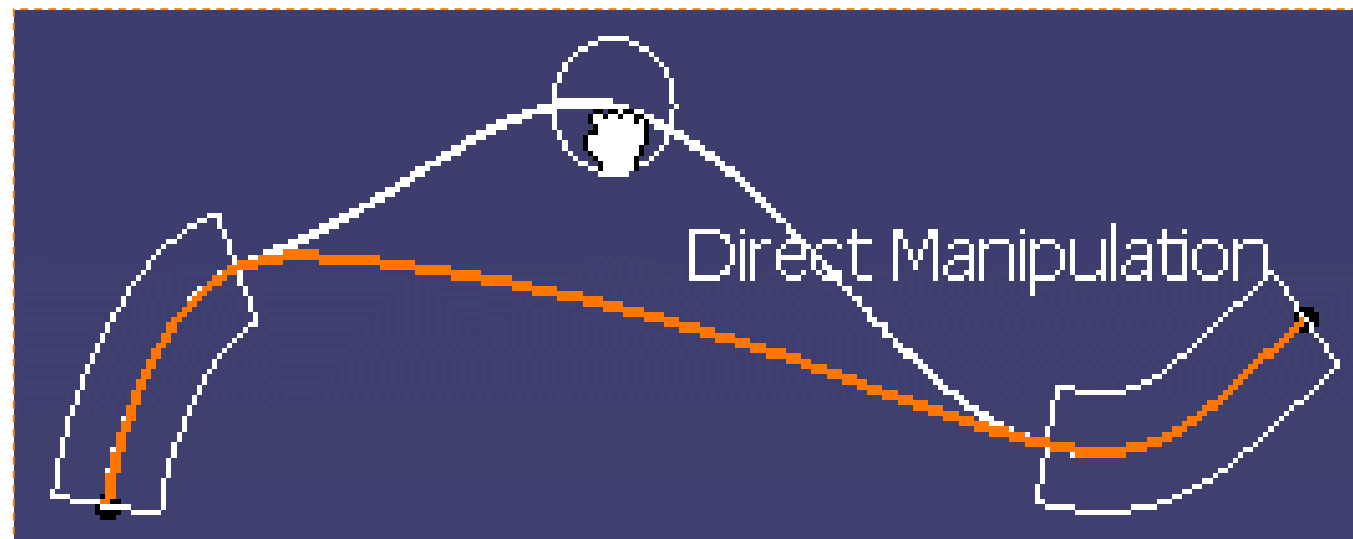
Using the same manipulation method as for the previous curve, modify this curve so that it best overlaps your drawing.



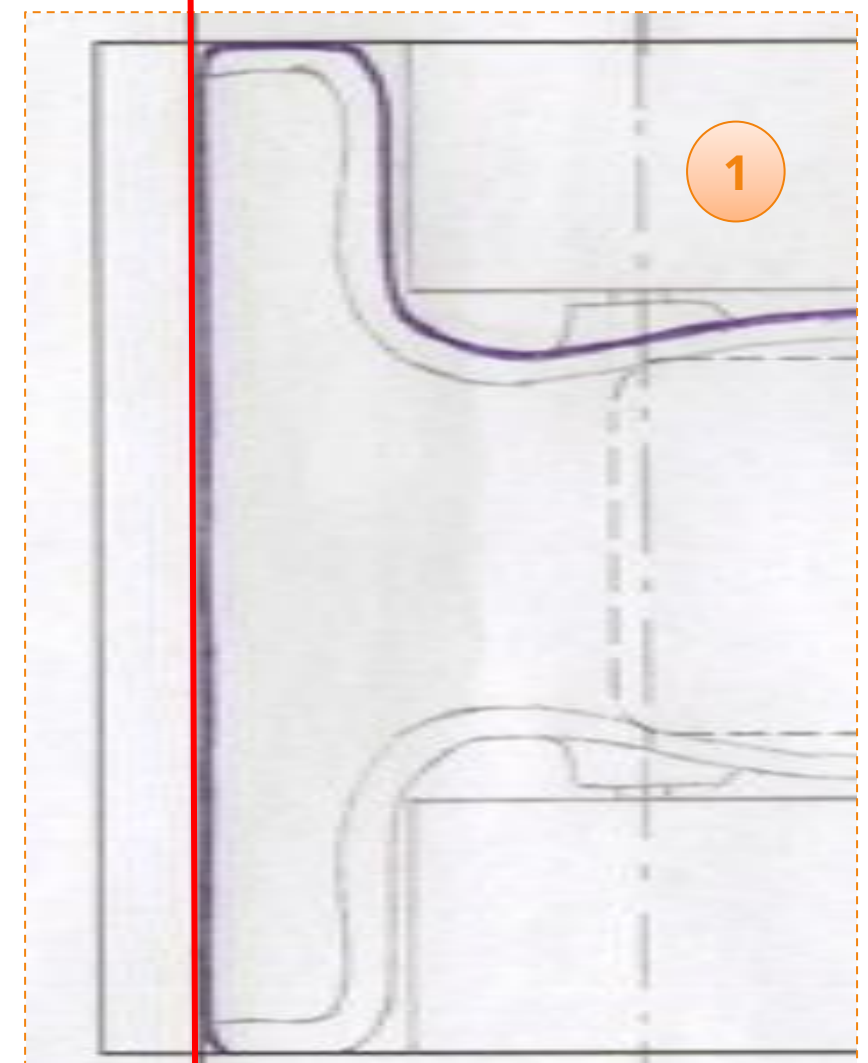
4. Remember to ensure that the curve does not exceed 290 mm.



You can define a selection area before modifying the curve. This will allow you to target your changes.



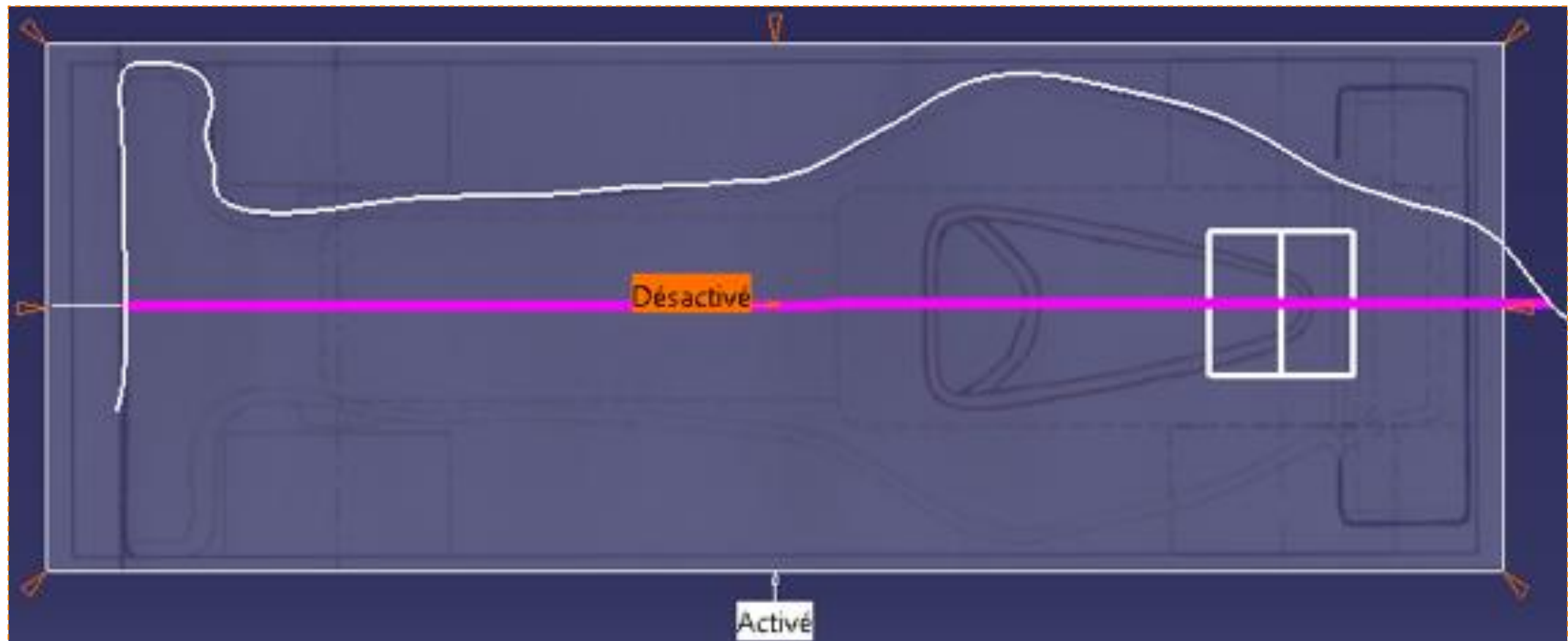
290mm





# Characteristic curves


5. When you are satisfied with your curve, exit the curve editor by pressing **two times** on the key < Esc >.



Don't forget to deactivate the table from the top view by clicking on the "enabled" box that is attached to the top view. Otherwise you will no longer be able to rotate in 3D space.

# Characteristic curves

To make the display more pleasant, we are going to modify the properties of this curve:

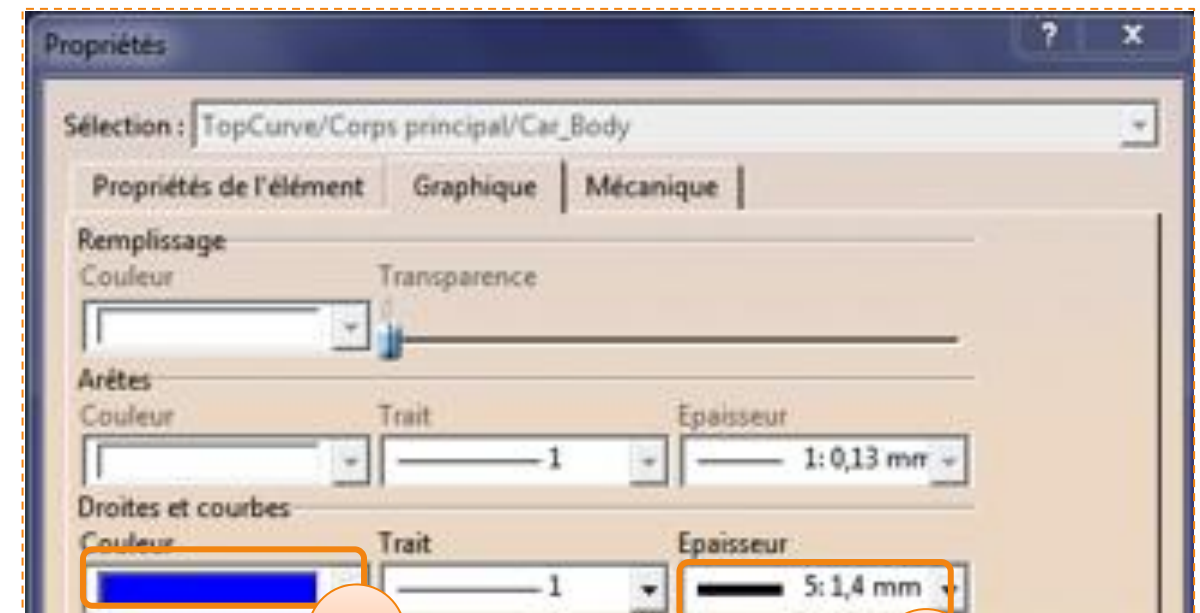
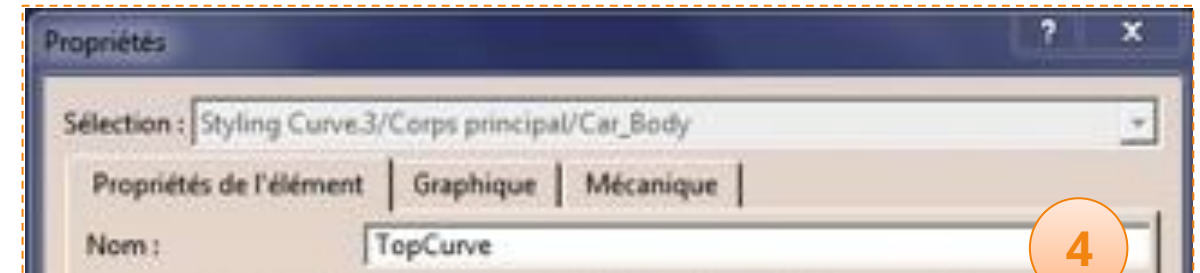
1. Click on the  of the branch **main body**.
2. Right click on **Styling Curve.2**.
3. Click on **Properties**.
4. In the **element properties** tab, rename the curve [TopCurve].
5. In the tab **Chart**,

**a.** Select color **Blue** for the curve.

**b.** Select thickness **5: 1.4mm**.



Save!



# Mechanical concept

In this step we will see how to create the base of your car body. For this we will use:

- ▶ The previously traced curves.
- ▶ The Part Design mechanical design workshop.
- ▶ Functions :
  - ▶ extruding,
  - ▶ Of drilling.

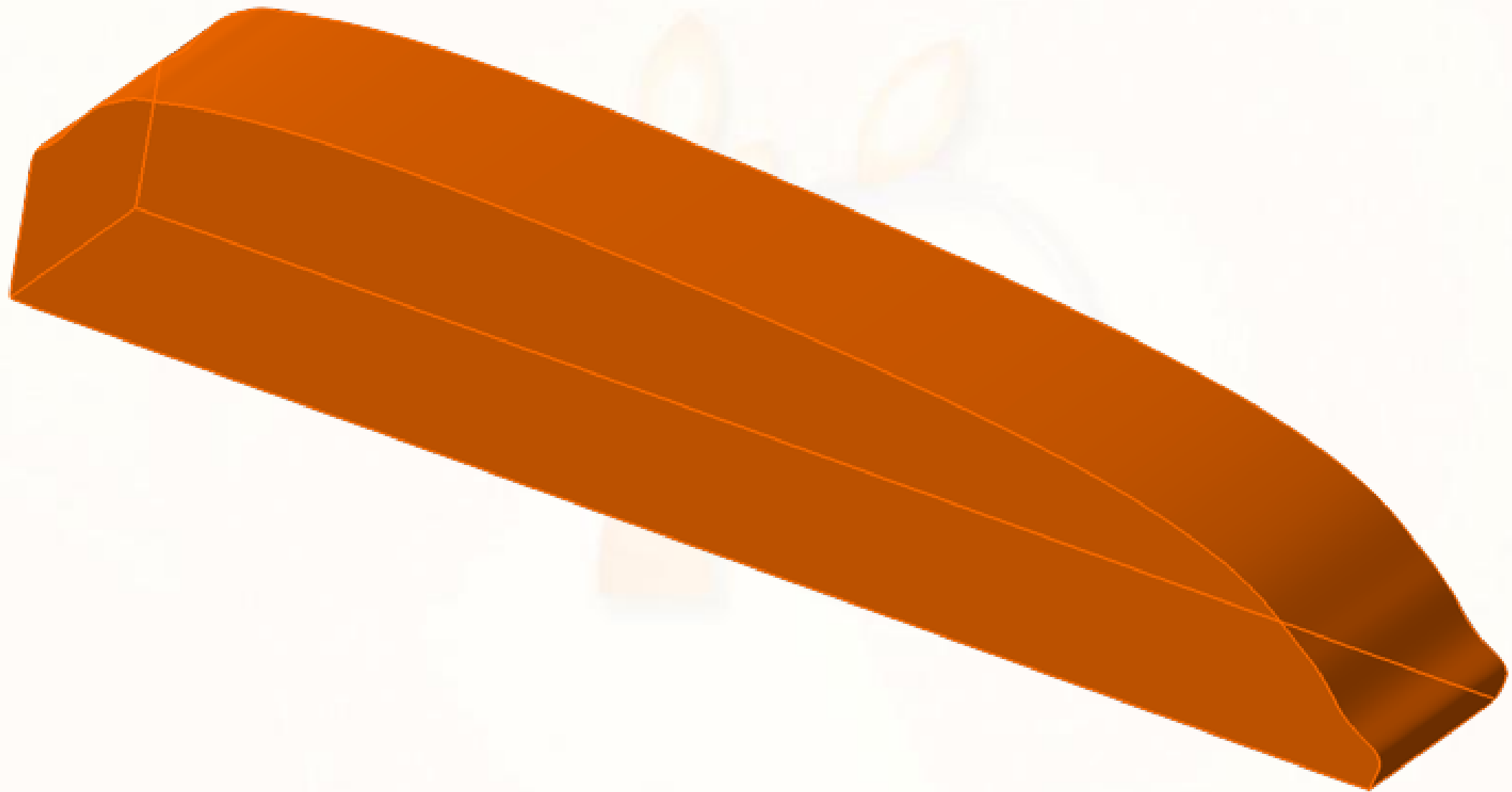


Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.**
    - i. Introduction.
    - ii. Settings.
    - iii. Model the body.**
      - I. Curves features.
      - II. Design mechanical.**
    - iv. Model the cockpit.
    - v. Model the rear wing.
    - vi. Finalization.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

## Mechanical concept

To start, you will use the previous curves. You will draw a first closed sketch, then you will extrude it.

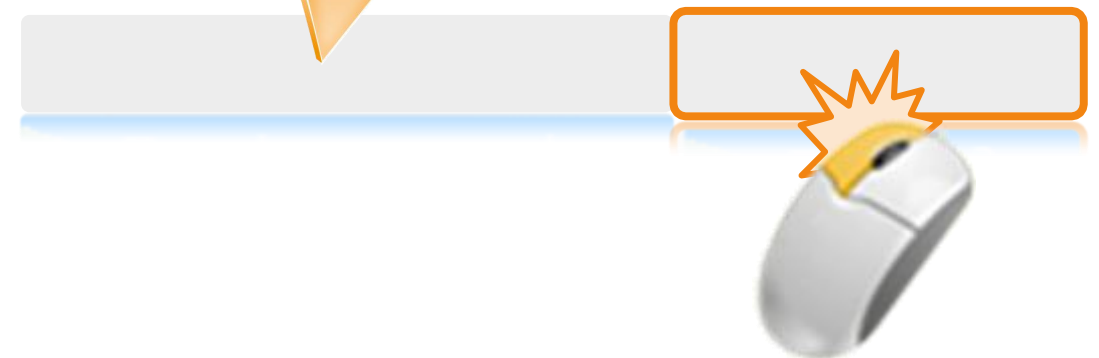
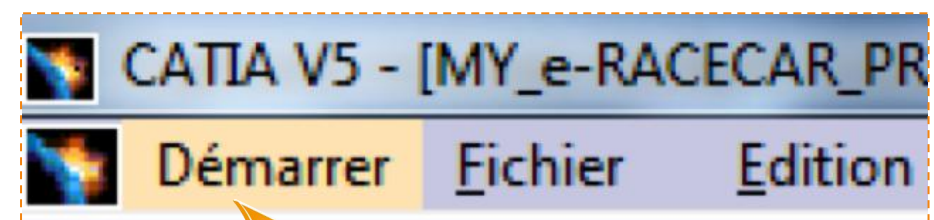
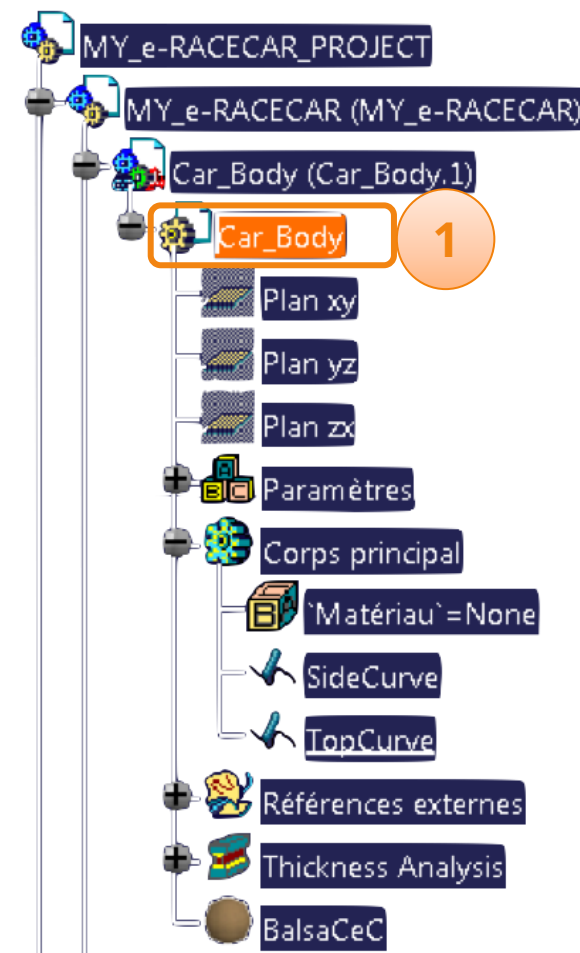


30 minutes

# Mechanical concept

We will use the previously drawn curves to model the frame. To do this, you must use the “Part Design” workbench:

1. Verify that the active part is still **Car\_Body**. If not, double-click it.
2. If your active workbench is already Part Design, skip steps 3-5.
3. Click on **To start up**.
4. Select **Mechanical concept**.
5. Click on **Part Design**.





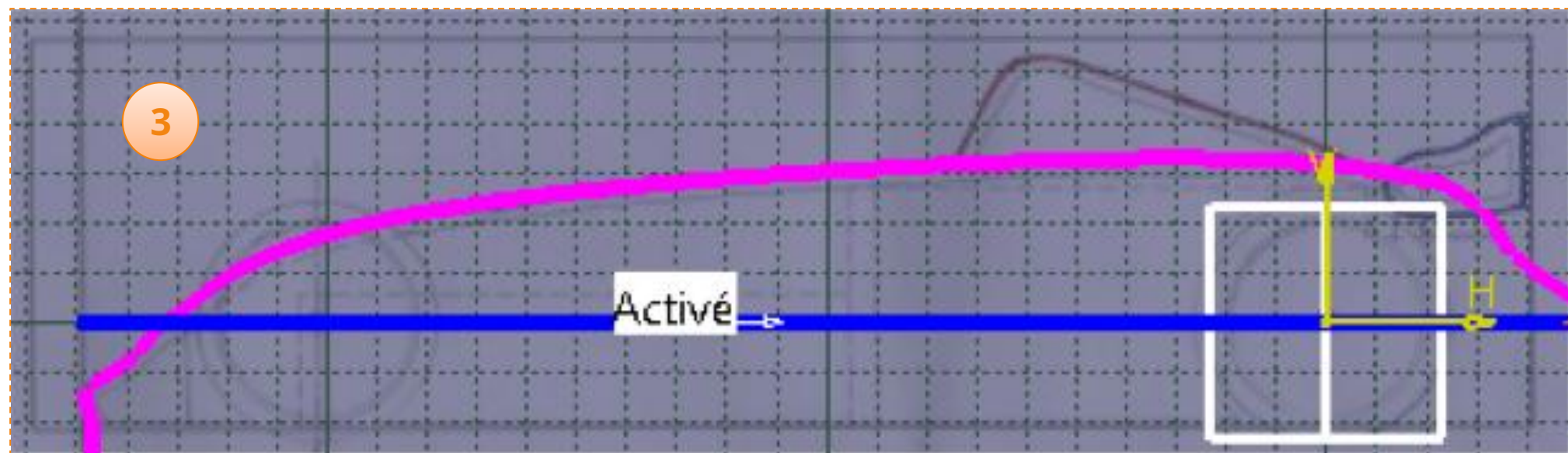
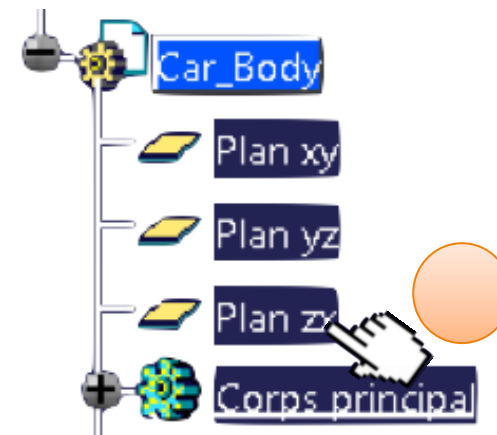
# Mechanical concept

We will go to the sketch creation workbench via the Part Design workbench:

1. Click the icon **Sketch**.

2. Click on the **ZX plane**.

3. A grid appears, indicating that you are in sketch creation mode.



## Sketcher workshop

Here are the different tools that we will be using in the Sketcher workbench.

### 1 Sketch tools:

This window includes the option of snapping to the grid, as well as the mode of creation of construction elements.

### 2 Constraint:

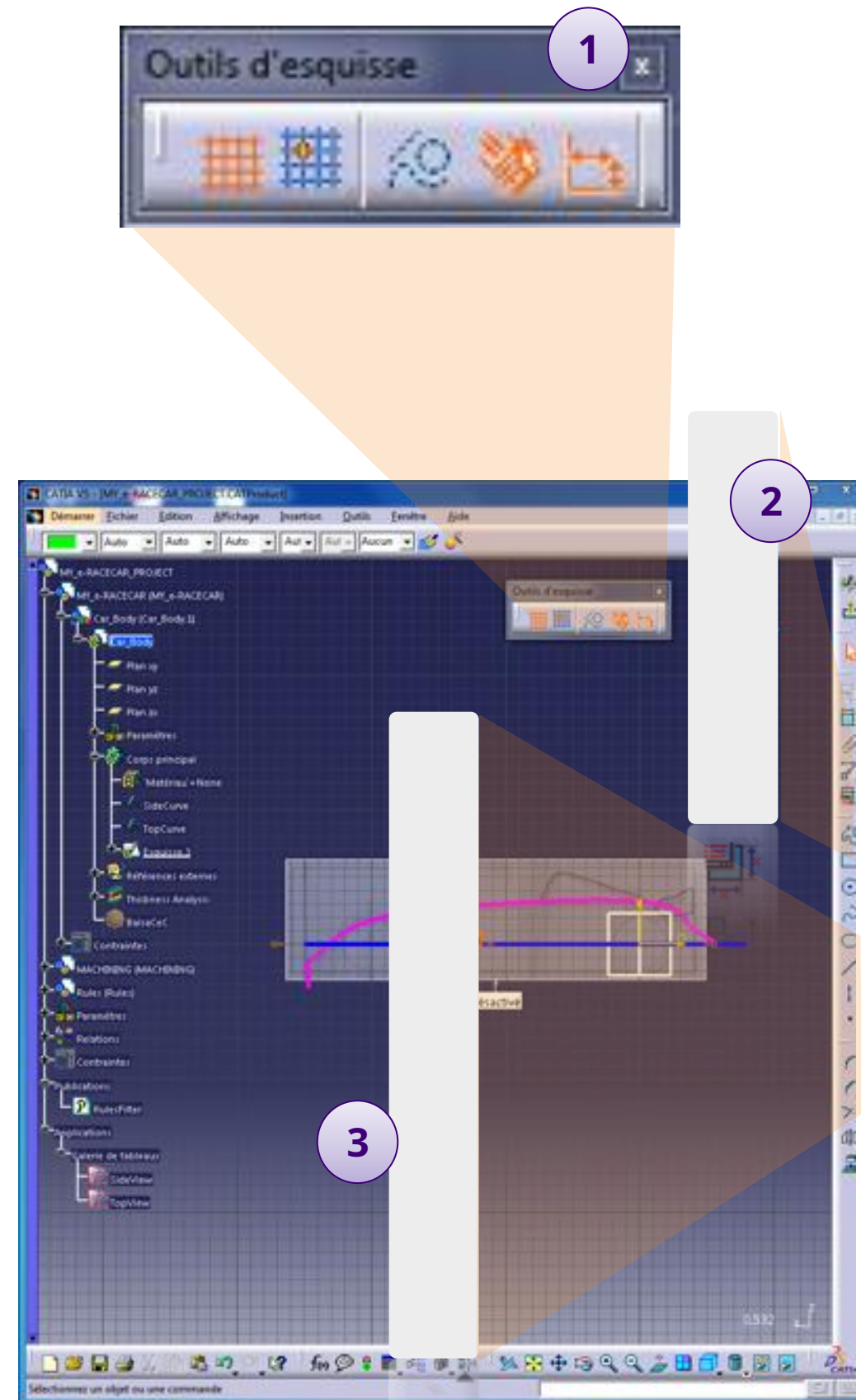
This window contains the icons that will allow us to size our sketches and constrain them geometrically.

### 3 Outline:

This window contains the different types of geometric elements that you can draw.

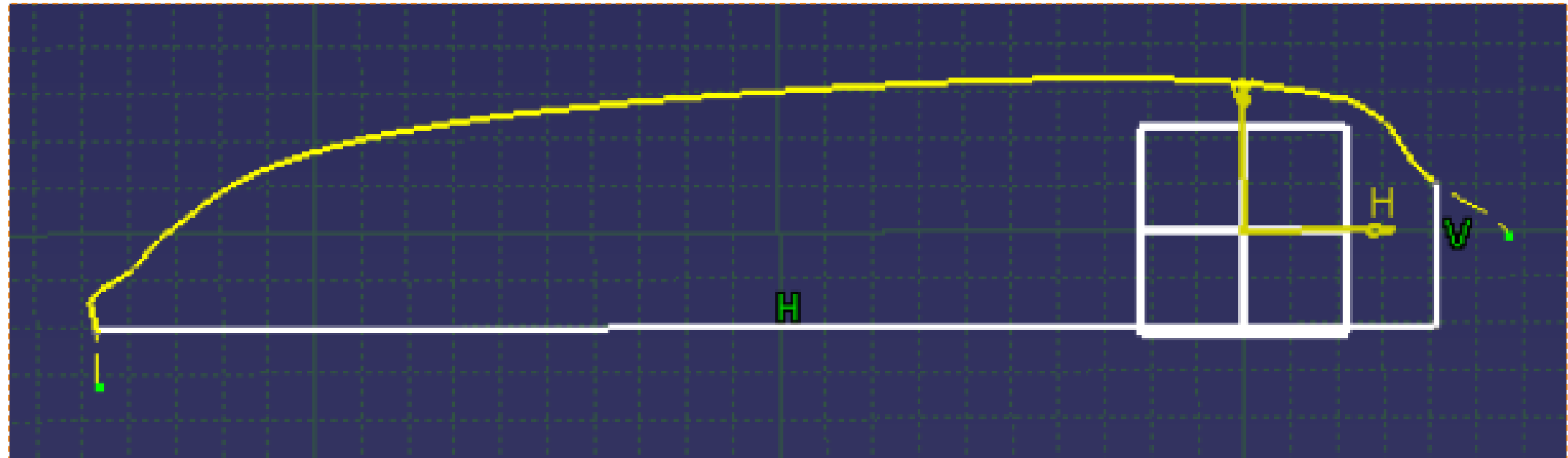


If you can't find the toolbars, consult the method sheet "CATIA Sketcher".



# Mechanical concept

We will create the following sketch:



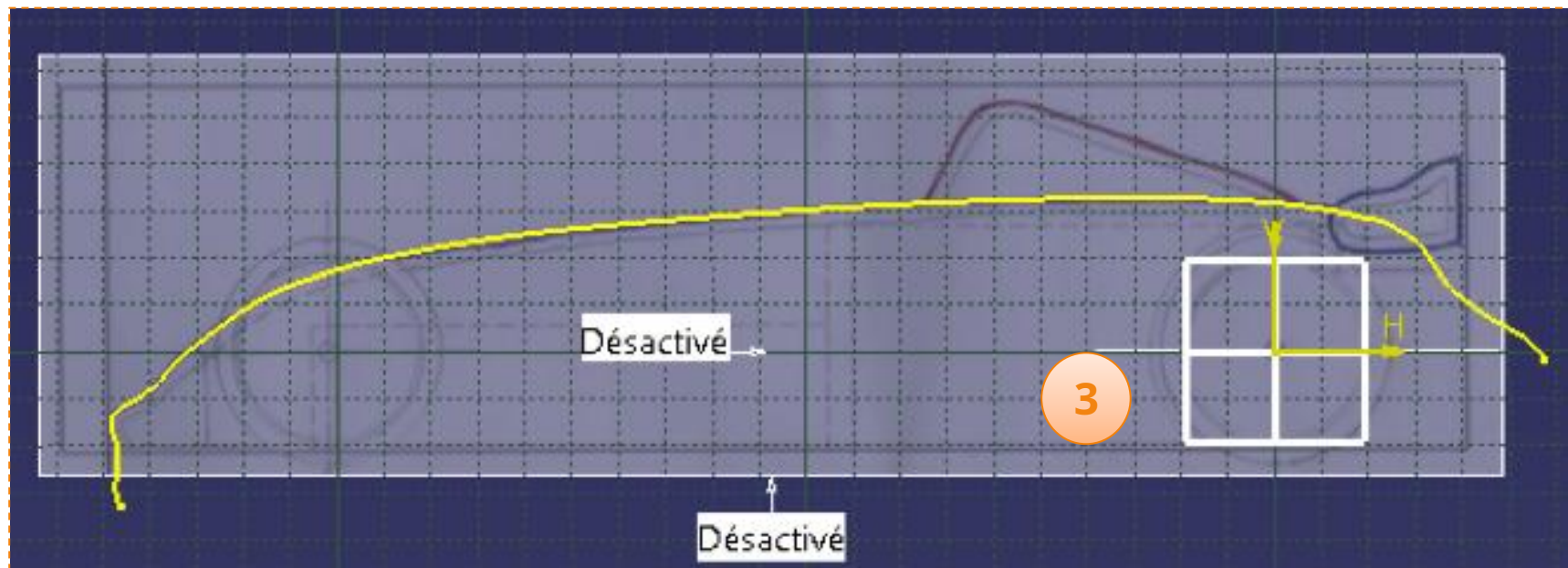
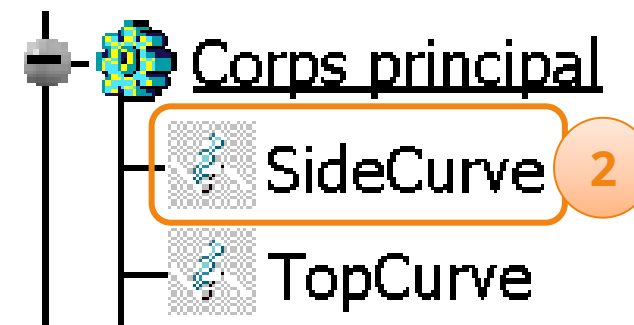
For this we must:

- ▶ Retrieve the SideCurve projection. Complete
- ▶ the sketch with the outline tool. Constrain
- ▶ the sketch.
- ▶ Re-trim the sketch.

# Mechanical concept

To begin, you must project the curve of the profile into the plane of the sketch:

1. Click the icon **Projection of 3D elements**.
2. Click on the pink curve **SideCurve**.
3. You can see the projection of the curve, above and thinner.
4. Hide curves **SideCurve** and **TopCurve**. This will save you from selection errors later.

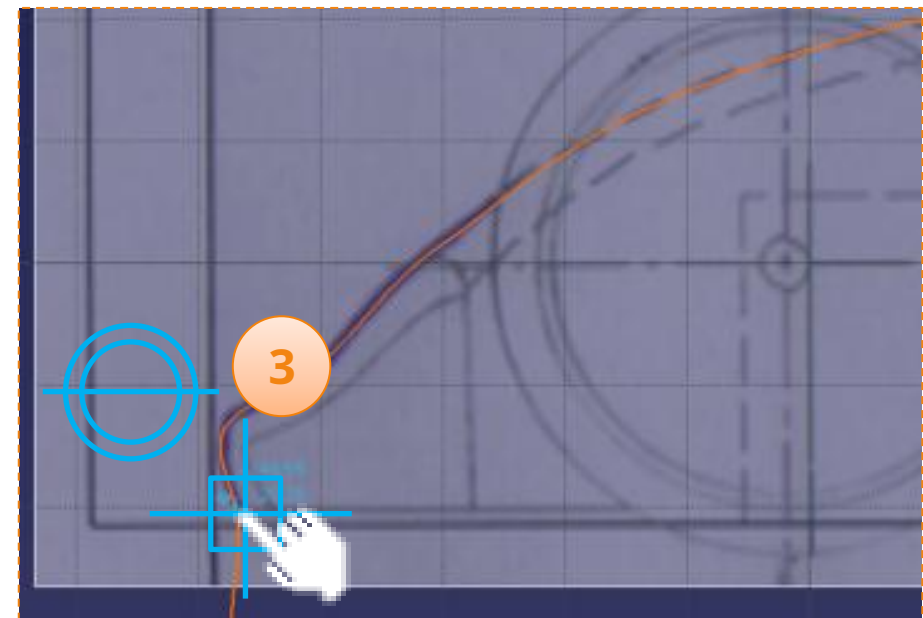




# Mechanical concept

We now need to complete the projection with two curves that will close the sketch:

1. First check that you are using the same settings as opposite in the **Sketch tools**. If a setting is incorrect, click it.
2. Click the icon **Outline**.
3. Click on the projected curve, at the location indicated in the image opposite. This point is the intersection of the projected curve with the bottom of the car.



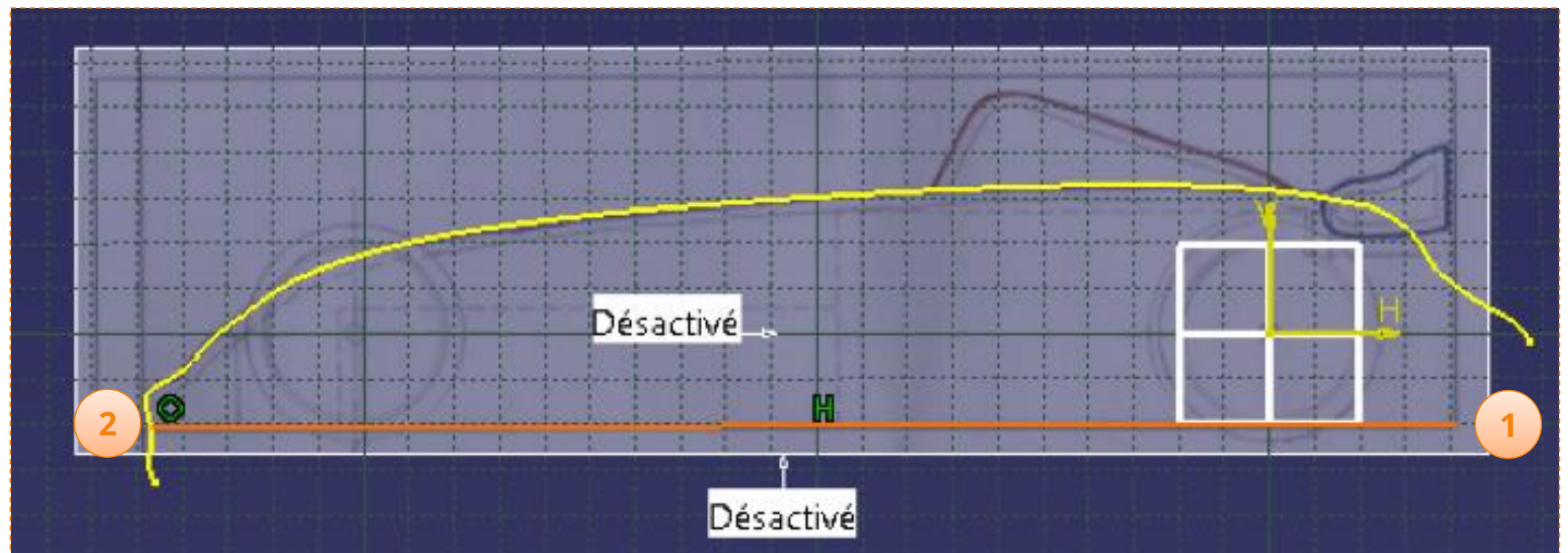
This icon means that your point will be on the curve you are pointing to.





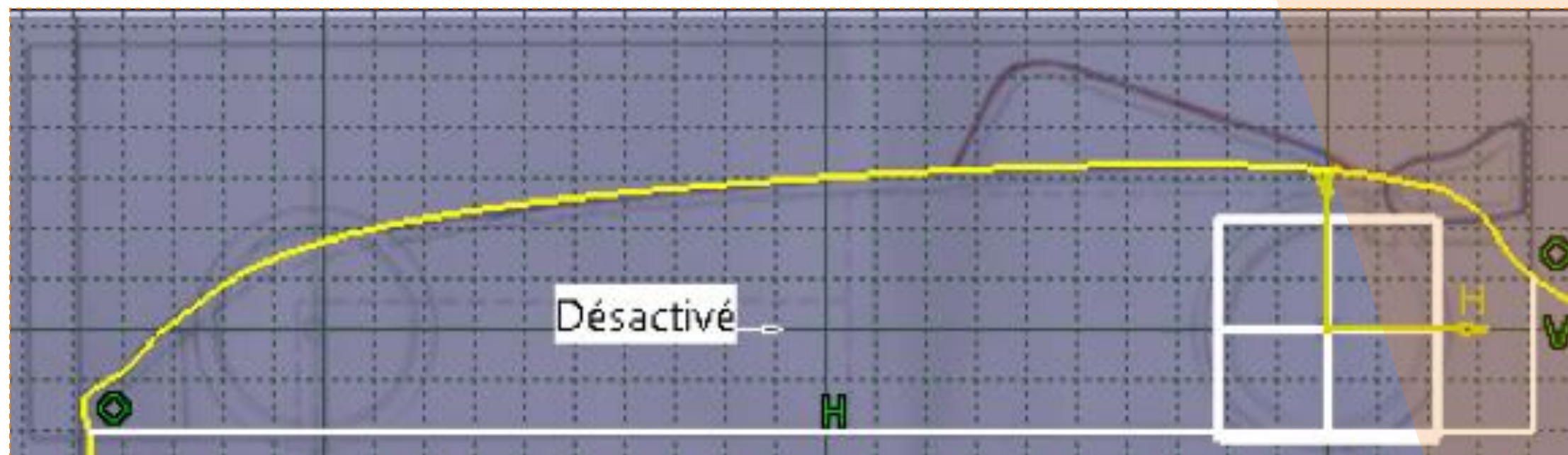
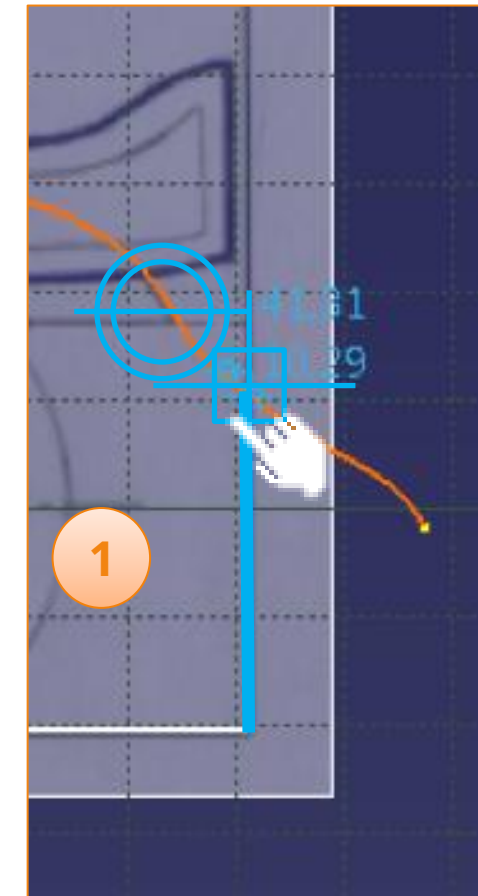
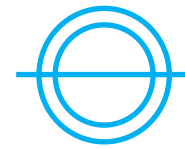
# Mechanical concept

4. Position the pointer below the rear end of the car to display the dotted lines.
5. Make sure the horizontal line is blue, and click to set the second point.



# Mechanical concept

6. Move the mouse over the projected curve, vertical to the center of the marker. The presence of the symbol opposite and the blue line allows this to be checked.
7. Double-click to complete the outline or single-click then **<Esc>**.



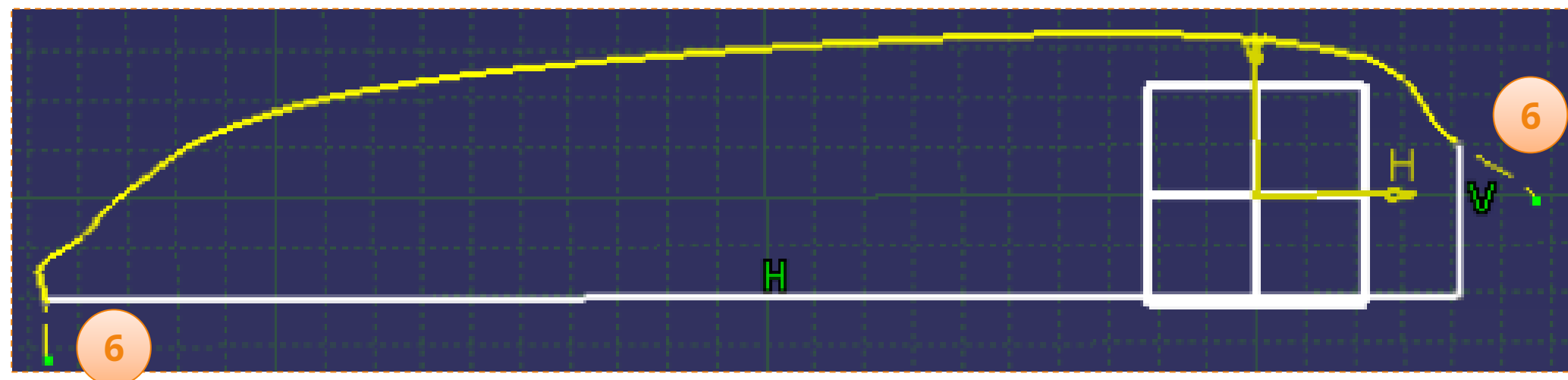
# Mechanical concept

The restriction takes place as follows:

1. Click on the **projected curve**.
2. Click on the **straight horizontal**.
3. Click the icon **relimitation**.
4. Click on the **projected curve**.
5. Click on the **straight vertical**.



6. Verify that you get the results below. You may still have both tables displayed. If so, hide them by right-clicking, then **Hide/Show**.



# Mechanical concept

The sketch drawing is now complete. We can leave the sketch mode, for that:

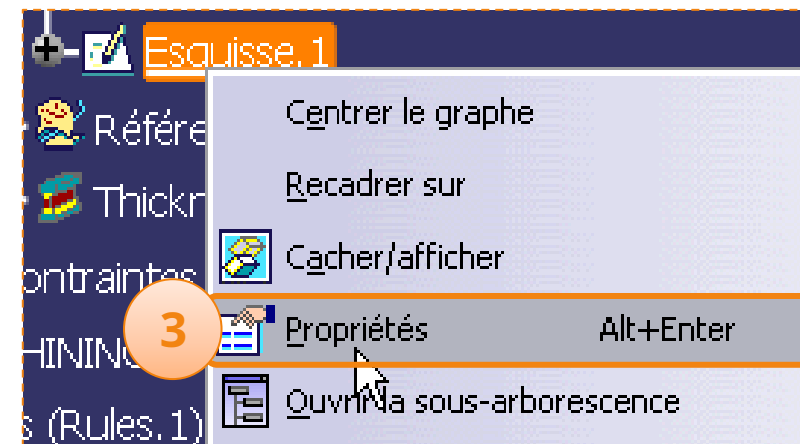
1. Click the icon **Leaving the workshop**.



To get a clear and readable tree, we will rename the sketch. This one is in the main body and is currently called Sketch.1.

2. Right click on **Sketch.1**.

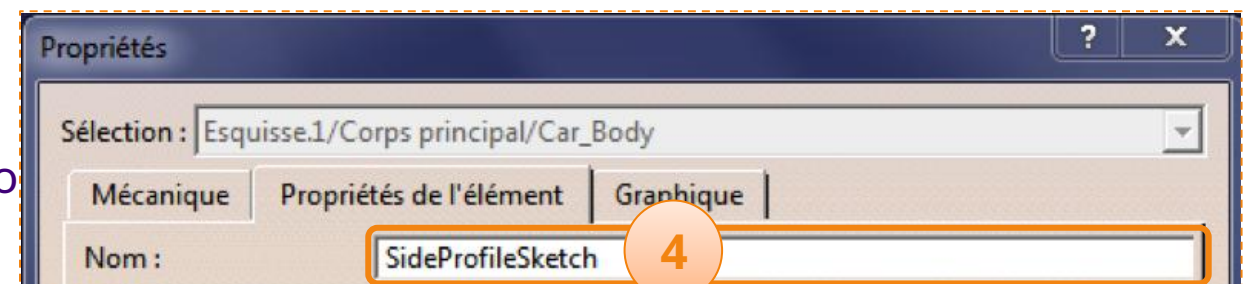
3. Click on **Properties**.



4. In the tab **Item Properties**, rename the name [SideProfileSketch].

5. Click on **okay**.

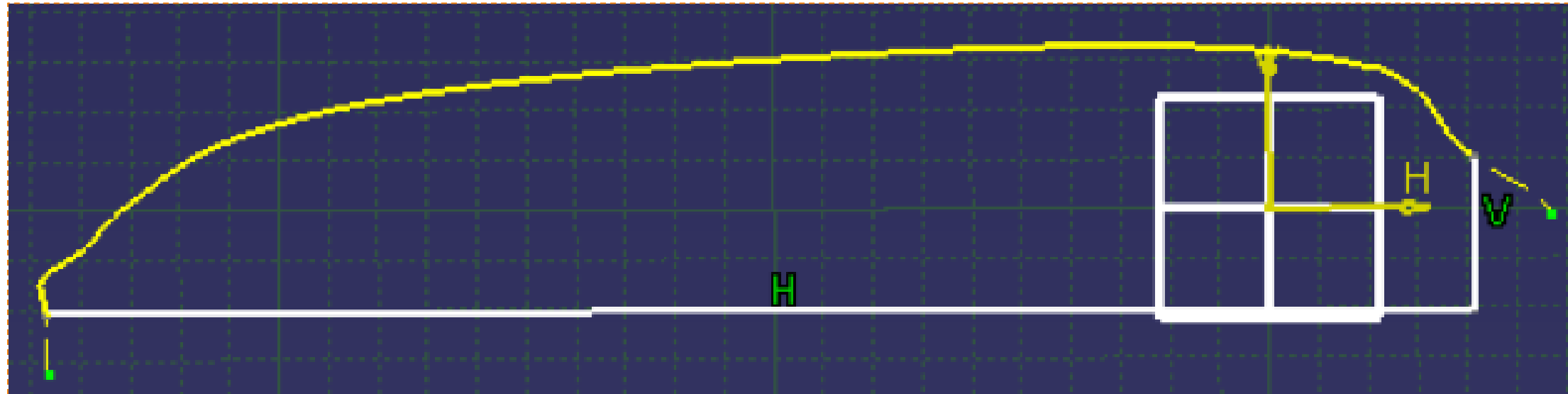
Why rename? When you create elements, CATIA gives them a generic name (ex: Extrusion.1). When you need to edit an element in your room, if you haven't renamed them, it will be difficult for you to spot which element to click on.





# Mechanical concept

- Congratulations, your first sketch is complete!



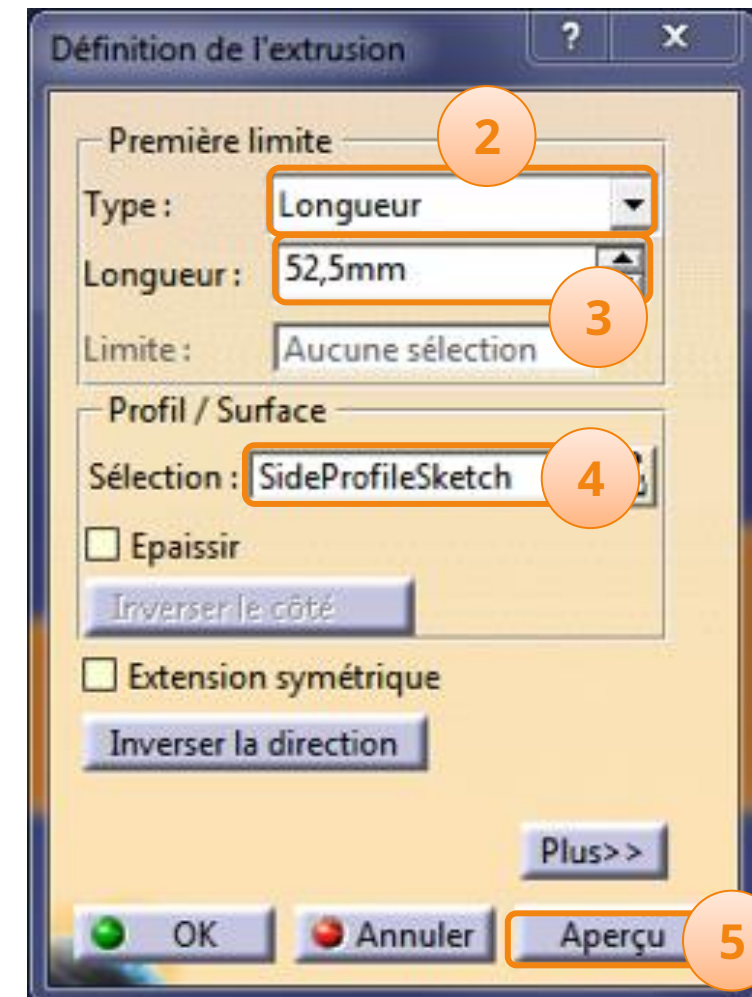
- To modify this sketch you can:
  - Directly modify the curve drawn in Imagine & Shape, the modification will be automatically updated.
  - Modify the values of the dimensions, the relimitations will be updated automatically.



# Mechanical concept

Now we are going to create the beginning of the body of the car thanks to the sketch previously drawn:

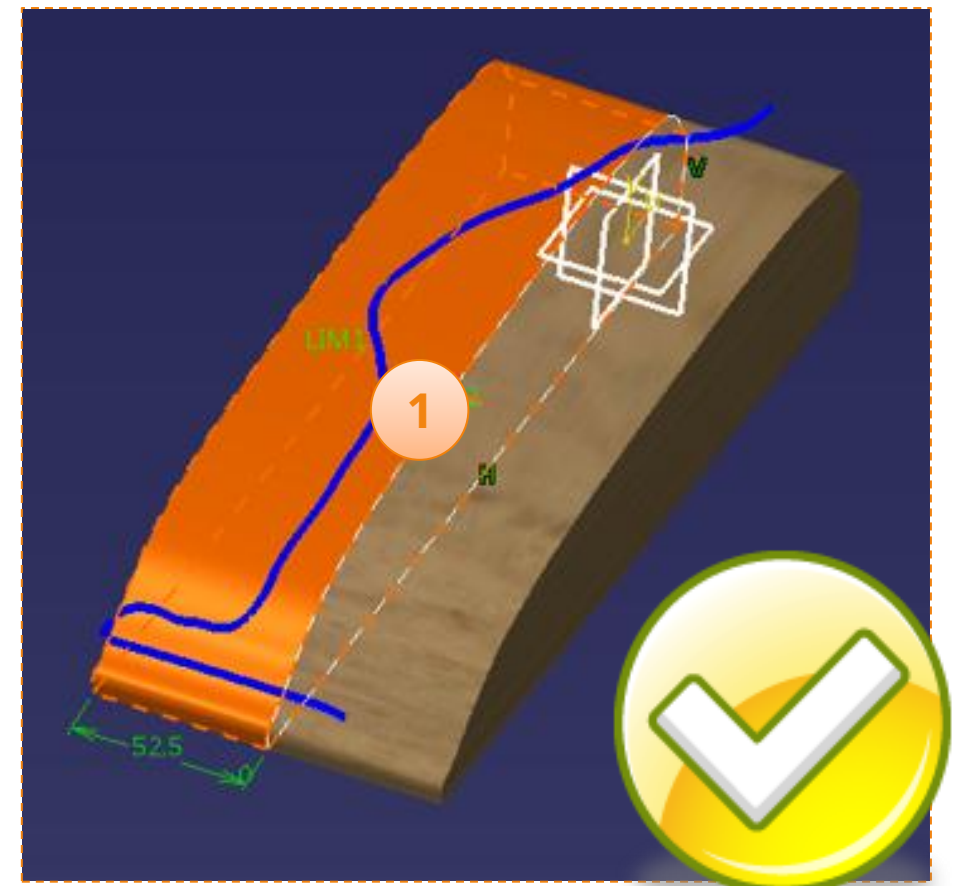
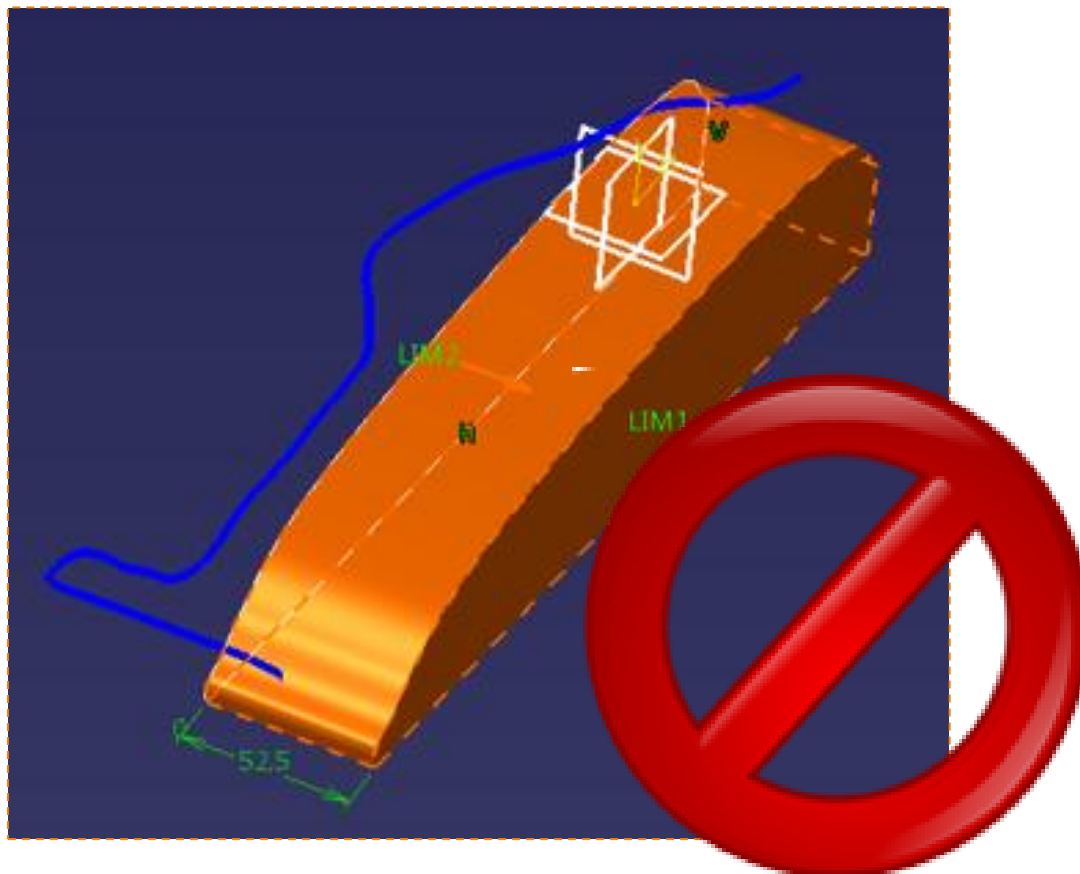
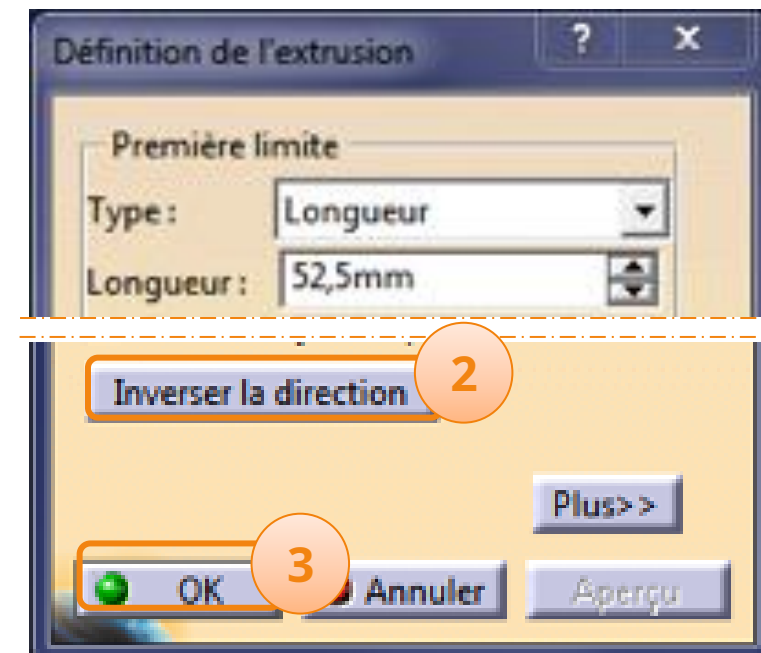
1. Click the icon **Extrusion**.
2. Check that the type is **Length**.
3. In the window that appears enter the value **[52.5mm]**.
4. Check that the selected profile is **SideProfileSketch**. If not, click on the sketch.
5. Click on **Insight**.



# Mechanical concept

The extrusion preview has been displayed, beware of the following:

1. You must find the extrusion on the side where you drew the TopCurve. **If so, go to step 3.**
2. If not as on the left, click **Reverse direction** in the dialog box. You will then see the extrusion on the other side.
3. Validate by clicking on **okay**.



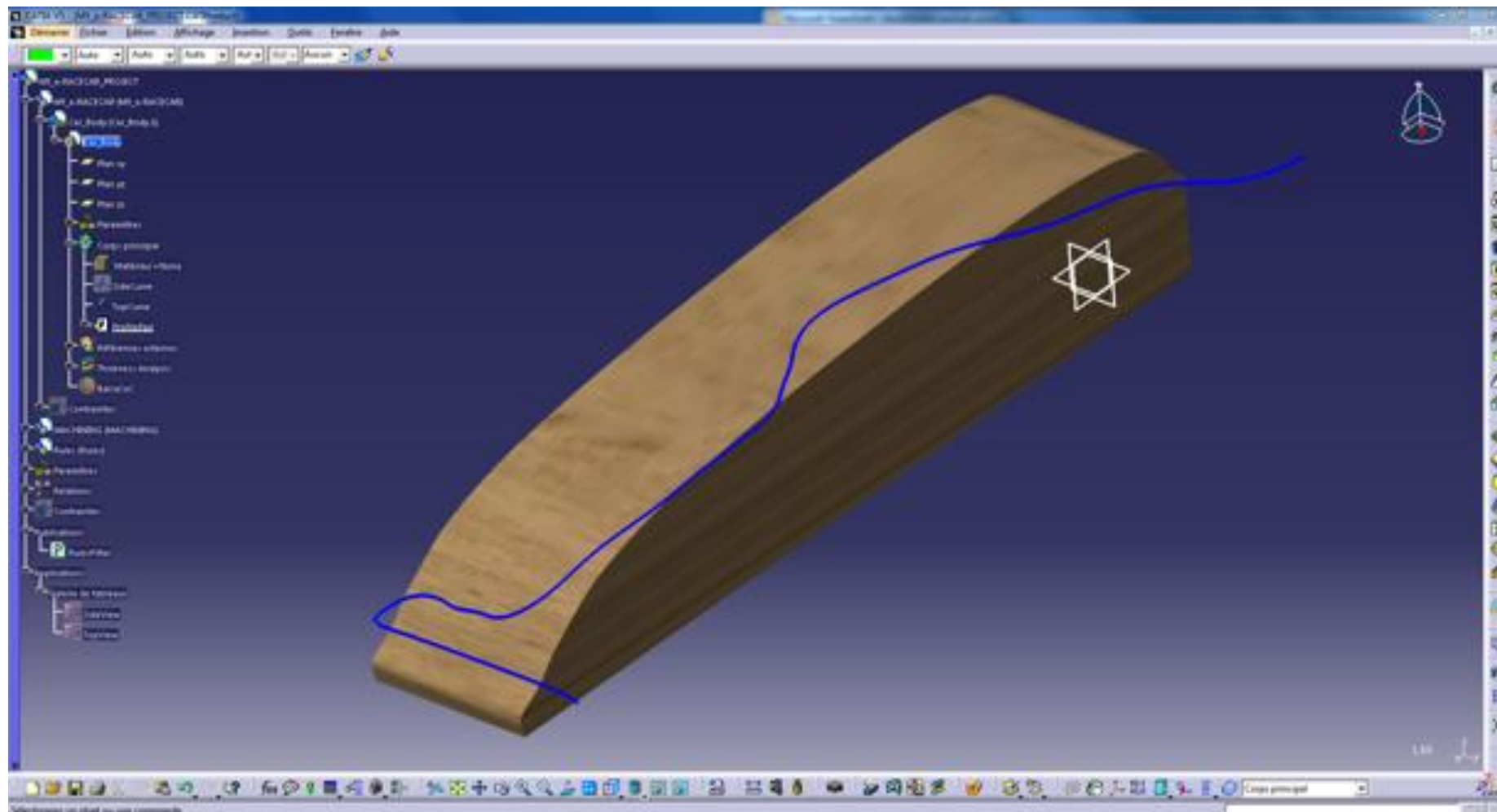
# Mechanical concept

We will rename this extrusion:

- 1.Right click on**Extrude.1**.
- 2.Click on**Properties**.
- 3.Rename it [ProfilePad].
- 4.Congratulations, you have modeled the chassis base!



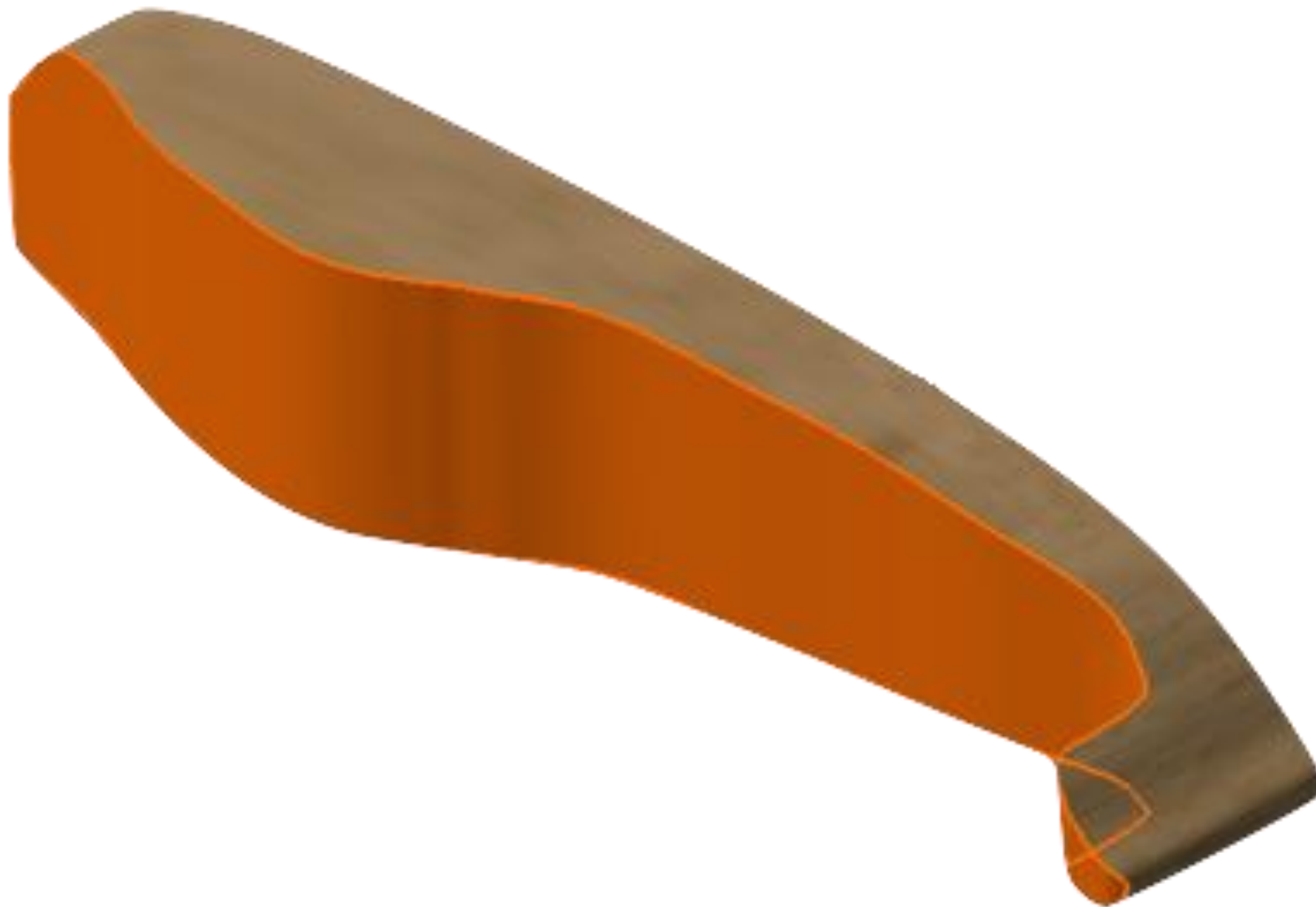
For more information on modifying the properties of an element, see the file method "Modify the Properties of Elements".



# Mechanical Design – Contour Cut

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Top C

ee





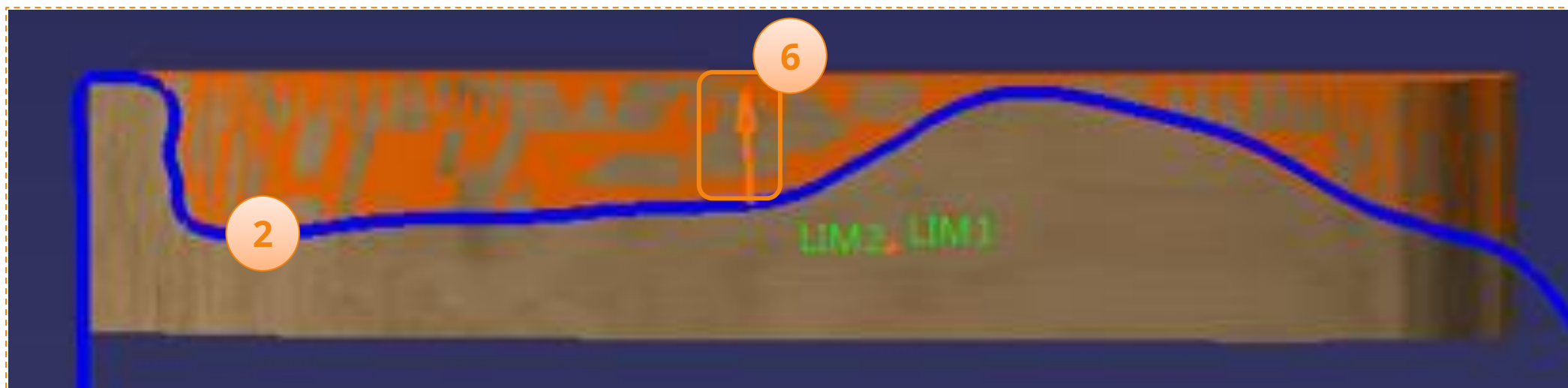
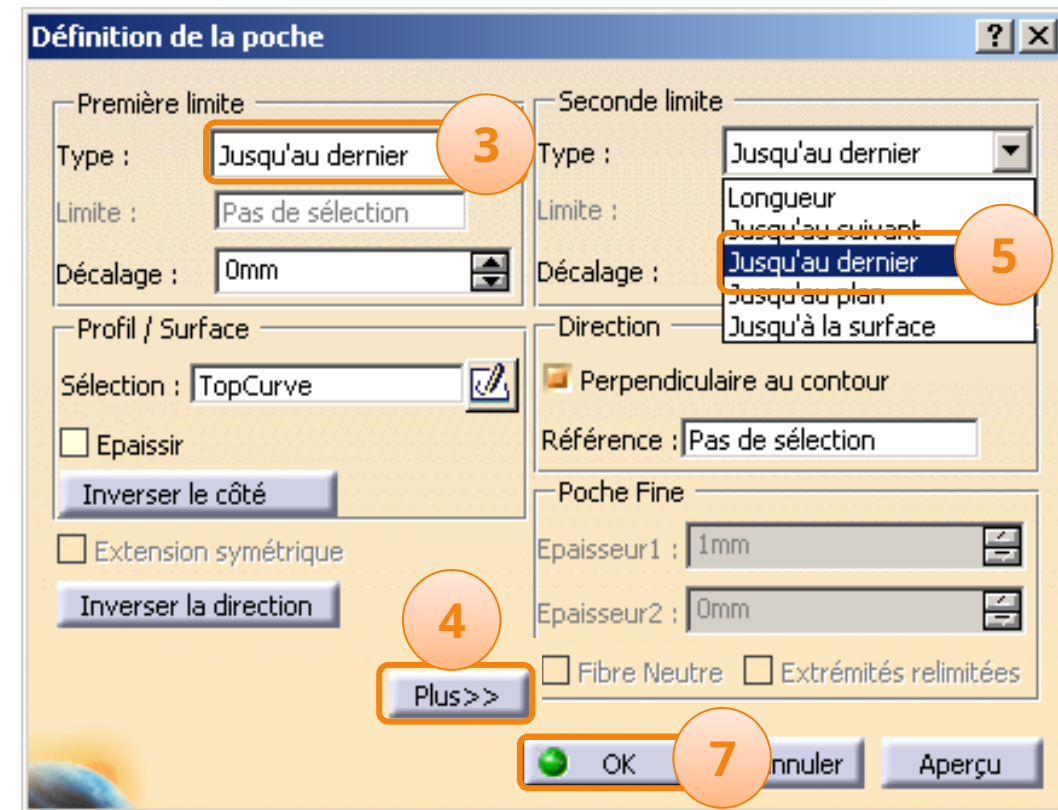
# Mechanical Design – Contour Cut

We are going to make the outline of the car, for this:

1. Click the icon **Poached**.
2. Click on the curve **TopCurve**.

To be sure to perform the operation on the entire body, select the following options:

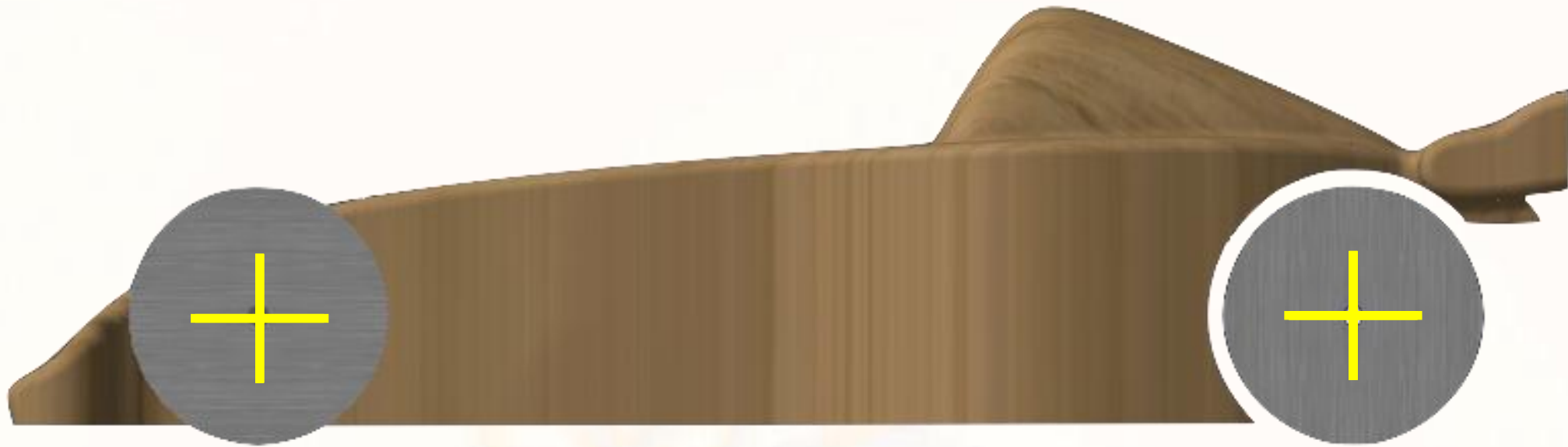
3. In option **Kind**, choose **Until the last one**.
4. Click on **More>>** to display the second part of the window.
5. In option **Kind** of the **Second limit** also choose **Until the last one**.
6. Check that the orange arrow is oriented as below.
7. Click on **okay**.
8. Rename the pocket [SidePocket].



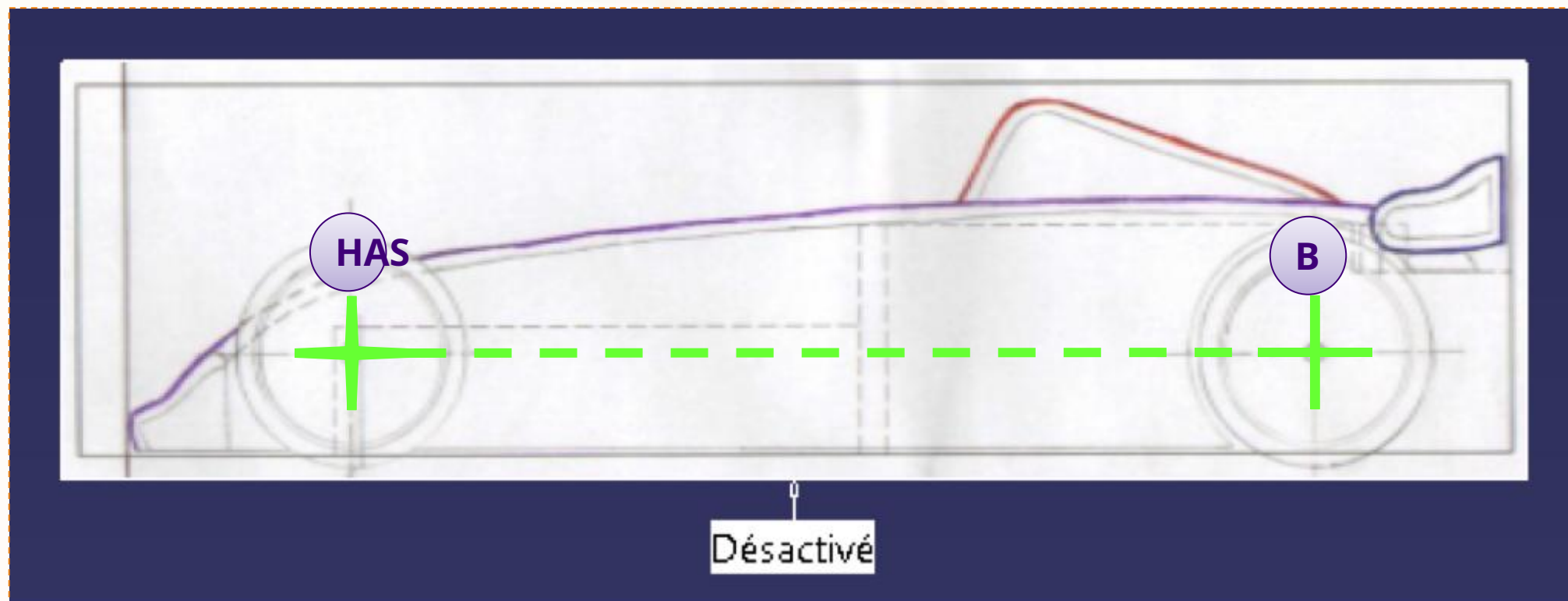


# Mechanical Design – Wheel Positions

1.We



2.Ci-d



10 minutes

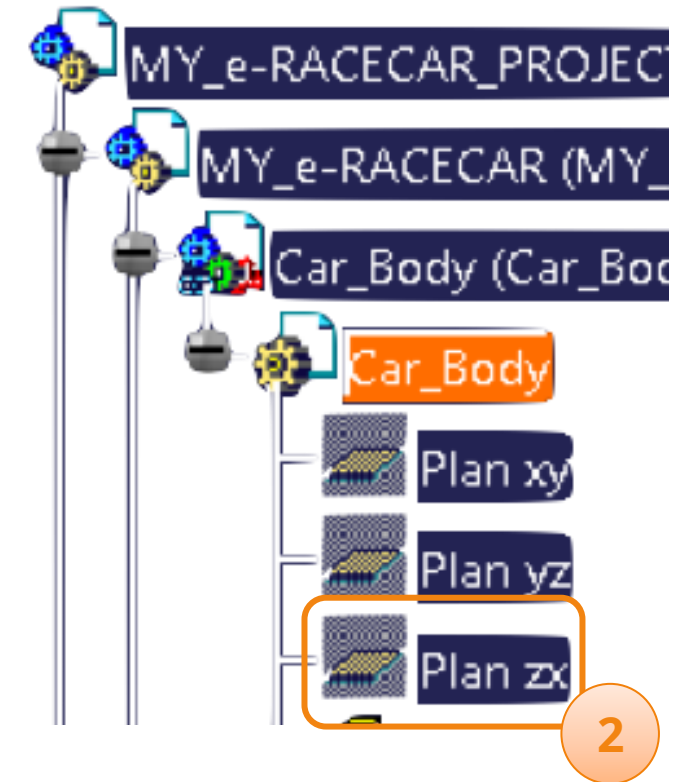
# Mechanical Design – Wheel Positions

We are going to create a sketch that will be used to position the axles of the wheels. This sketch will not be used to create material, but to define construction elements.

1. Click on **Sketch**.



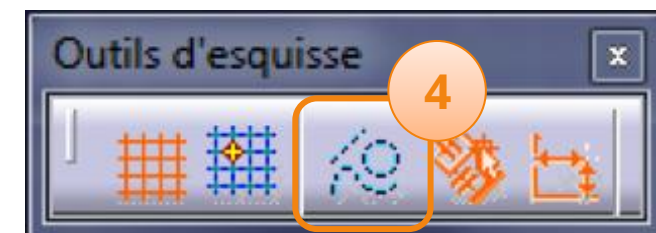
2. Click on the **ZX Map** which is located in the element **Car\_Body**. You switch to the Sketcher workbench.



3. Click the icon **Right**.

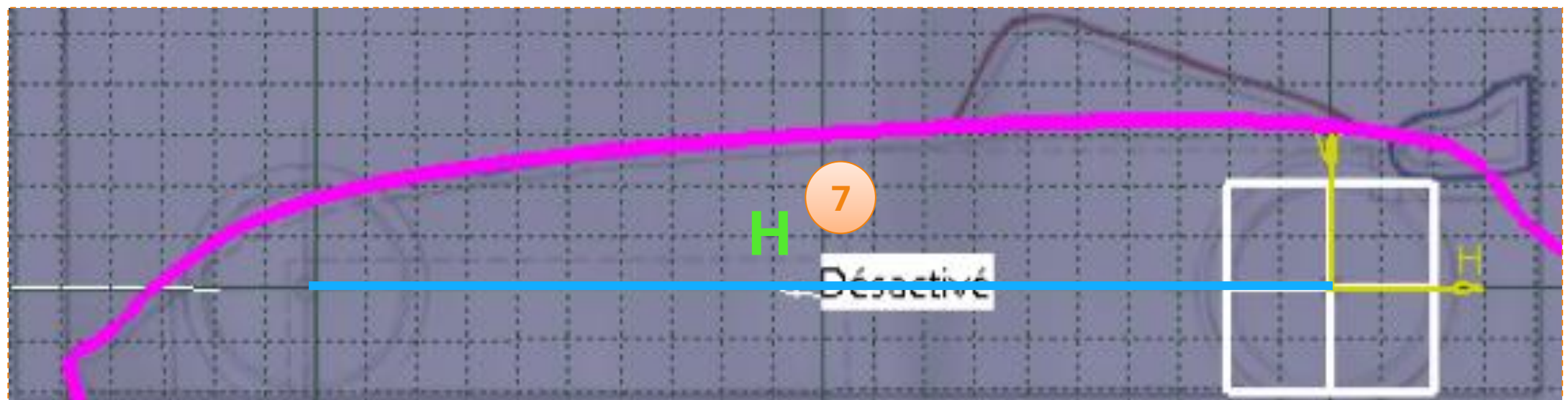
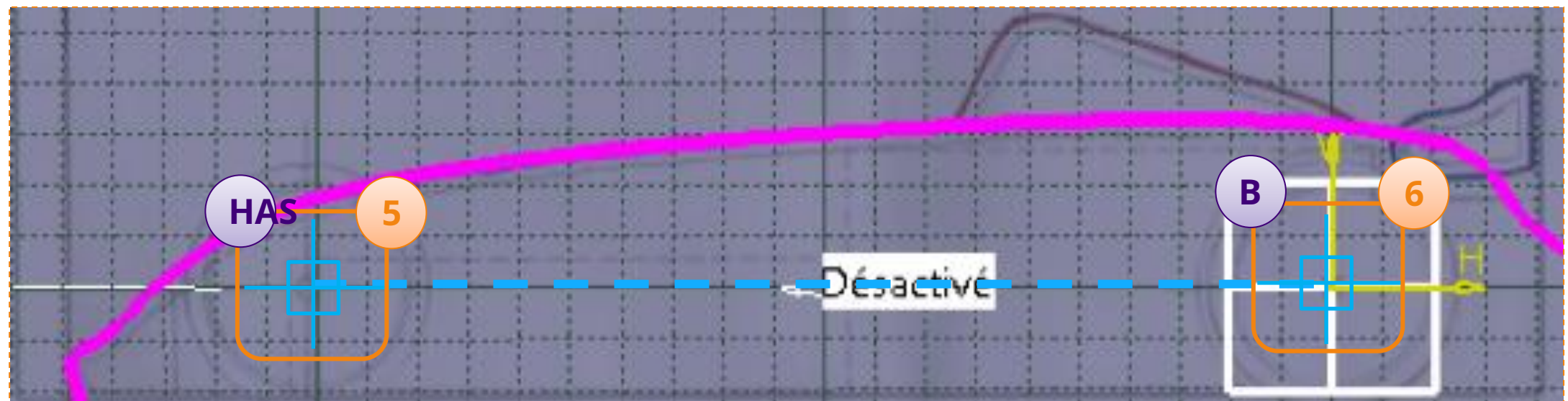


4. Click the icon **Standard elements/Construction elements** to switch to construction elements mode.



# Mechanical Design – Wheel Positions

5. Click on the **ITEM A**, which represents the position of the front axle.
6. Make sure the line is horizontal (blue dotted line) before clicking on the **B-POINT** which represents the position of the rear axle.
7. Check the presence of **H** on the right which means it is horizontal.

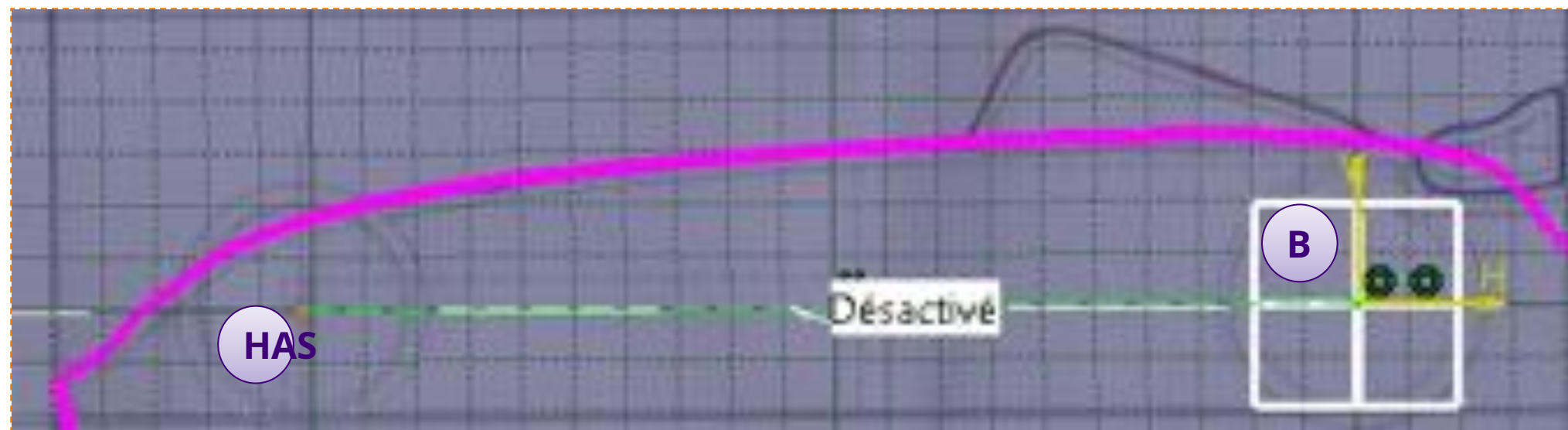
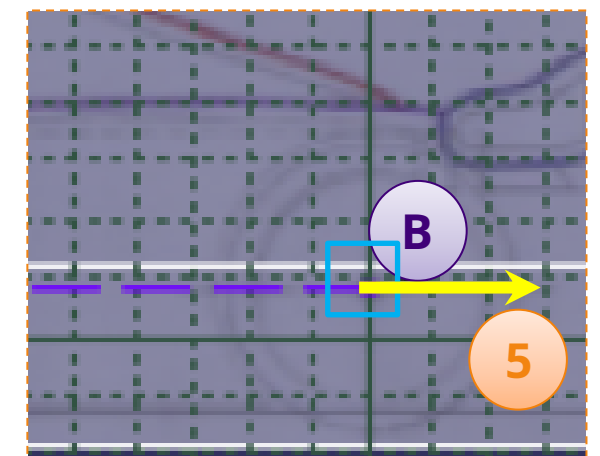
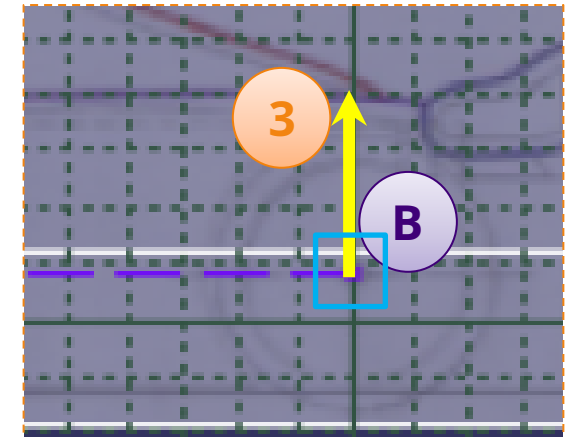




# Mechanical Design – Wheel Positions

Then we are going to position the wheelbase in relation to the reference of the car:

1. Click the icon **Constraint**.
2. Click on the DOT **B**.
3. Click the **Vertical axis**.
4. Right-click in a blank area to validate the dimension, then select **Coincidence**.
5. Using the same method create a dimension between the POINT **B** and the **horizontal axis**.
6. Check that the POINT **HAS** is still on the axis of the front wheels (position to the eye on the SideView drawing).



# Mechanical Design – Wheel Positions

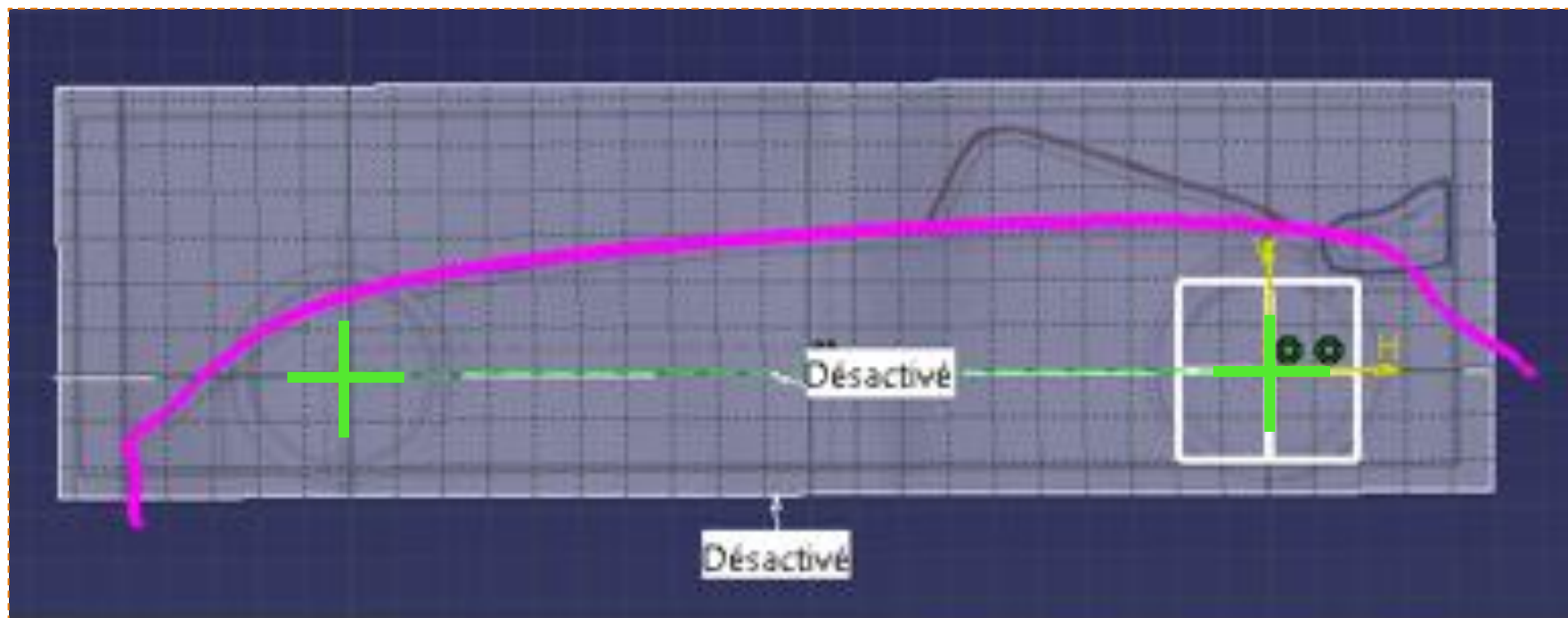
What interests us in this sketch are the extremities of the line, the points **H** and **B**. To be able to rely on them for the next operations, they must not be construction elements.

For that :

1. Select **one end**.
2. Click the icon **building elements**.
3. Select **the other extremity**.
4. Click the icon **building elements**.



Note that the dots are now represented by small crosses.





# Mechanical Design – Wheel Positions

1.To exit sketch mode, click the icon **Leaving the workshop.**

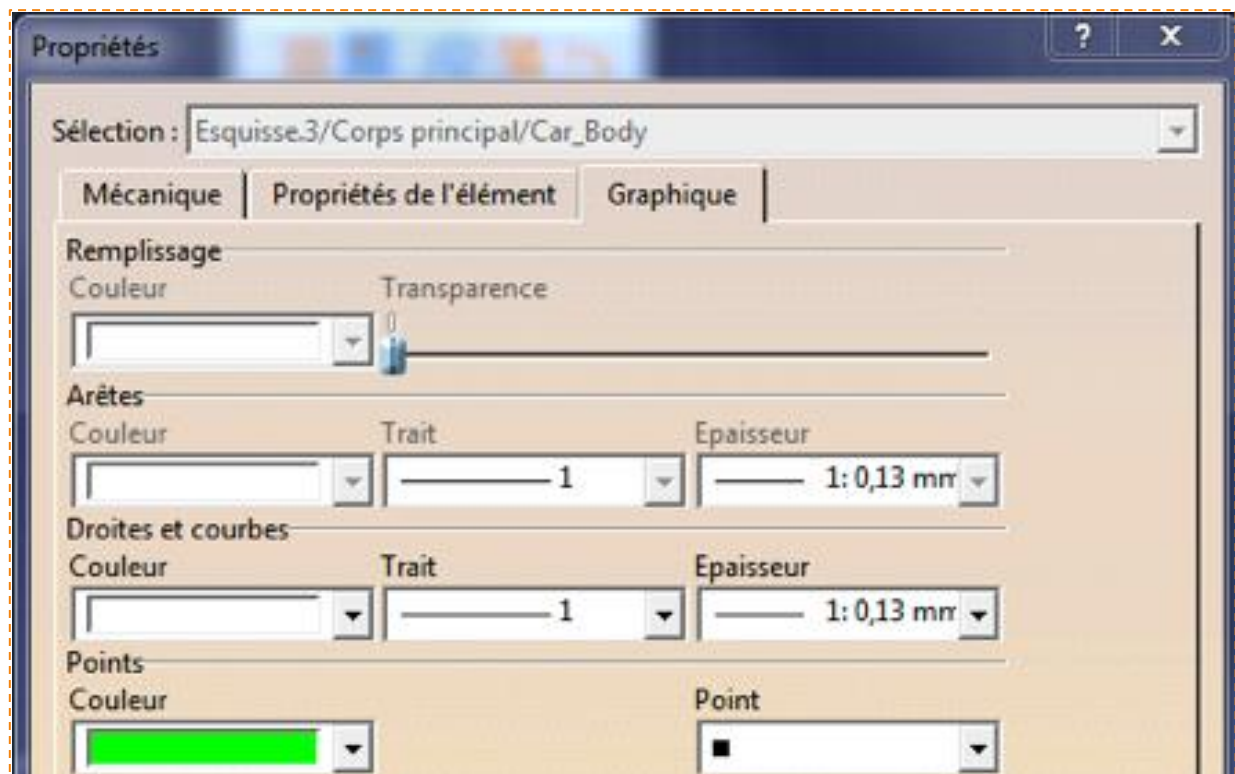
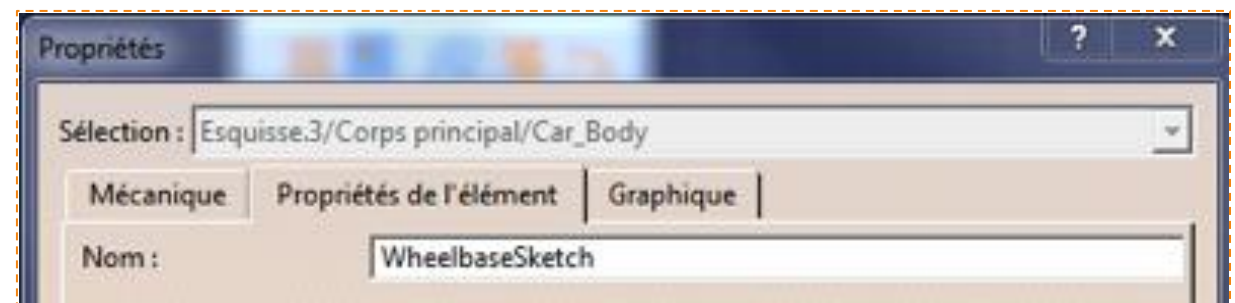
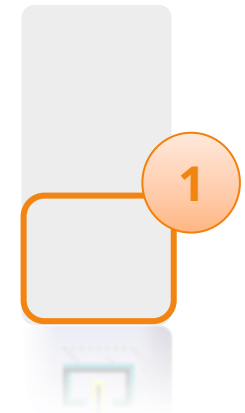
To rename the sketch:

2.Right-click in the tree on**Sketch.2**. This must appear in the tree of specifications below the ProfilePad extrusion.

3.Click on**Properties**.

4.Rename the [WheelbaseSketch].

5.In the tab**Chart**, configure as opposite.



## Mechanical Design – Wheel arches

For the modeling of the wheel arches we will use the previous sketch. This method will allow us to easily modify their positions if necessary.



**10 minutes**

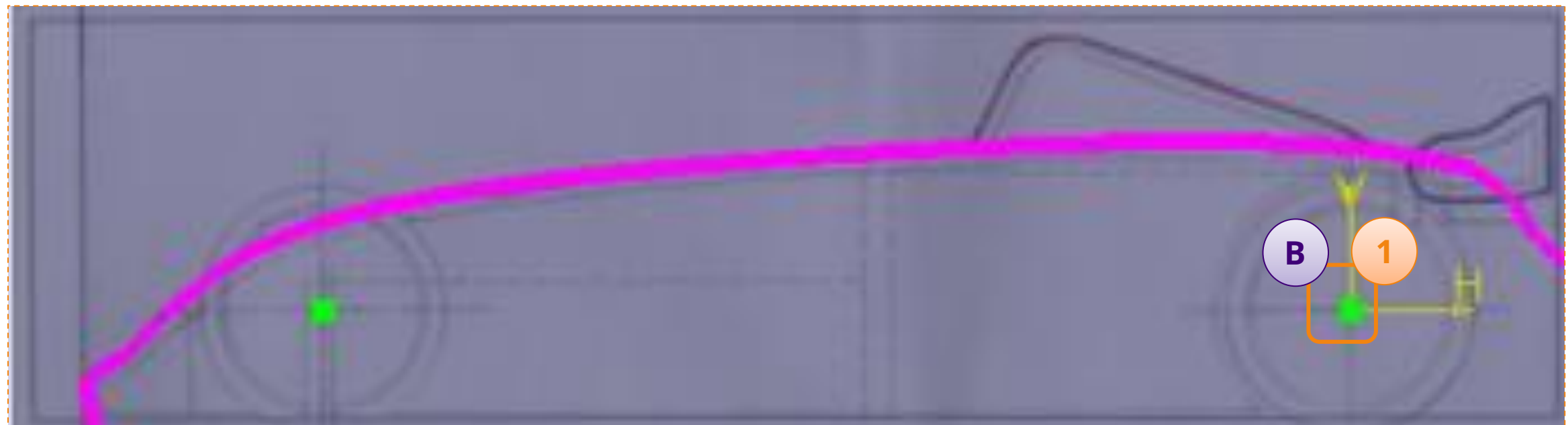
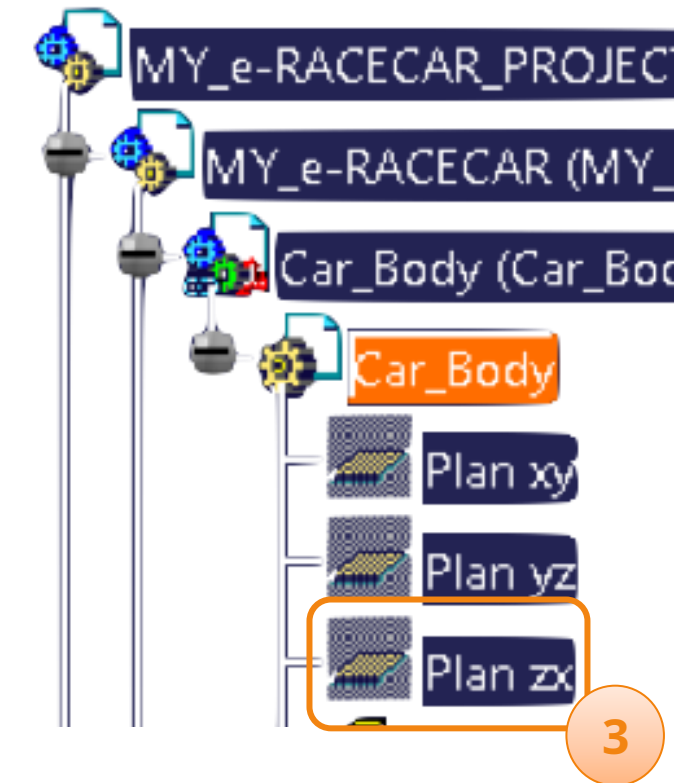
# Mechanical Design – Wheel arches

We are going to do the rear wheel arch. For that :

1. Click on the DOT **B** representing the location of the rear axle.

2. Click the icon **Hole**.

3. Click on the **ZX Map**.



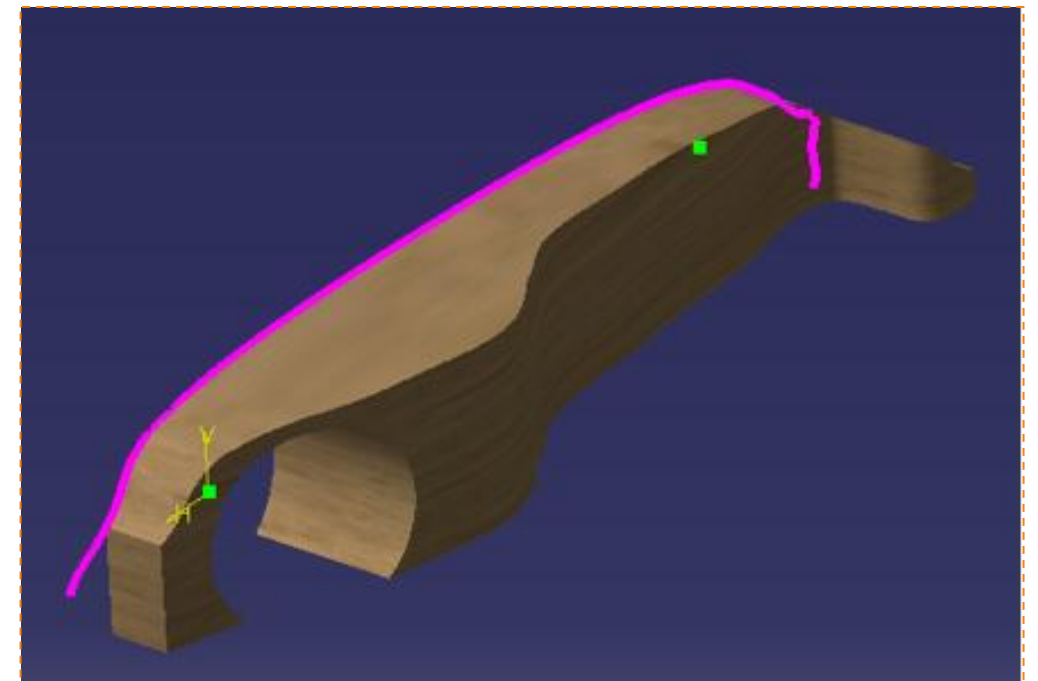
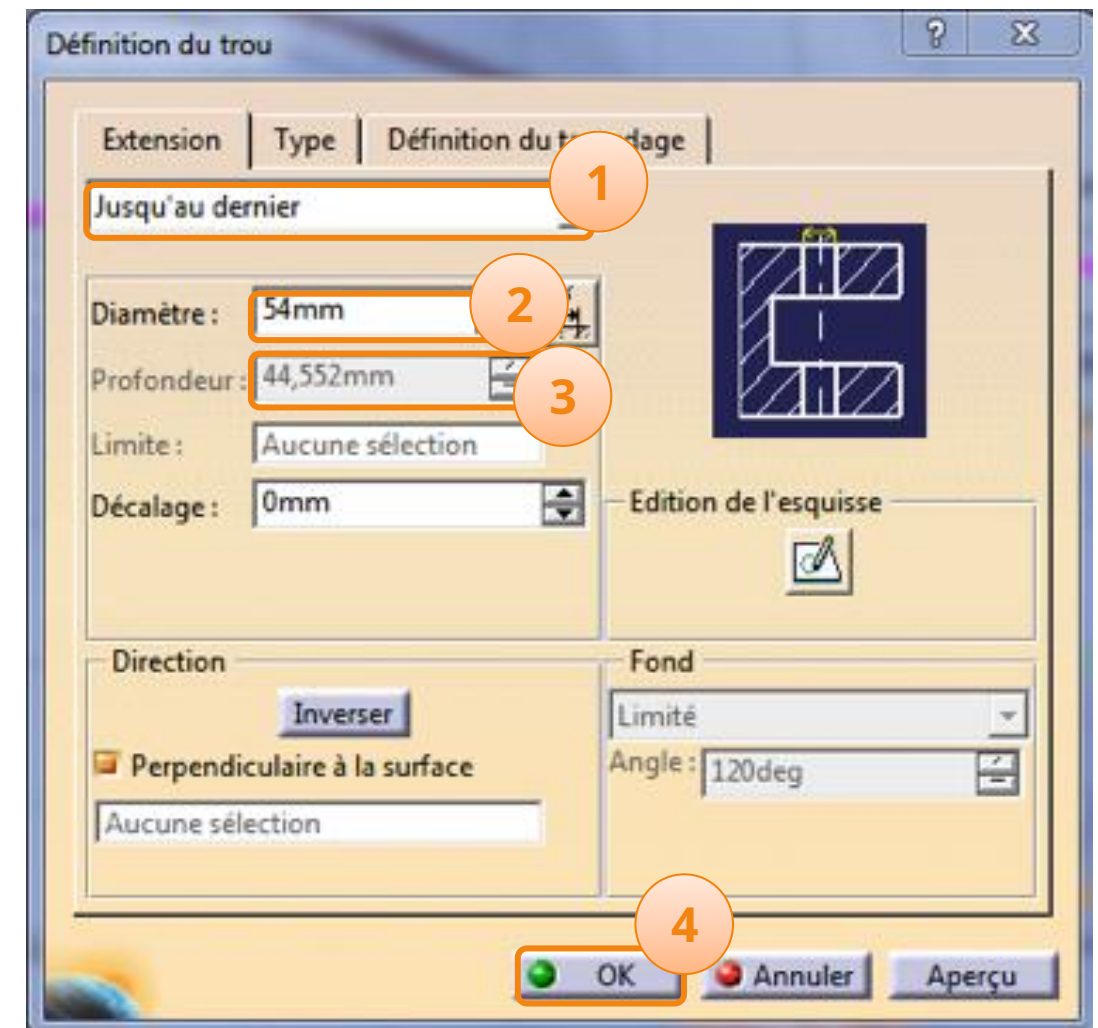
# Mechanical Design – Wheel arches

We will now define the properties of the hole:

1. Select **Until the last one**.
2. Enter [54 mm] for the diameter.
3. Verify that the dimension for the depth is grayed out.
4. Validate by clicking on **okay**.
5. Rename the hole [RearWheelHousing].



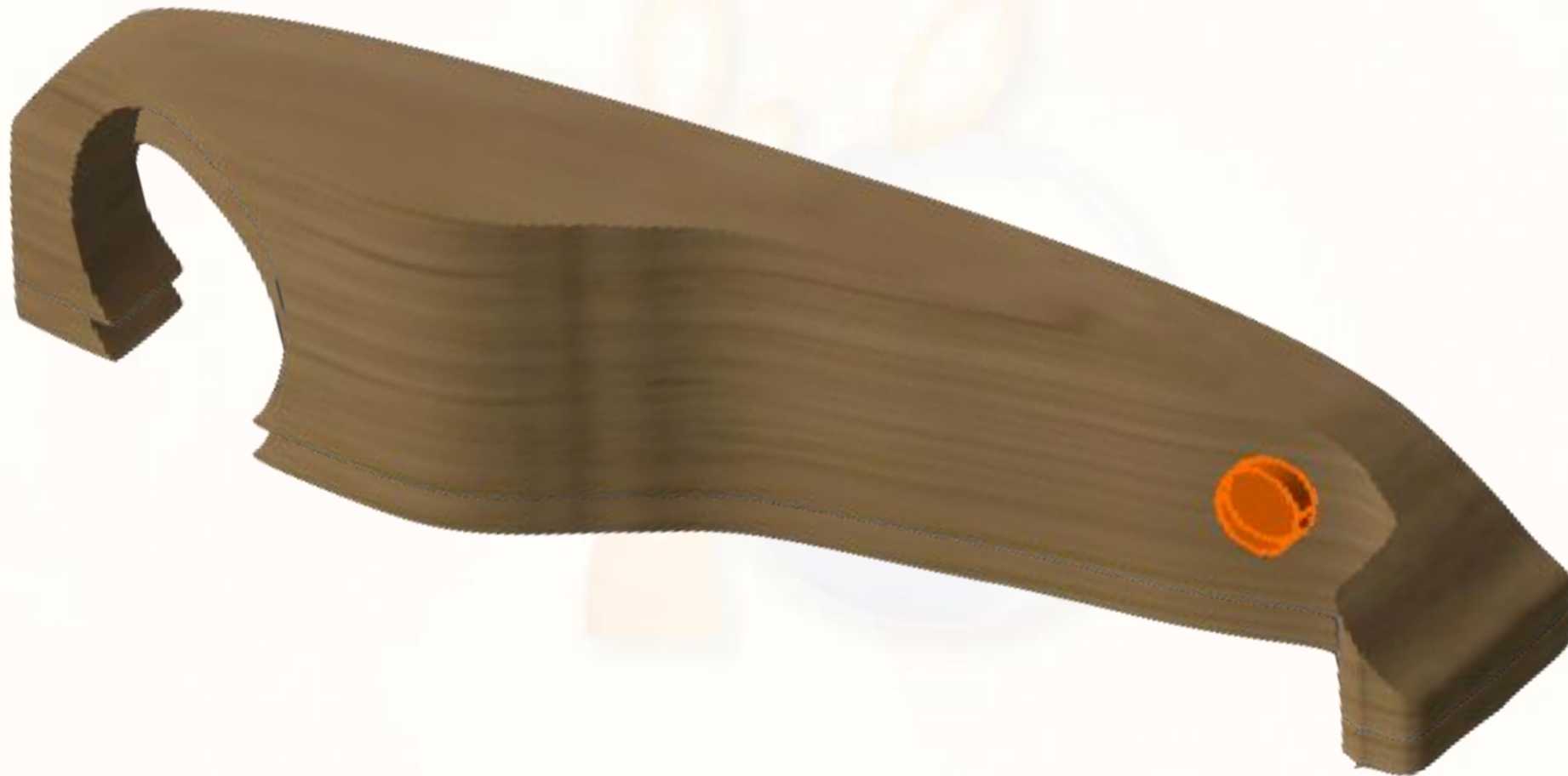
For more information on editing of the properties of an element, consult the method sheet "Modify the element properties."





# Mechanical Design – Shaft Support

We are going to make the support for the front axle.



15 minutes



# Mechanical Design – Shaft Support

Let's start with the sketch of the axis support, for this:

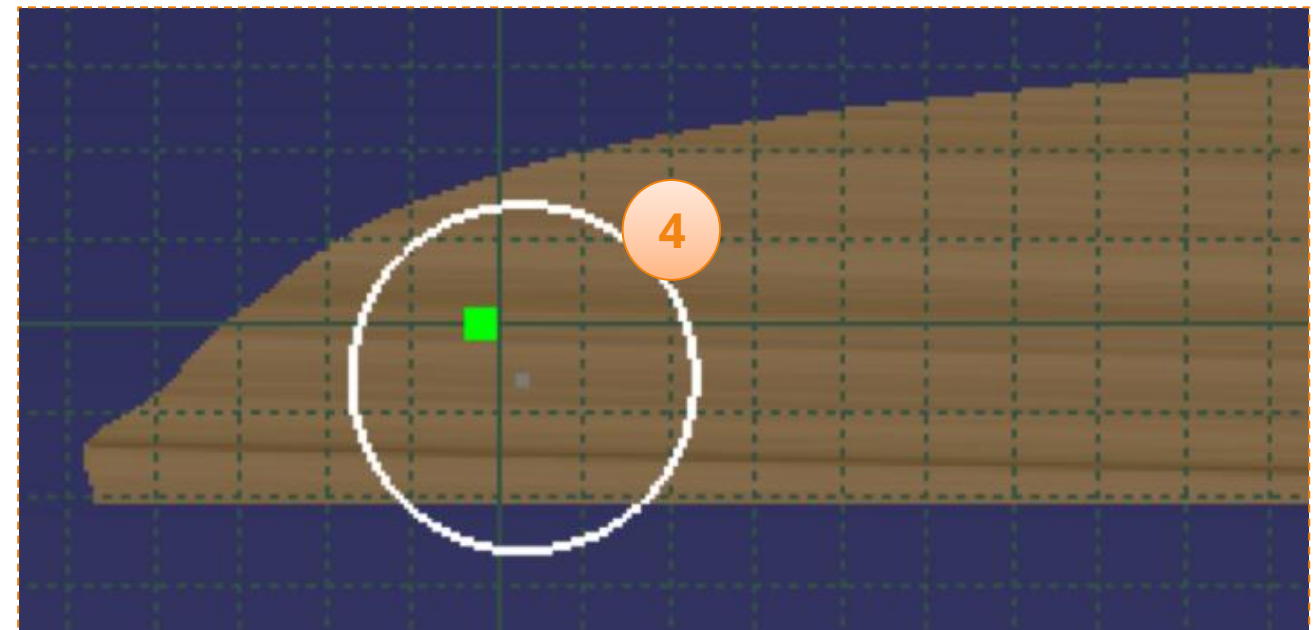
1. Click the icon **Sketch**.

2. Select the **ZX Map**.

3. Click the icon **Circle**.

4. Click once to place the approximate center of the circle.

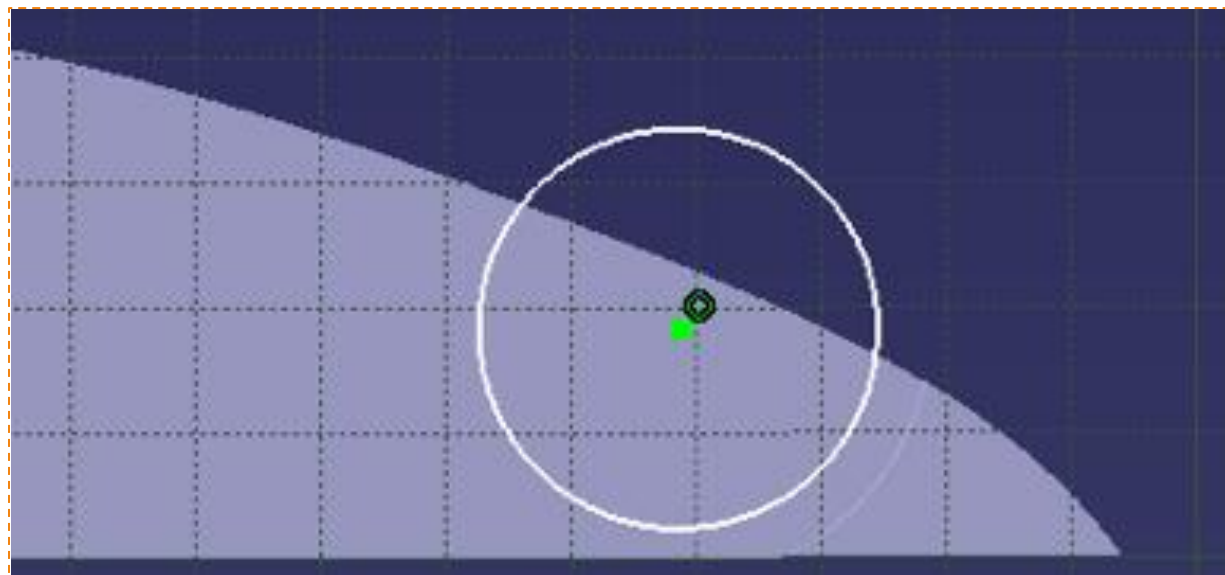
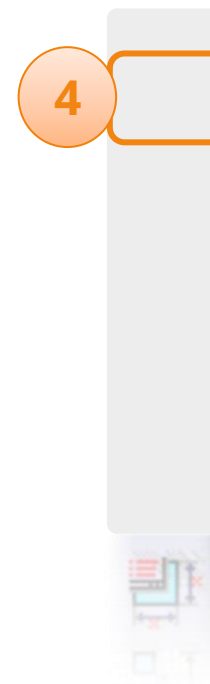
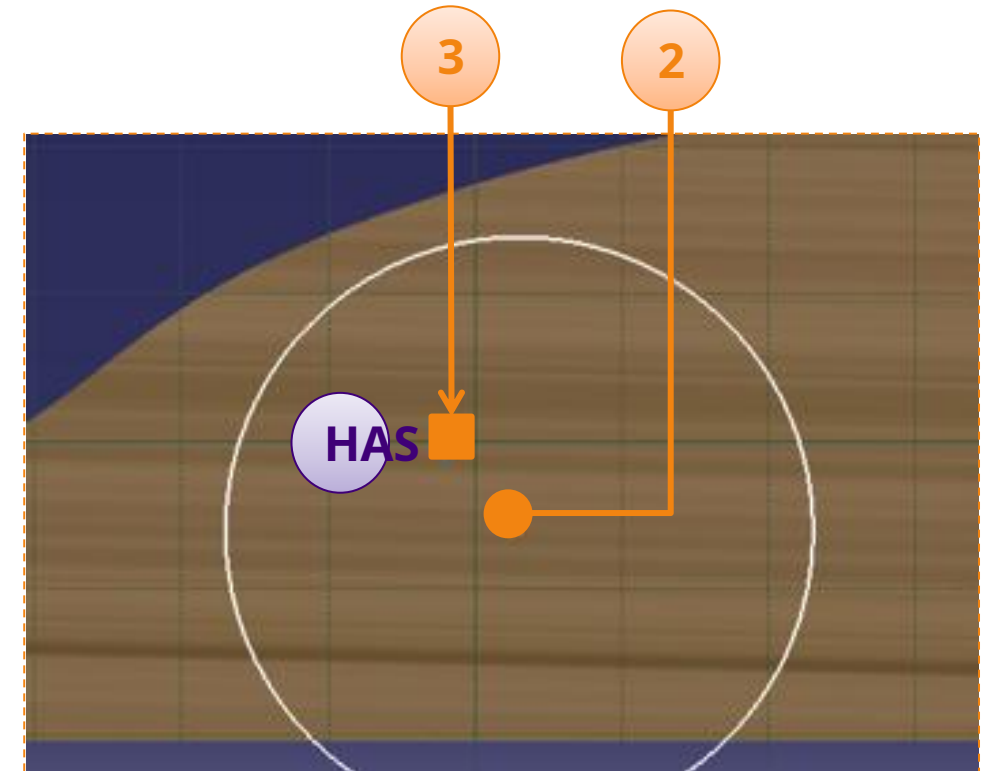
5. Click a second time to set the radius of the circle.



# Mechanical Design – Shaft Support

We are going to make the center of the circle coincide with the point **HAS**. For that :

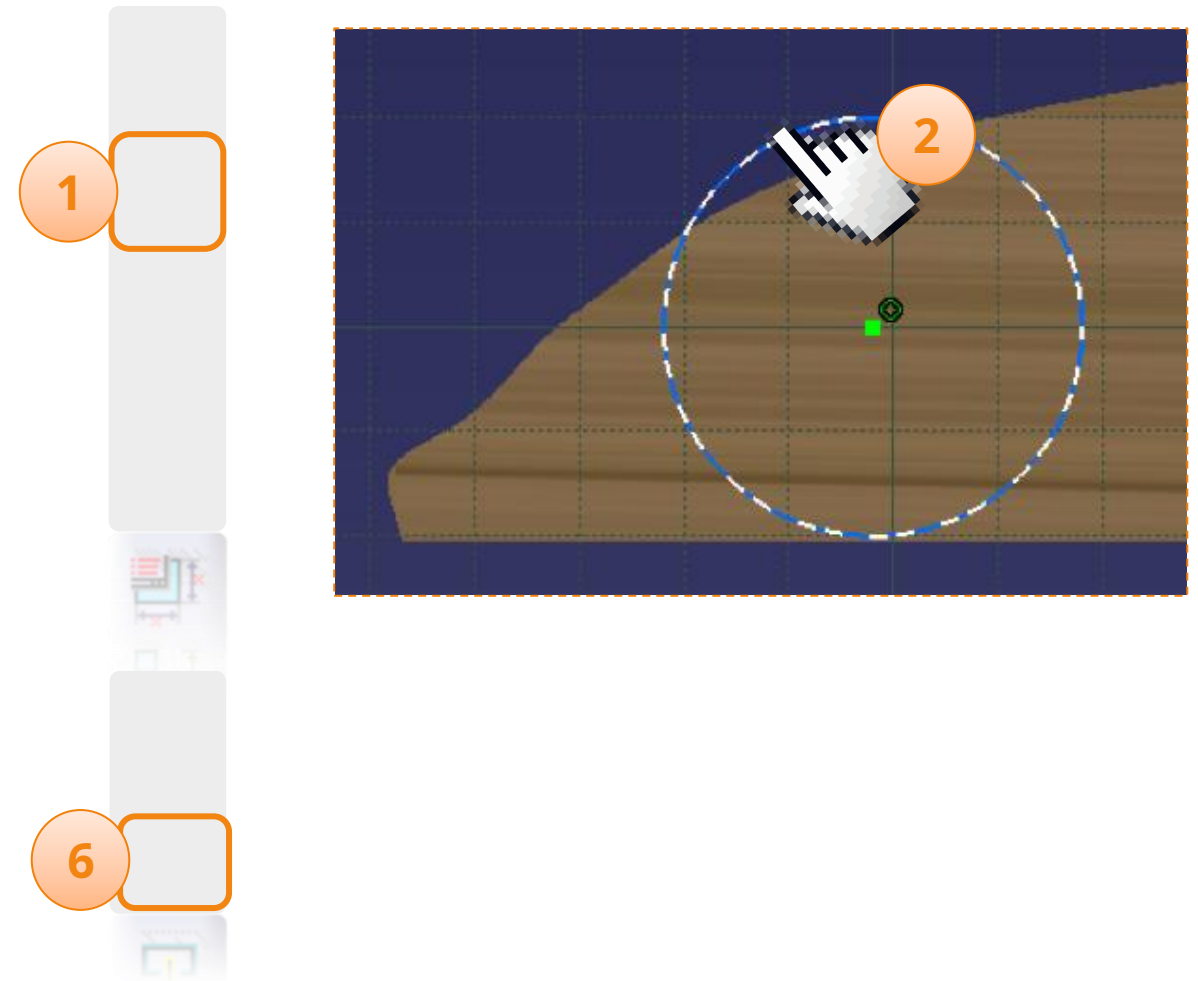
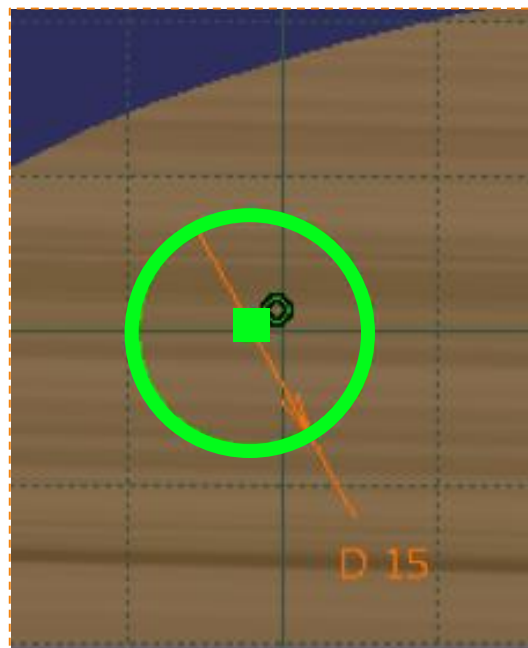
1. Hold down **<CTRL>** depressed.
2. Select the center of the circle.
3. Select stitch **HAS**. This will select both items at the same time.
4. Click the icon **Constraints** chosen in a dialog box.
5. Activate the constraint **Coincidence**.
6. Click on **okay** to get the result below.



# Mechanical Design – Shaft Support

To size the circle:

1. Click the icon **Constraint**.
2. Click on **The circle**.
3. Move the mouse then click to place the dimension.
4. Double click on the value to edit it.
5. Enter a diameter of [15mm].
6. Exit the sketch.
7. Rename the sketch [FrontAxleSupportSketch].

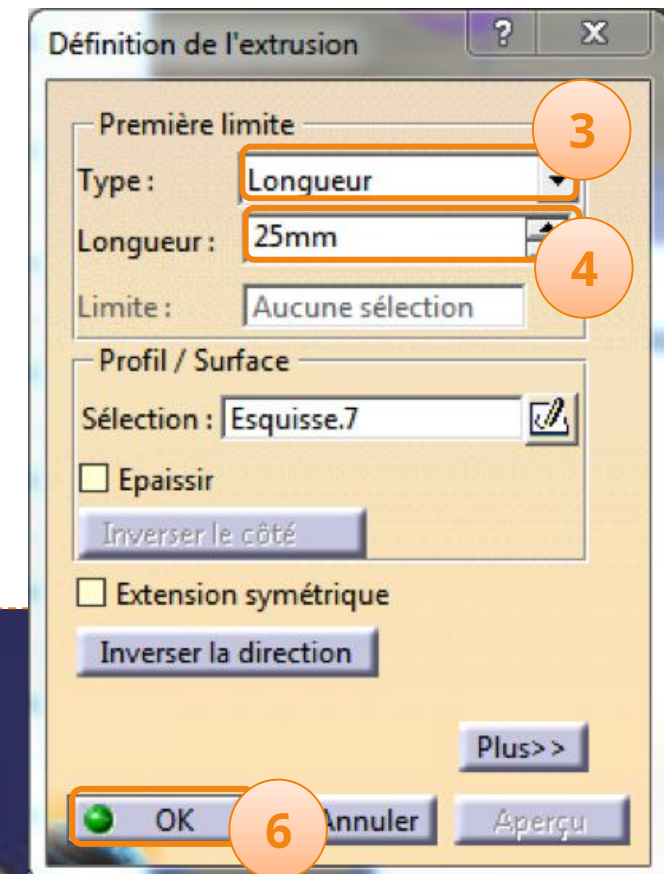


For more information on creating constraints, see the method sheet corresponding.

# Mechanical Design – Shaft Support

We are going to extrude the front axle support:

1. Click the icon **Extrusion**.
2. Click on **the sketch of the circle** only if it is not already selected.
3. Select Type **Length**.
4. Enter the length [3mm].
5. Choose the opposite direction using the button **Reverse direction**.
6. Validate by clicking on **okay**.
7. Rename the extrusion [FrontAxleSupport].





## Mechanical Design – Drilling

We are going to make the hole for the front axle.



30 minutes

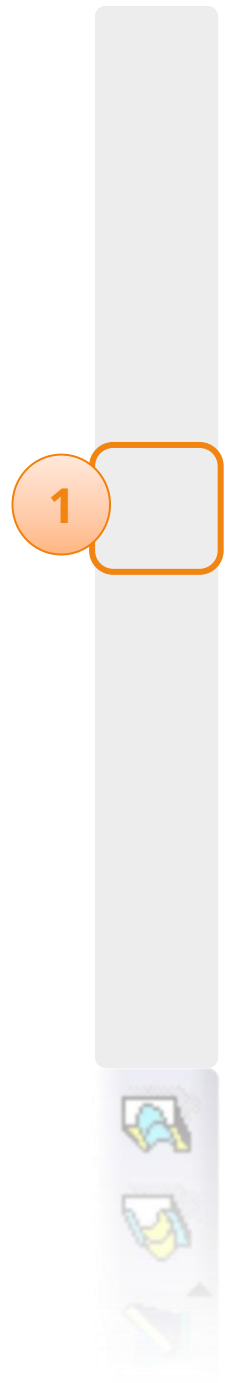
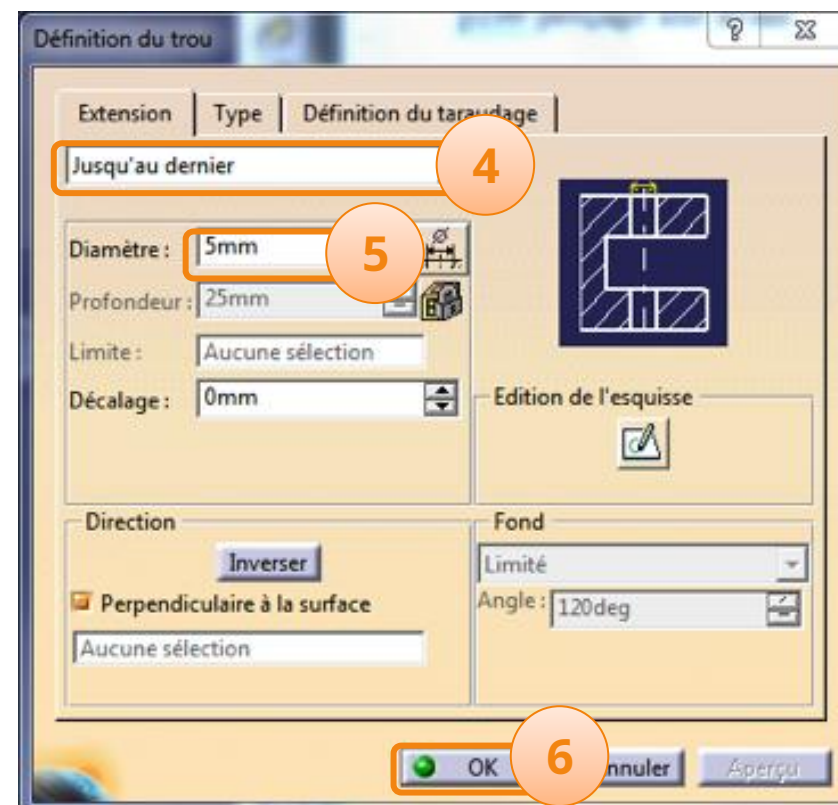
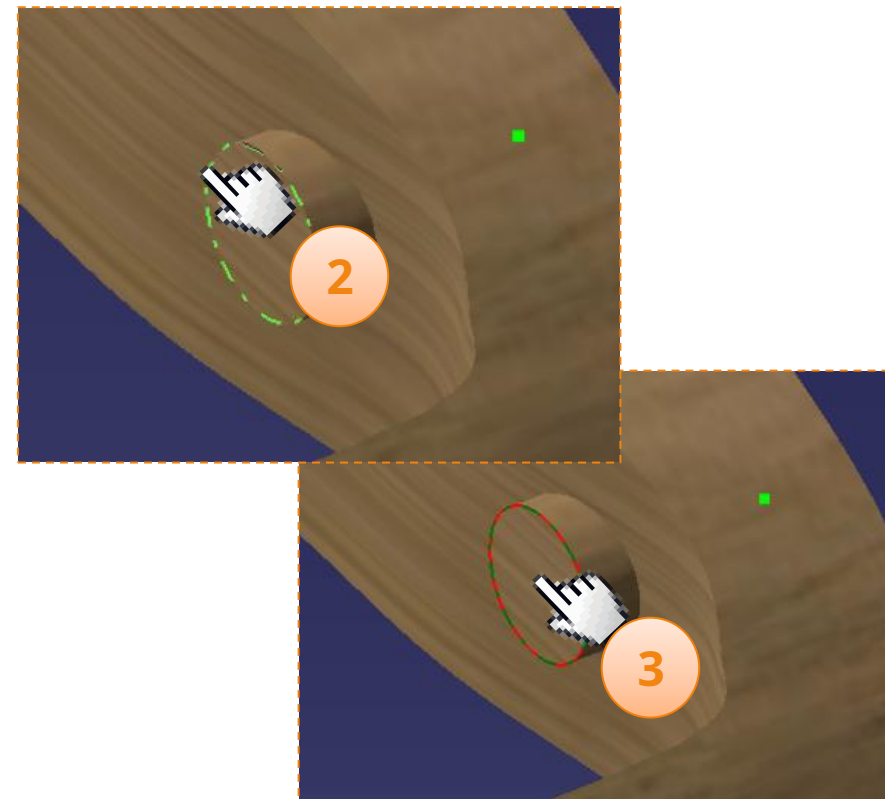
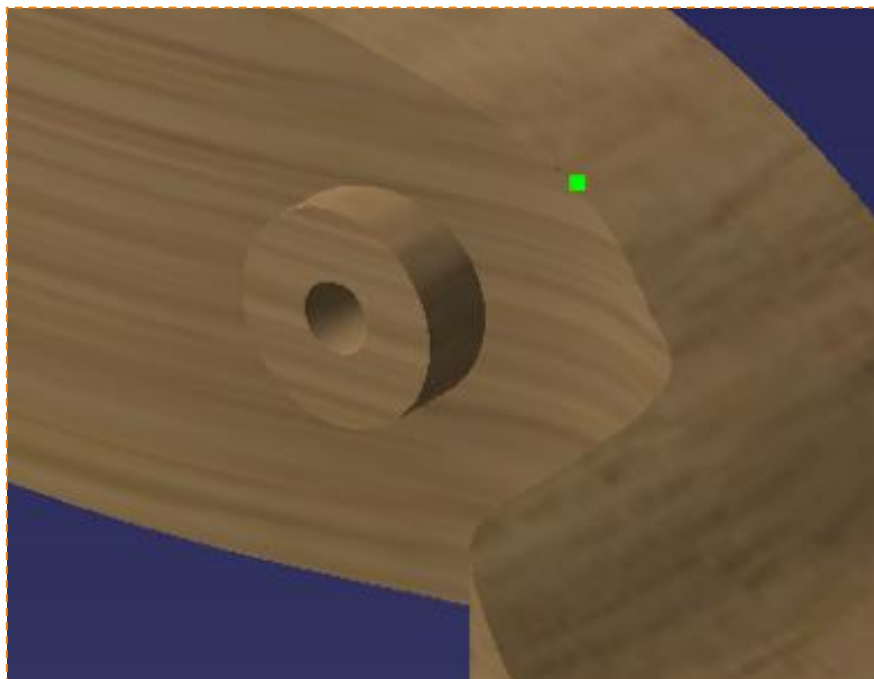


# Mechanical Design – Drilling

1. Click the icon **Hole**.
2. Click on **the ridge** of FrontAxleSupport as opposite.
3. Click on **the face** from FrontAxleSupport. This creates a hole whose center coincides with the center of the circle clicked in step 2.

In the window that appears enter the following parameters:

4. Select **Until the last one**.
5. Enter the diameter [6 mm].
6. Click on **okay**.
7. Rename the [FrontAxleHole].

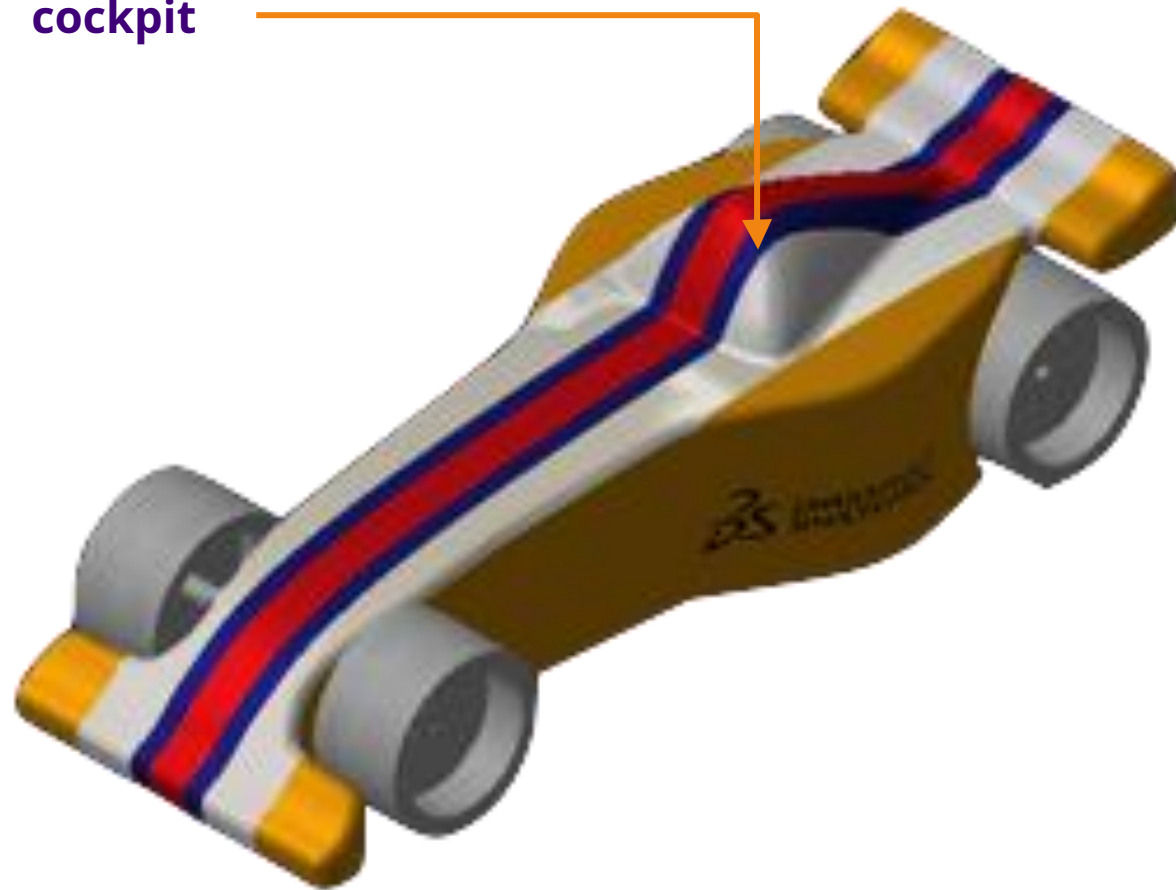


# Model the cockpit

We will create the cockpit using:

- ▶ The Imagine & Shape workshop.
- ▶ A subdivision surface.

cockpit

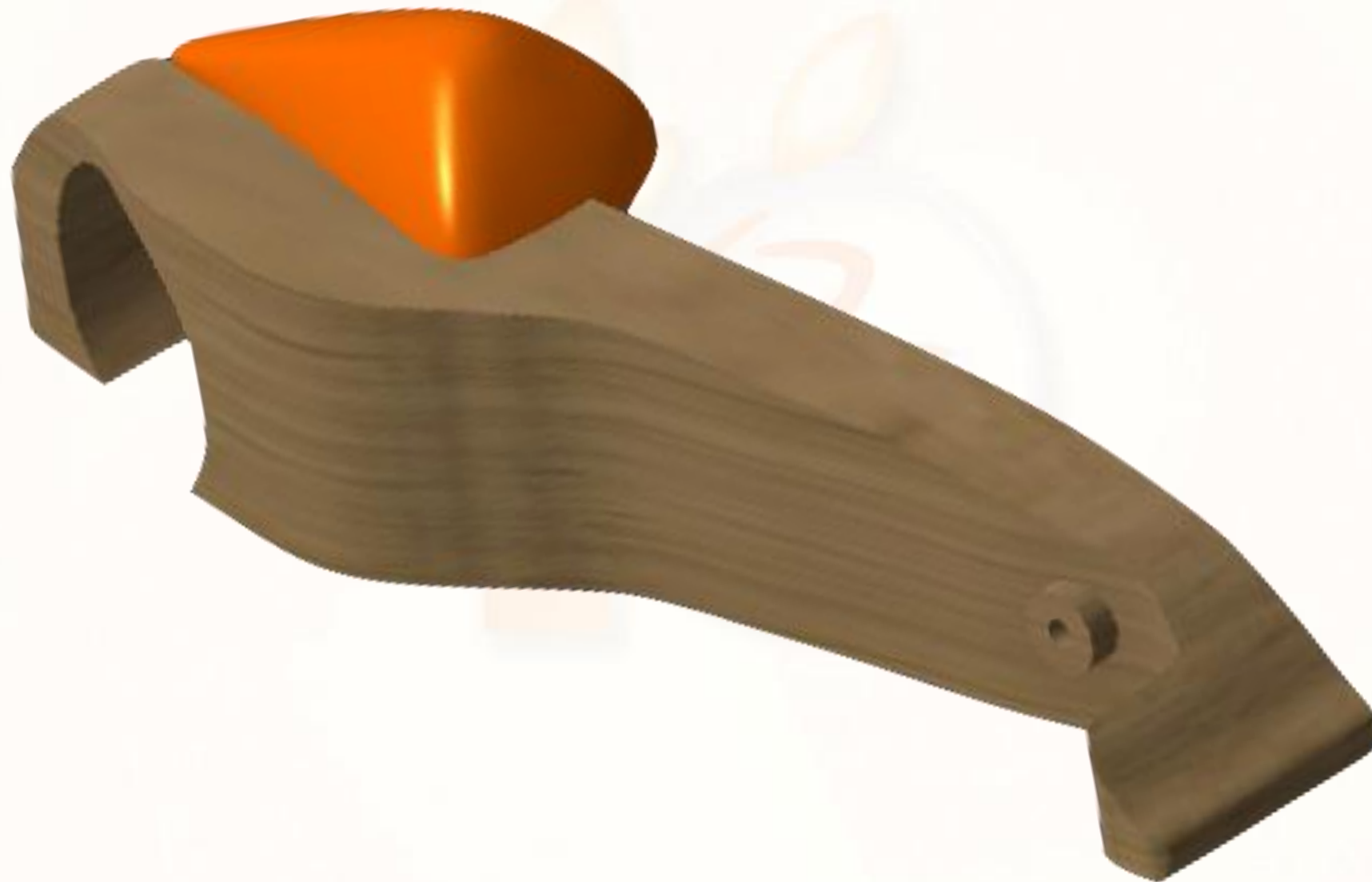


Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.**
    - i. Introduction.
    - ii. Settings.
    - iii. Model the body.
    - iv. Model the cockpit.**
    - v. Model the rear wing.
    - vi. Finalization.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

## Model the cockpit

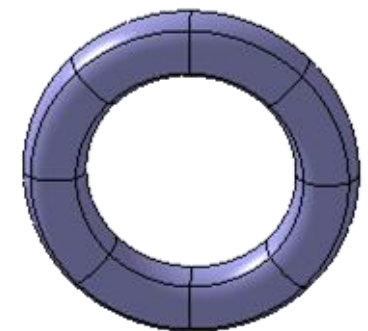
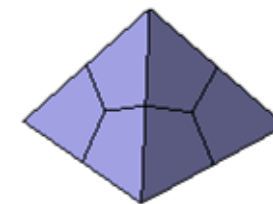
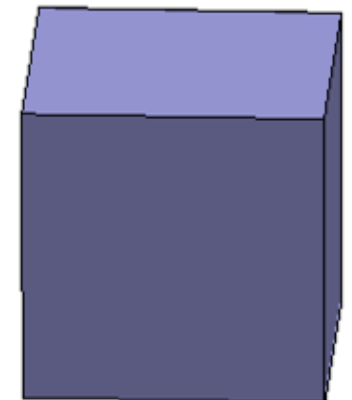
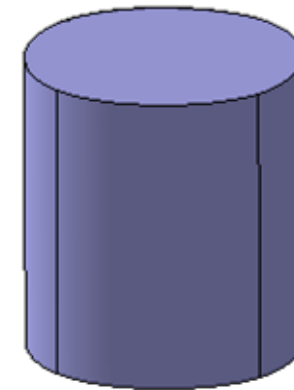
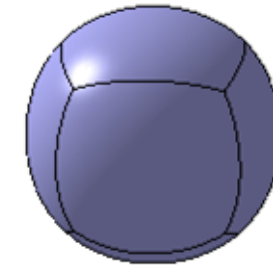
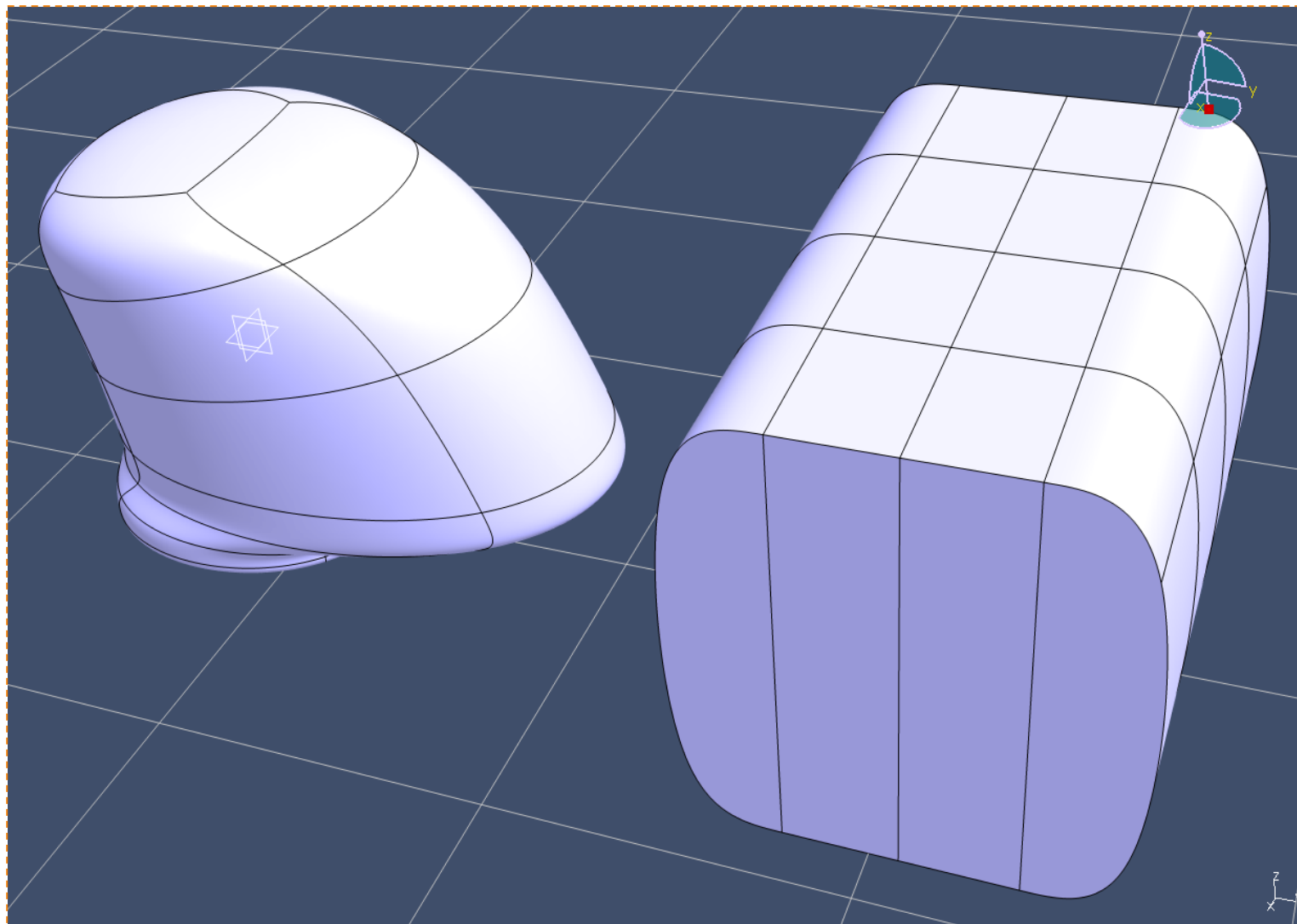
We will create the cockpit using the Imagine & Shape workbench which contains very powerful and easy to use tools.



**30 minutes**

## Presentation Imagine & Shape

We used the Imagine & Shape workbench to draw curves. We will now use the same workbench to create form surfaces: subdivision surfaces.





# Model the cockpit

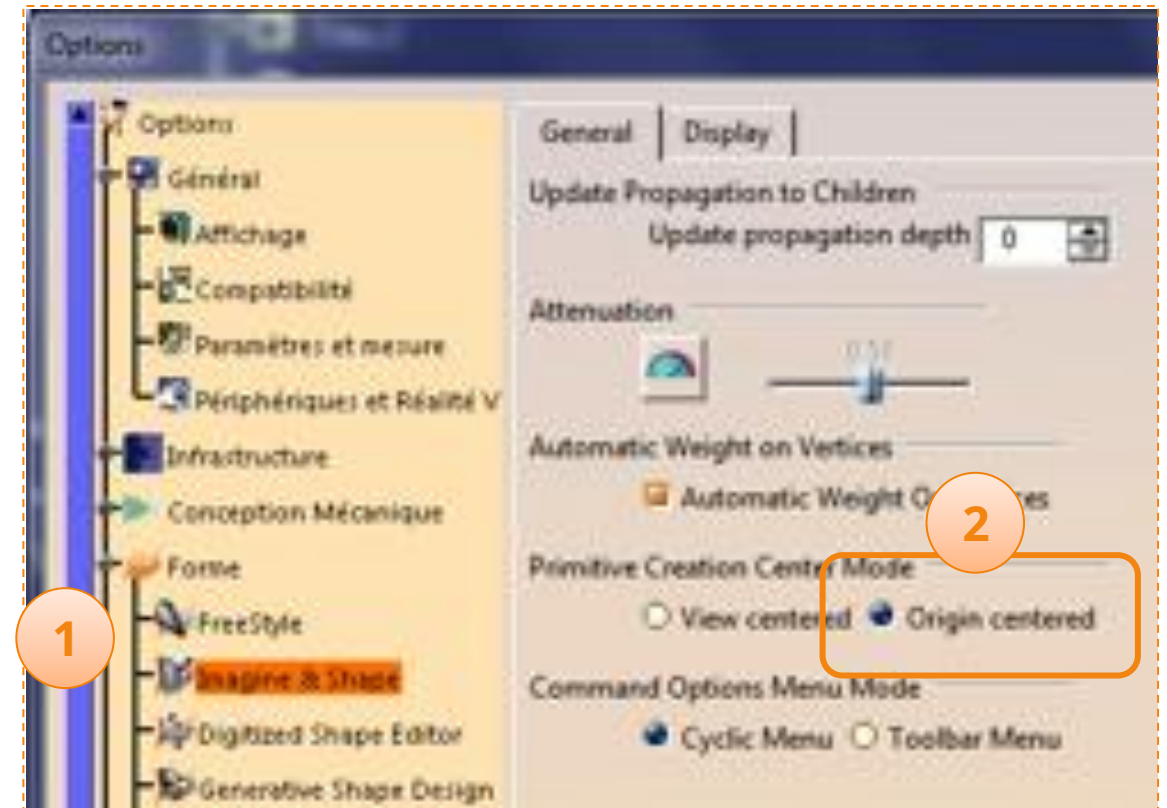
Before creating the cockpit we will check the following settings:

1. Check the setting opposite in: **Tools > Options > Shape > Imagine & Shape.**



2. Check the option **Origin-centered** to **Primitive Creation Center Fashion**. This option guarantees a centered positioning with respect to the origin.

3. Click on **okay** to apply these settings.

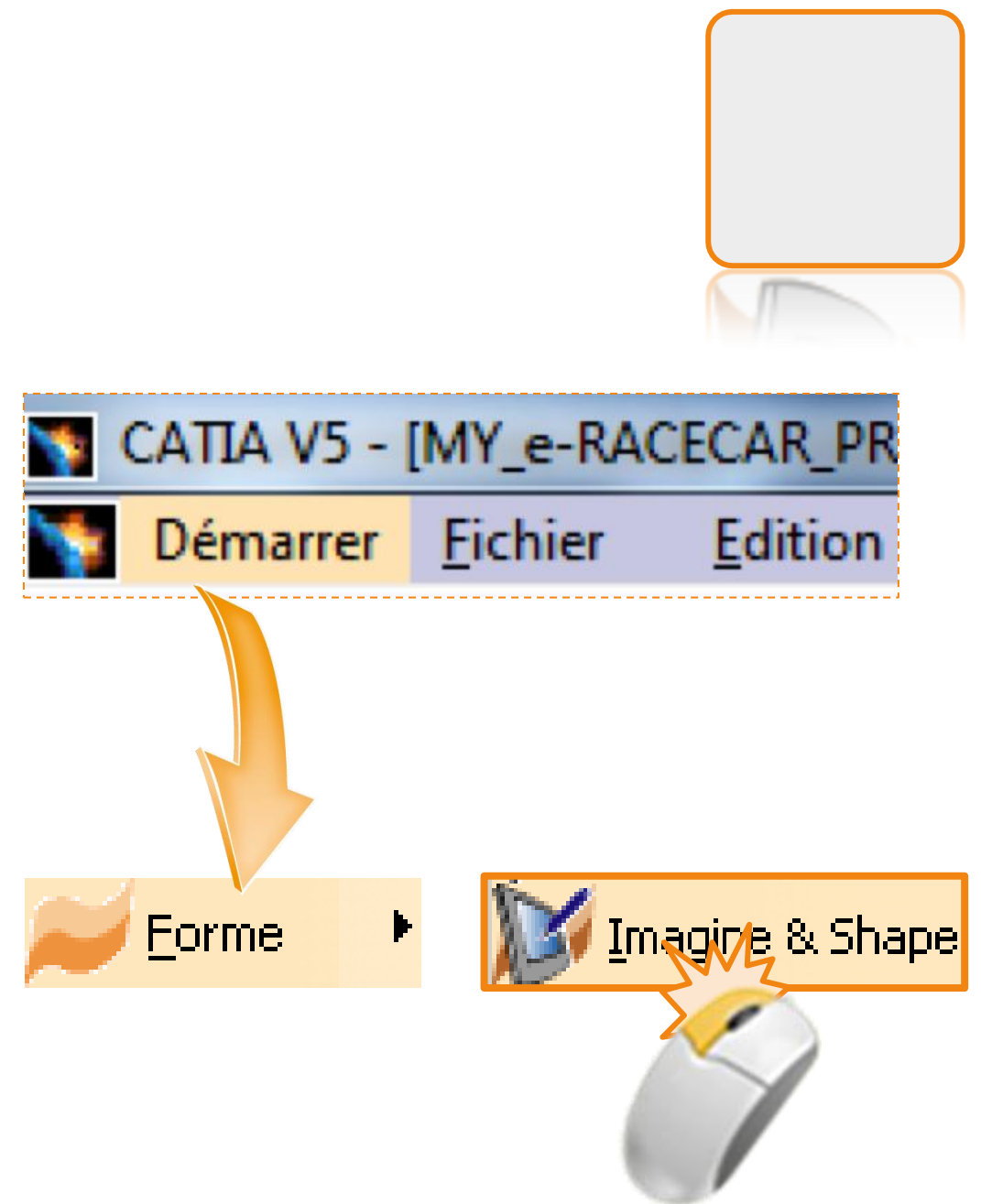




# Model the cockpit

We are going to switch to the Imagine & Shape workbench. To change workshop click on:

**1.Start > Shape > Imagine & Shape.**



# Model the cockpit

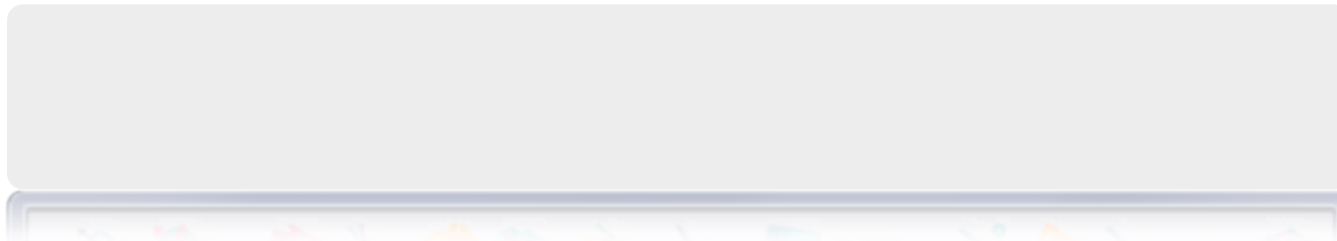
To create the cockpit:

1. Display the **Profile view** by right-clicking on **Side View**, then **Hide/Show**.

1. Click on **Top view**.

2. Click the icon **Sphere**.

You then get the following window. This includes all the tools needed to refine your cockpit and make it faithful to your design.



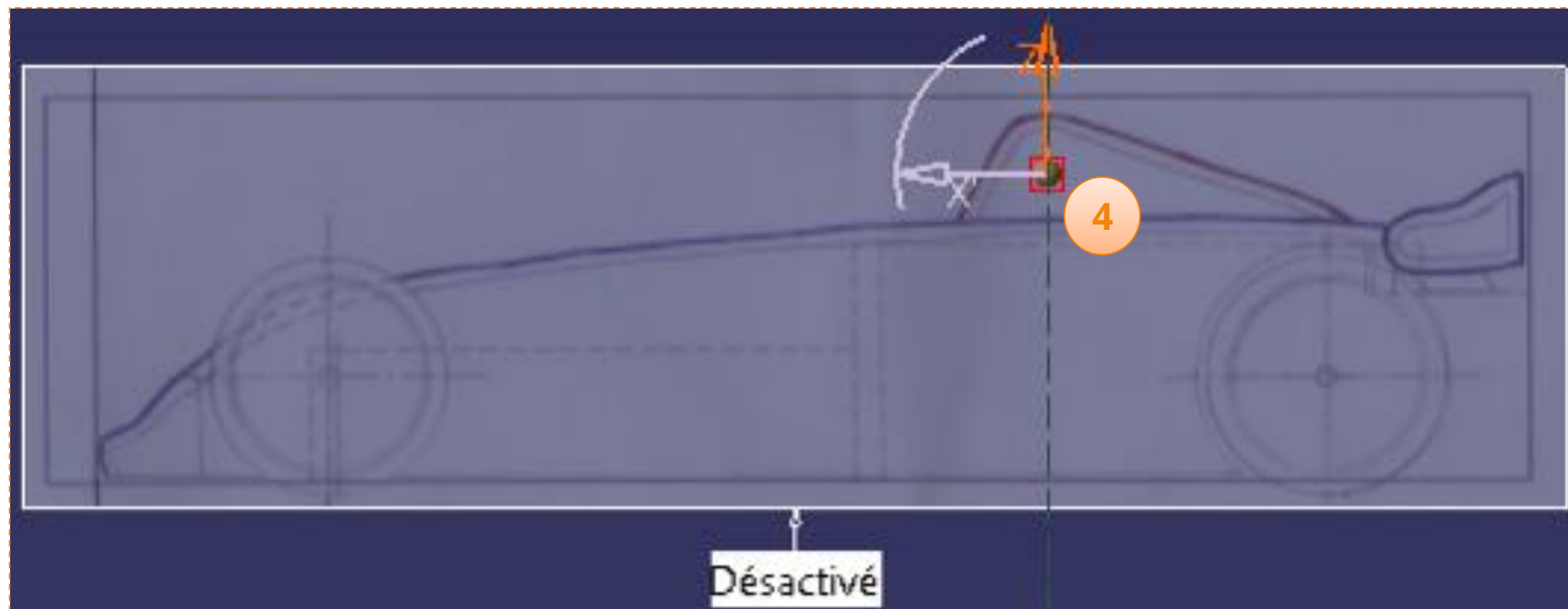
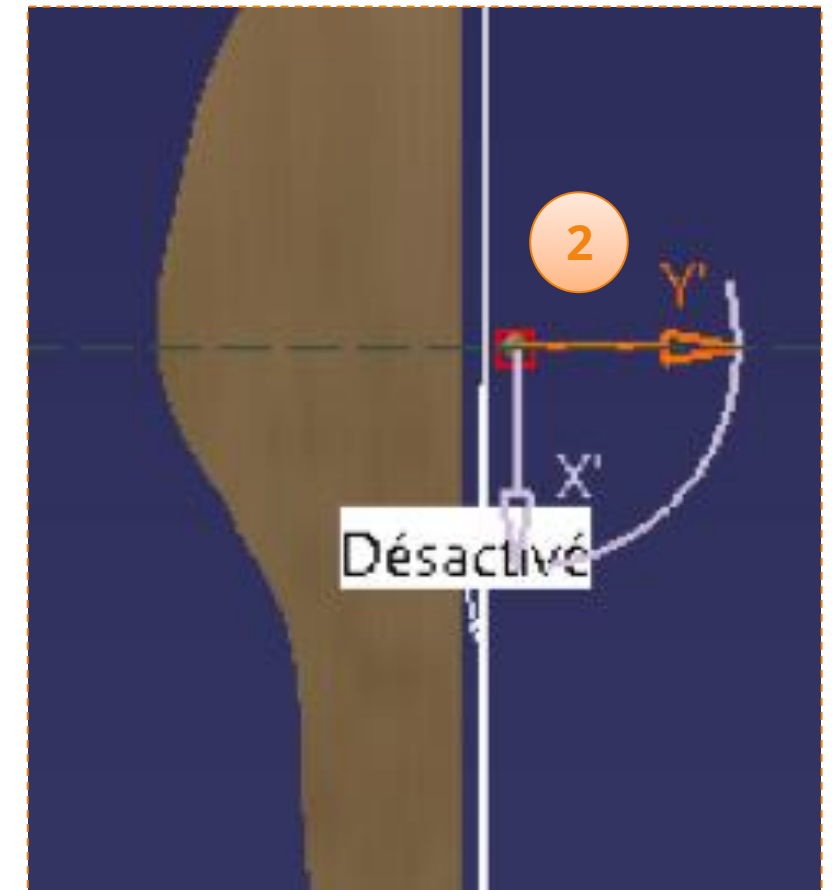
You also obtain the display opposite for the sphere created.



# Model the cockpit

We will now place this ball in the right place. For that :

1. Always in **Top view**, click on **Translation**.
2. Place the ball very slightly in front of the drawing, the goal being to be able to work the ball as close as possible to the drawing.
3. Then go to **Right view**.
4. Then move the ball on the drawing of your cockpit as below.



# Model the cockpit

You will notice that the sphere does not fill the red space on the drawing. It is therefore necessary to enlarge it and give it the shape of the cockpit:

1. Click on **All Elements Selection**.

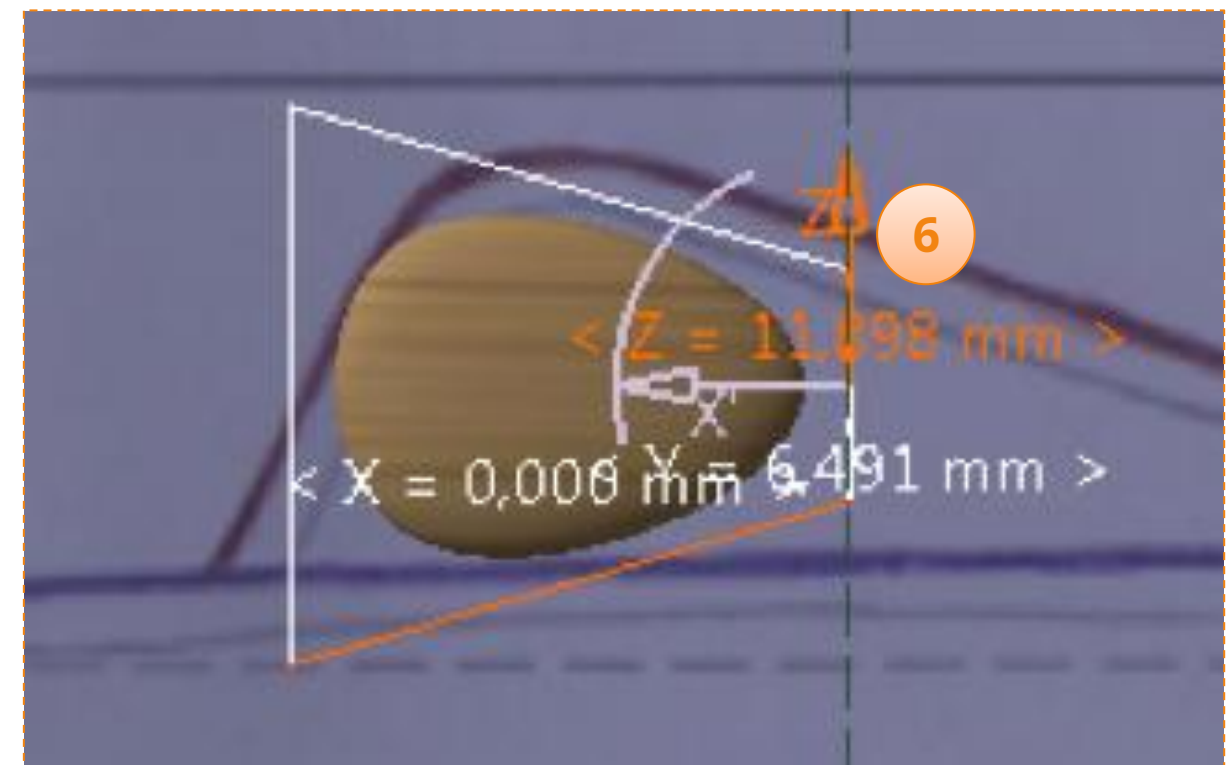
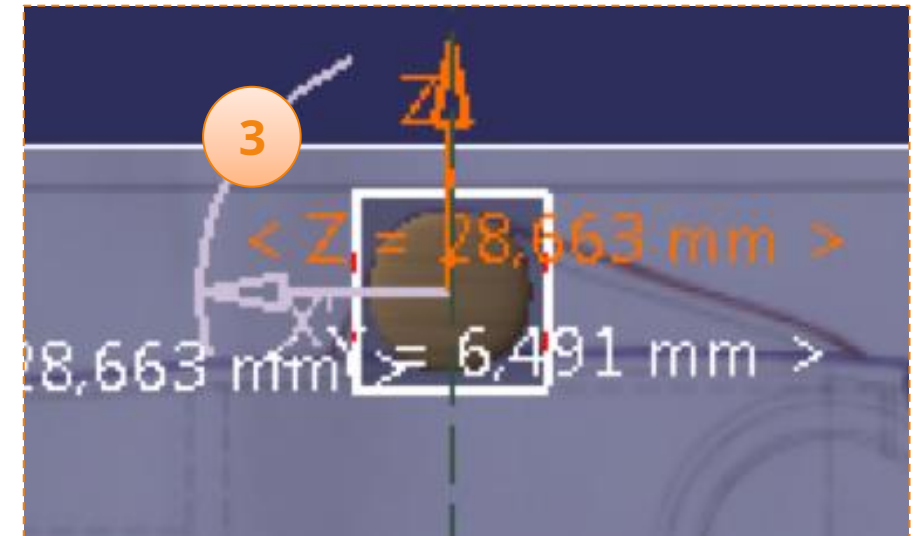
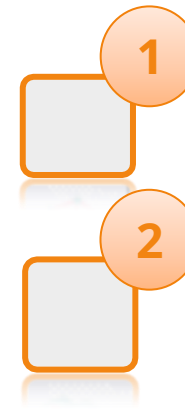
2. Click on **Affinity**.

3. A marker appears. Use it to enlarge the sphere.

4. Select the back side with **Face Selection**. **Affinity** still needs to be enabled.

5. Hold down **<CTRL>** depressed

6. Edit the shape using the guide.



# Model the cockpit

7. Hold **<CTRL>** depressed.

8. Then select the front and top faces.

9. Click on **Attraction**.

10. A cursor is displayed on the screen, allowing to modify the attraction of the volume towards the axes. Modify this slider until you get the desired result.

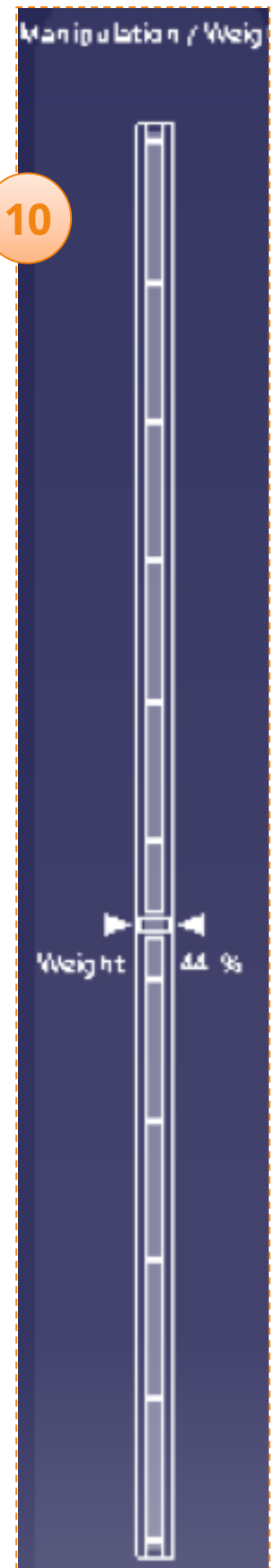
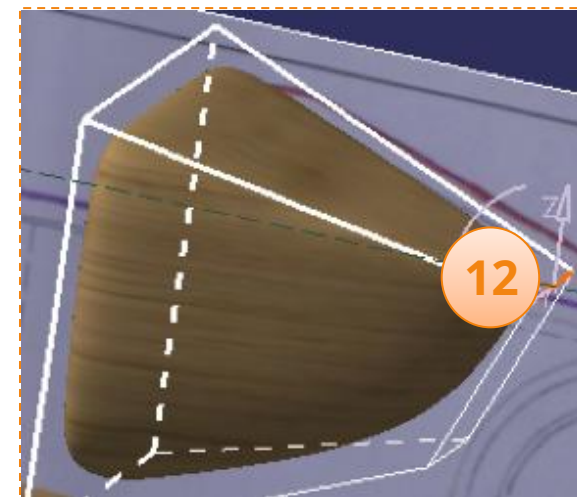
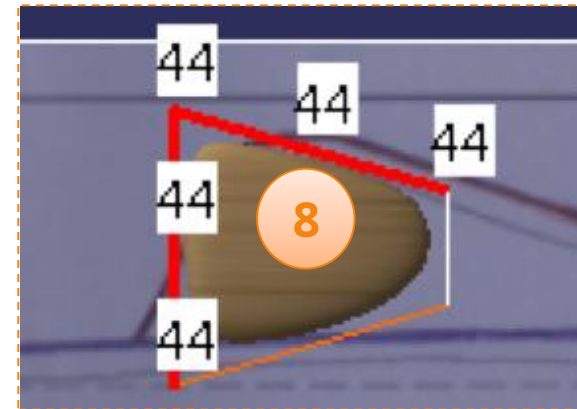
11. Click on the tool **Edge Selection**.

12. Select the top back edge.

13. Click on the tool **Translation**.

14. Use the mark to match the volume with the drawing.

15. Repeat the steps **10** at **13** for the two edges at the front. You should get the following result:





# Model the cockpit

We are now going to thicken this volume so that it also matches the shape of the cockpit seen from above:

1. Switch to **Front view**.



We then observe that the sphere is far from the extrusion and that the cockpit is too thin.

2. Click the icon **All Elements Selection**.



3. Click the icon **Translation**.



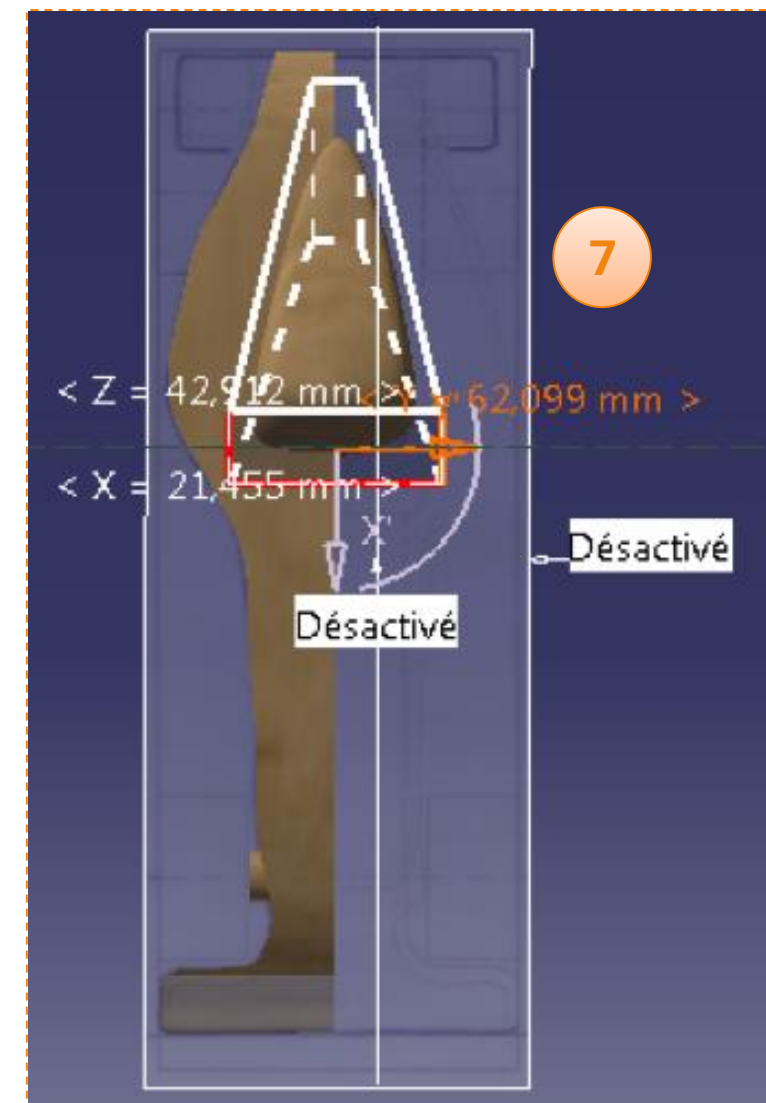
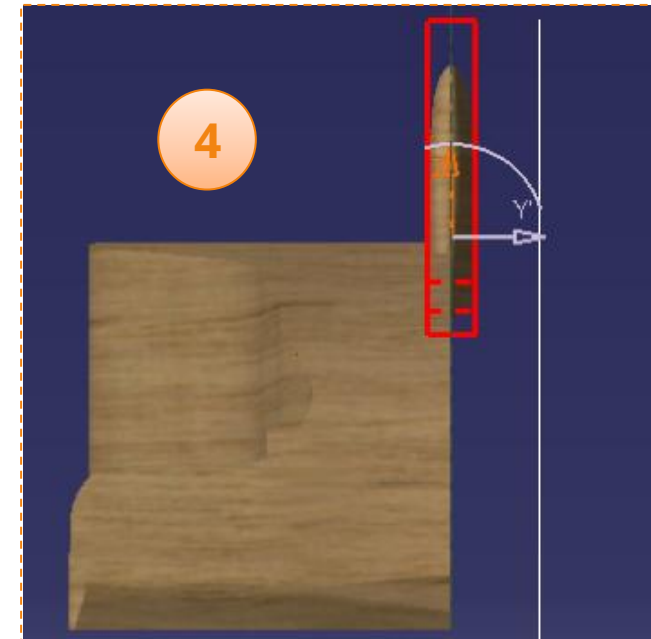
4. Move the cockpit as shown opposite.

5. Then hold **<CTRL>** and select the two vertical edges of the front of the cockpit.

6. Click on the tool **Affinity**.

7. Finally, use the guideline to widen the cockpit to match the red area of the drawing.  
Display the table if necessary **Top View** by right-clicking on it and then **Hide/Show**.

8. Play with the affinity of faces for aerodynamics.



# Model the cockpit

The surface we have created is an envelope, the interior contains no material. So let's fill it in:

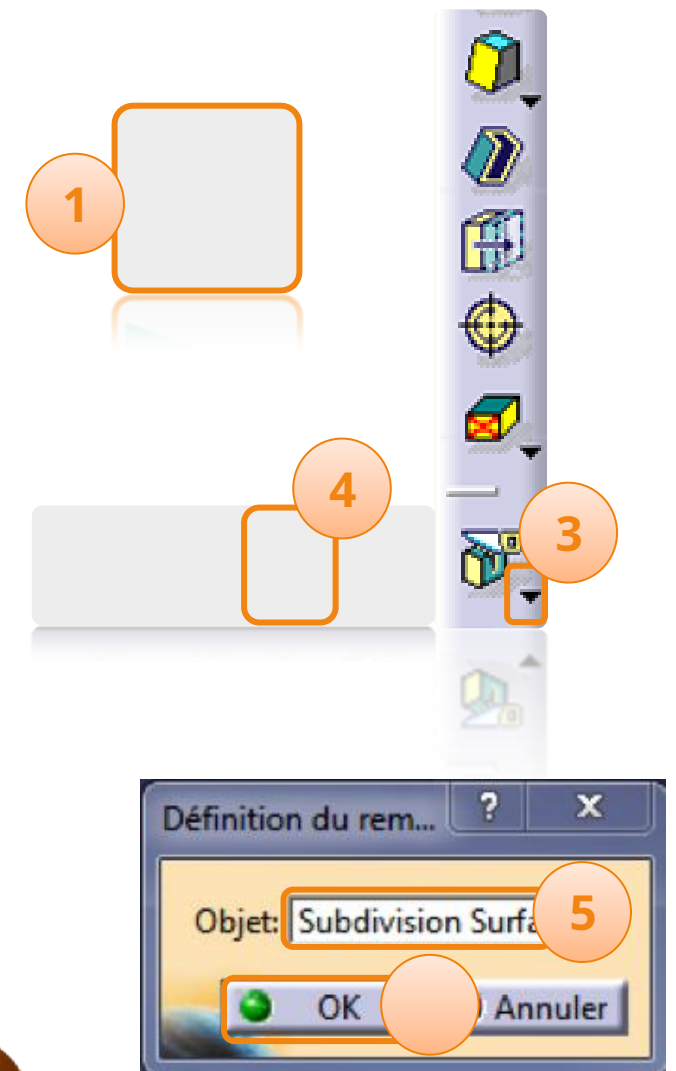
1. Return to the workshop **Part Design**. (**Start > Mechanical Design > Part Design**)
2. Rename Subdivision Surface.1 to [Subdivision Surface]. (Right-click then **Properties...**)
3. Click on the **black arrow** icon **Chopped off**. This will bring up other tools.
4. Click the icon **Filling**.
5. In the called window **Infill Definition**, check that the object is **Subdivision Surface**.
6. Click on **okay**.
7. Rename Fill.1 to [Cockpit]. (Right-click then **Properties...**)

You must obtain as opposite an overlap of the surface of subdivision, and the material created by the filling.

8. Hide Subdivision Surface. (Right-click then **Hide/Show**)



For more information about: changing the properties of an element. Consult the method sheet.



## Model the cockpit

Congratulations you have made the Cockpit of the car!



# Model the rear wing

In this part we will model the rear wing of your car.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. *Your racing car.***
  - has. Draw your car.
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  - vs. Create an immersive sketch.
  - d. *Model your car.***
    - i. Introduction.
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    - iii. Model the body.
    - iv. Model the cockpit.
    - v. *Model the rear wing.***
    - vi. Finalization.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

## Model the rear wing

We will model the rear wing using two curves that we will draw by hand in Imagine & Shape.

The junction of the upper and lower surfaces of the aileron will be ensured by a fillet.



45 minutes



# Model the rear wing

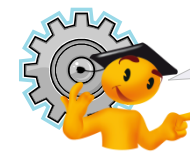
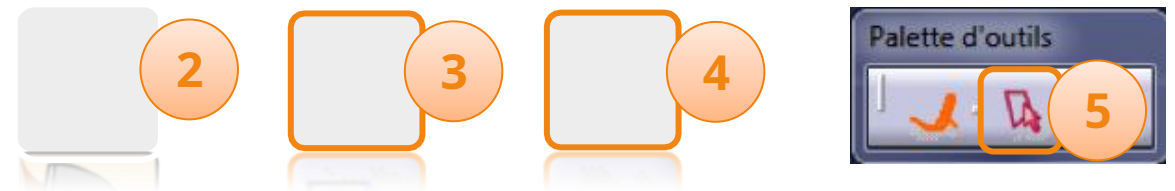
We are going to draw the first curve, which is the extrados curve of the aileron:

1. First go to **Car\_Body** by double-clicking on it.
2. Enter the workshop **Imagine & Shape**.
3. Click the icon **Right view**.
4. Click the icon **Sketch Curve**.
5. Click the icon **Plane Selection** in the tool palette.

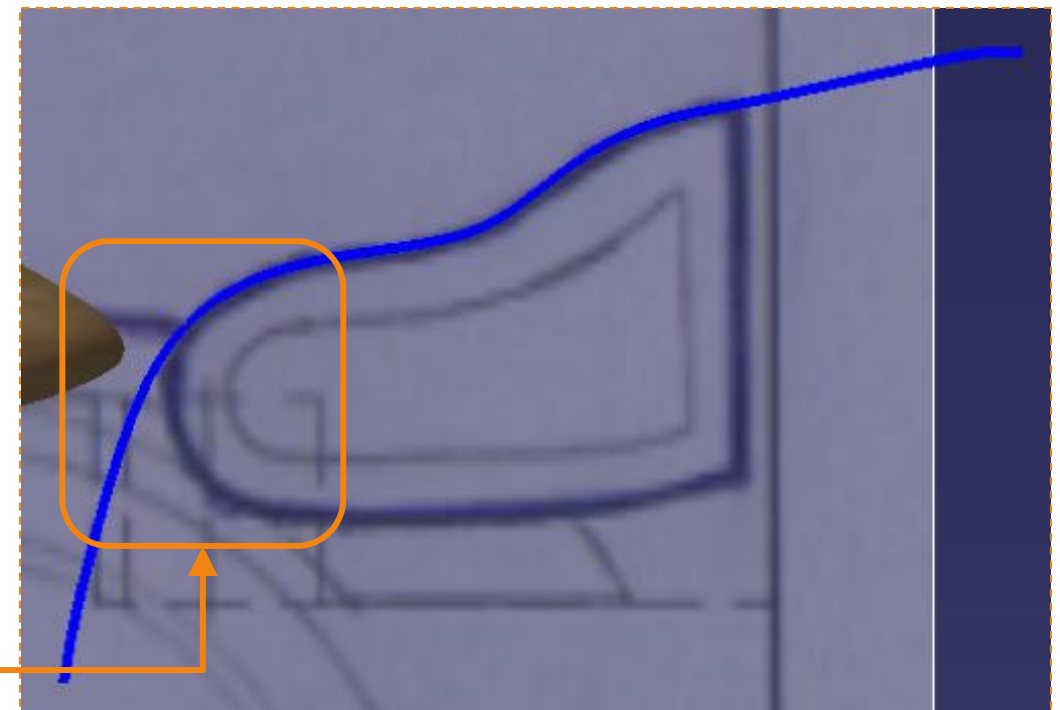
6. Click on the **ZX Map**.
7. Trace the curve of the top of the rear wing (in blue opposite).

You can use the manipulation tools we saw earlier to refine the drawing of your curve.

8. Rename the curve [ExtradosCurve] and change its color to blue and its thickness to 5.



Characteristic curves in  
Aero Dynamics

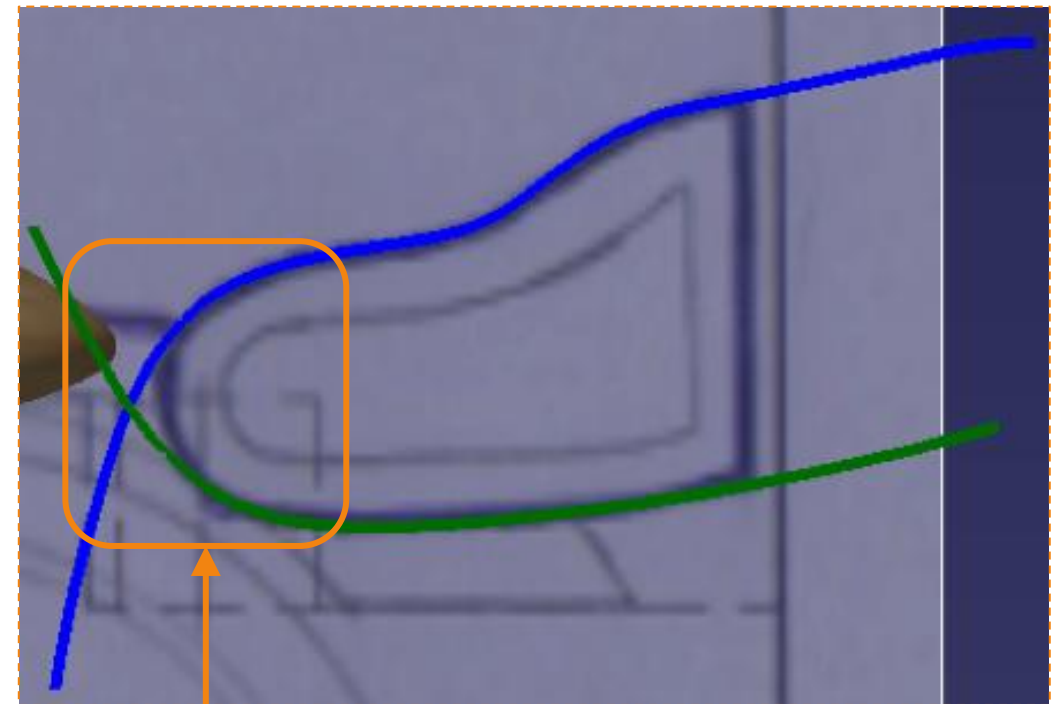
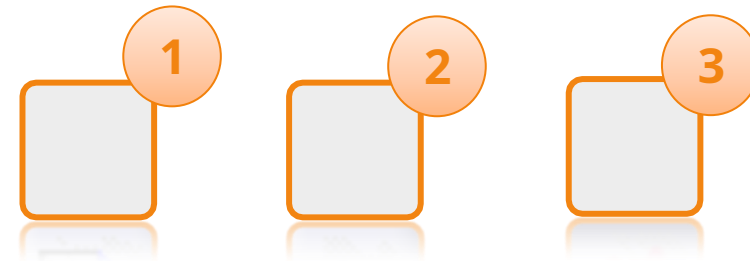


Deliberately leave the leading edge, so that the junction between the two curves will be done later with a more precise way.

# Model the rear wing

Let's draw the second curve, the underside curve of the fin:

1. Click the icon **Right view**.
  2. Click the icon **Sketch Curve**.
  3. Click the icon **Plane Selection** in the tool palette.
  4. Click on the **ZX Map**.
  5. Trace the curve of the underside of the rear wing (in green opposite).
- You can use the manipulation tools we saw earlier to refine the drawing of your curve.
6. Rename the curve [ExtradosCurve] and change its color to green and its thickness to 5.

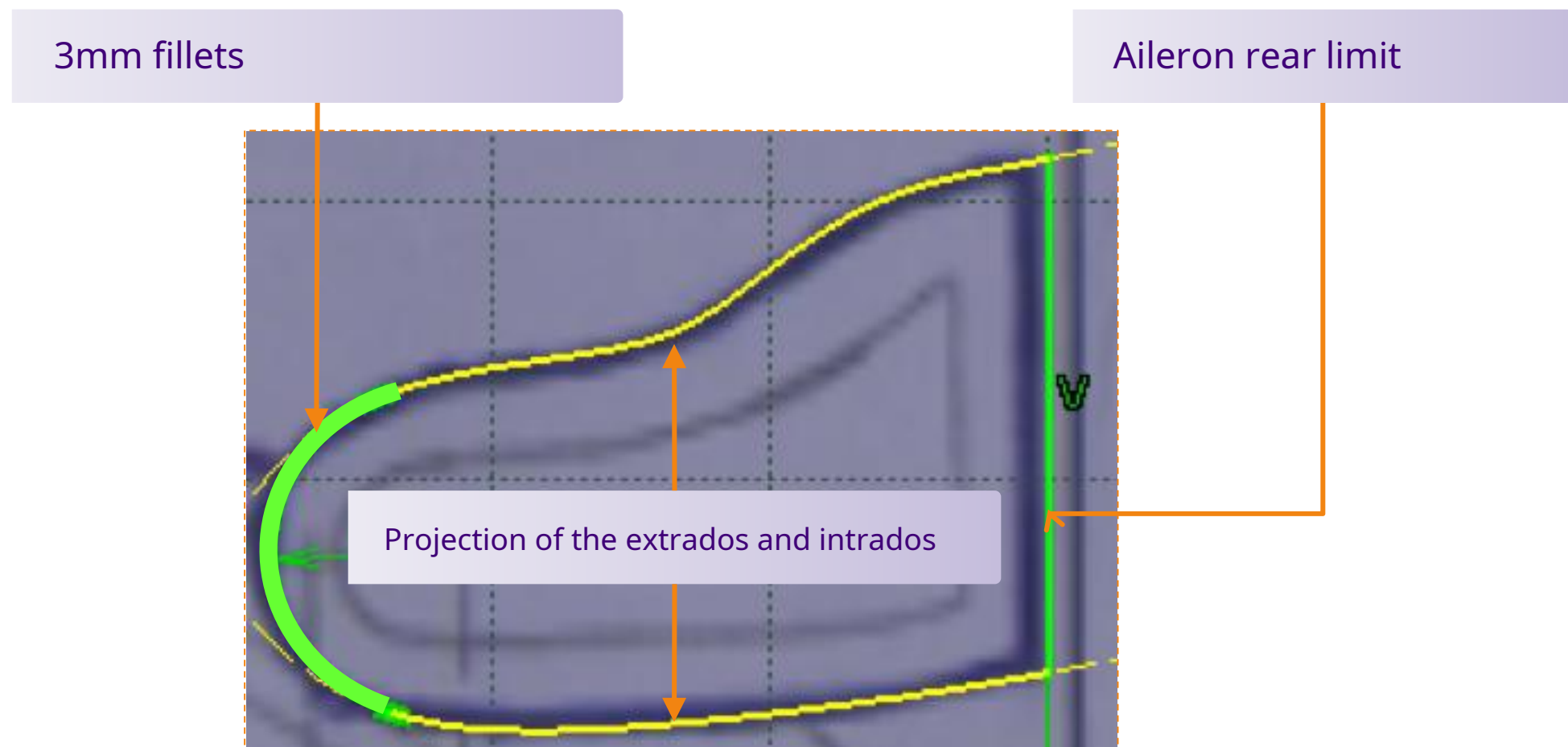


Deliberately leave the leading edge, so that the junction between the two curves will be done in a more precise.

# Model the rear wing

We will now use these two curves to sketch the rear wing below:

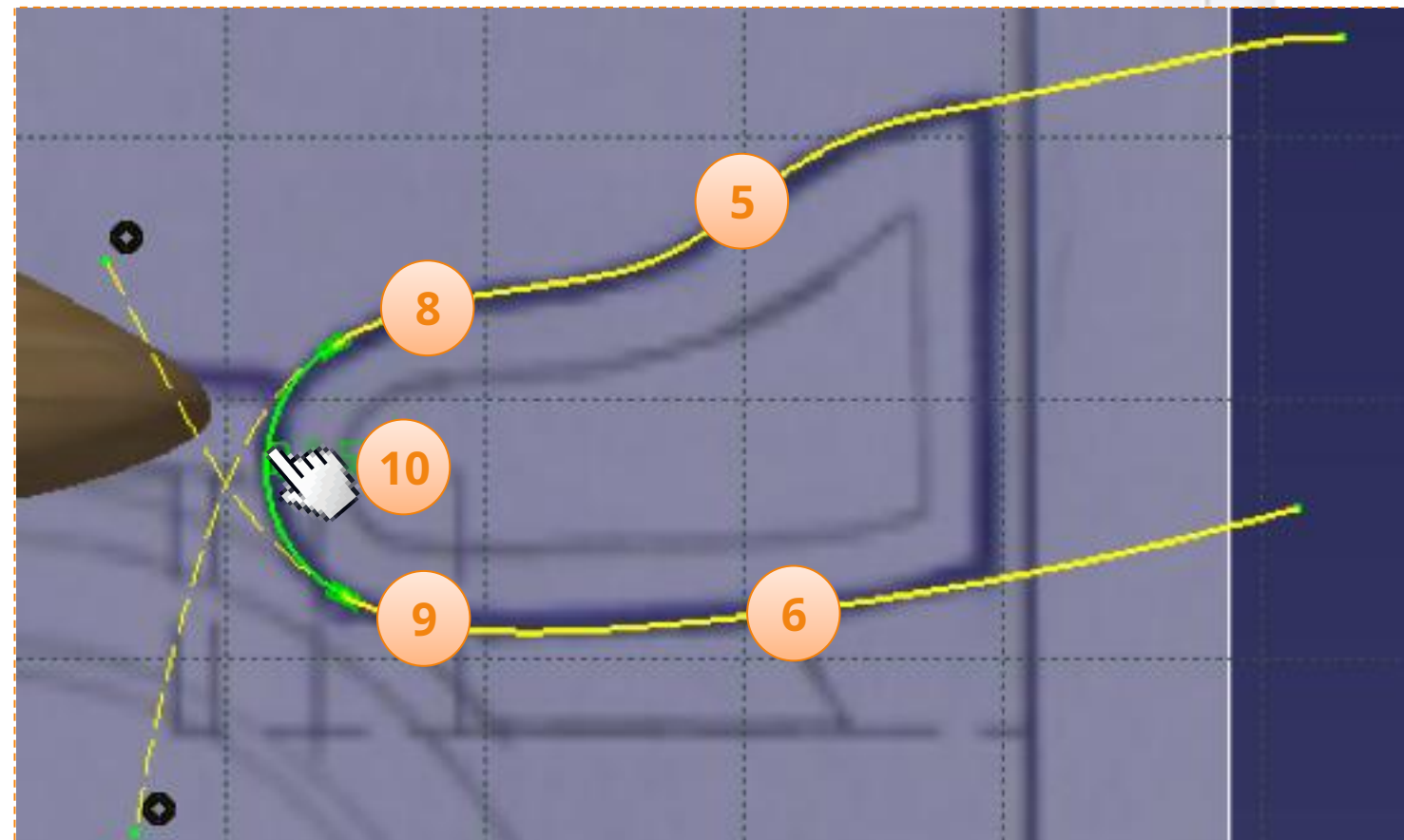
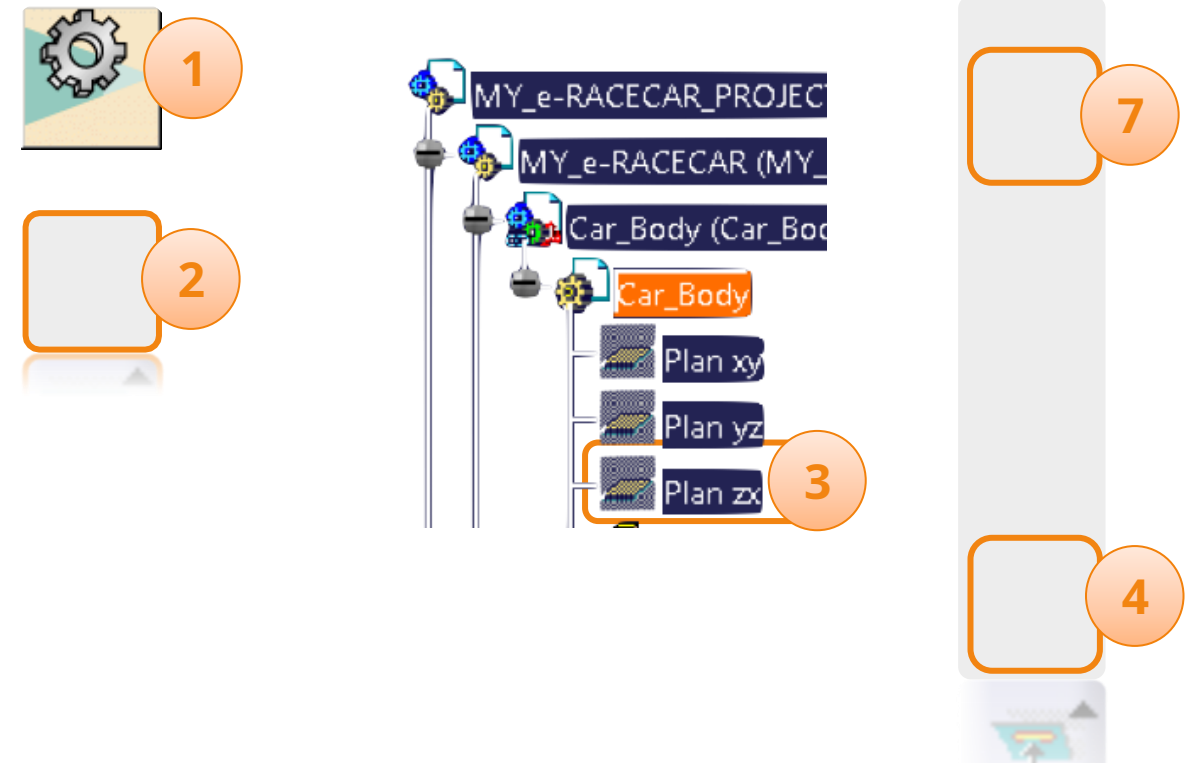
- ▶ Projection of the extrados and intrados.
- ▶ Connection leave.
- ▶ Aileron rear limit.



# Model the rear wing

To draw the sketch:

1. Go to the workshop **Part Design**.
2. Click the icon **Sketch**.
3. Click on the **ZX plane**.
4. Double-click the icon **Projection of 3D elements**.
5. Click the **ExtradoCurve**.
6. Click the **IntradoCurve**.
7. Hide the intrados and extrados curves (right-click then **Hide/Show**).
8. Click the icon **Rounded** which is a little above the projection icon.
9. Click on **one of the projections**.
10. Click on **the other of the projections**.
11. Click to define the roundness.



# Model the rear wing

1. Edit the dimension of the rounding and enter the value [6 mm].

2. Click the icon **Right**.

3. Click the **ExtradosCurve**.

4. Click the **IntradosCurve**.

5. Select the **right**.

6. Click the icon **Constraints chosen in a dialog box**.

7. Select **Verticality**.

8. Click on **okay**.

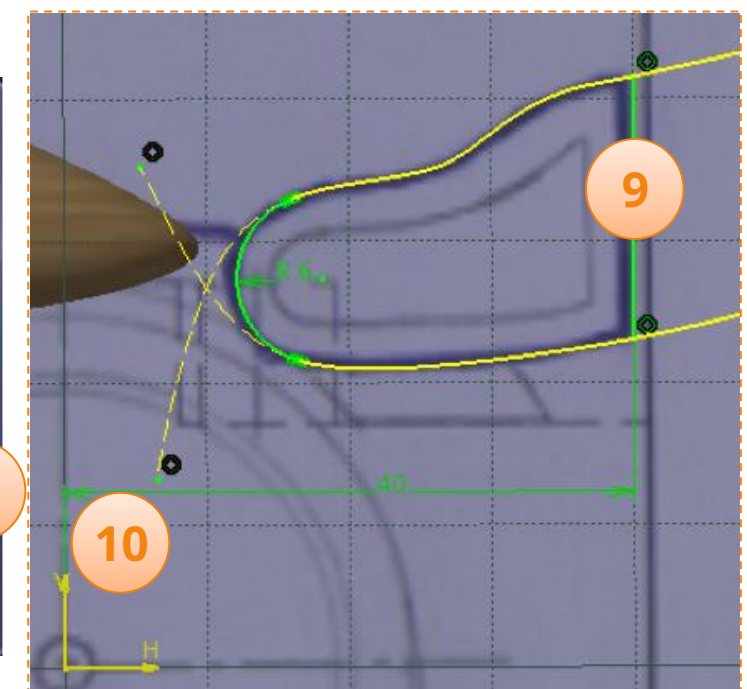
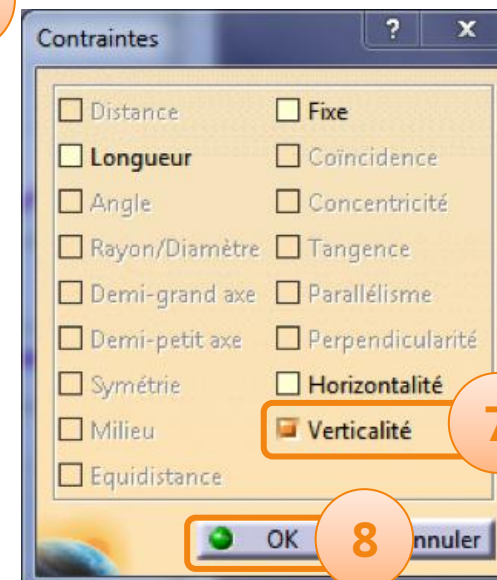
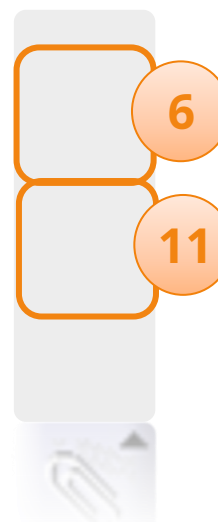
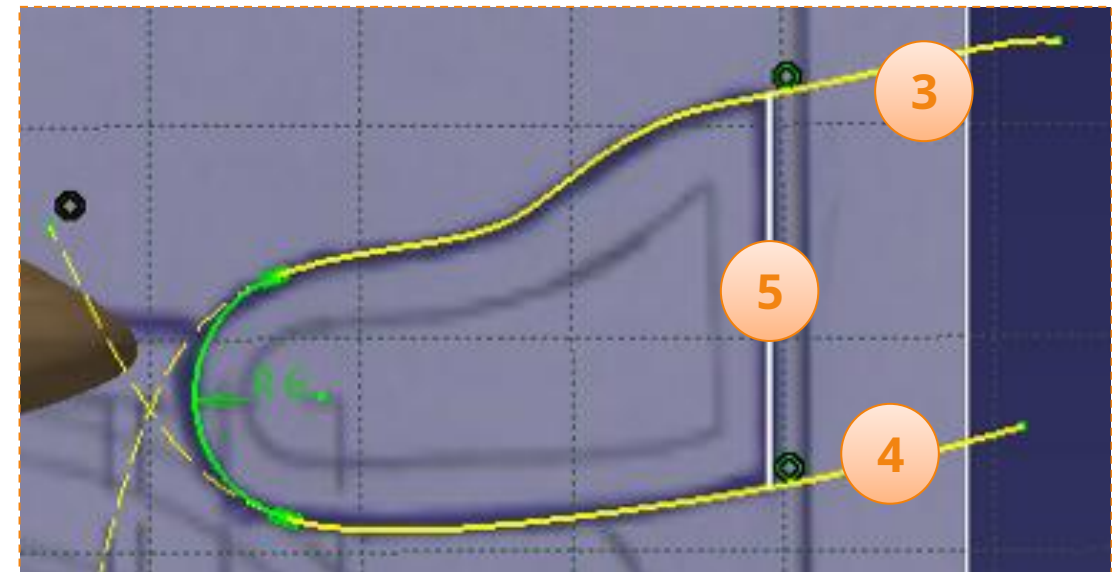
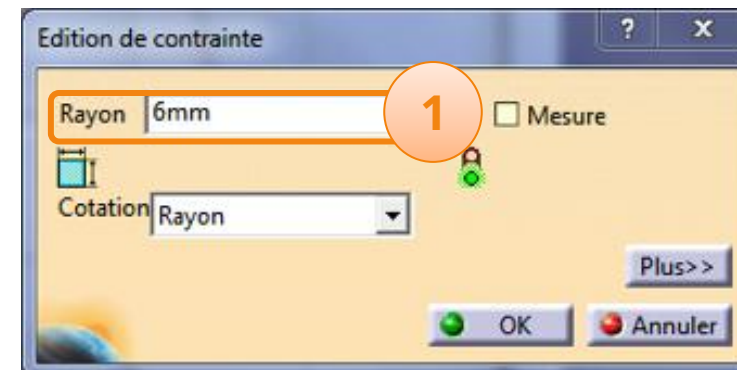
9. Select the **right**.

10. Select the **vertical axis**.

11. Enter a **Distance Constraint** of [40]



For more information on creating constraints between two elements, see the corresponding method sheet.

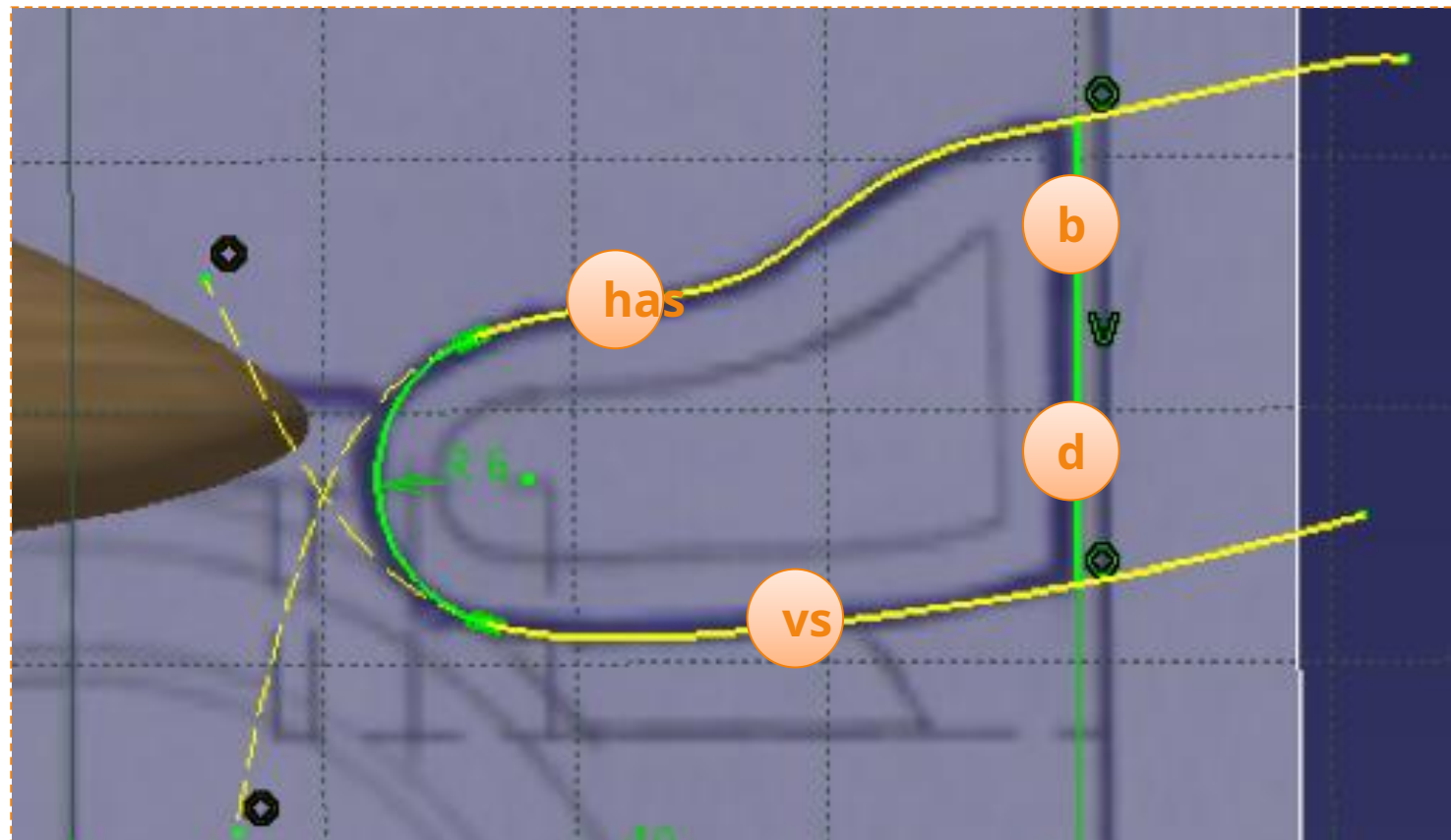
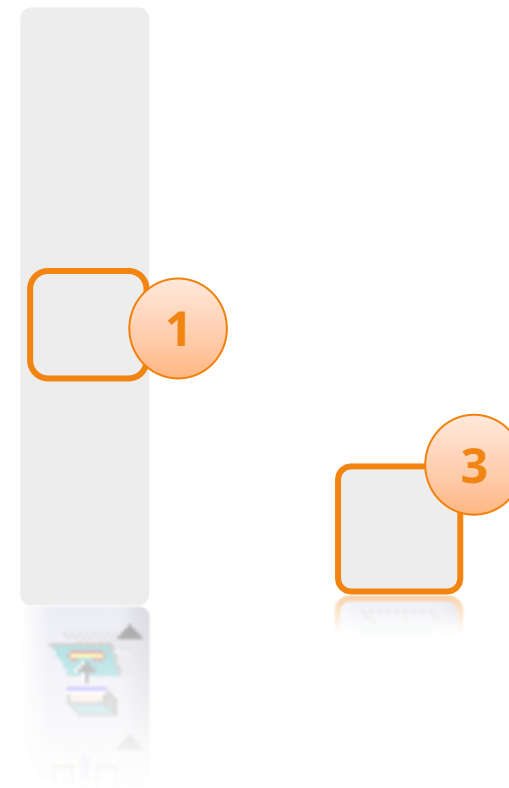




# Model the rear wing

To complete the sketch, it must have only one closed contour. For that :

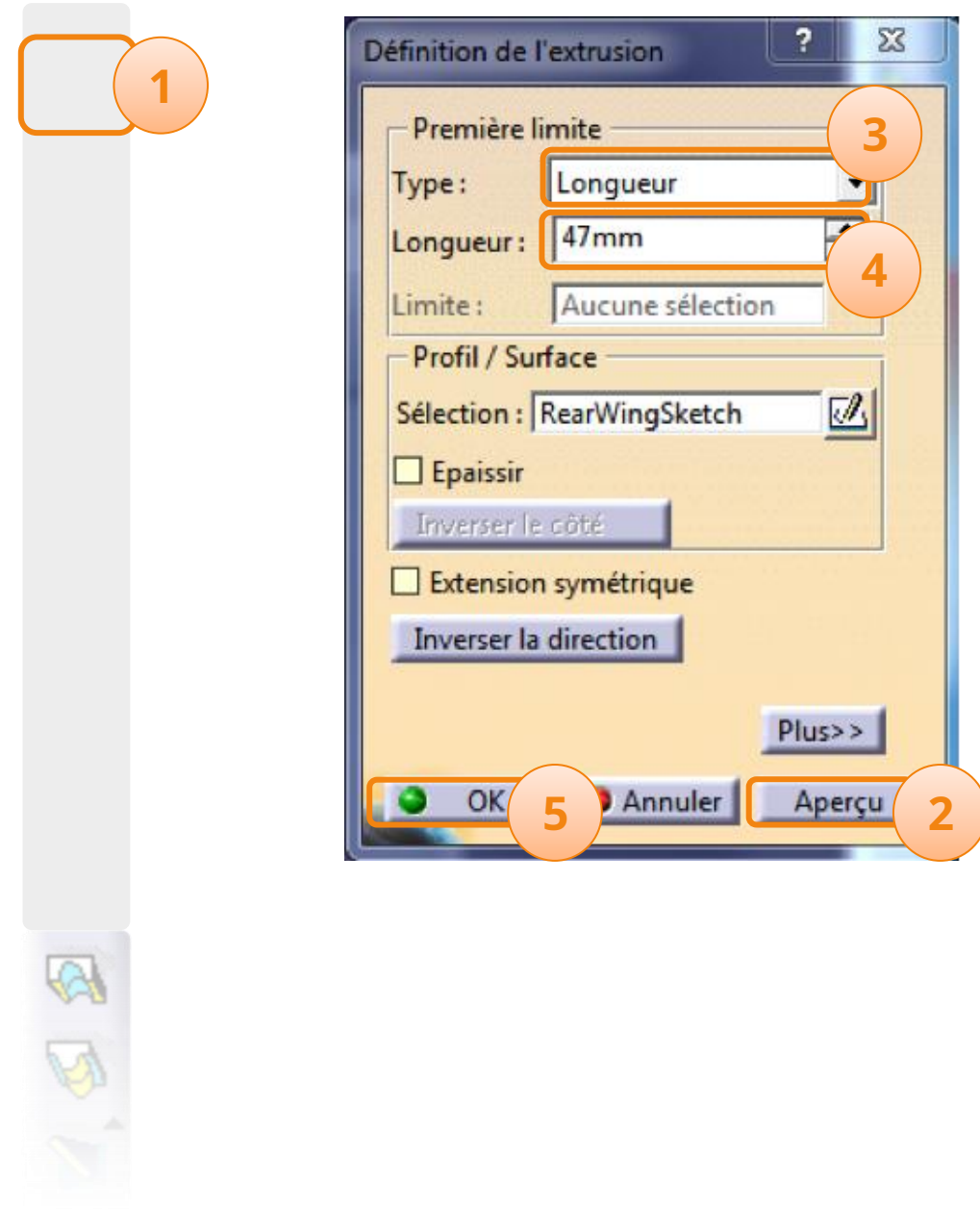
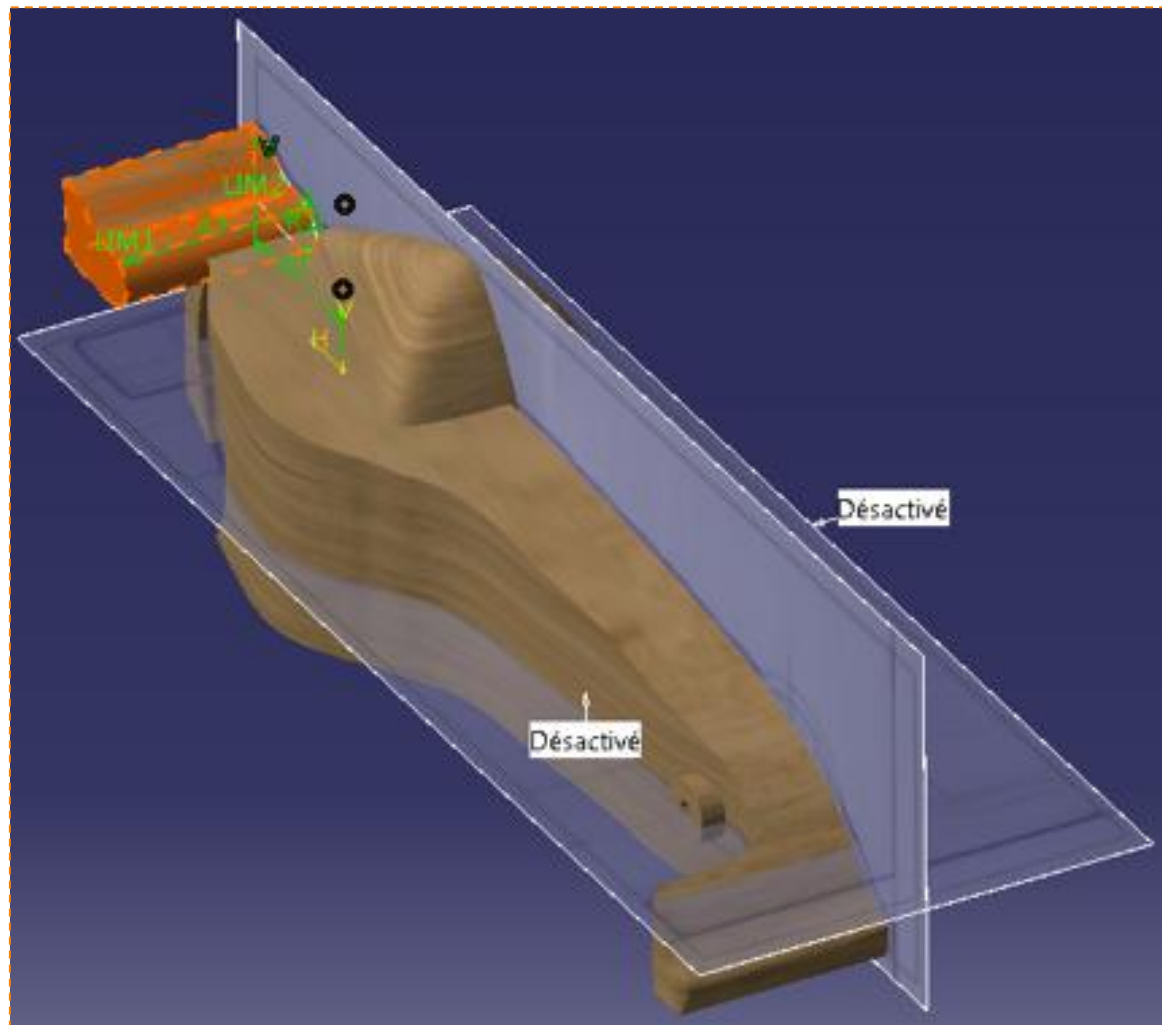
1. Double-click the icon **Limitation**.
2. Click the items following the guide below.
3. Click the icon **Leaving the workshop**.
4. Rename the sketch [RearWingSketch].



# Model the rear wing

All that remains is to create the material. For that :

1. Click the icon **Extrusion**.
2. Click on **Insight** and verify that the extrusion is on the correct side, as shown below. If not, click **Reverse direction**.
3. Choose Type **Length**.
4. Enter a length of [47mm].
5. Click on **okay** to validate.
6. Rename the extrusion [RearWing].





# Finalization

In this part, we are going to carry out the different operations to finalize the modeling of the bodywork:

- ▶ The location of the propulsion system.
- ▶ Edge fillets.
- ▶ The central section of the car.
- ▶ The symmetry.

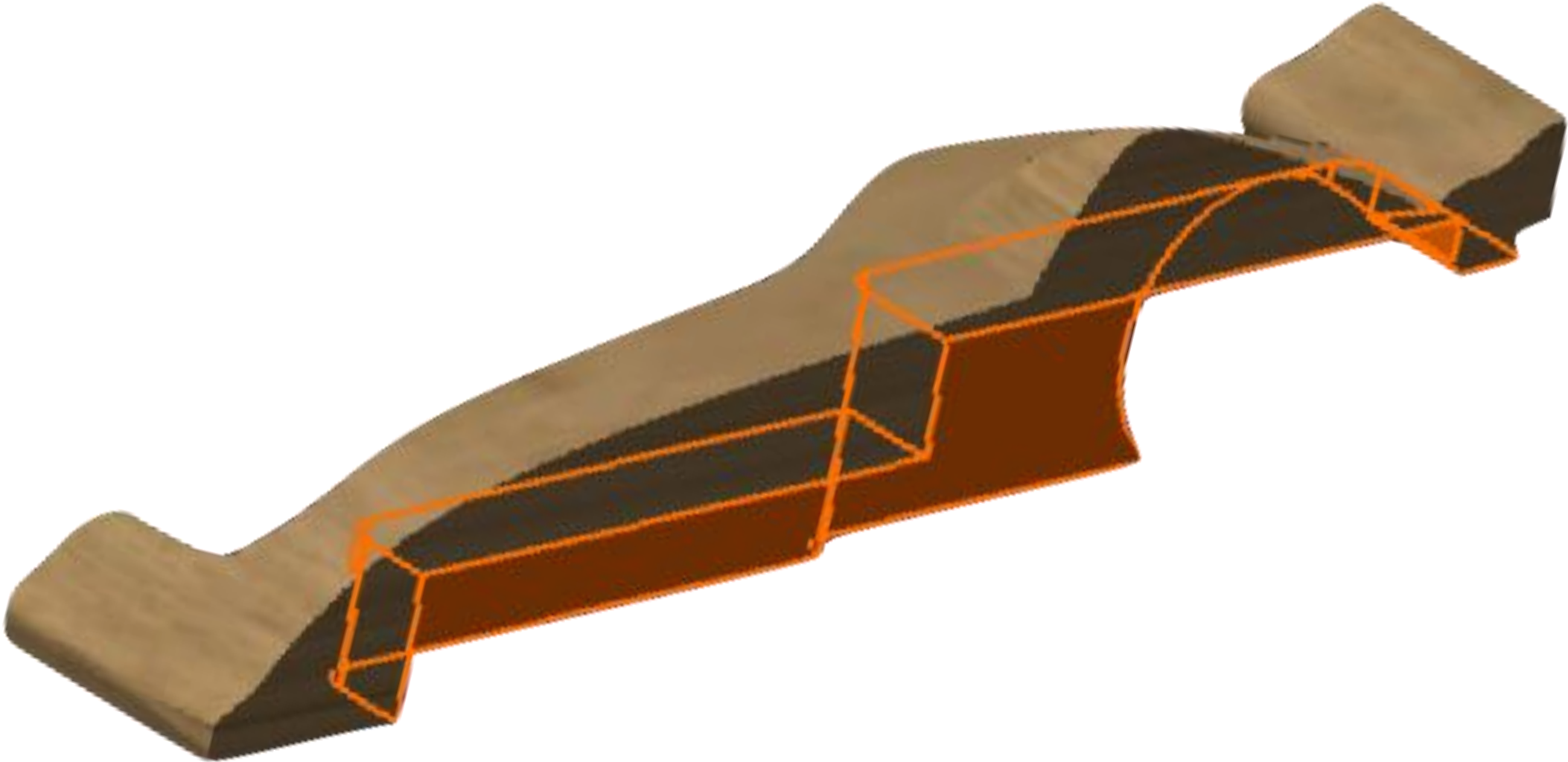


Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.**
    - i. Introduction.
    - ii. Settings.
    - iii. Model the body.
    - iv. Model the cockpit.
    - v. Model the rear wing.
    - vi. Finalization.**
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

NOT

land and pod.

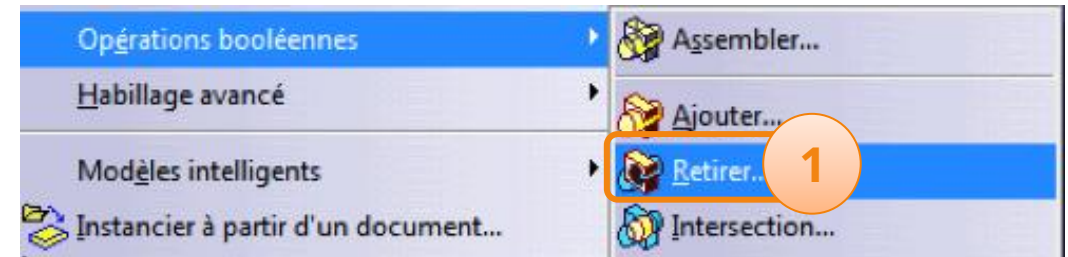




# Finalization – Propulsion System Location

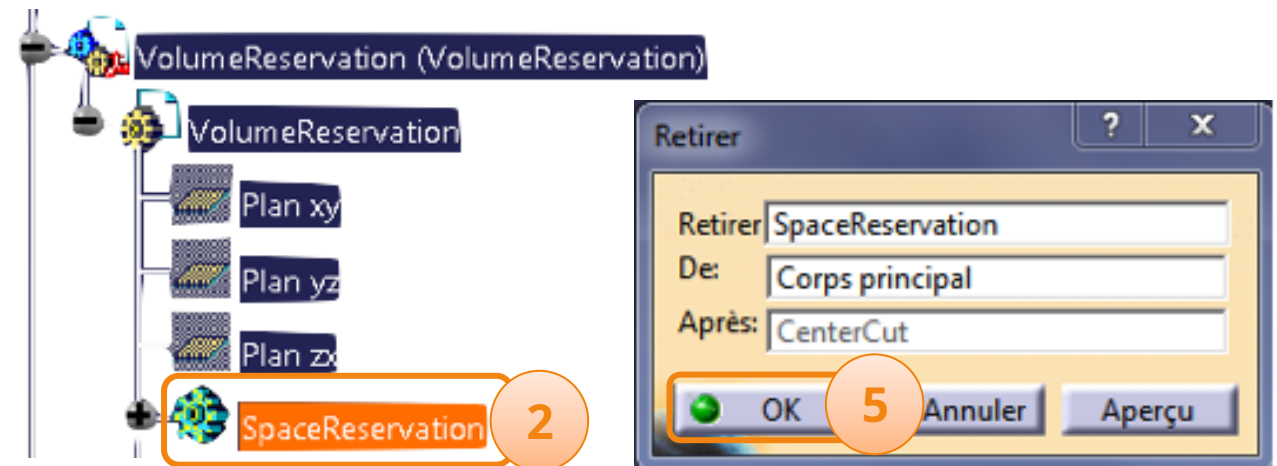
We are going to remove some material to place the motor, the battery and the pod. For that :

1. Select **Insertion** > **Boolean operations** > **Withdraw**.



2. Click on **+** of the branch **Rules**.

3. Click on **+** of the branch **VolumeReservation**.



4. Click on **SpaceReservation**.

5. Validate by clicking on **okay** in the window that appears.

6. Rename Remove to [Remove\_SpaceReservation].

You should get a result similar to this:



## Finalization – Leave

We are going to make the following holidays.

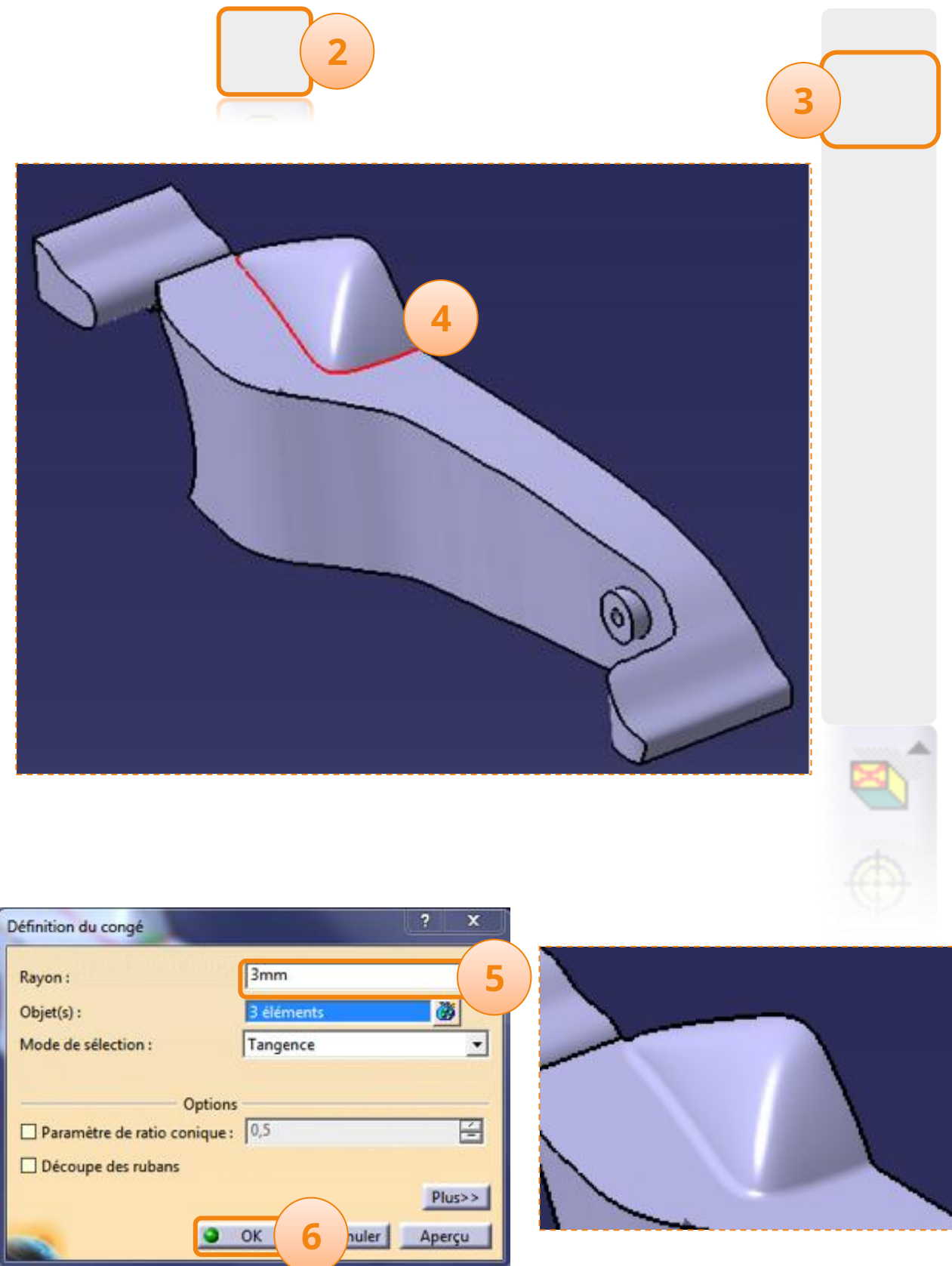


# Finalization – Leave

1. First check that **Car\_Body** is the active level.
2. Switch to mode **Realistic rendering with edges without dead edges**.
3. Click the icon **Edge fillet**.
4. Click on the edges shown opposite.
5. Enter [3 mm] for the radius value.
6. Validate by clicking on **okay**.



You can give back to the main body its original appearance by disabling transparency. This will make it easier to select ridges.  
To deselect an item, click a second time on it.



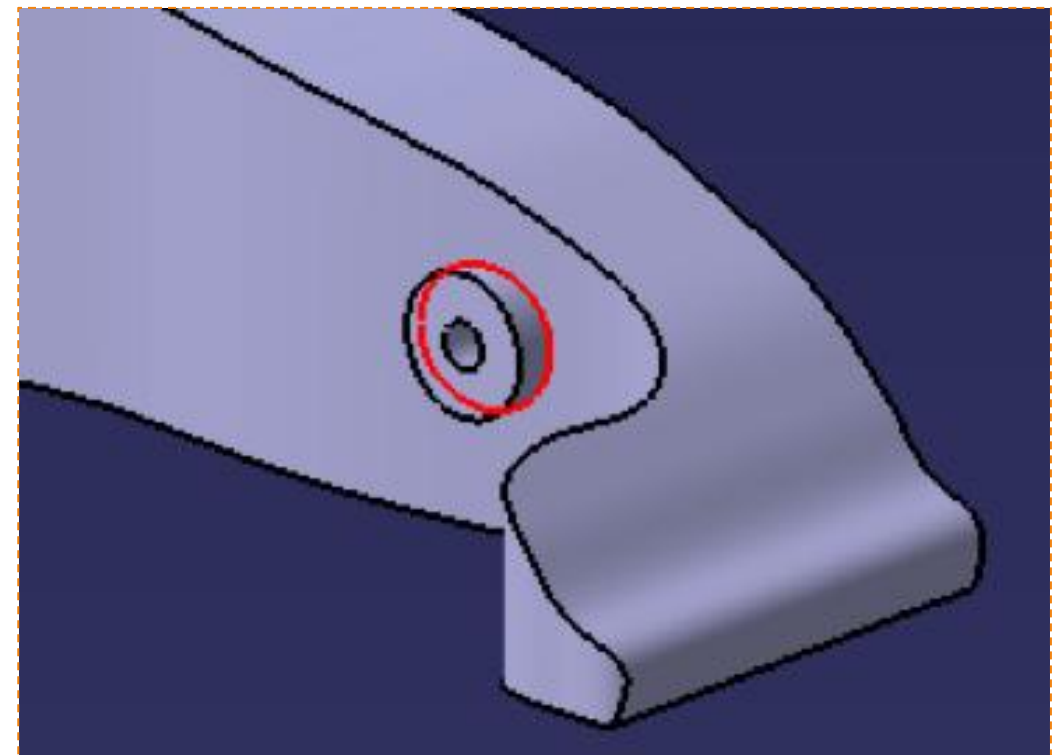
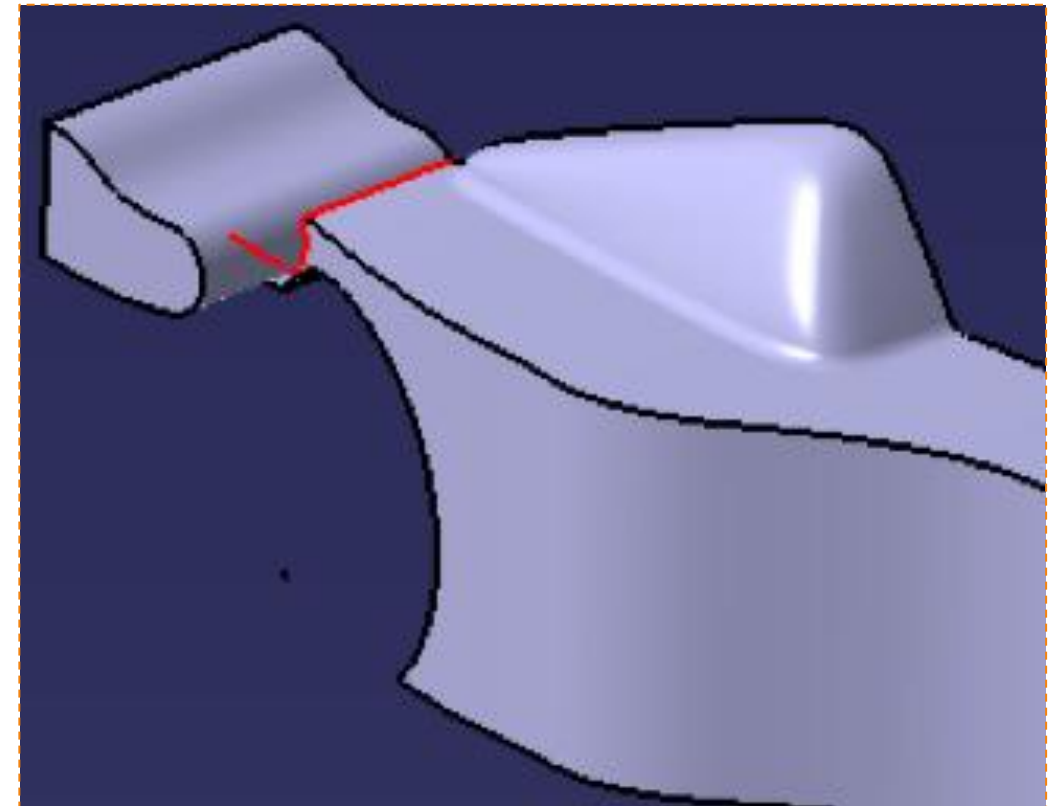
## Finalization – Leave

Let's move on to the holidays. First of all :

1. Click the icon **Edge fillet**.
2. Select the edge opposite.
3. Enter the value [3mm].
4. Validate by clicking on **okay**.

Then we will use the edge opposite.

5. Click the icon **Edge fillet**.
6. Select the edge opposite.
7. Enter the value [3mm].
8. Validate by clicking on **okay**.

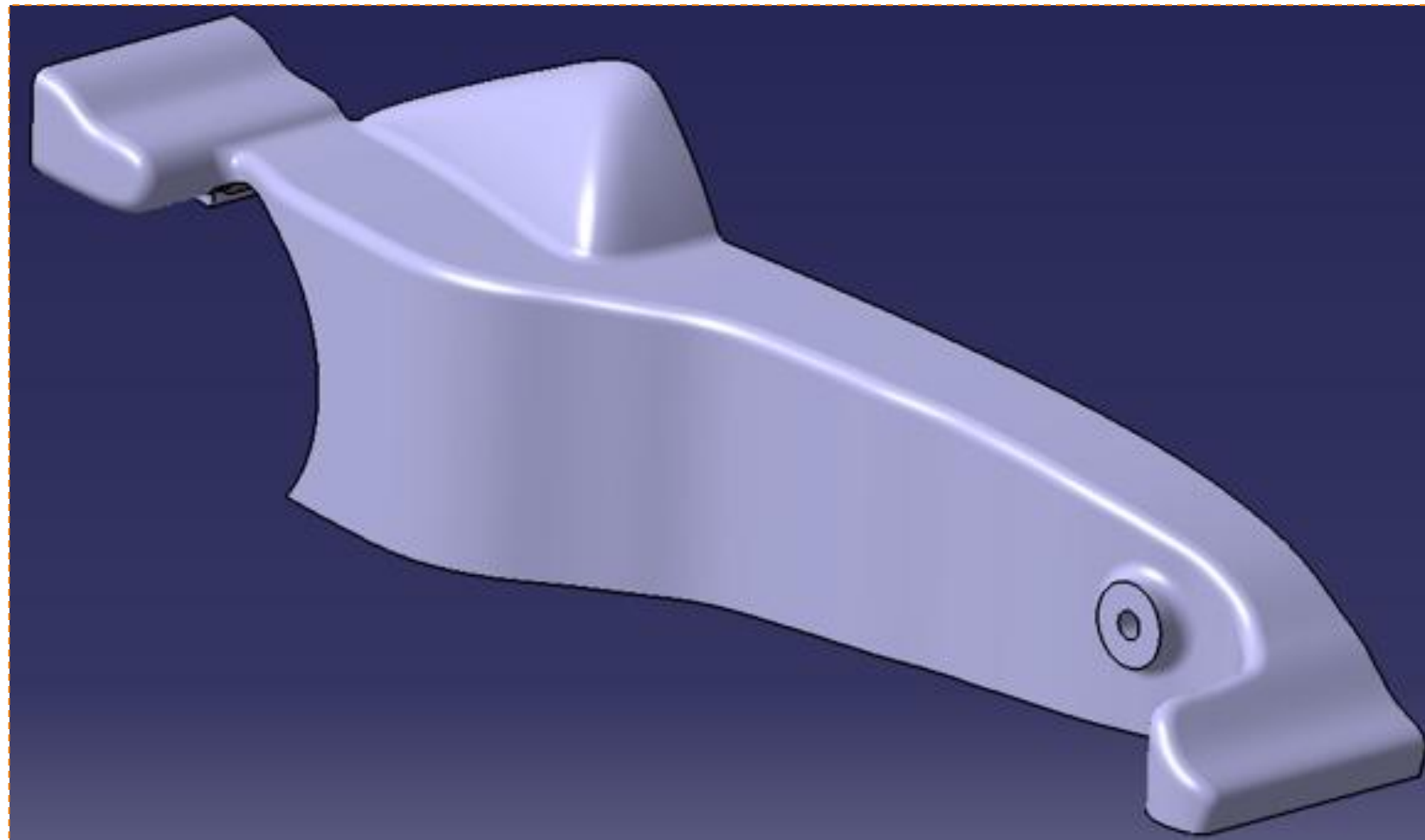
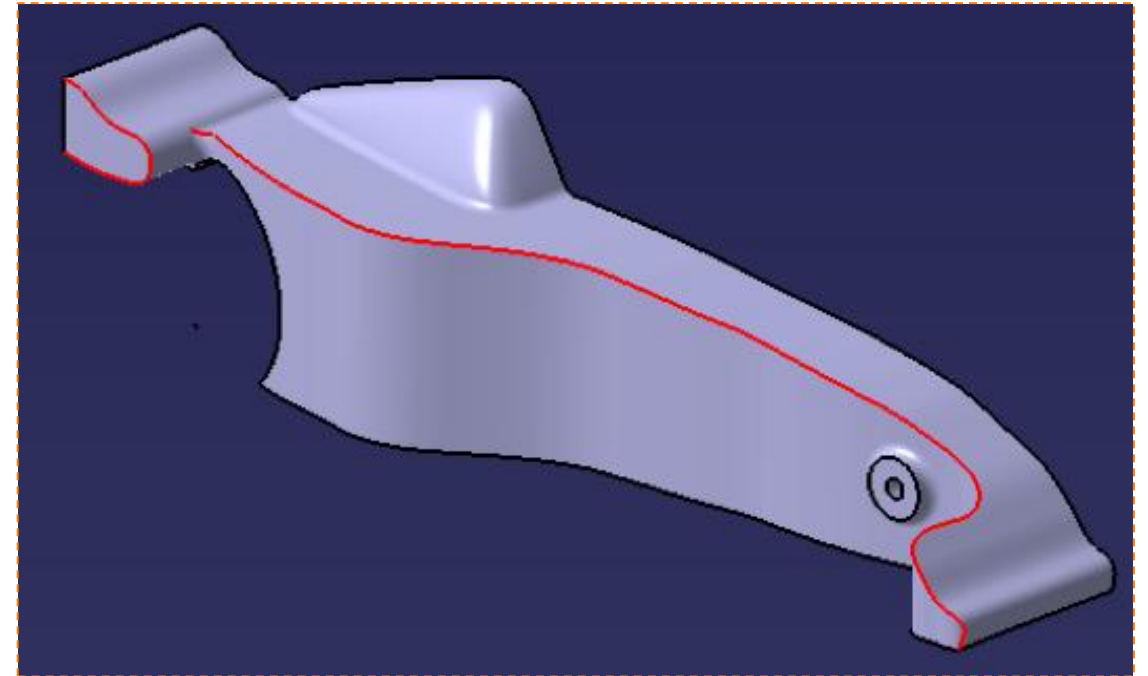


## Finalization – Leave

For the last holiday, we are going to perform a grouping:

1. Click the icon **Edge fillet**.
2. Select the edges opposite.
3. Enter the value [3mm].
4. Validate by clicking on **okay**.

You should get the result below:

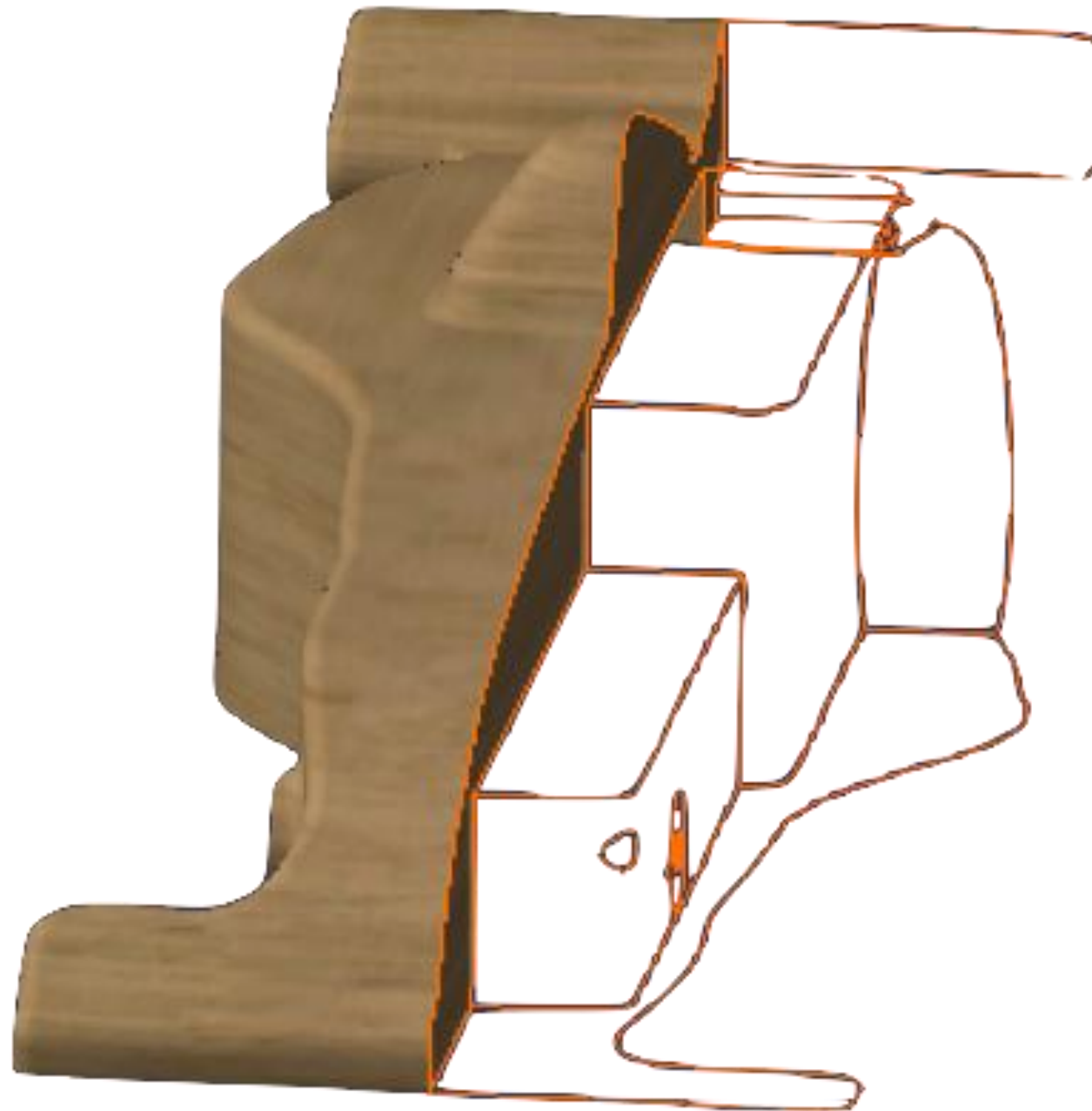




## Finalization – Respect of the symmetry constraint

The car must be symmetrical after machining, we must ensure that it is perfectly symmetrical on the digital model.

We have



## Finalization – Central Cup

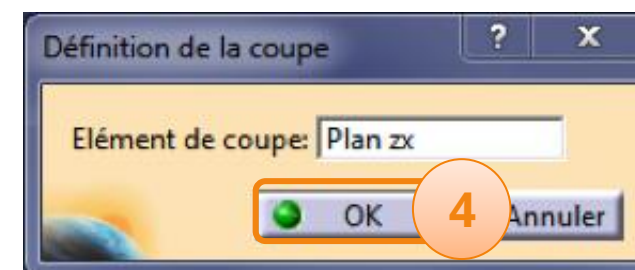
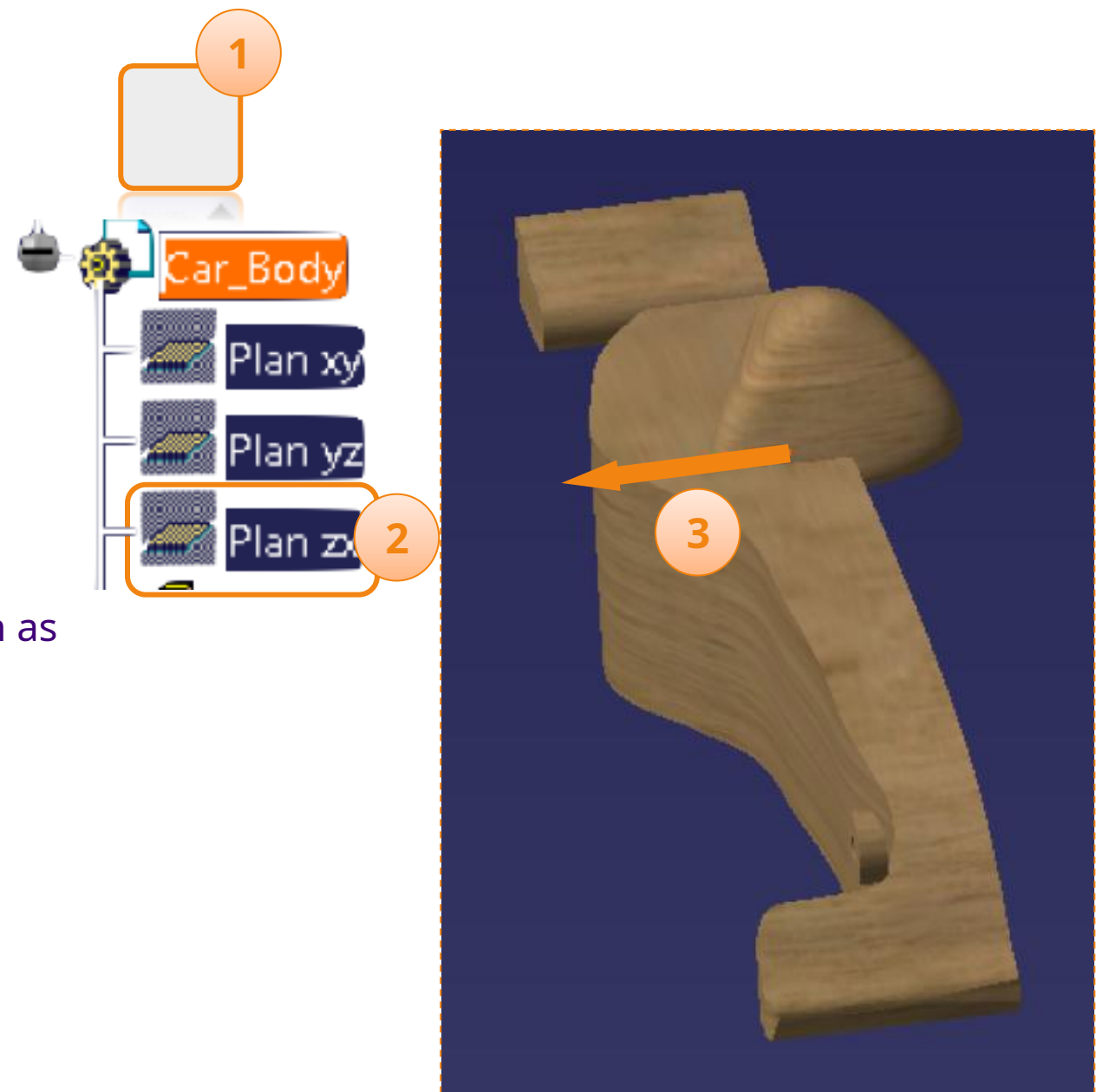
1. Click the icon **Chopped off**.

2. Click on the **ZX Map**.

3. Check that the orange arrow is in the same direction as in the image opposite. If not, click on it.

4. Validate by clicking on **okay** in the window that opened.

5. Rename Cut.1 to [CenterCut].

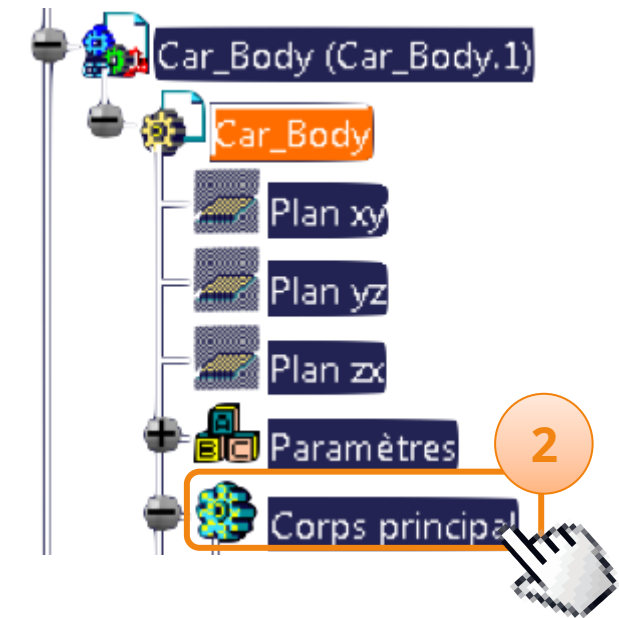


# Finalization – Respect of the symmetry constraint

1. In order to better observe the result, return to mode **Realistic rendering with texture**.



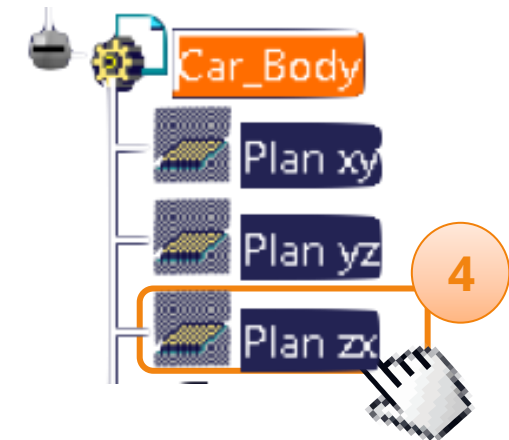
2. Click on **main body** to select it.



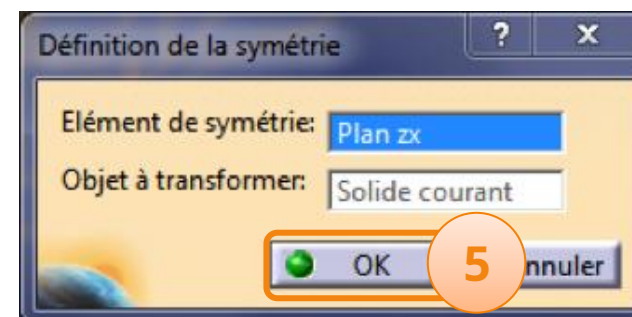
3. Click the icon **Symmetry**.



4. Click on the **ZX Map**.



5. Validate by clicking on **okay** in the window that opened.



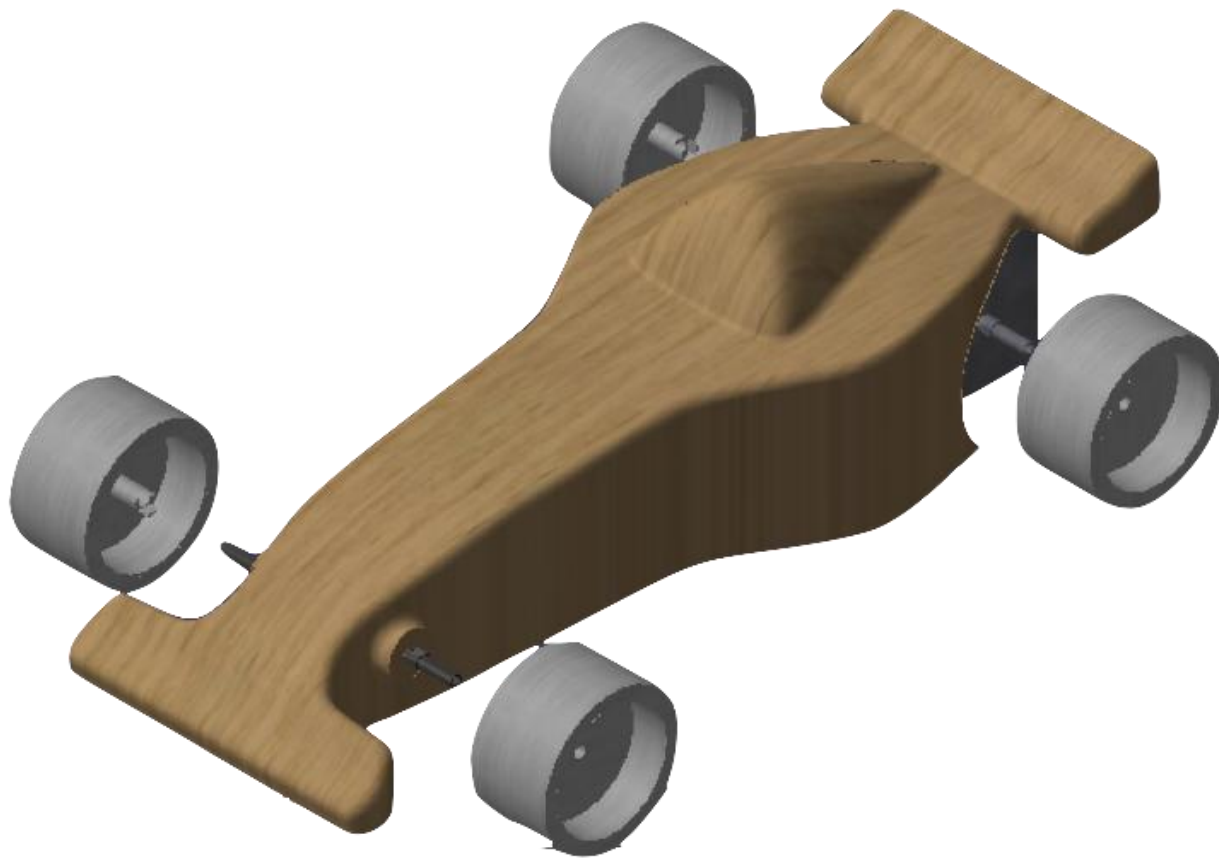
## Finalization – Respect of the symmetry constraint

Congratulations ! Your car is now perfectly symmetrical.



# Assemble your car

In this part we will assemble the different parts that make up the car.

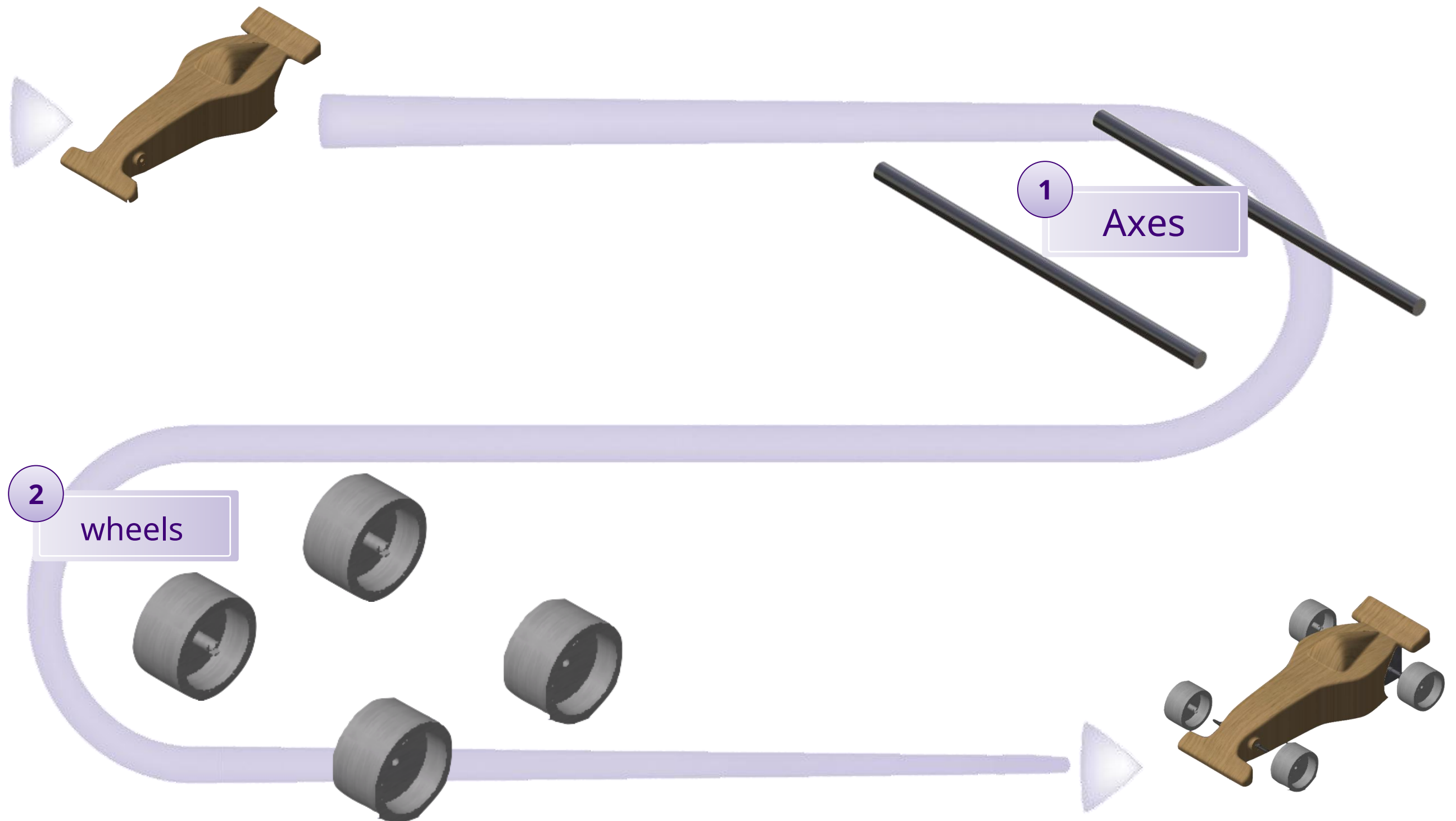


Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. *Your racing car.***
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.***
    - i. Introduction.***
      - ii. Insert the axes.
      - iii. Insert the wheels.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.



# Introduction – Assembly Process

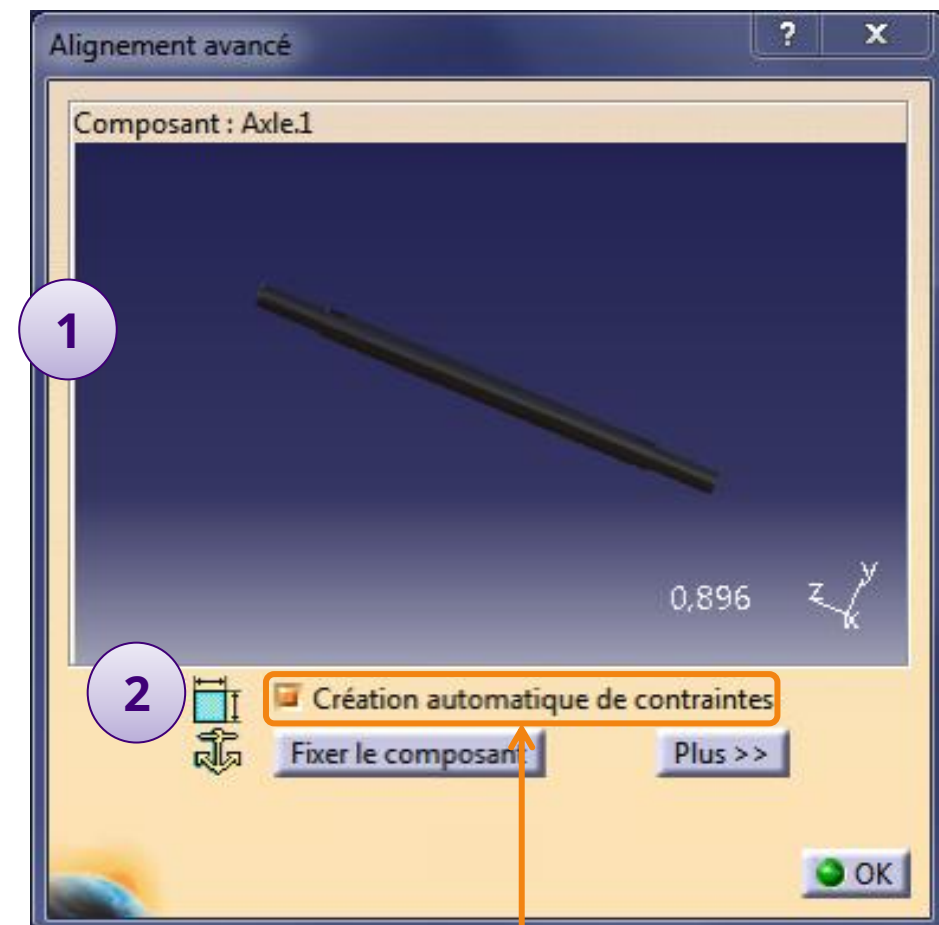
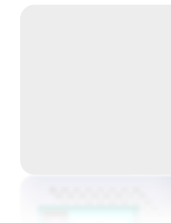


## Existing component with positioning

In this part we will insert the components with the positioning tool.

- 1 This tool displays a small window containing the part we want to insert. This view is handled in the same way as in the CATIA environment. Thanks to this we have easy access to the surfaces of the part without being bothered by the rest of the assembly.
- 2 The interest of this tool lies in the option of automatic creation of constraints. You will be able to create the constraints using this tool, which makes the assembly more ergonomic.

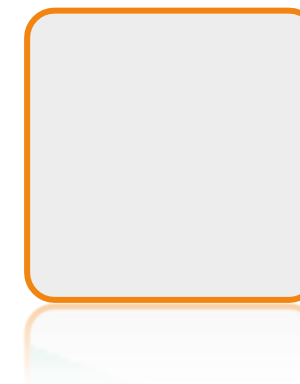
Don't forget to select the creation automatic constraints



# Introduction

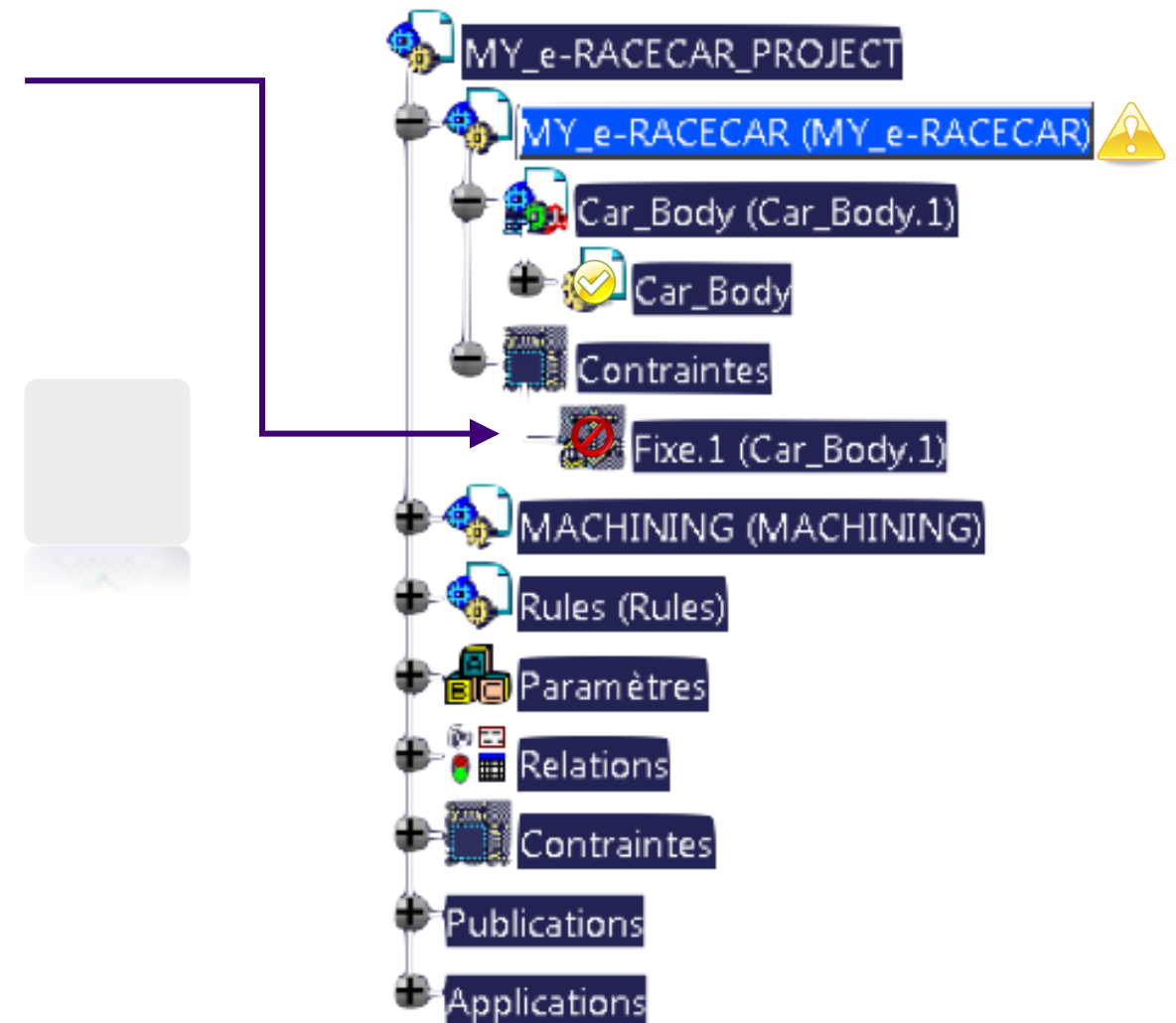
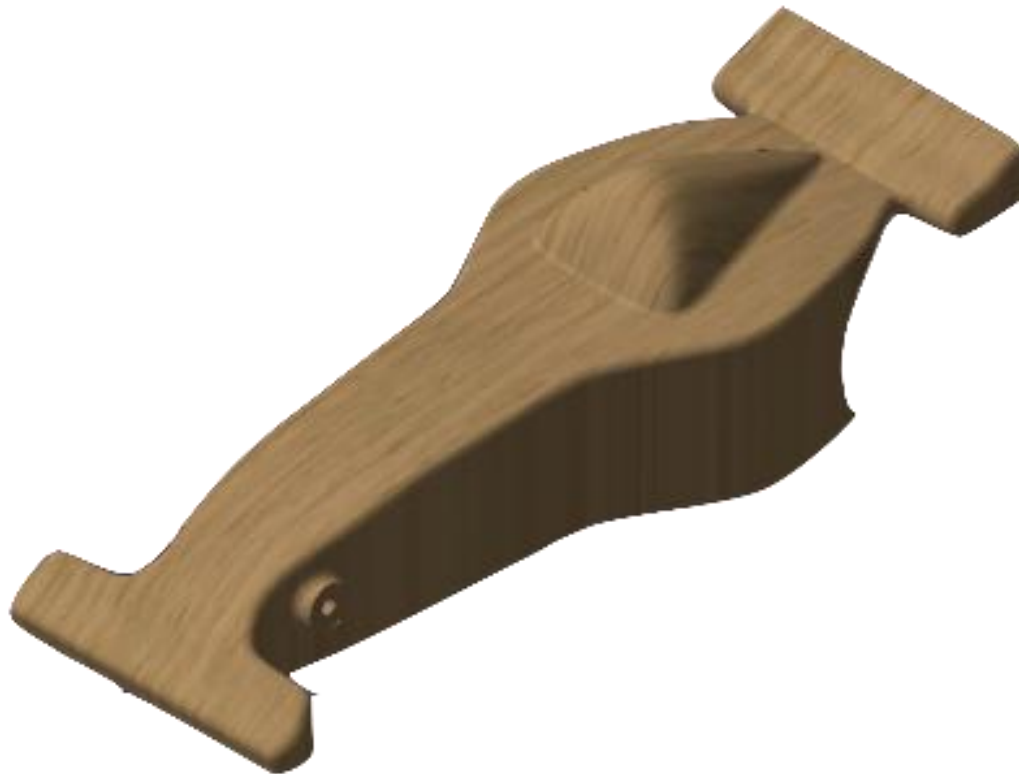
In order to add elements to the car, we must first activate the product that will receive the different elements.

1. Double-click **MY\_e-RACECAR**. The product is highlighted in orange to indicate that it is selected.
2. Verify that you are in the Assembly Design workbench. To do this, look at the active workshop icon (located in the upper right corner).  
If not, click **Start > Mechanical Design > Assembly Design**.



# Introduction

- ▶ You will notice that a constraint is already present in the product **MY\_e-RACECAR**. It is used to fix a component, and is applied to the part **Car\_Body**.
- ▶ This constraint must be applied to the element that serves as the frame in your assembly.
- ▶ Any assembly must begin with the definition of the parts that will remain fixed.
- ▶ Check that **MY\_e-RACECAR** is indeed the active product.



# Insert the axes

In this part we will add the front axle of the car.



Here are the steps to follow:

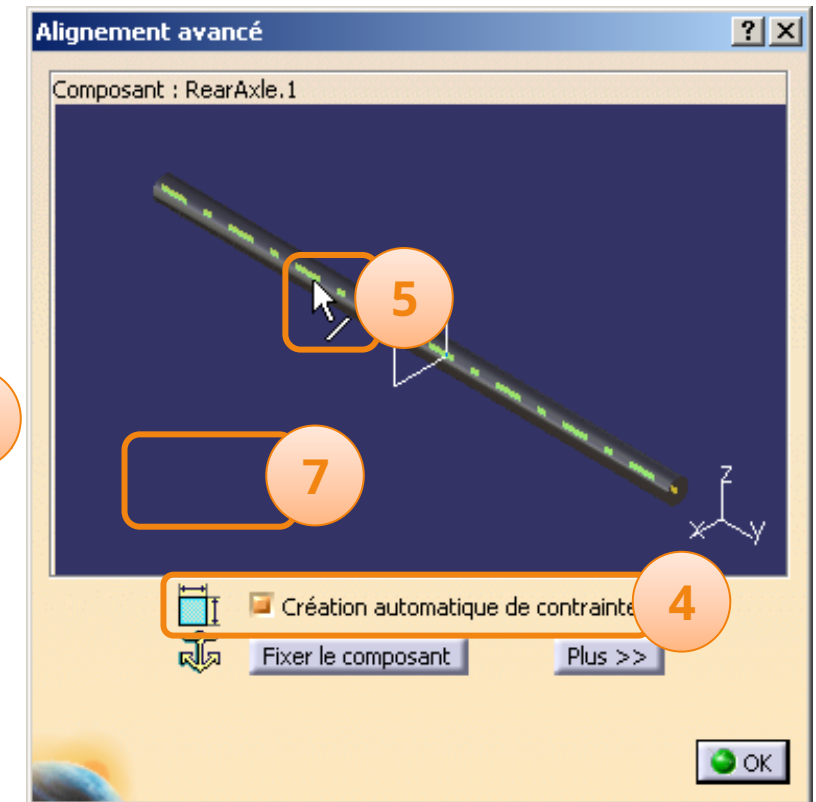
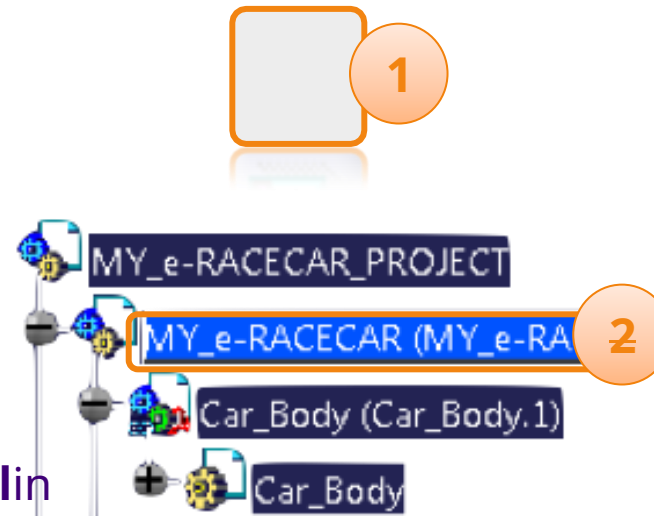
1. Introduction.
2. CATIA V5 QuickStart.
3. *Your racing car.*
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. *Assemble your car.*
    - i. Introduction.
    - ii. *Insert the axes.*
    - iii. Insert the wheels.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renders.
  - i. Simulate machining.
4. Method sheets.



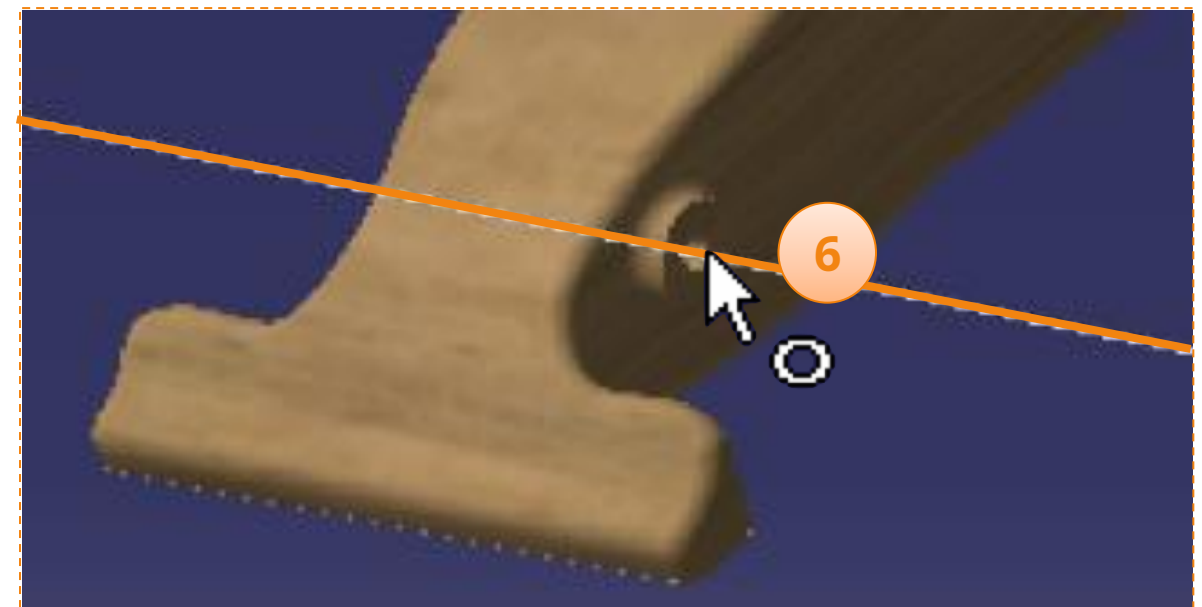
# Insert the axes

To start, we will constrain the axis with respect to the hole for the passage of the axis.

1. Click the icon **Existing component with positioning**.
2. Click on the product **MY\_e-RACECAR**.
3. A file selection window opens. Select file **Axe** in the directory **MiniF1e 2010 - StarterPackage**.

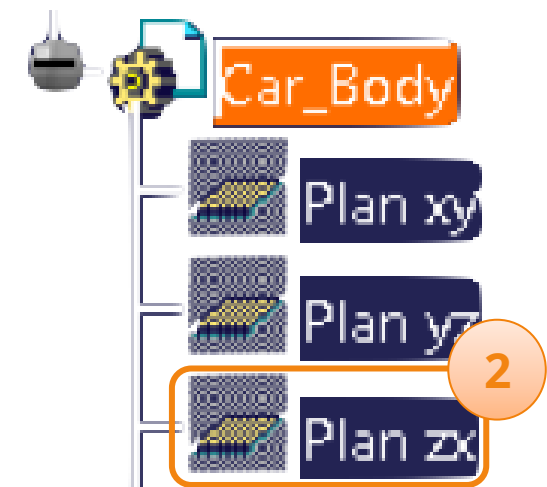
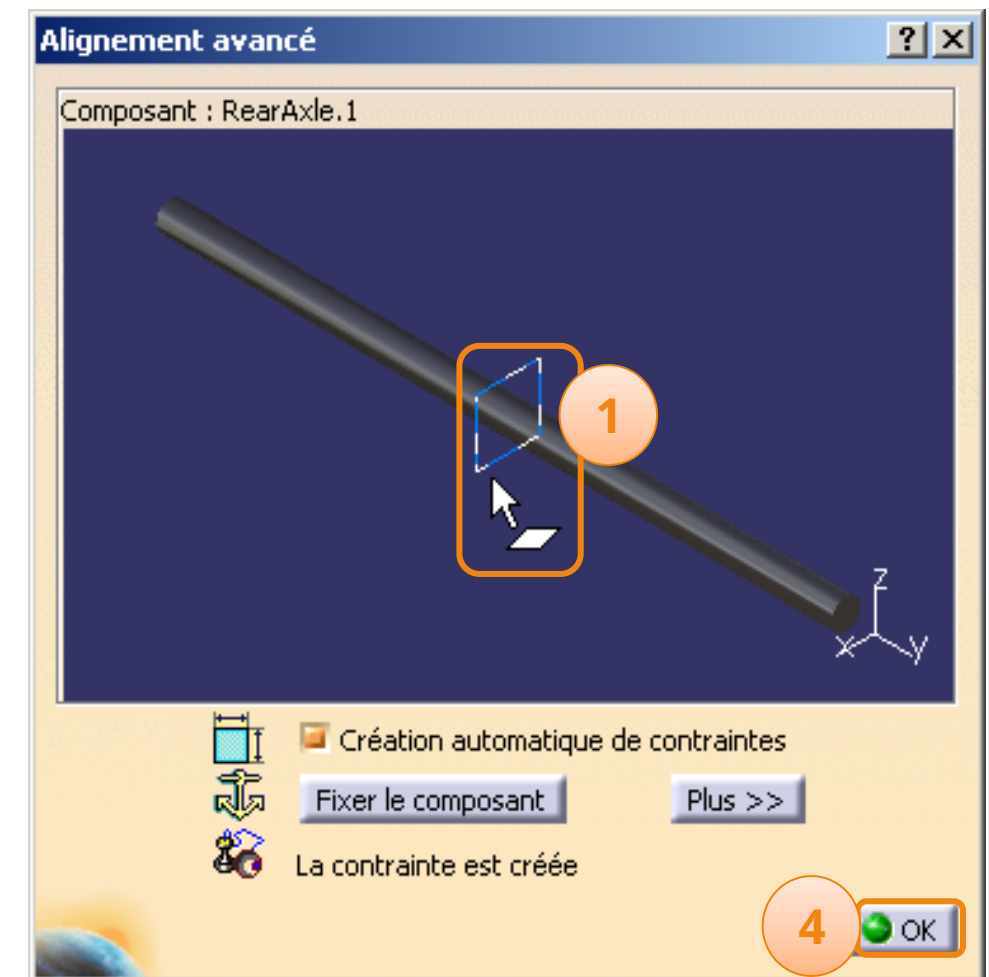
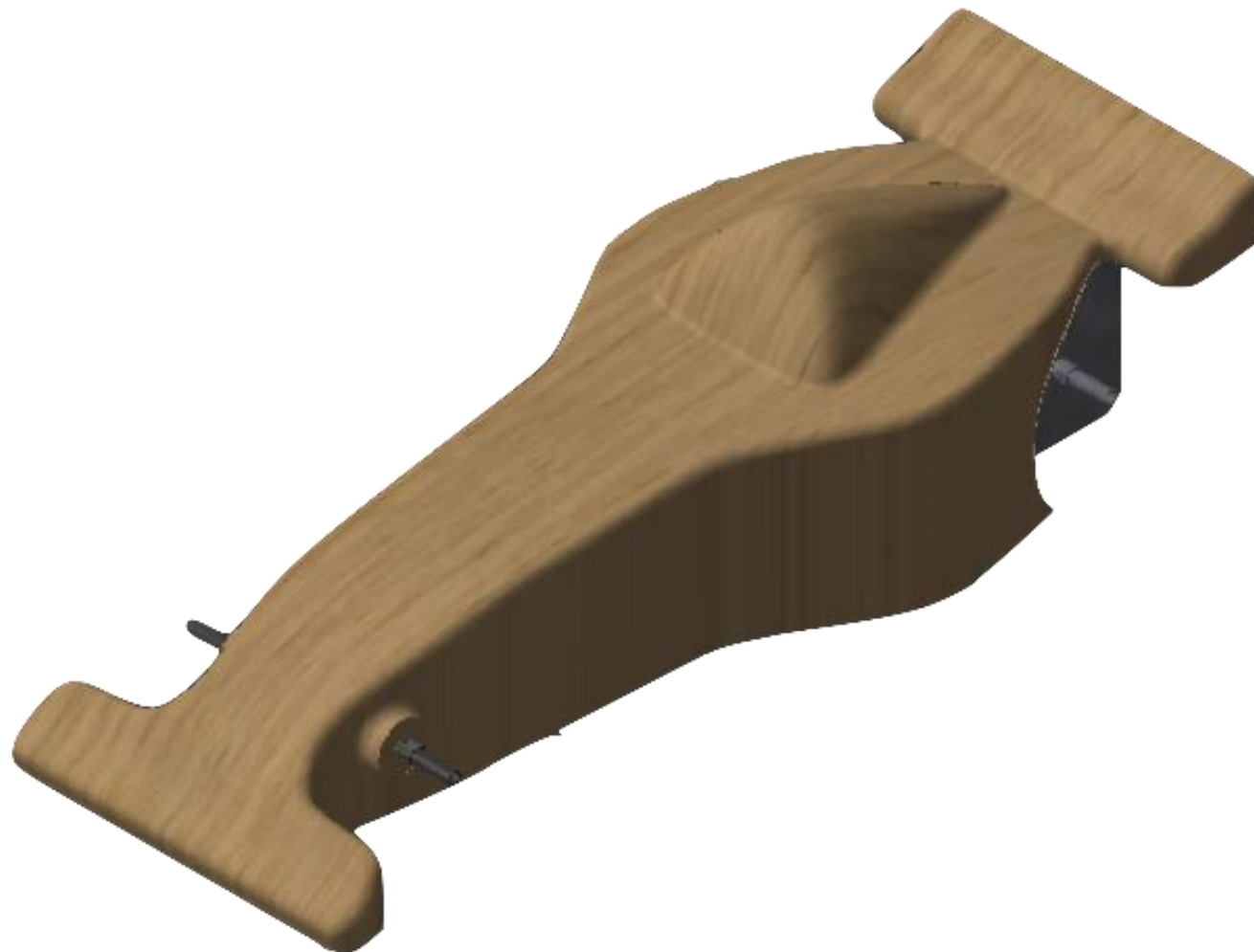


4. Verify that automatic constraint creation is still enabled.
5. Click the **axis**.
6. Click the **car front axle**. This appears when you bring the mouse closer to the landing.
7. Click in the **blank part** of the viewing window. This will create the constraint between the two parts.



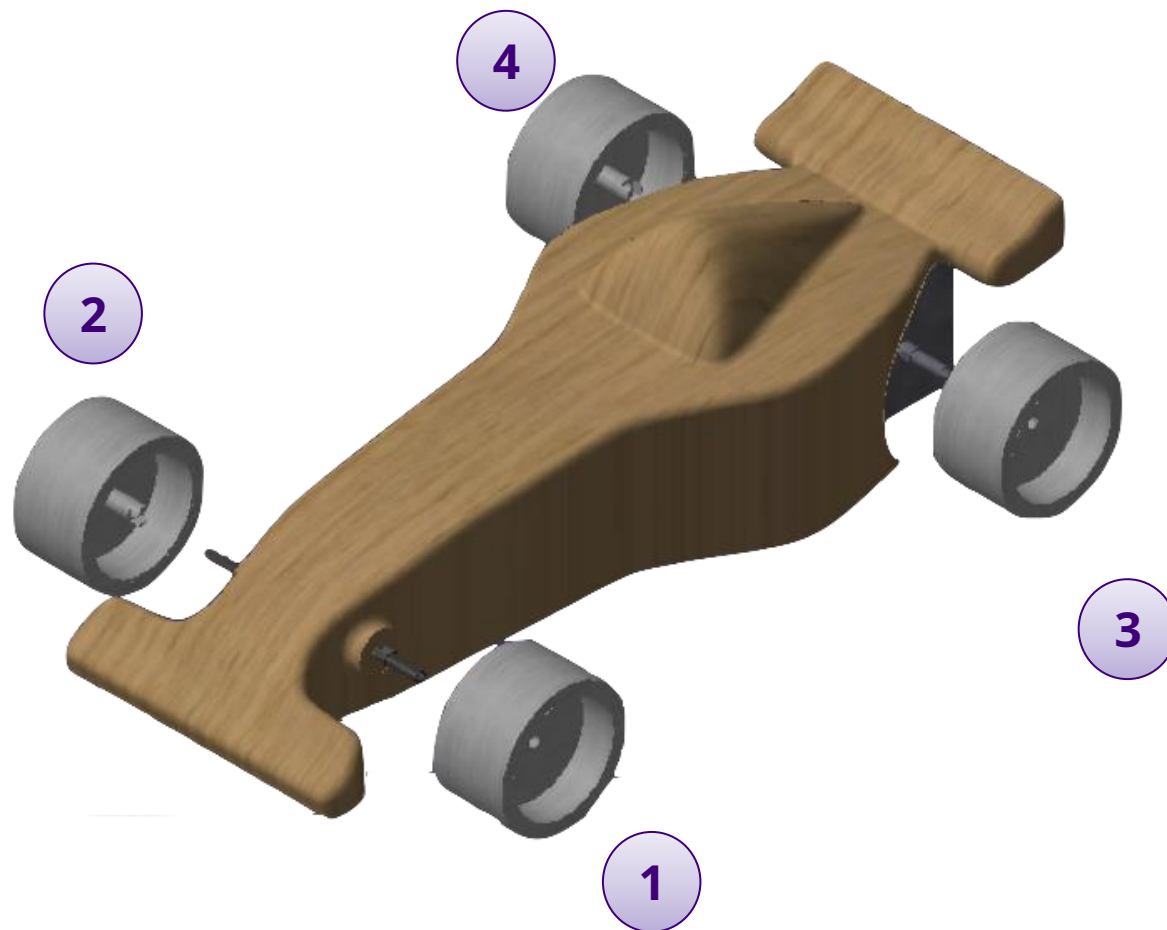
## Insert the axes

1. Then click on **the plan** which is displayed in the viewing window.
2. Then click on the **ZX plane** present in **Car\_Body**.
3. Click in the **blank part** of the viewing window.
4. Click on **okay**.



# Insert the wheels

In this part we will add the four wheels of the car in the order below.



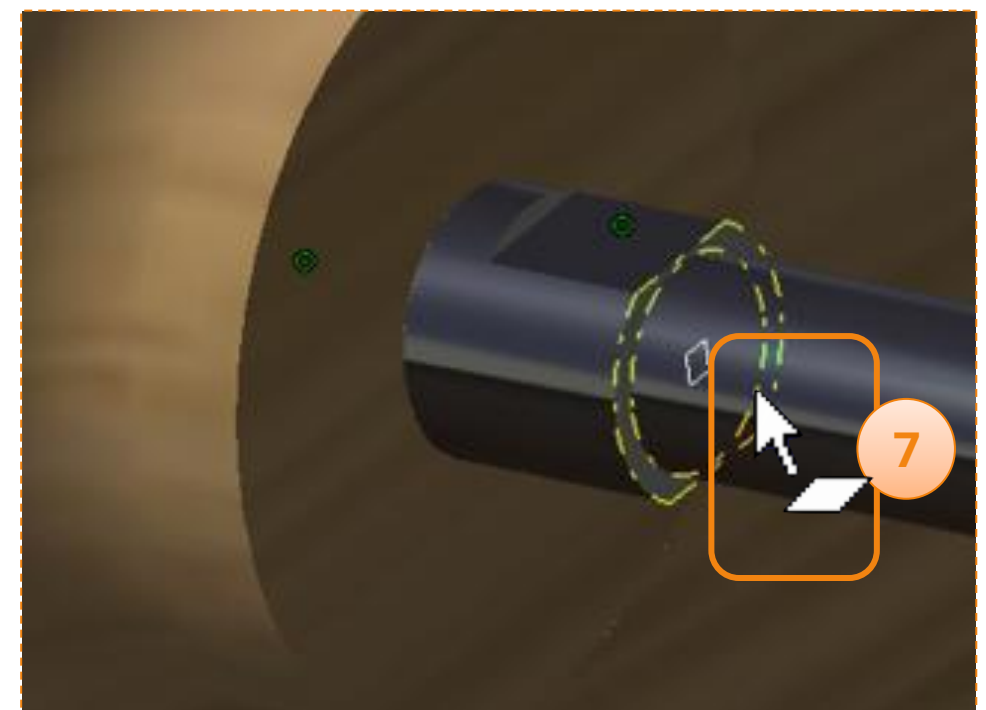
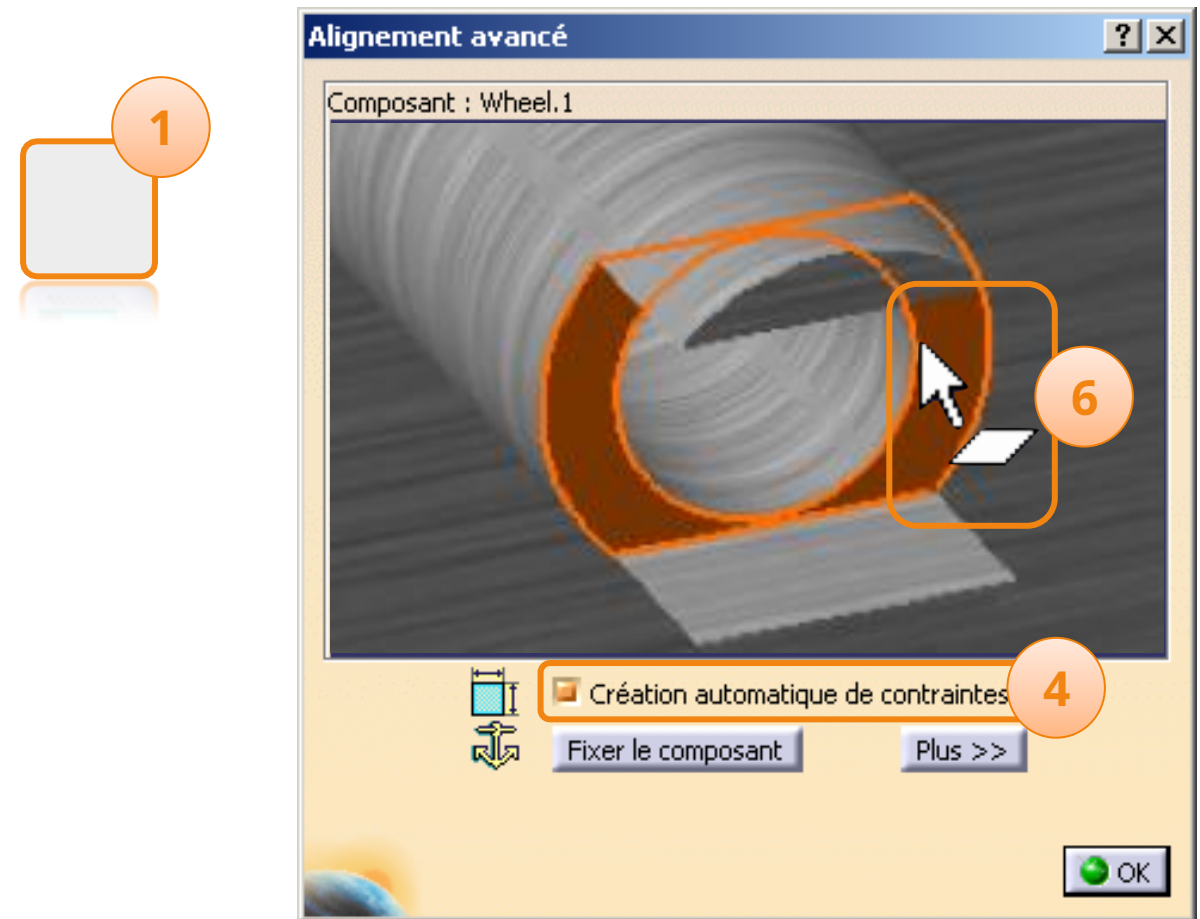
Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.**
    - i. Introduction.
    - ii. Insert the axes.
    - iii. Insert the wheels.**
  - f. Create a plan.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

# Insert the wheels

We will start by integrating the left front wheel. For this we are going to constrain the wheel in abutment on the shoulder of the front axle.

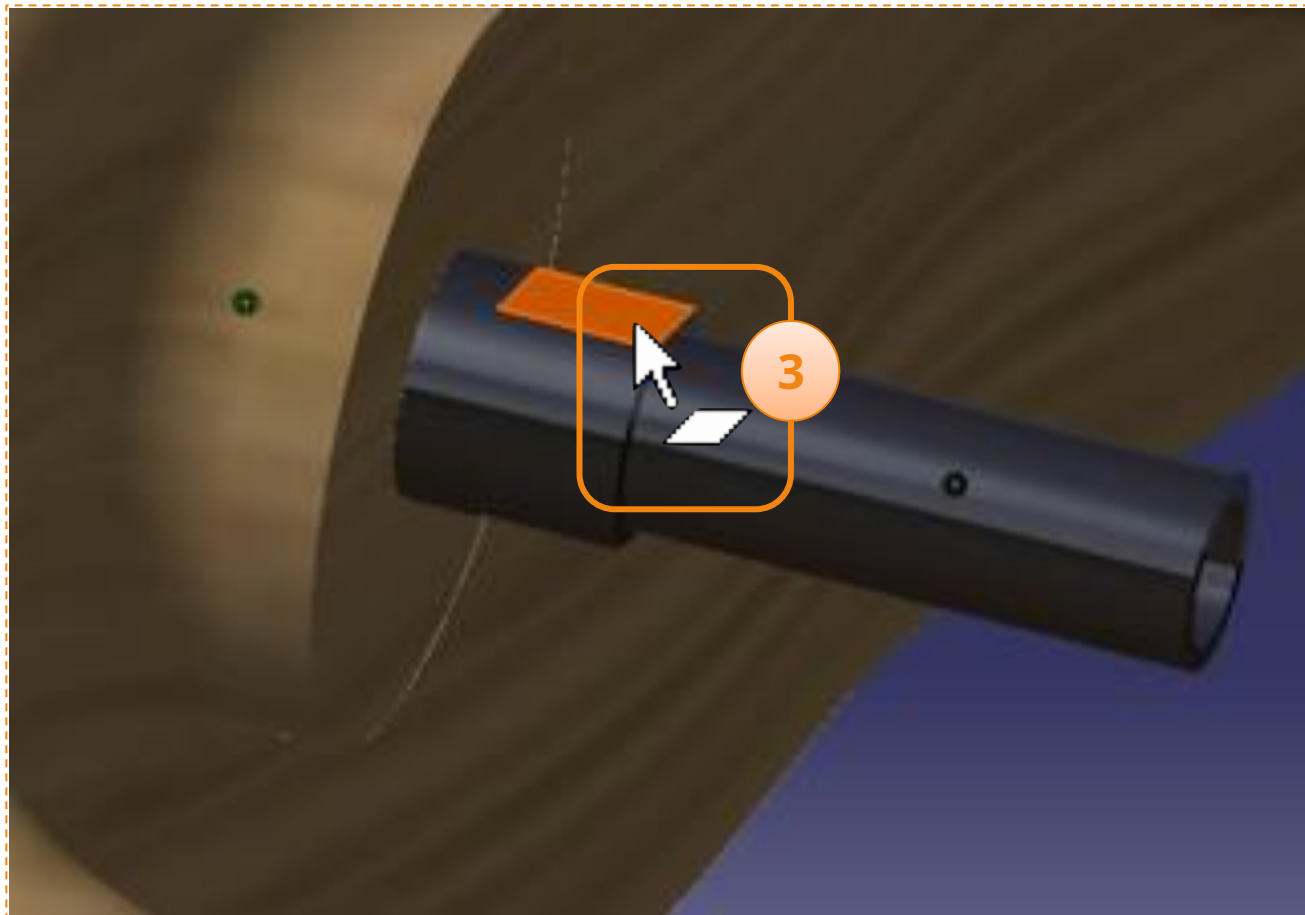
1. Click the icon **Existing component with positioning**.
2. If nothing happens, click on the product **MY\_e-RACECAR**.
3. A file selection window opens. Select file **wheel** in the directory **MiniF1e 2010 – StarterPackage**.
4. Verify that automatic constraint creation is still enabled.
5. Orient the wheel so you can see the inside.
6. Click on the **plane of the wheel**.
7. Click on the **axis plane**.
8. Click in the **blank part** of visualization.



## Insert the wheels

We are now going to force the rotation of the wheel with the axle. For that :

1. Orient the visualization of the wheel so that you can click on the map as shown opposite.
2. Click on the **plan**.
3. Click on the **corresponding plane on the axis**.
4. Click in the **blank part** of visualization.

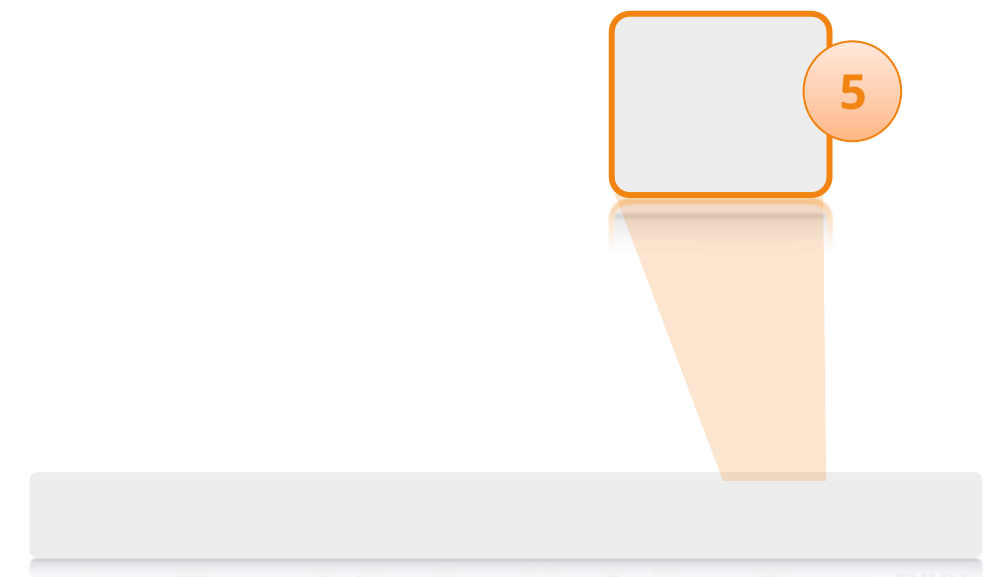
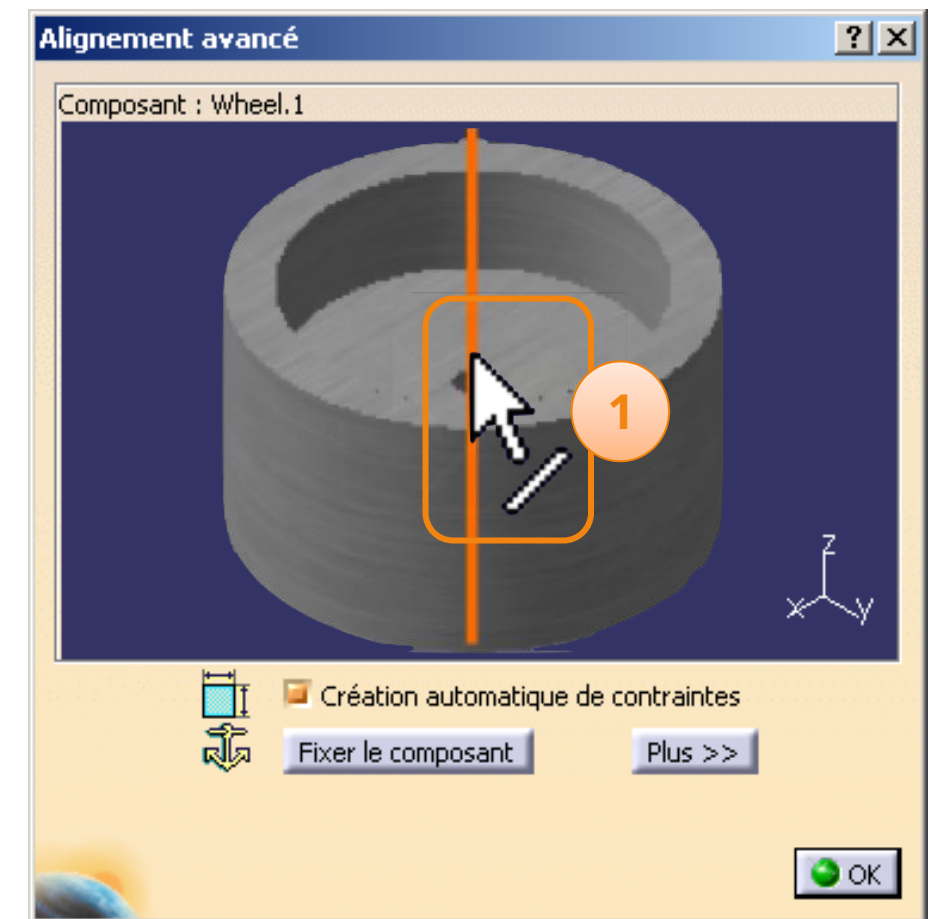
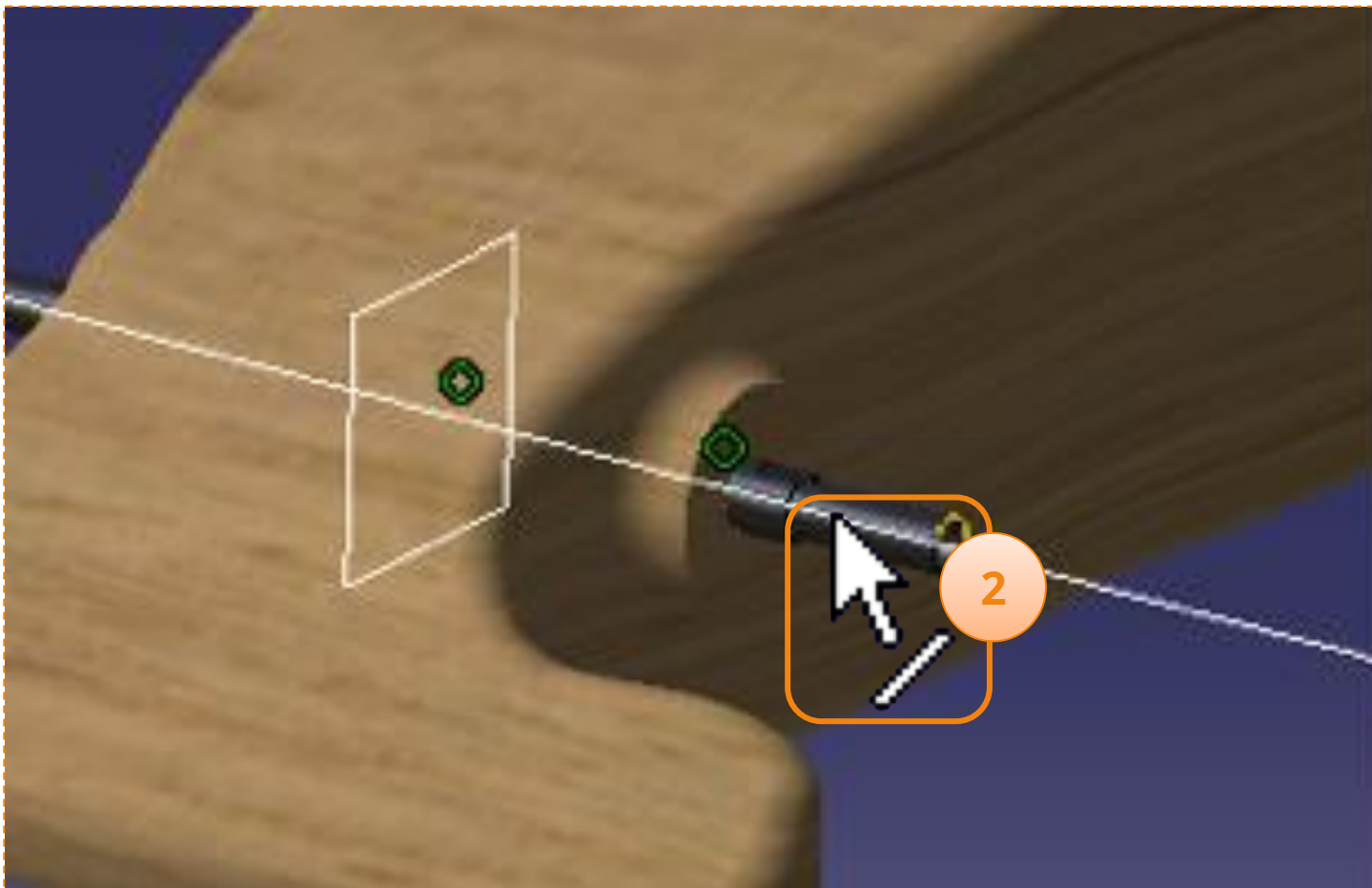




## Insert the wheels

We are now going to constrain the wheel axially with respect to the front axle of the car. For that :

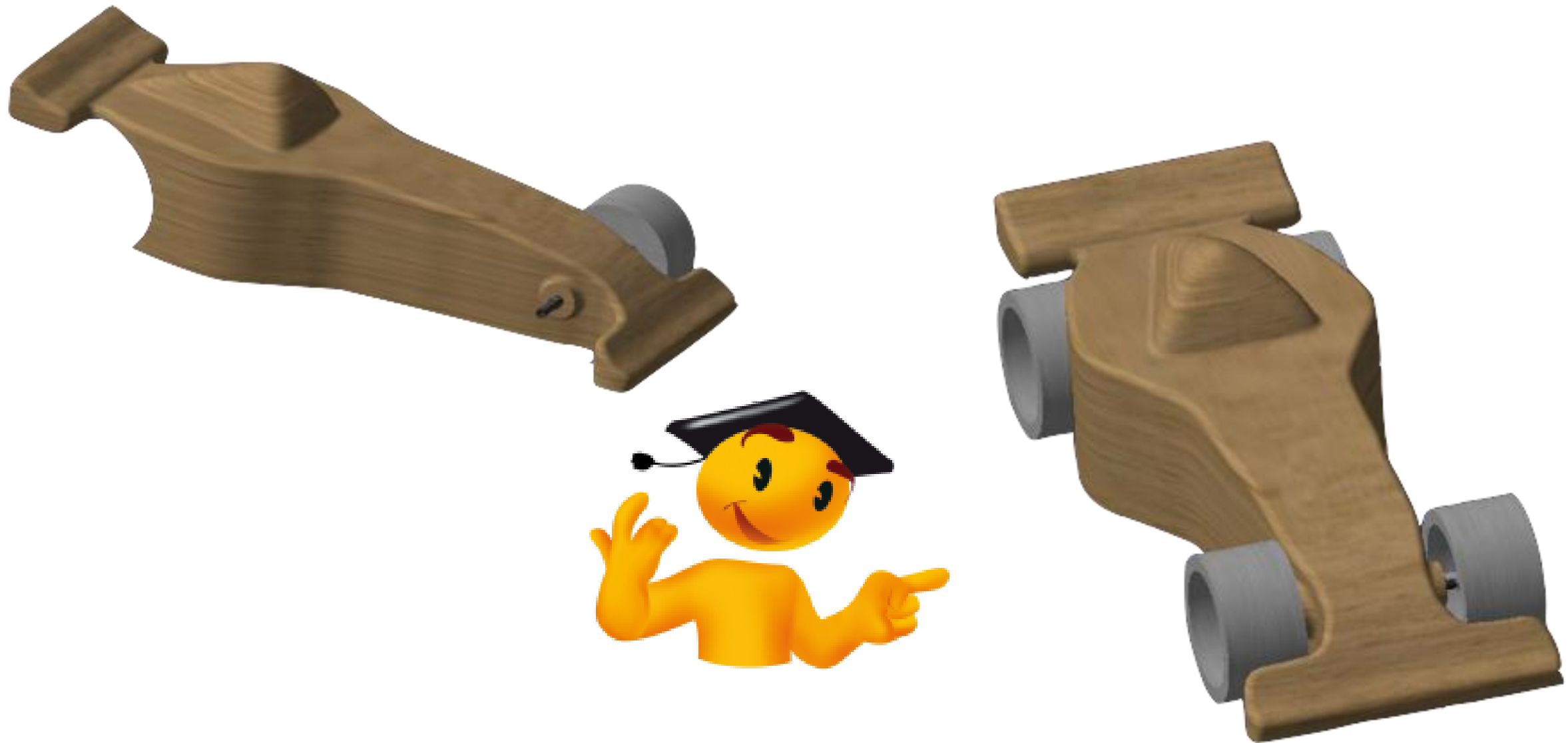
1. Click the **wheel axle**.
2. Click the **car front axle**.
3. Click in the **blank part** of visualization.
4. Click on **okay**.
5. Click on **Edit > Update**. Or on the icon of **update**.



## Insert the wheels

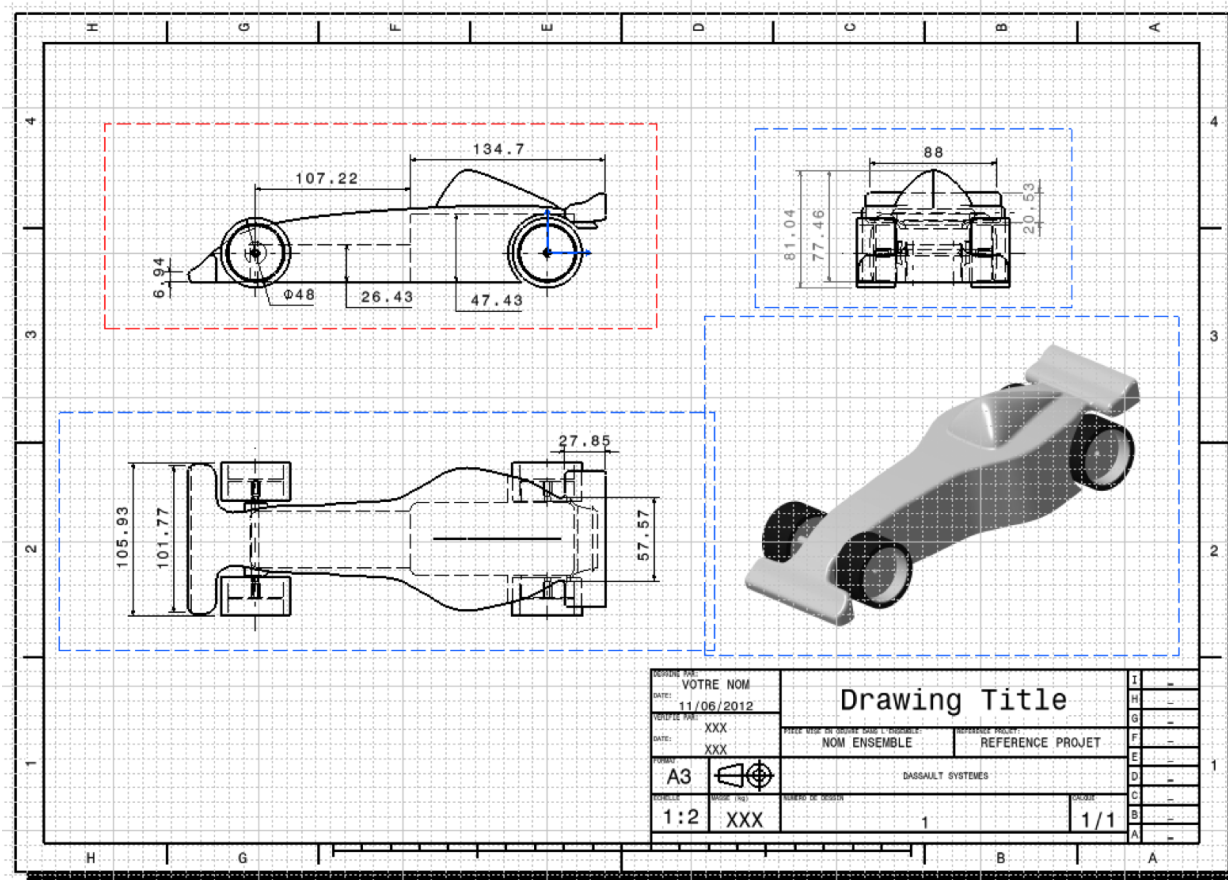
You have inserted the left front wheel of the car.

1. Use the same method to insert the other three wheels, first displaying the motor in **Rules > HPP\_ElecMotor** (Right-click then **Hide/Show**).



# Create a drawing

In this part we will see how to draw up your digital model. This will allow you to check and list the dimensions of the specifications.



Here are the steps to follow:

1. Introduction.

2. CATIA V5 QuickStart.

**3. Your racing car.**

has. Draw your car.

b. Open your project.

vs. Create an immersive sketch.

d. Model your car.

e. Assemble your car.

**f. Create a drawing.**

**i. Open a drawing.**

ii. Personalize your drawing.

iii. Define the main view.

iv. Define projected views.

v. Quoting.

vi. Insert a realistic view.

g. Control and modify your car.

h. Create realistic renderings.

i. Simulate machining.

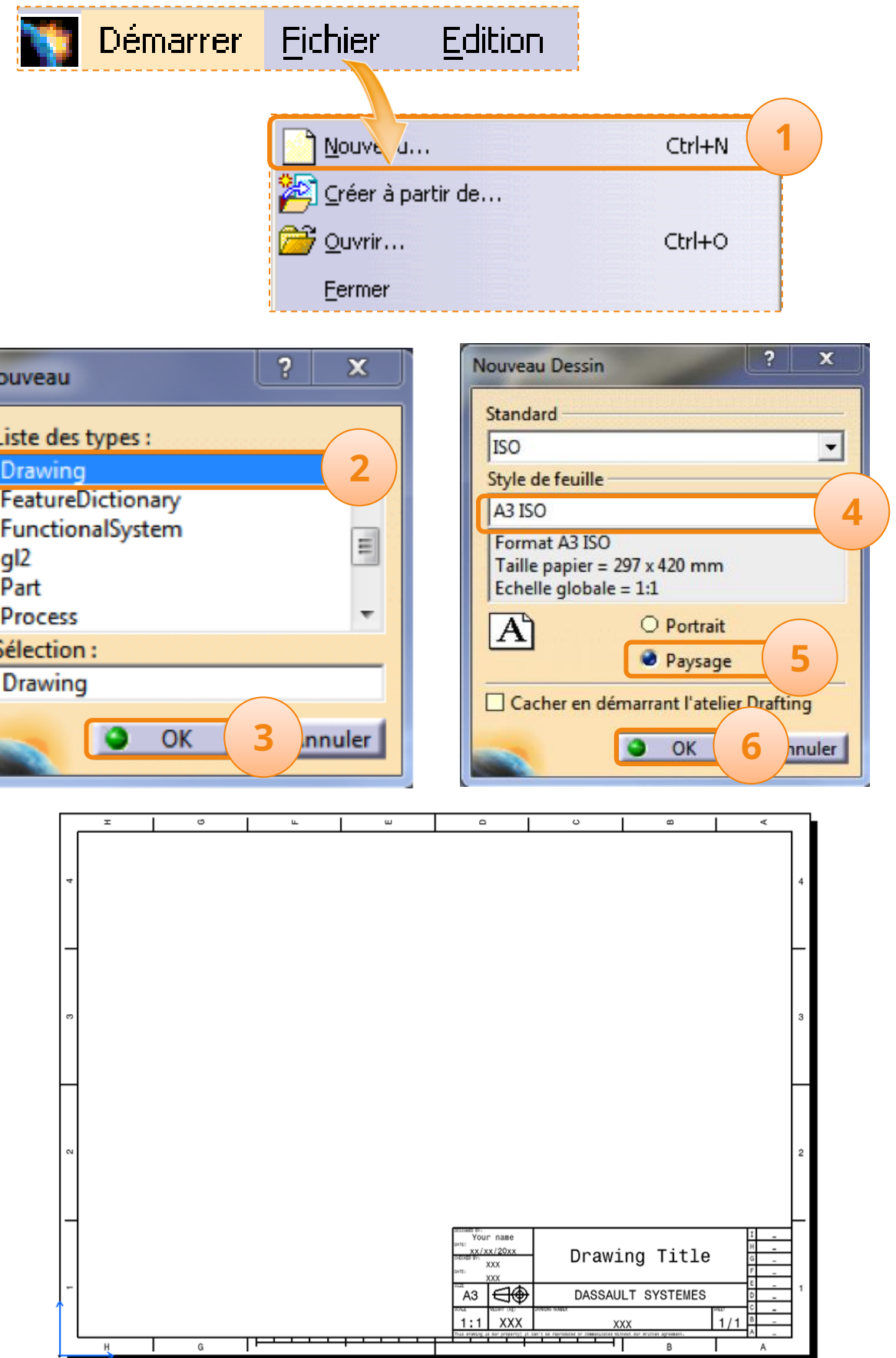
4. Method sheets.

# Open a drawing

We are going to use a drawing sheet with a title block. For that :

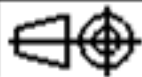
1. Click on **File** > **New....**
2. Select **drawing** in the window that appears.
3. Validate by clicking on **okay**.
4. Define **A3 ISO** as a sheet style.
5. Also change the orientation to **Landscape**.
6. Validate by clicking on **okay**.

The drawing sheet opposite opens:



# Customize your drawing

We are going to configure the drawing, by choosing the projection mode, and by modifying the text in the title block.

DESIGNED BY: Your name		Drawing Title		I	-
DATE: XX/XX/20XX				H	-
CHECKED BY: XXX				G	-
DATE: XXX				F	-
SIZE A3		DASSAULT SYSTEMES		E	-
SCALE	WEIGHT (kg)	OBJECT NUMBER	SHEET	D	-
1:1	XXX	XXX	1/1	C	-
This drawing is our property; it can't be reproduced or communicated without our written agreement.				B	-
				A	-



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a drawing.**
    - i. Open a drawing.
    - ii. Personalize your drawing.**
    - iii. Define the main view.
    - iv. Define projected views.
    - v. Quoting.
    - vi. Insert a realistic view.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.



# Customize your drawing

Now we are going to modify the text present in the title block. For that :

1. Click on **Edit > Layer Background**.

The background changes from white to gray. Now by double-clicking on a text, you can modify it.

Example :

2. Double-click **Drawing Title**.

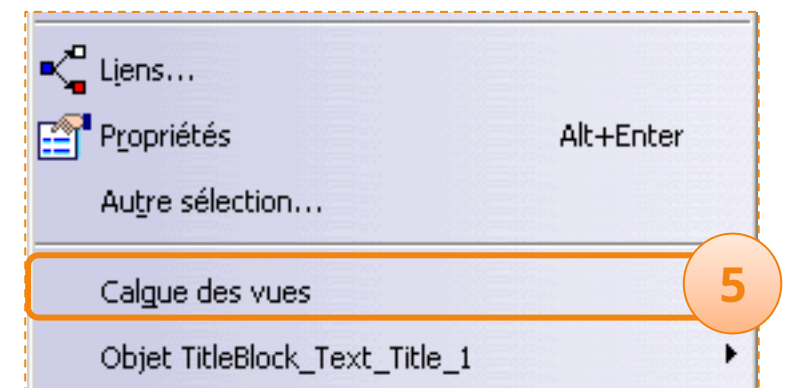
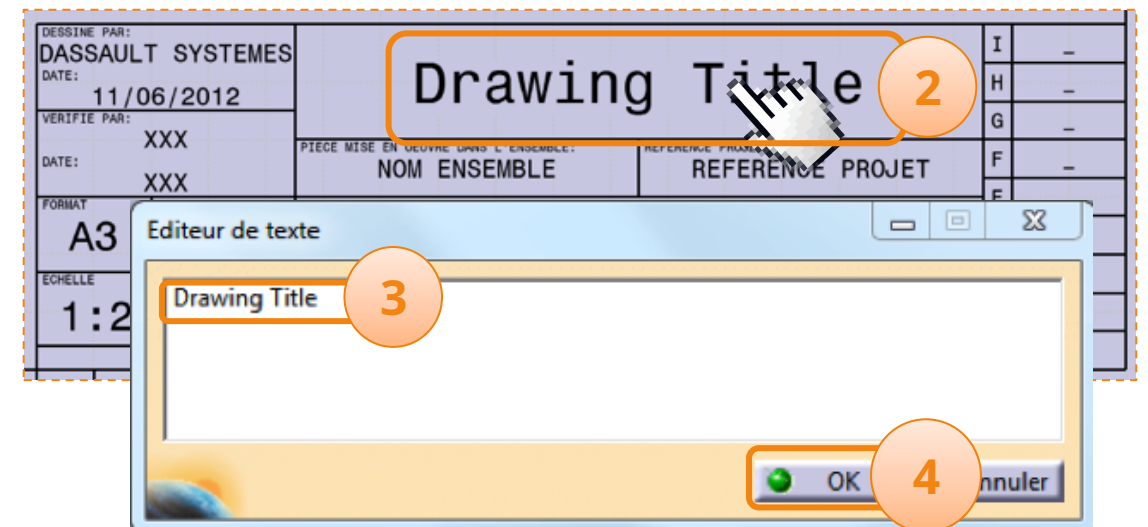
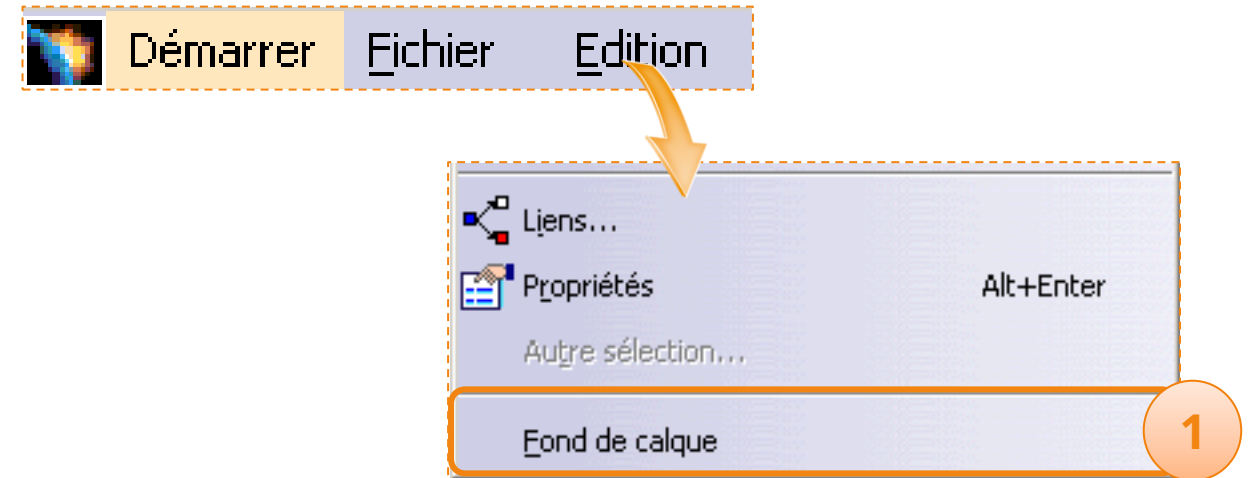
3. Enter the title of your drawing.

4. Click on **okay**.

Modify the elements of the title block to your liking.

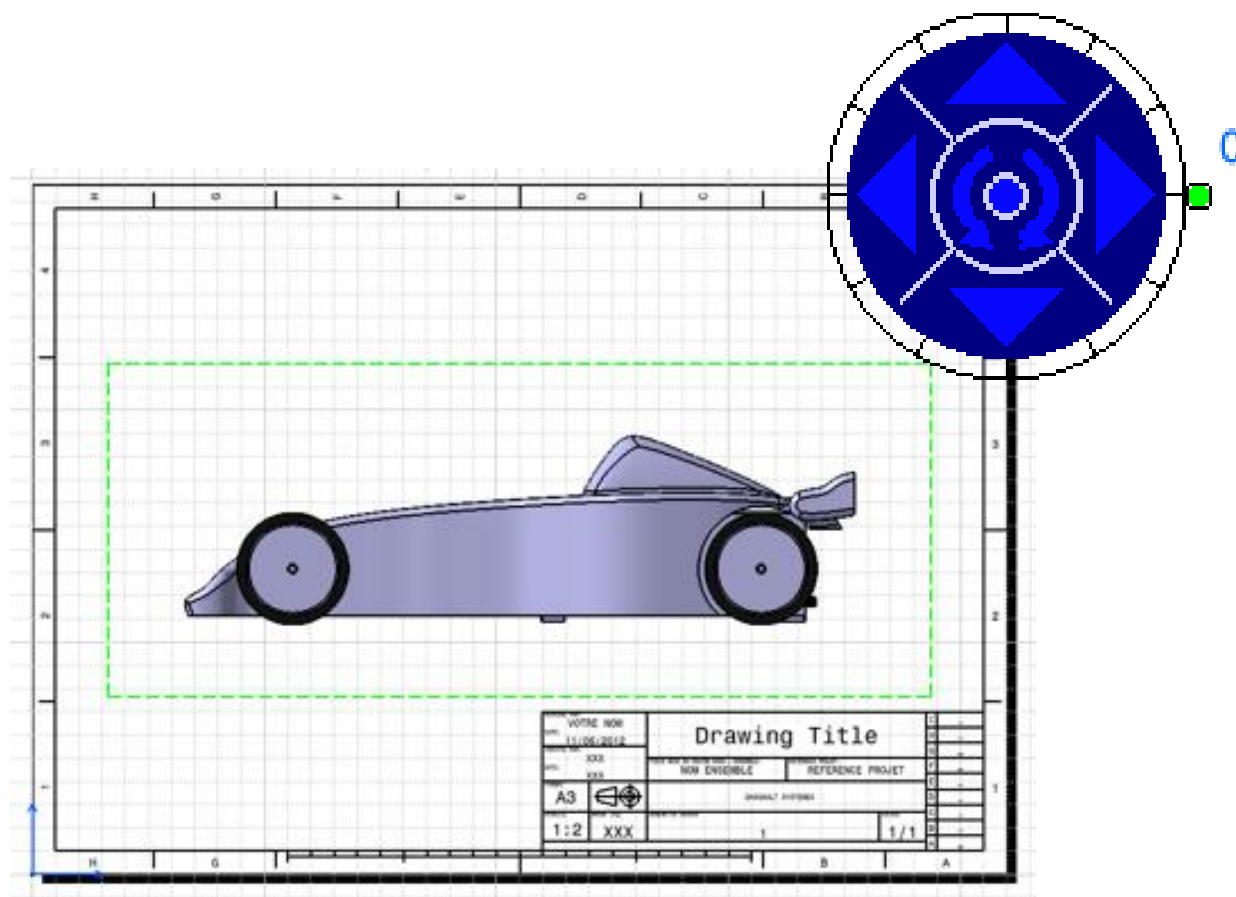
5. To quit the "Layer background" mode, click on **Edit > View Layer**.

You have configured your drawing, we can start drawing.



# Define the Main View

We will start by inserting the main view of our drawing. We will use the side view as the main view.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. *Your racing car.*
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. *Create a drawing.*
    - i. Open a drawing.
    - ii. Personalize your drawing.
    - iii. *Define the main view.*
    - iv. Define projected views.
    - v. Quoting.
    - vi. Insert a realistic view.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

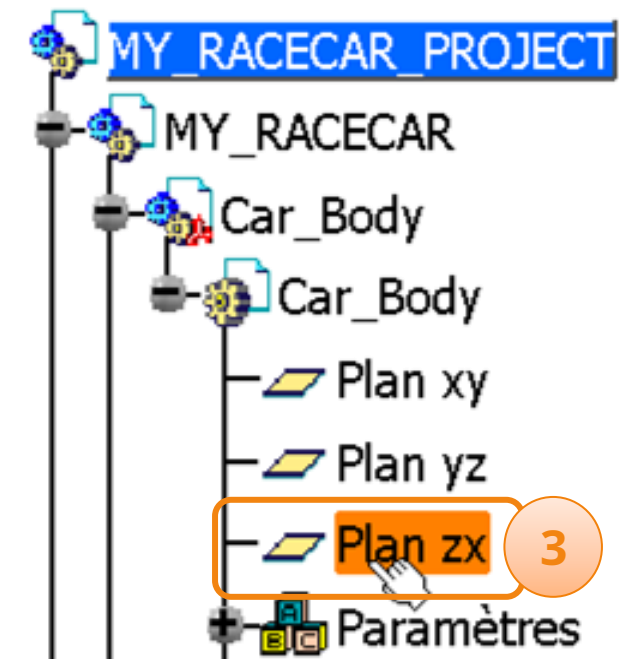
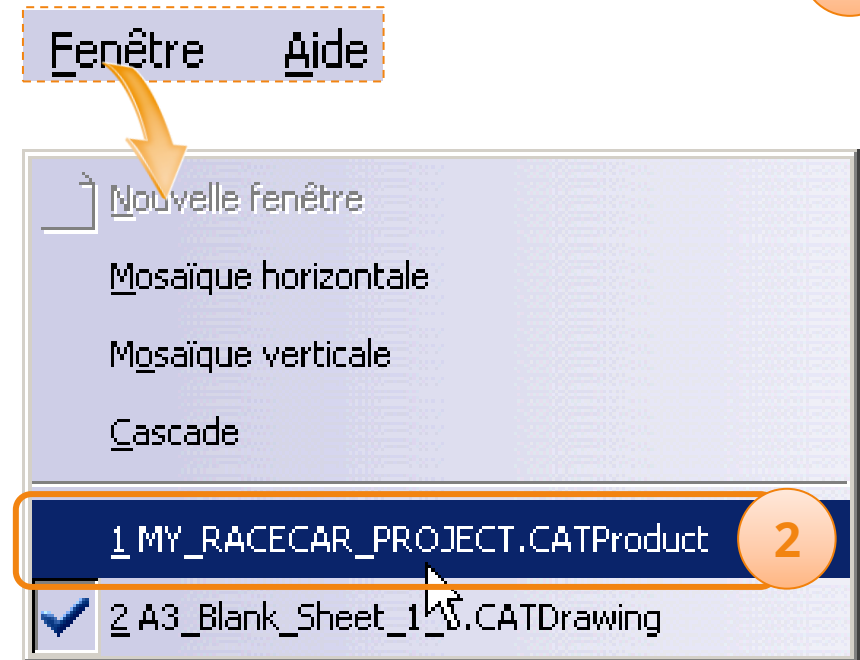
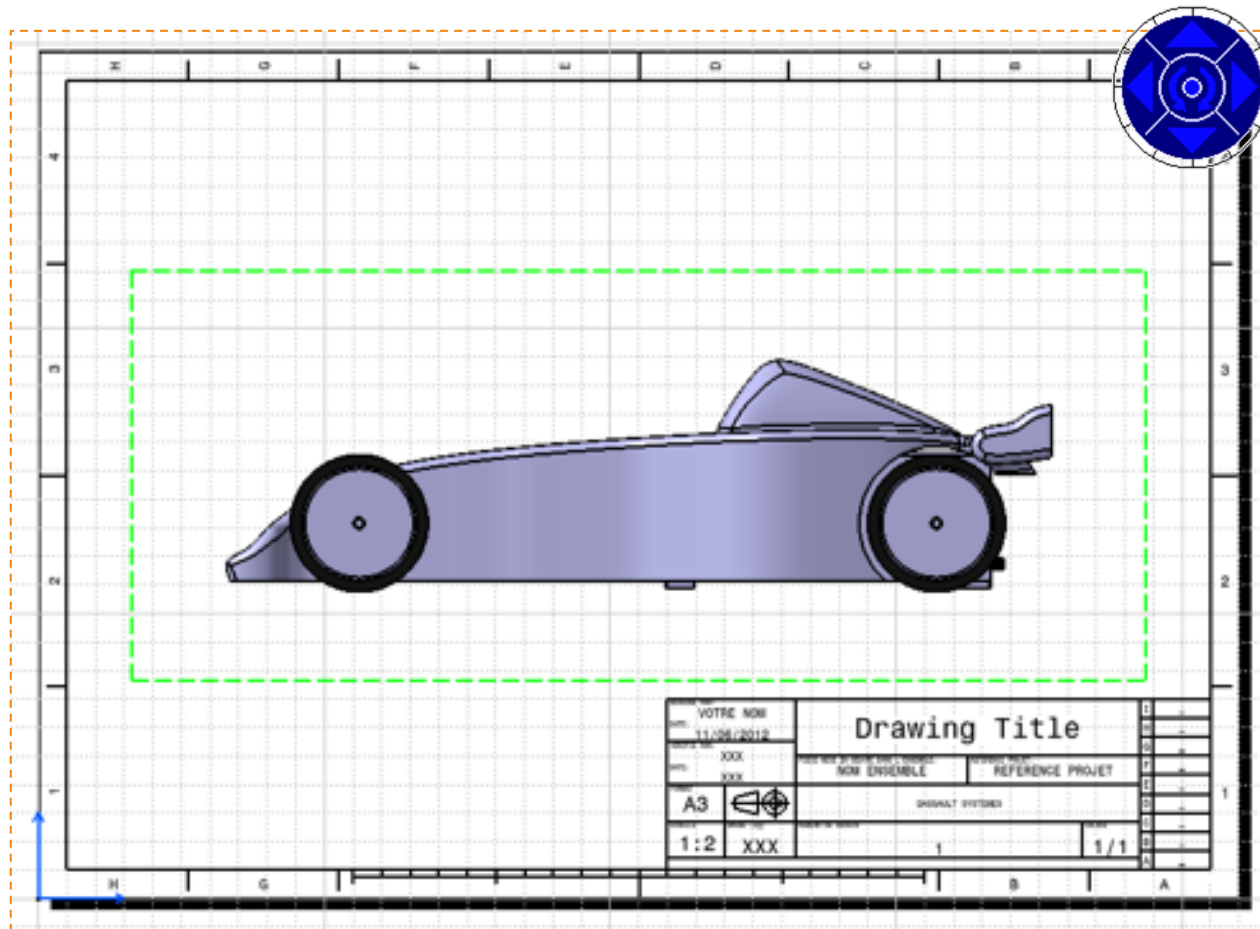
# Define the main view

The main view serves as a reference to create the other views. To set it:

1. Click the icon **Front view**.
2. Then in the menu bar, click on **Window > MY\_e-RACECAR\_PROJECT.CATProduct**.

You will display the assembly of the car.

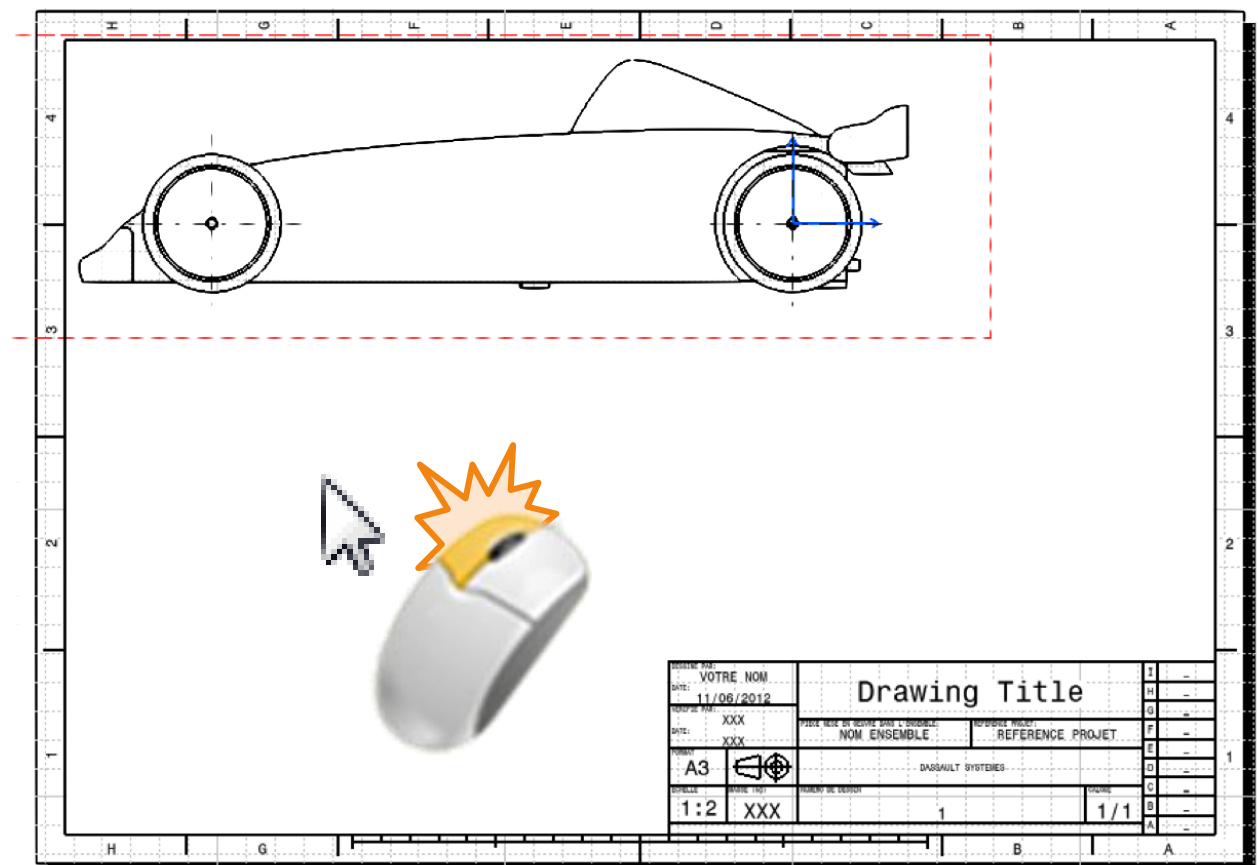
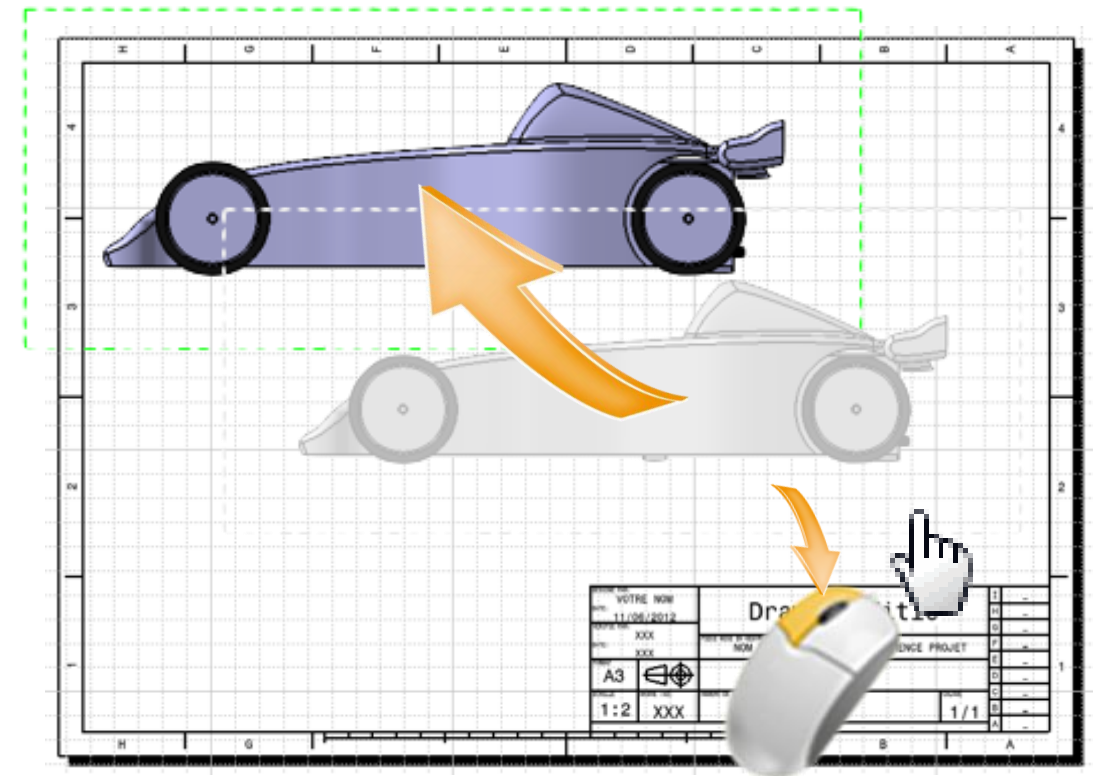
3. Click on the **ZX plane**, which cuts the car in half.  
This will open the drawing again with a preview of your car as below.



# Define the main view

Now we will place the car as below. For that :

1. Click and hold the green frame surrounding the car, then move the view to the desired location.
2. Click outside the green box to validate the drawing of the car.

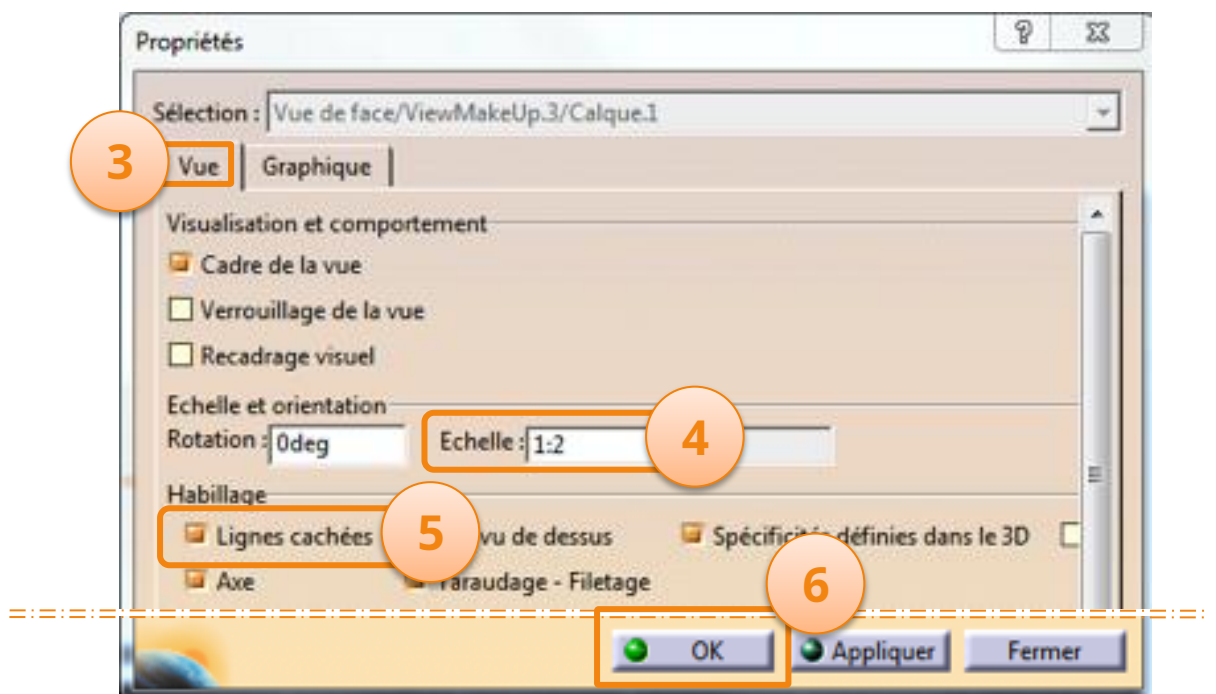
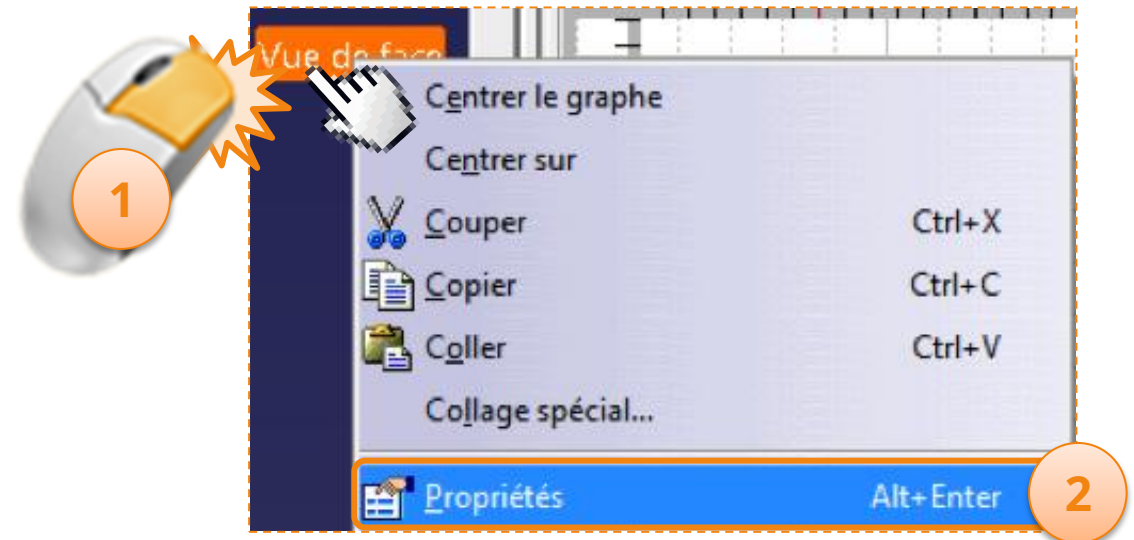




# Define the main view

Finally, you will modify the properties of the view. For that :

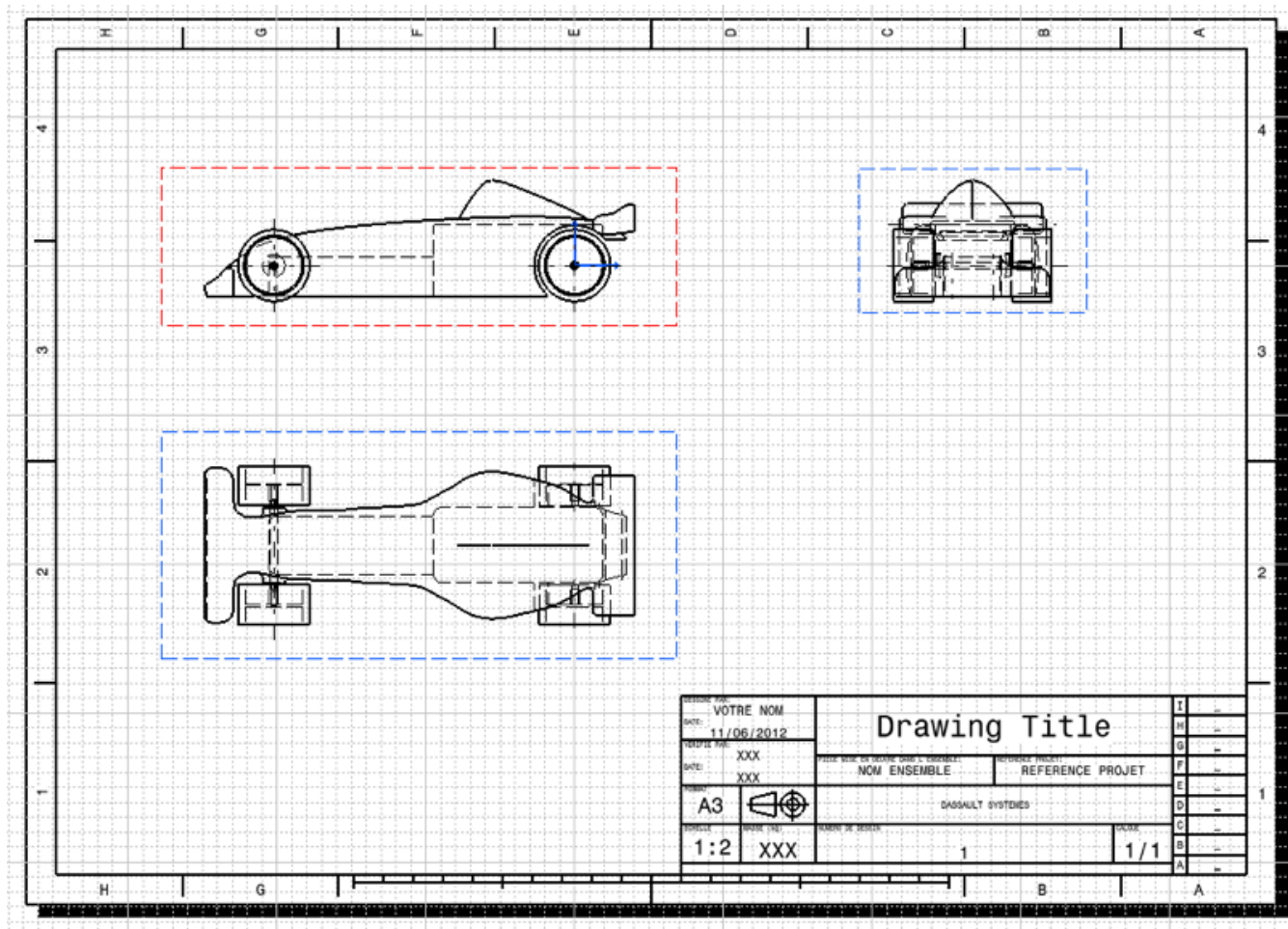
1. Right click on **Front view**.
2. Select **Properties**.
3. Select tab **View**.
4. Change the scale to [1:2].
5. You can also, for example, display the **hidden lines**.
6. Click on **okay** to validate.





# Define projected views

We will now define the auxiliary views from the side view. In this way we will add the top view, and the front view.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. *Your racing car.*
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. *Create a drawing.*
    - i. Open a drawing.
    - ii. Personalize your drawing.
    - iii. Define the main view.
    - iv. *Define projected views.*
    - v. Quoting.
    - vi. Insert a realistic view.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

# Define projected views

Auxiliary views are defined by main ra.  
To set the top view

1. Click on **the little black arrow face**.

e



2. Click the icon **Projected view**.

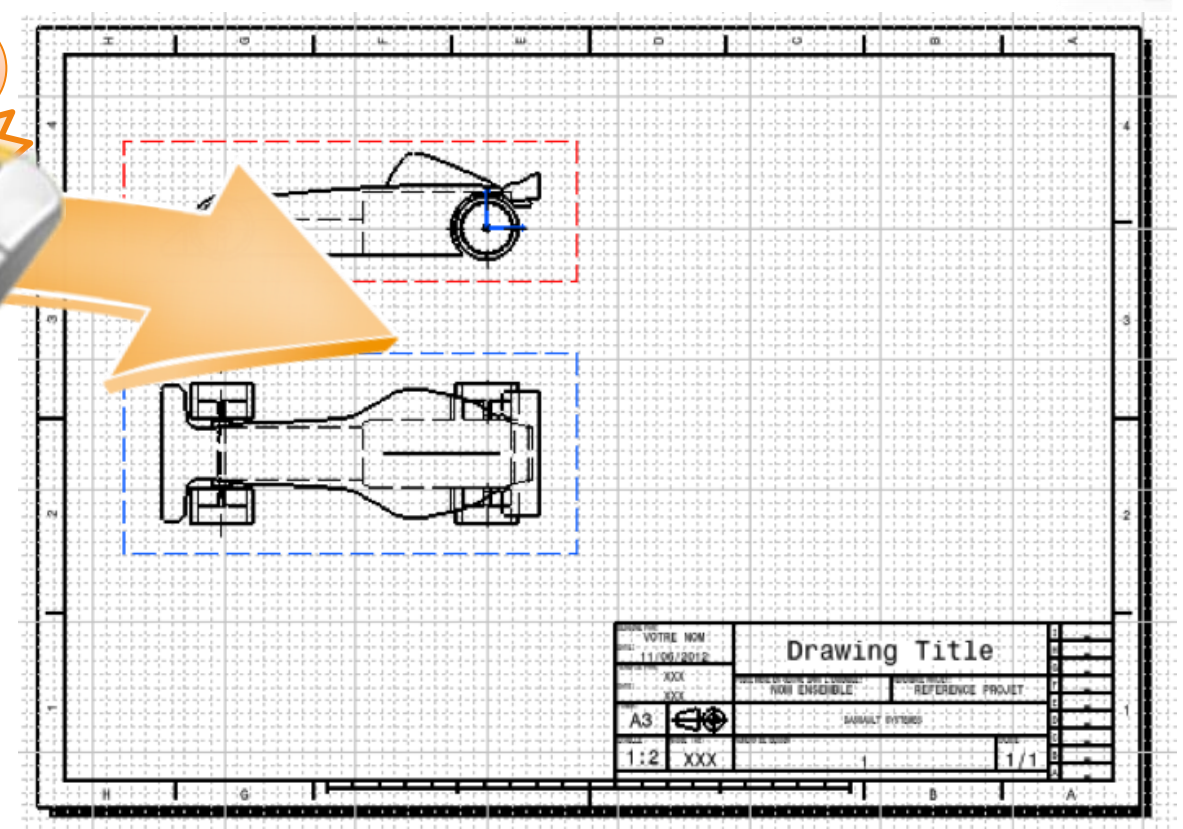
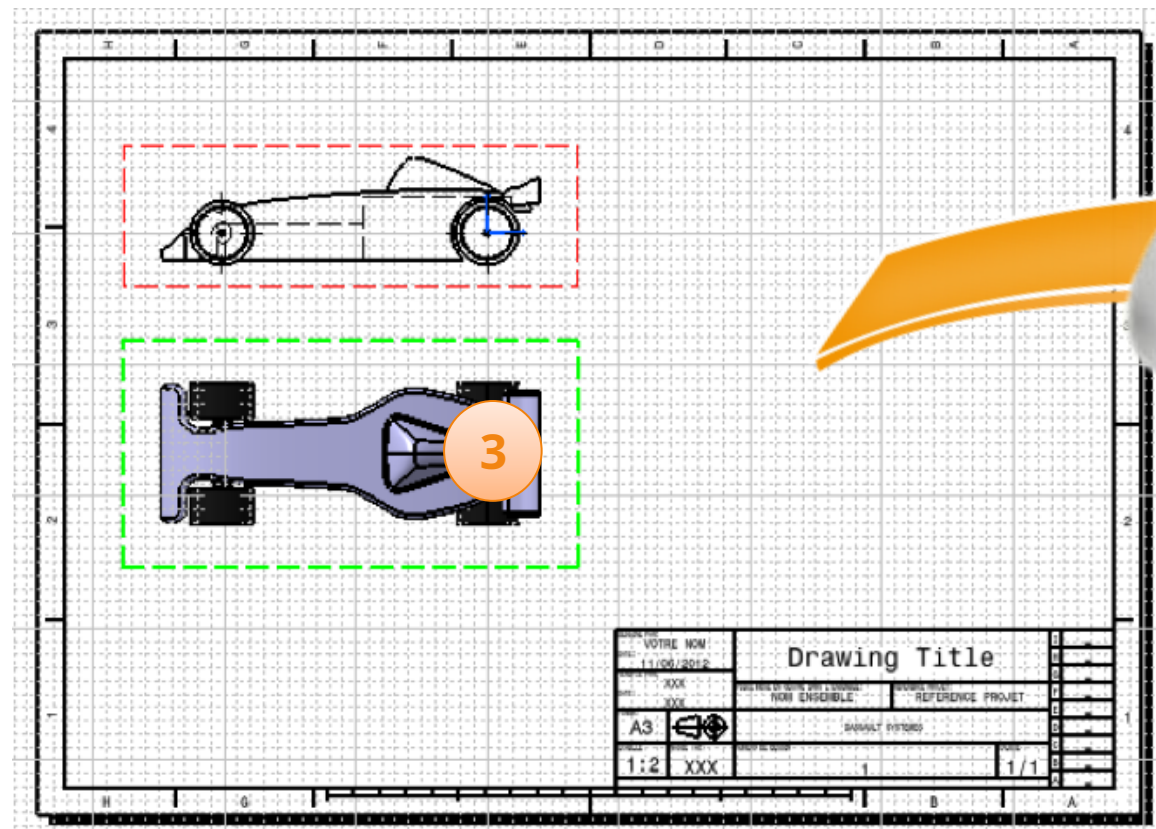
3. Move your mouse below you can view  
the preview of

e,

.

4. Click to insert the drawing of the

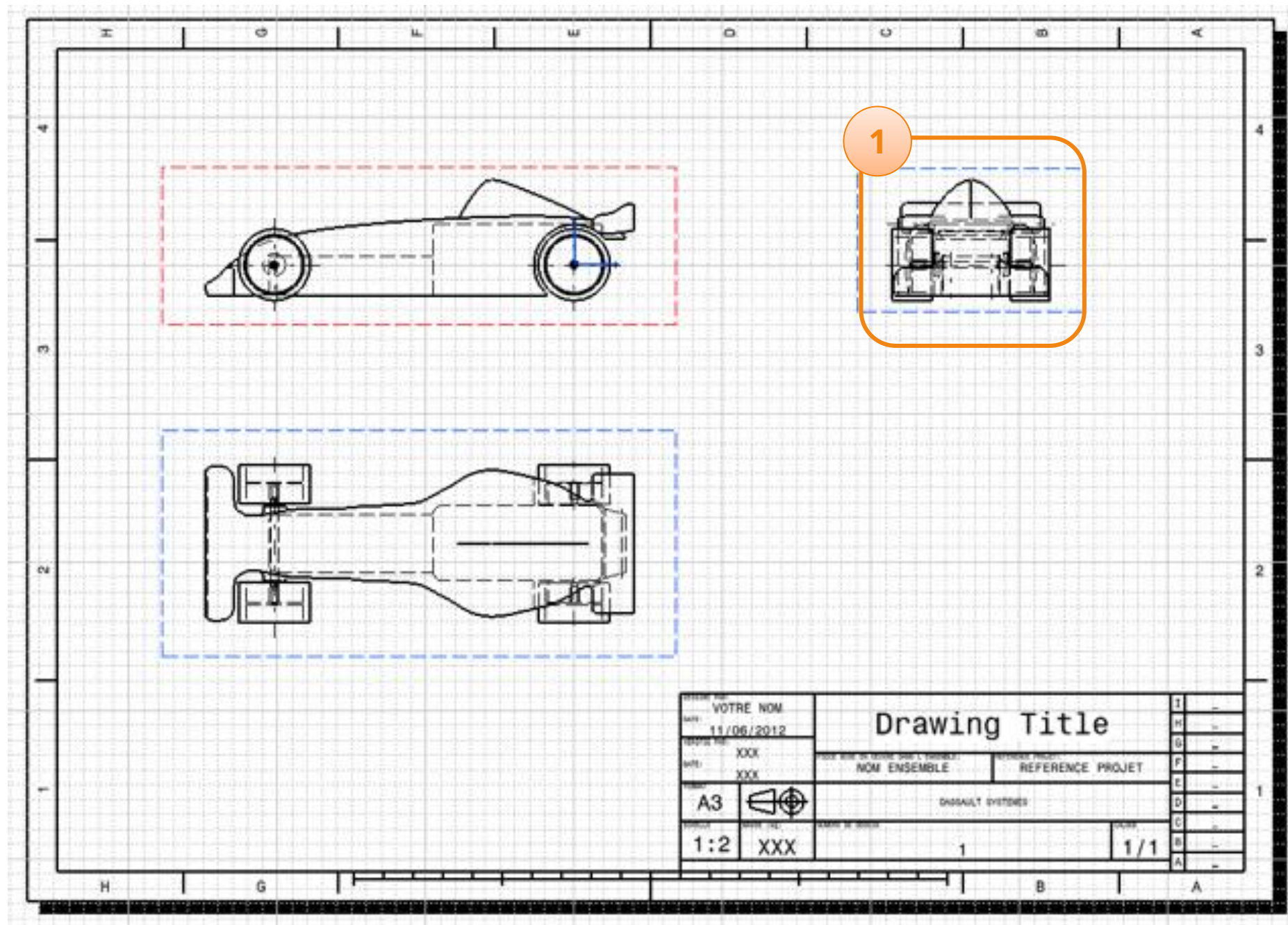
.





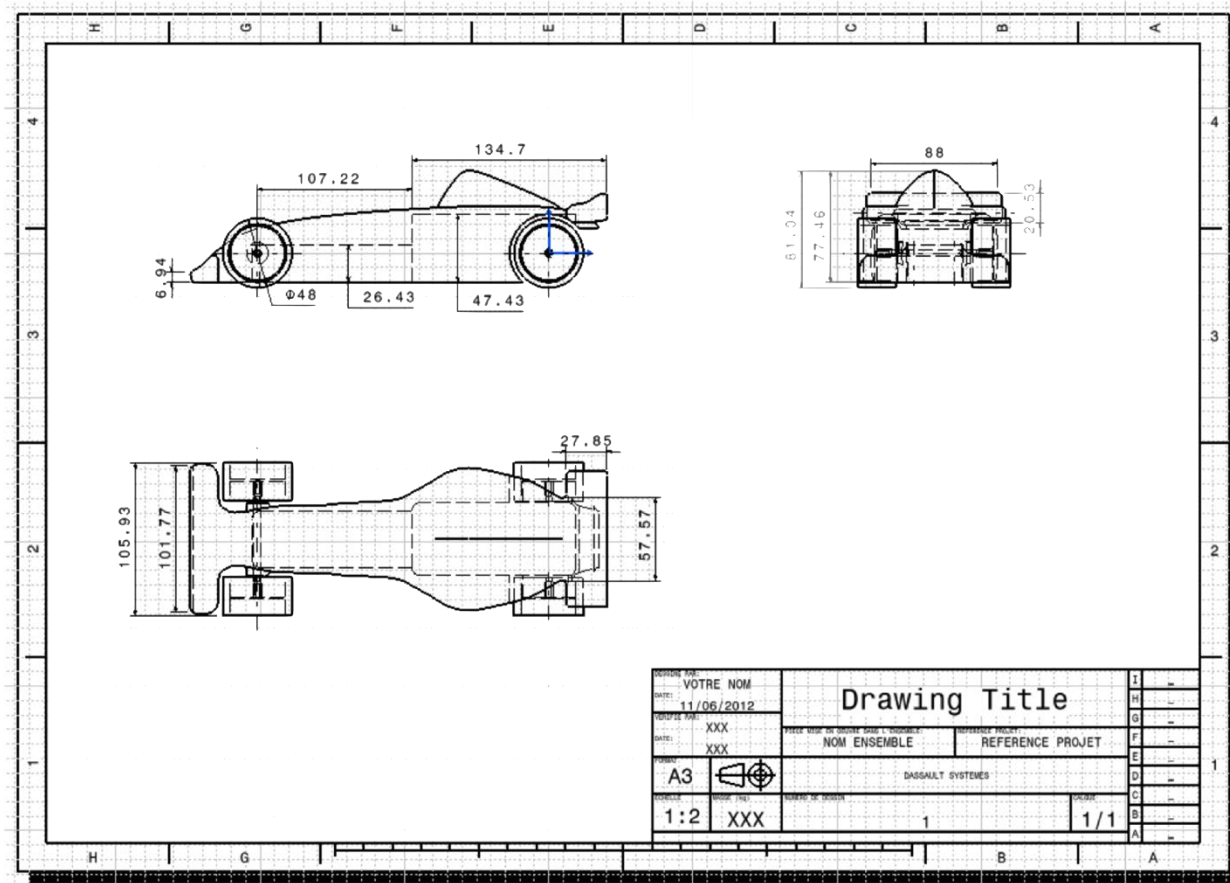
# Define projected views

1. You can now insert the front view using the same method.



# Quoting

We are going to finish the drawing by adding some dimensions.



Here are the steps to follow:

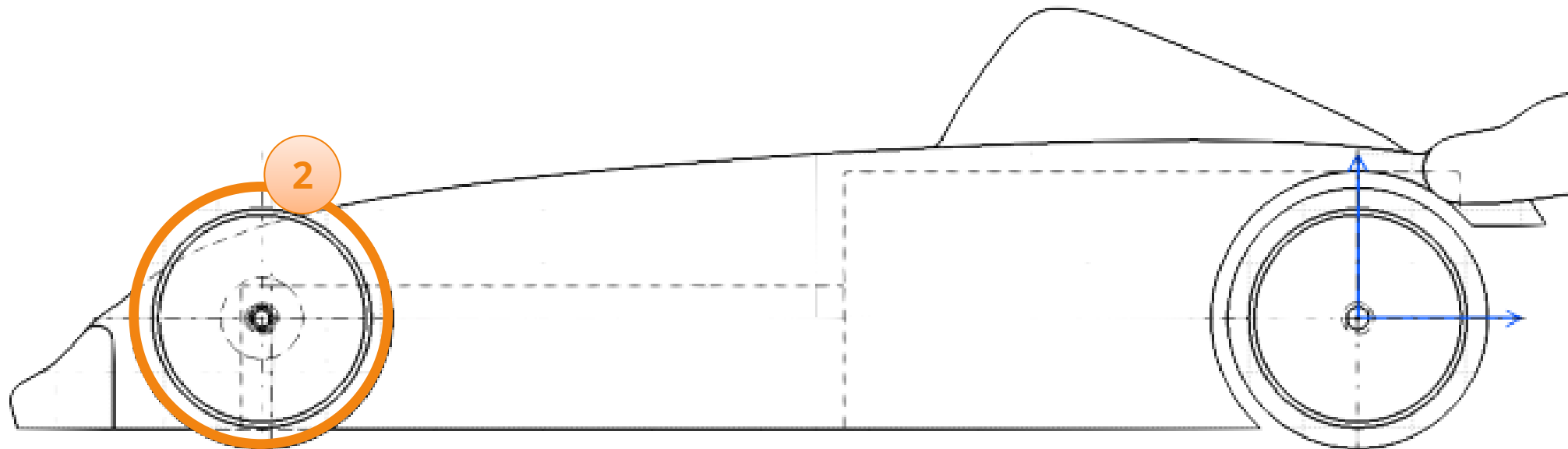
1. Introduction.
2. CATIA V5 QuickStart.
3. *Your racing car.*
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. *Create a drawing.*
    - i. Open a drawing.
    - ii. Personalize your drawing.
    - iii. Define the main view.
    - iv. Define projected views.
  - v. *Quoting.*
    - vi. Insert a realistic view.
  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

# Quoting

We will start by creating the dimensions on the side view.  
For the diameter of a wheel:

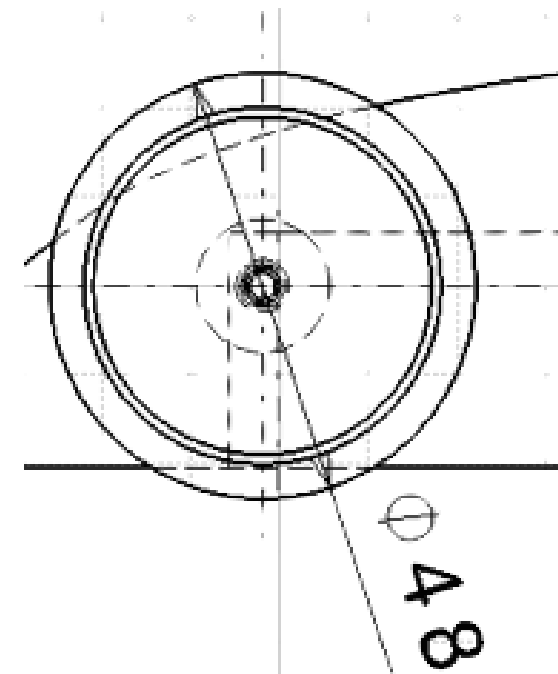
1. Click the icon **Quotes**.

2. VS



A dimension of the diameter type appears, you can move it by moving the mouse. When the location suits you:

3. Click to fix it on the drawing.





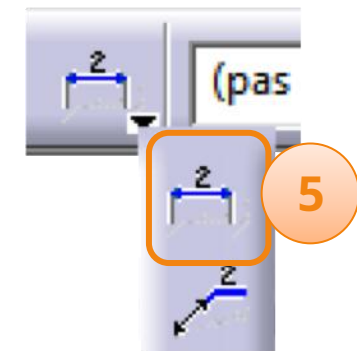
# Quoting

Let's continue the dimensioning by the depth of the housing of the propulsion system. For that :



1. Click on the little arrow icon **Quotes** to bring up other icons.

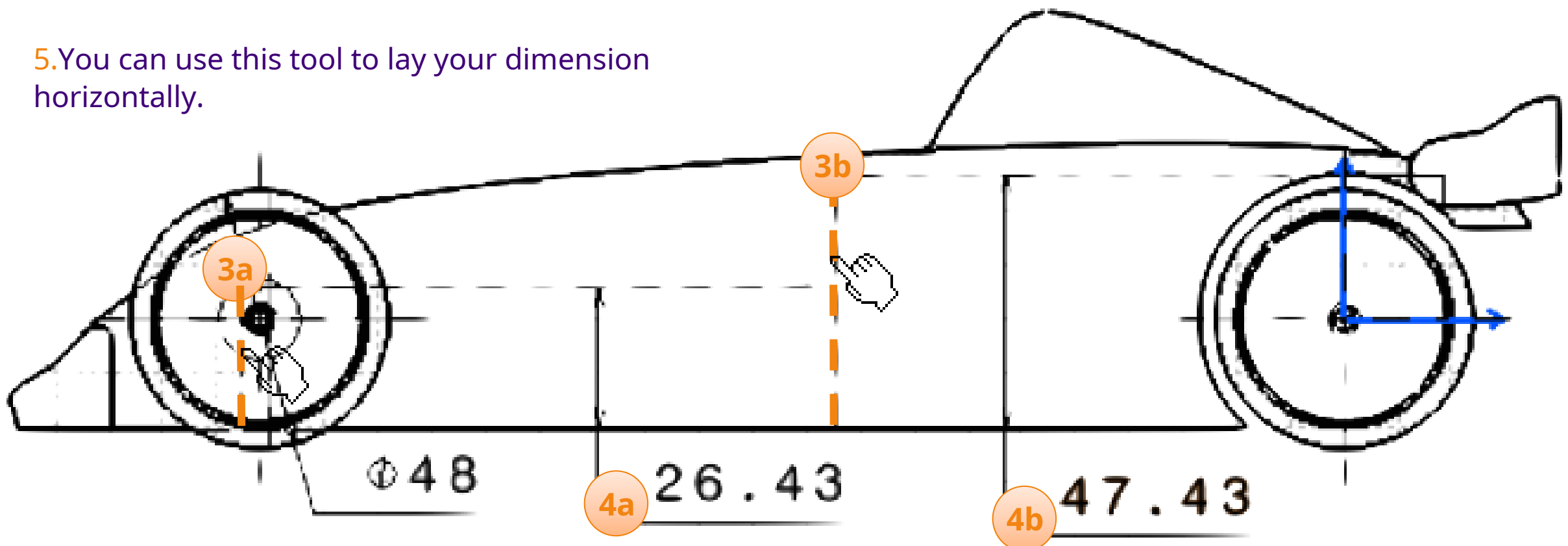
2. Click the icon **Dimensioning of diameters**.



3. Click on the edges below.

4. Click to fix the dimensions at the desired places.

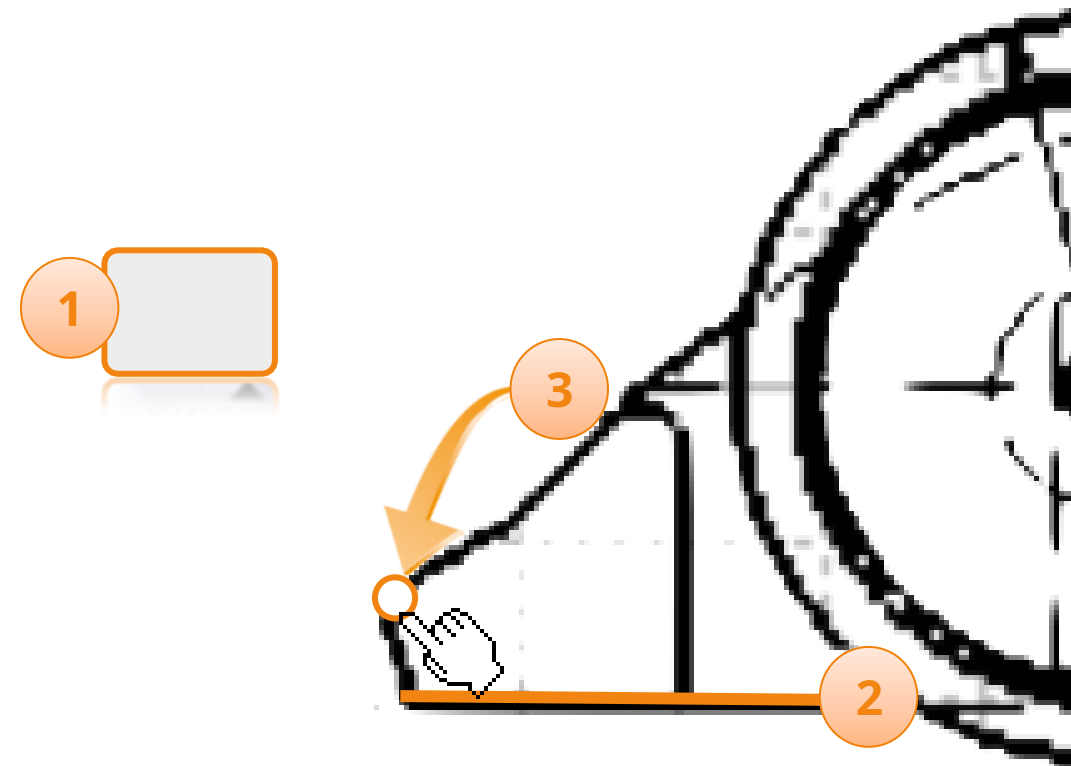
5. You can use this tool to lay your dimension horizontally.



# Quoting

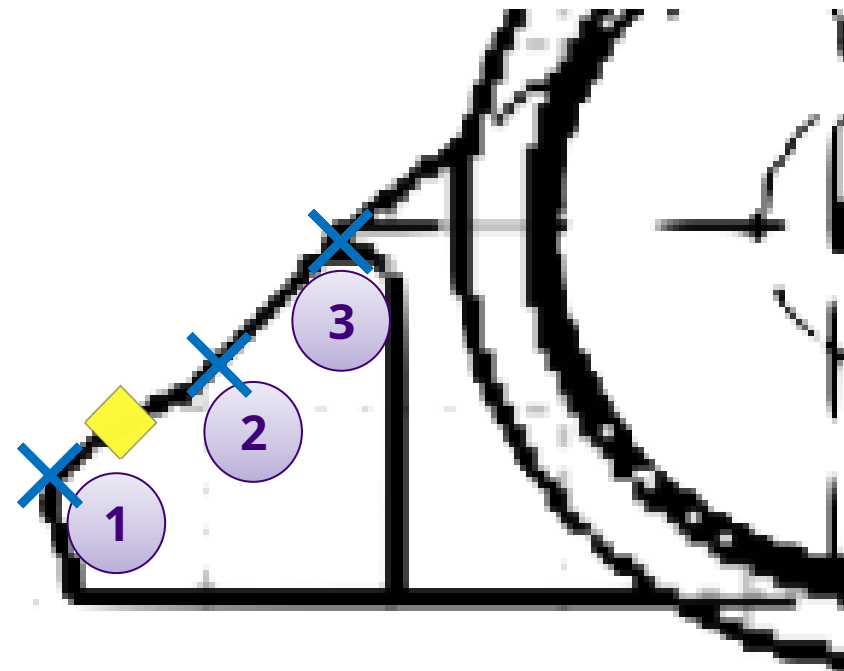
We will now see how to place a dimension between two elements. We will take the example of the minimum thickness of the front wing.

1. Click the icon **Quotes**.
2. Click on the lower part of the aileron.
3. Click on the upper part of the fin, where you think it is thinnest.



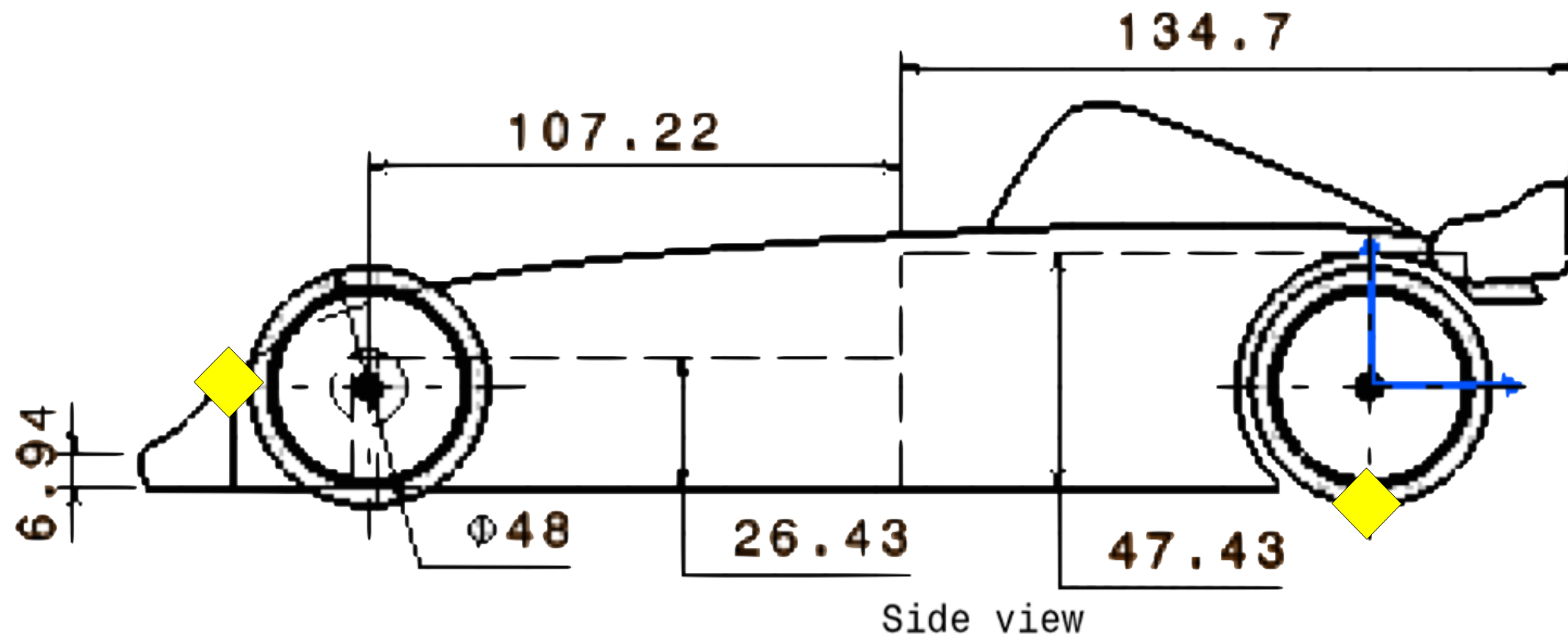
A yellow diamond appears telling you that the anchor point on the top of the fin is here.

4. If the anchor point is not the correct one, hold down the key **<CTRL>** pressed to show the different anchor points (blue cross).
5. Move the yellow diamond (Drag and Drop) to point 1.
6. Click again to place the dimension.









## Quoting

8. You are now able to complete the side view, as shown below.



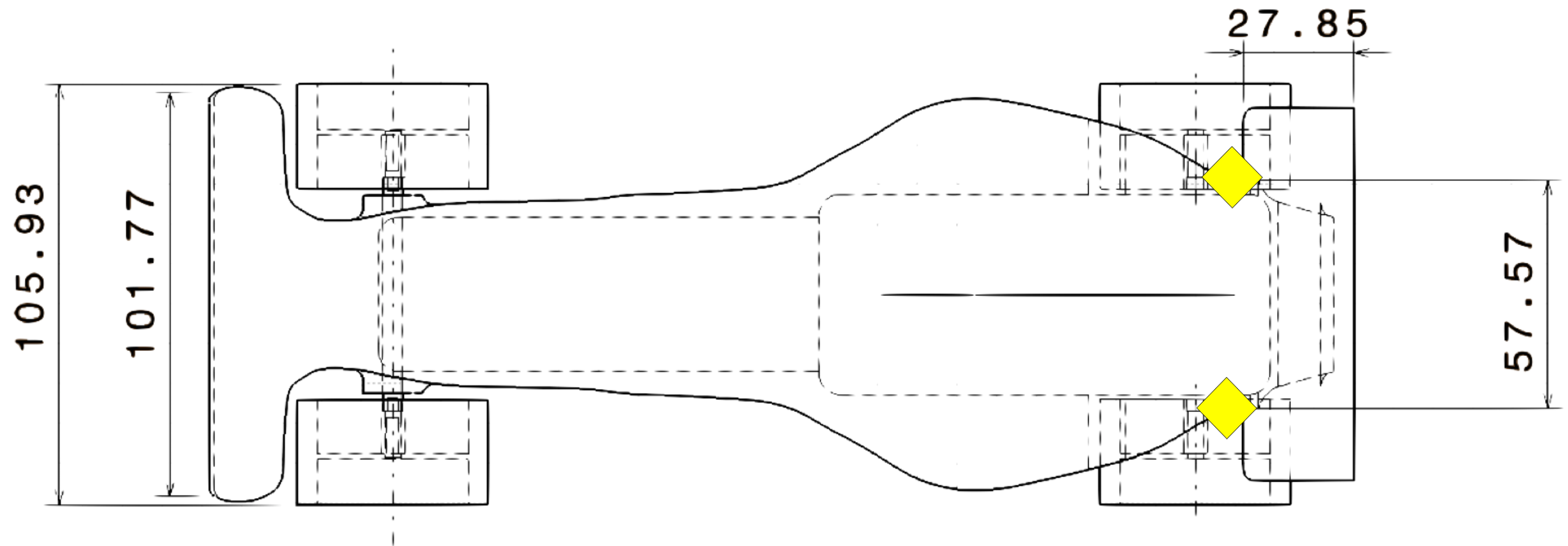
◆ Dimension requiring the use of anchor points.

When creating a dimension between several elements, the direction may not be correct. In this case, right-click then **Representation of the dimension**, then choose from the options opposite.

-  Cote projetée
-  Forcer une cote sur l'élément
-  Forcer une cote horizontale dans la vue
-  Forcer une cote verticale dans la vue
-  Forcer une cote selon direction
-  Cote vraie grandeur

# Quoting

9. Continue to complete the top view as shown below.

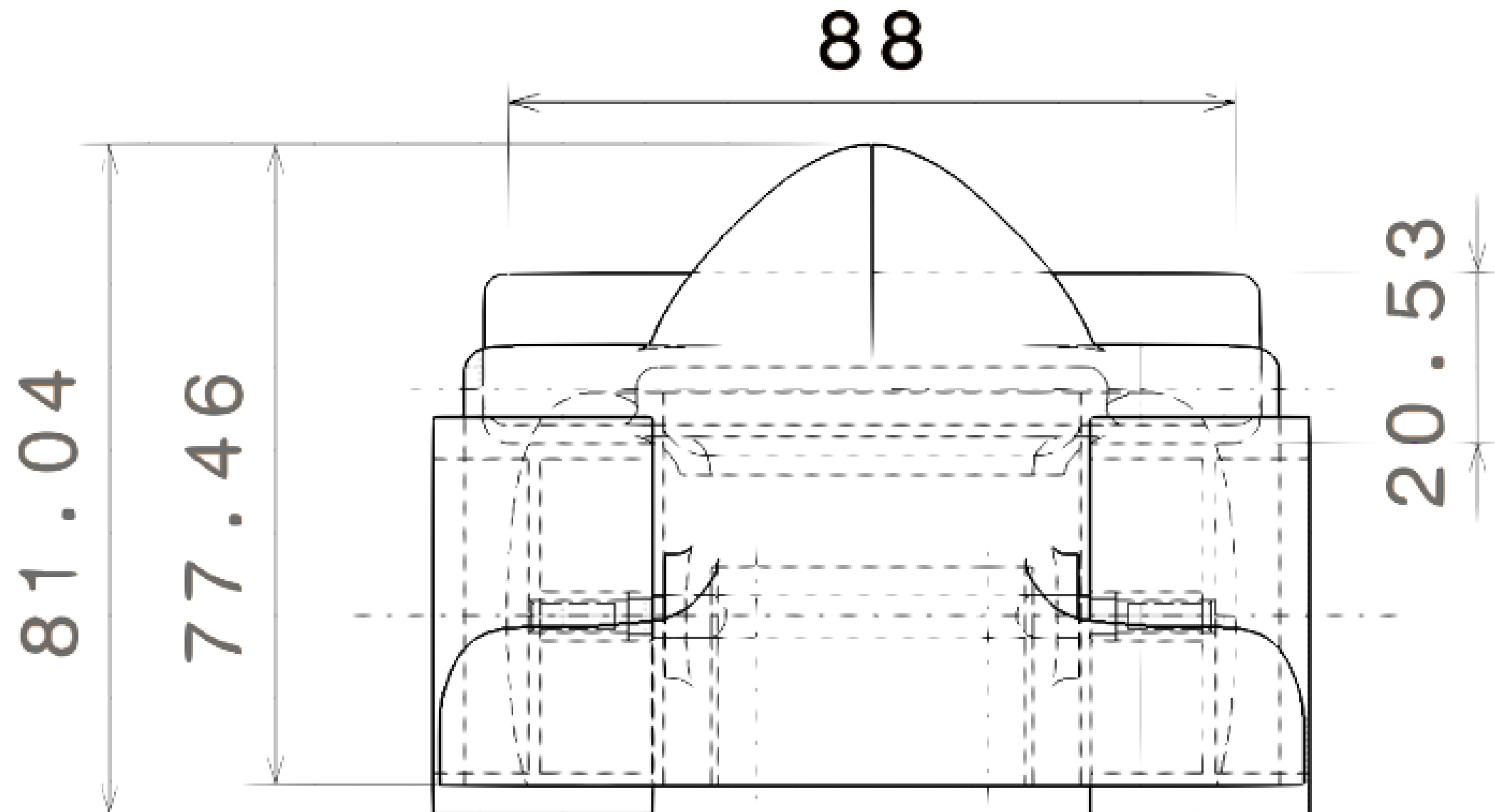


Top view

◆ Dimension requiring the use of anchor points.

## Quoting

10. Finally, complete the front view as below.

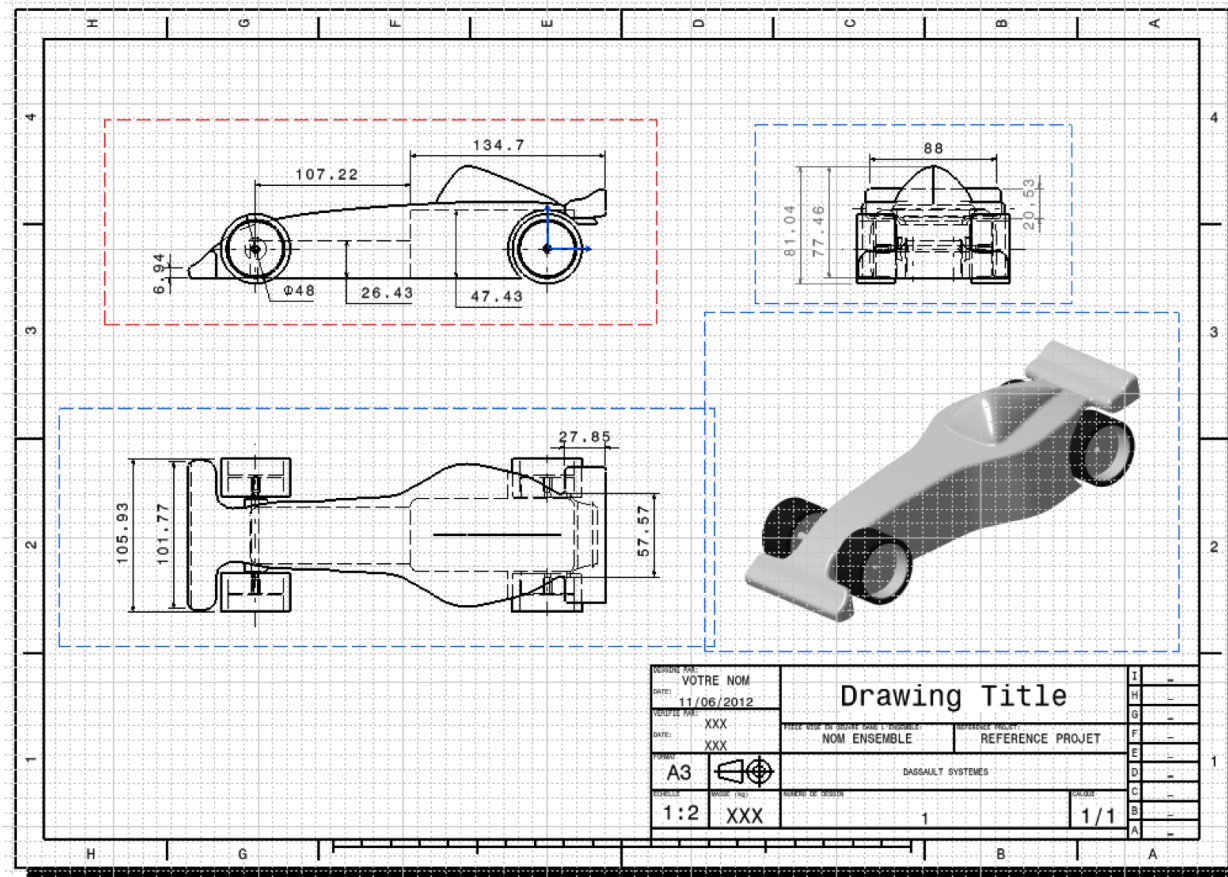


Front view



# Insert a realistic view

We are going to add a realistic view of the car to our drawing.



Here are the steps to follow:

1. Introduction.

2. CATIA V5 QuickStart.

**3. Your racing car.**

has. Draw your car.

b. Open your project.

vs. Create an immersive sketch.

d. Model your car.

e. Assemble your car.

**f. Create a drawing.**

i. Open a drawing.

ii. Personalize your drawing.

iii. Set the Main View.

iv. Define projected views.

v. Change properties.

vi. Quoting.

**vii. Insert a realistic view.**

g. Control and modify your car.

h. Create realistic renderings.

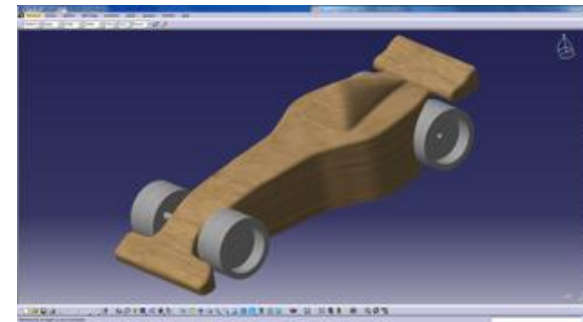
i. Simulate machining.

4. Method sheets.

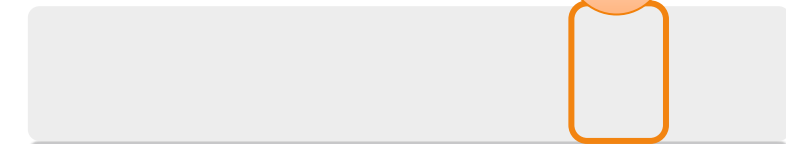
# Insert a realistic view

The realistic view depends on the view currently used in your assembly. Before carrying out this manipulation, orient your car as you wish. Next :

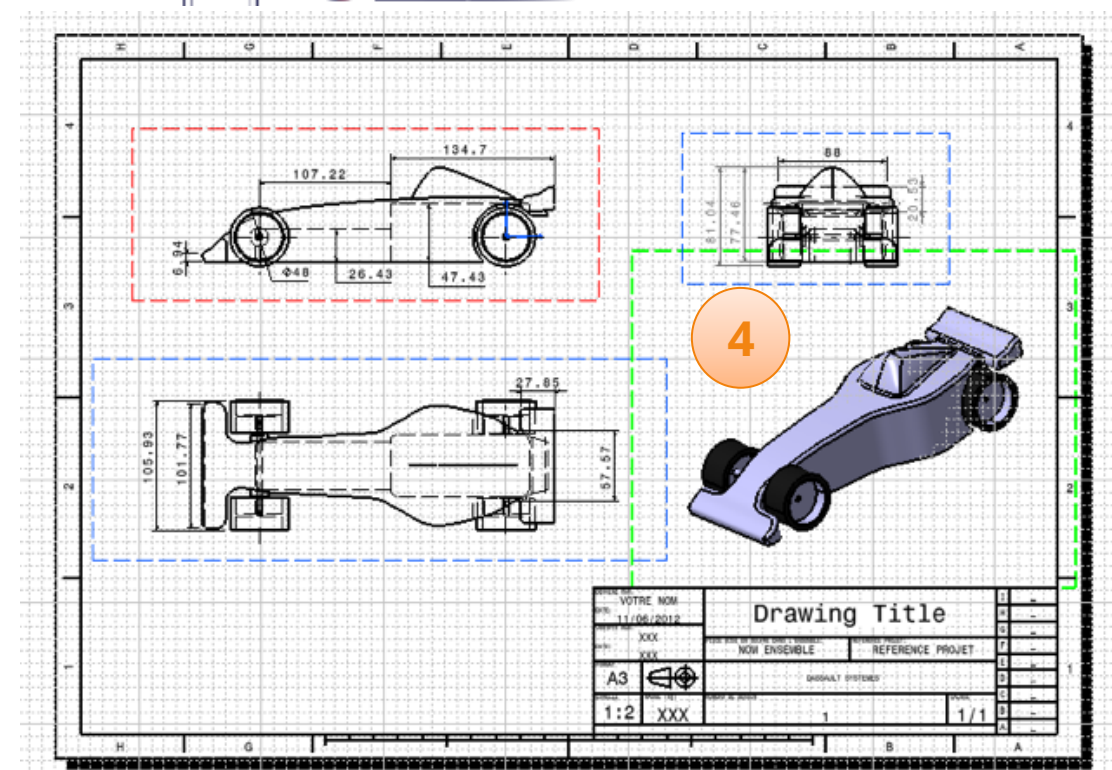
1. Click the icon **Isometric view** located in the same toolbar as the projected view.
2. Click on **Window > MY\_e-RACECAR\_PROJECT**.
3. Click on the part **Car\_Body**, making sure that the **My\_e-RACECAR\_PROJECT** product is activated.
4. This operation returns you to the drawing window with a view equivalent to that of the assembly.
5. Without paying attention to its location, which we will modify later, click anywhere on the drawing to validate the view.



1



3



4

# Insert a realistic view

We are now going to modify the properties of this new view to integrate it perfectly into the drawing.

1.Right-click on the blue frame surrounding the view.

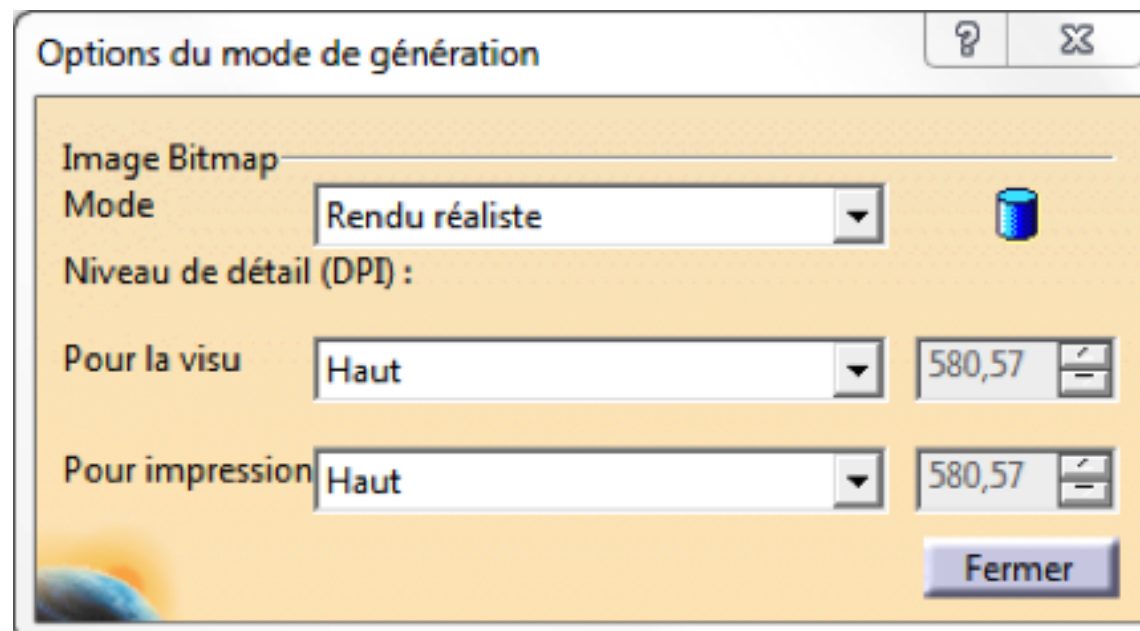
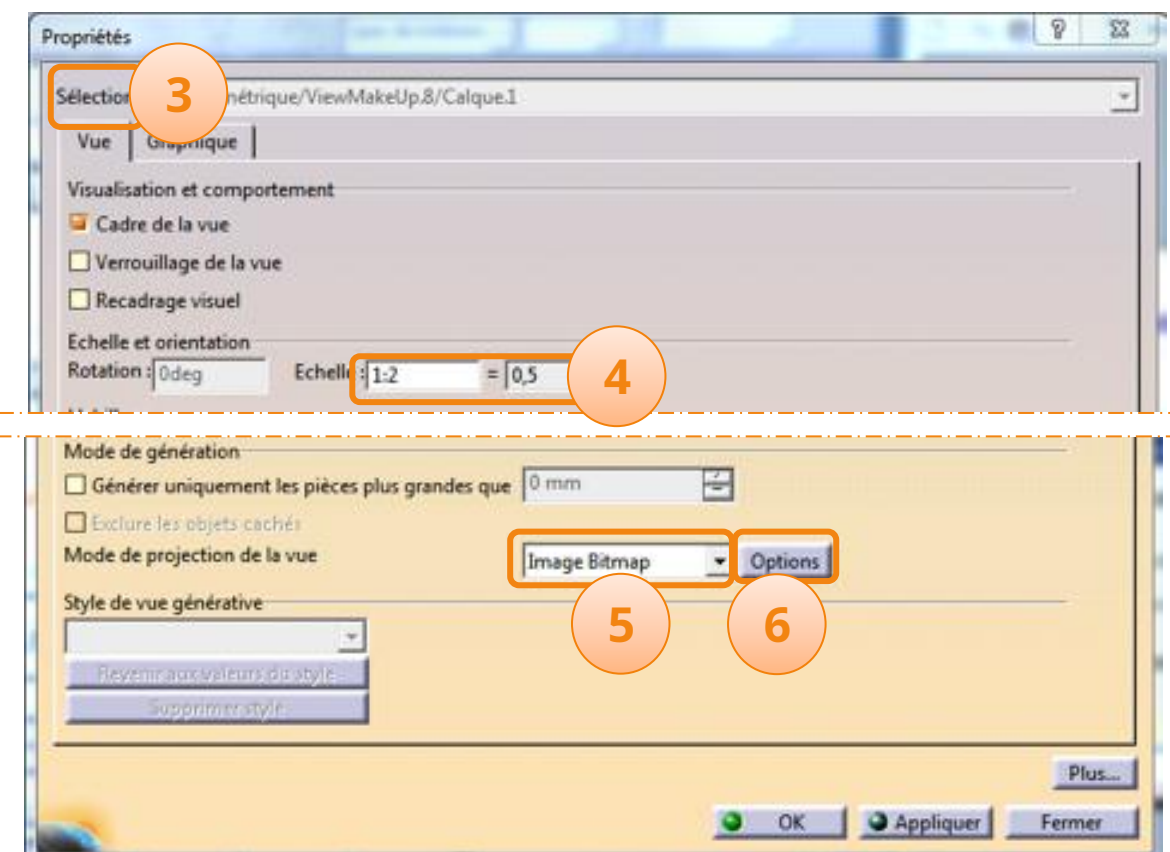
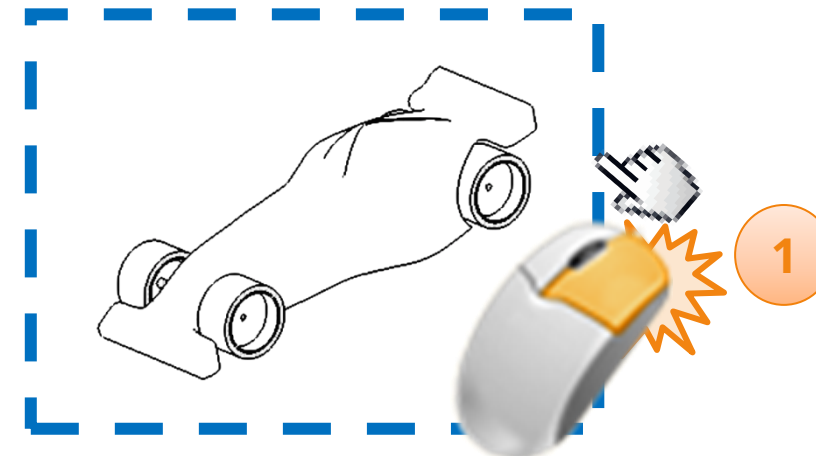
2.Select **Properties**.

3.Select tab **View**.

4.Check that the scale is [1:2].

5.At the bottom choose **Bitmap-Picture**.

6.Click on **Option** and choose the settings below, then click **Apply**, then **okay**.



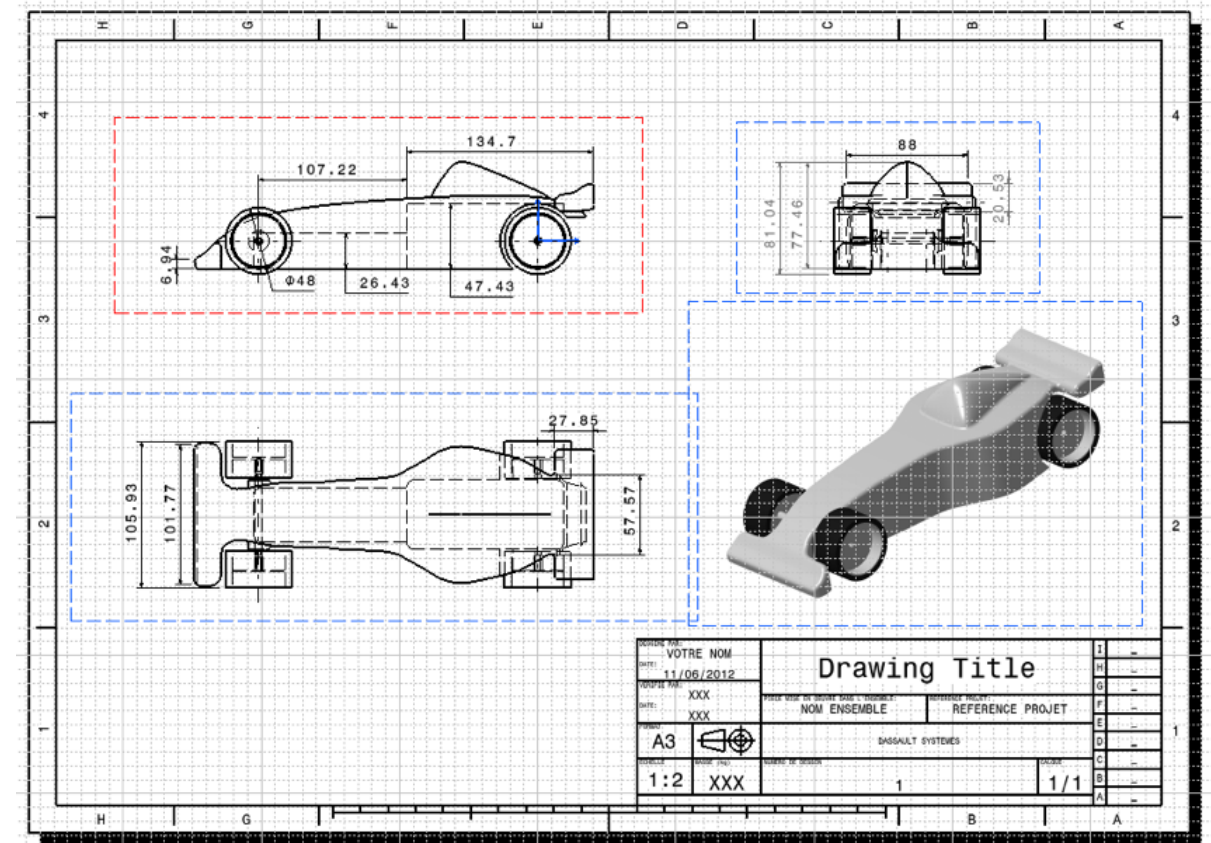


# Insert a realistic view

- Congratulation ! You have completed the drawing part.
- You can now save the drawing in Drawing format, or by using the command, **File > Save As**, save it as a PDF.



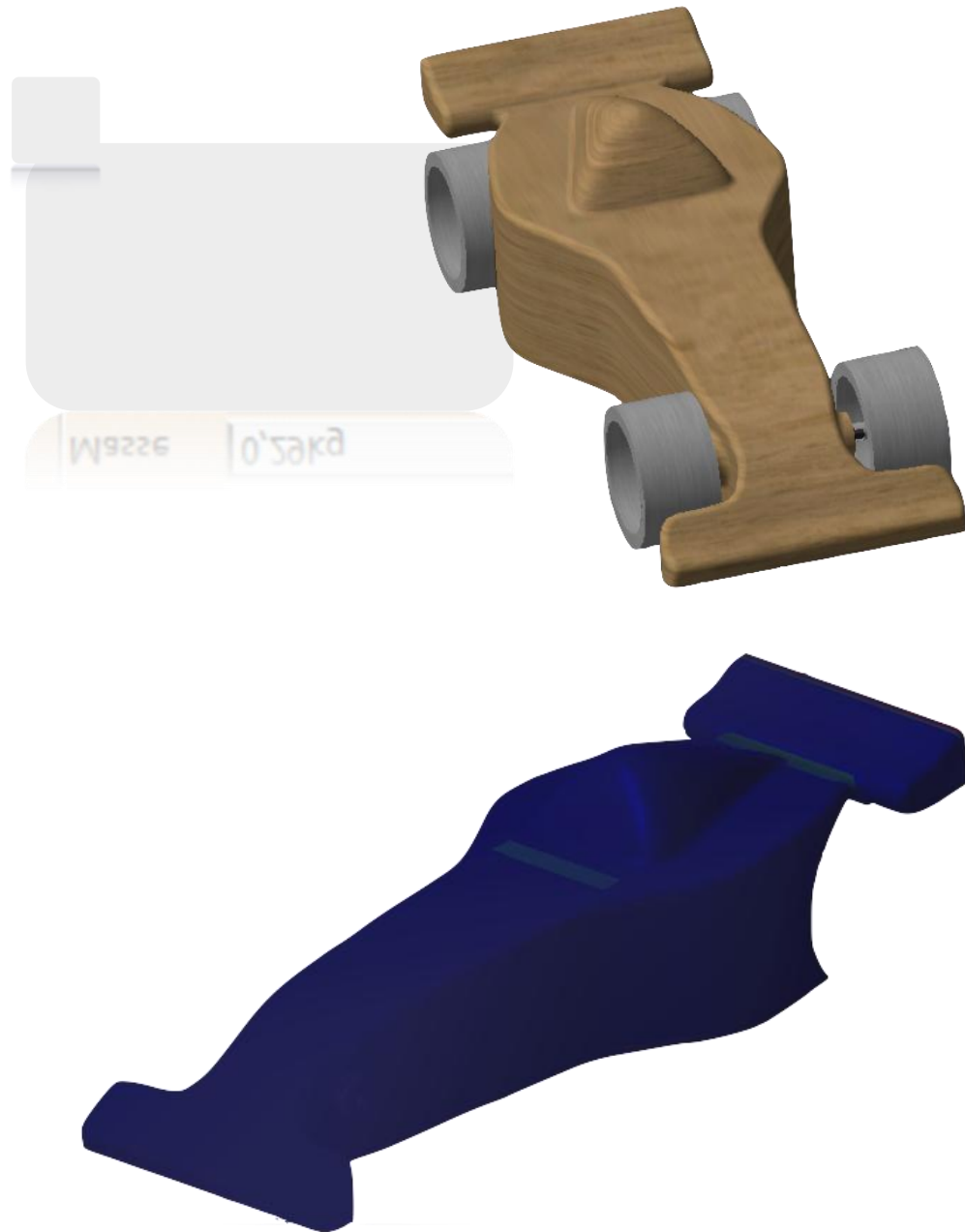
CATDrawing File



# Control and modify your car

In this part we will see how to control the car. These checks will identify a desi problem.

next.



Here are the steps to follow:

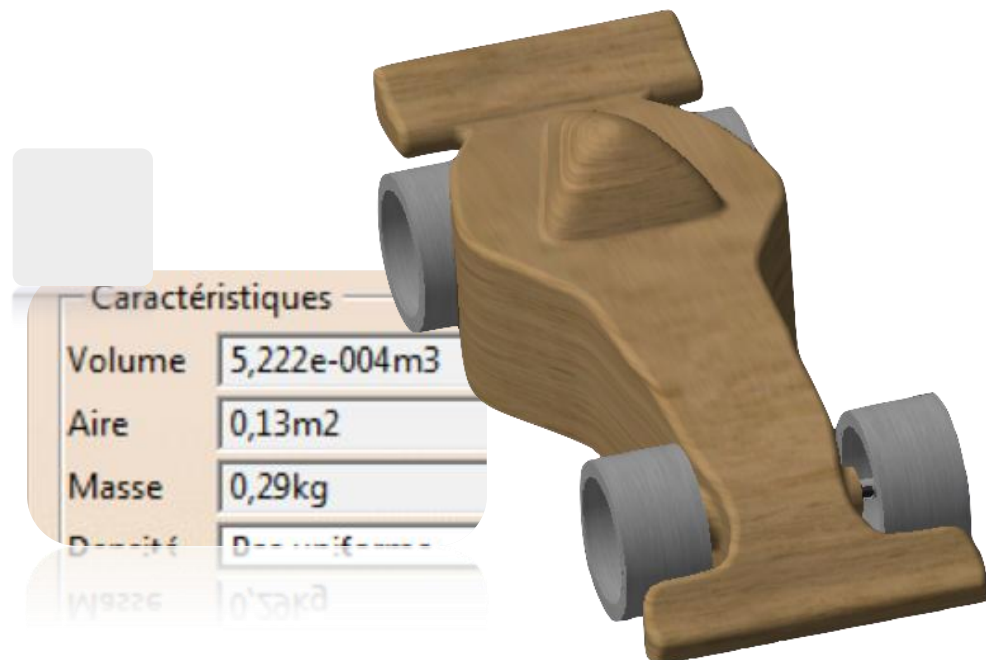
1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.**
    - i. Mass control.
    - ii. Thickness analysis.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.



# Control and modify your car

In this part we will see how to control the car. These checks will identify a problem, which we will then correct.

We will start by controlling the mass of the car assembly. This control will also allow you to visualize its center of gravity.



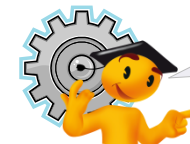
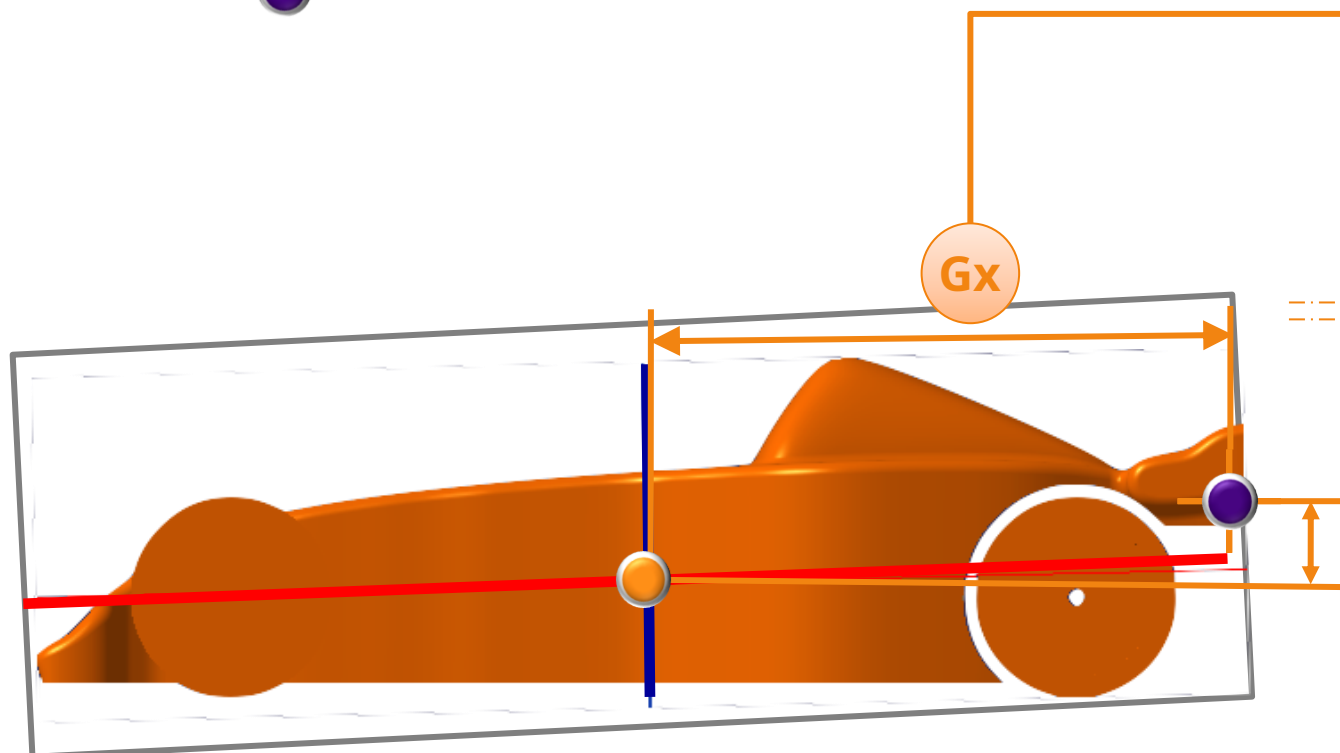
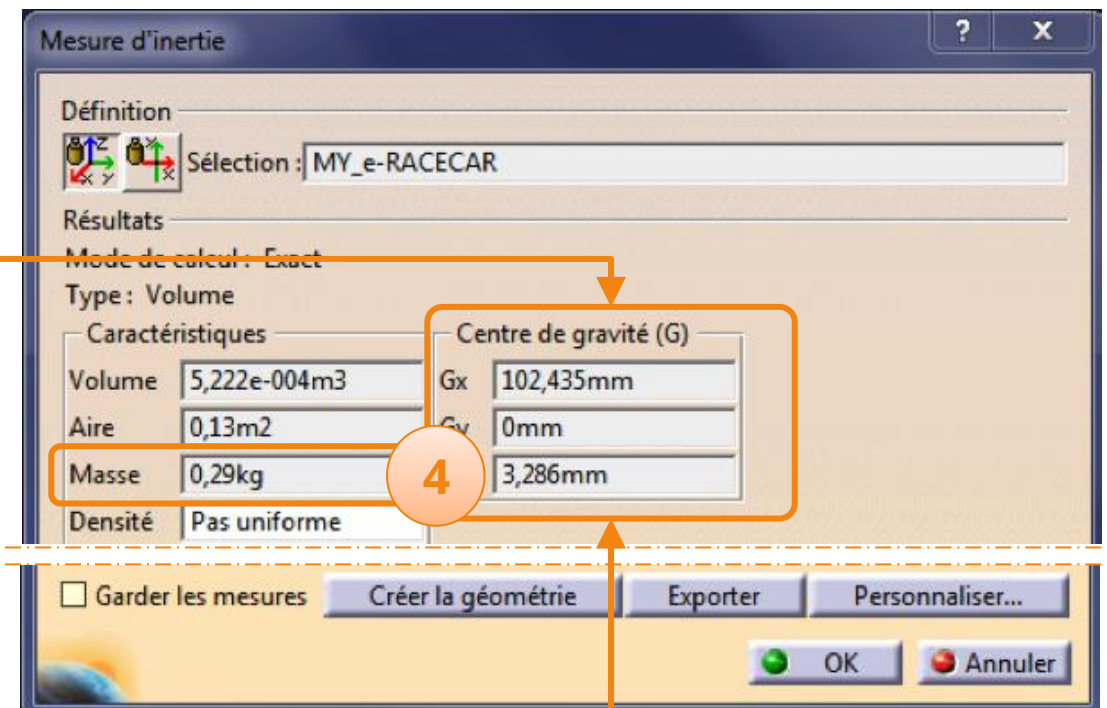
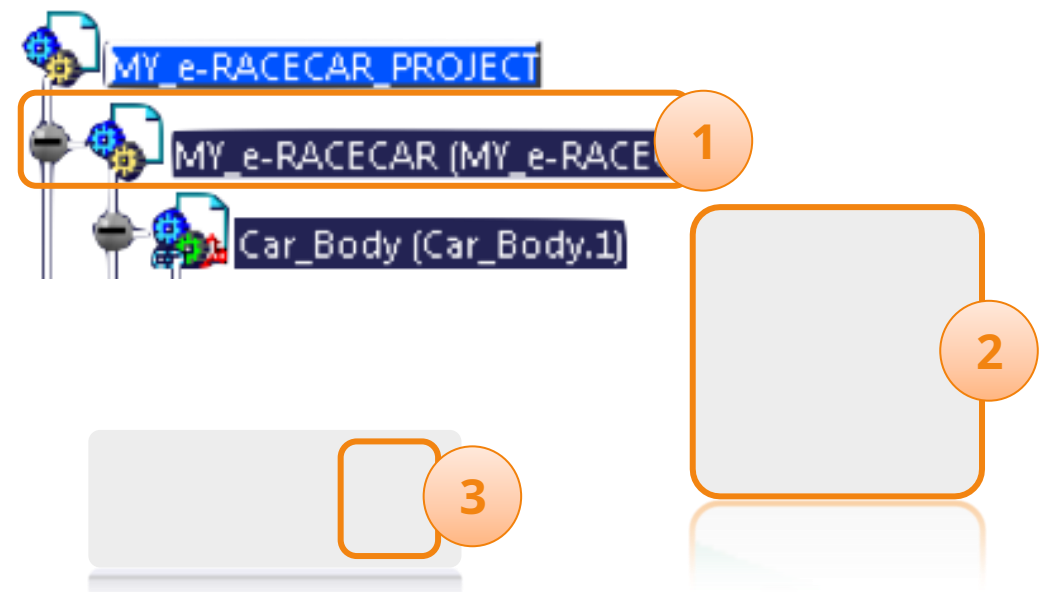
Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.**
    - i. Mass control.**
    - ii. Thickness analysis.
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

# Mass control

CATIA allows you to calculate the mass of your car based on the materials applied to it. For that :

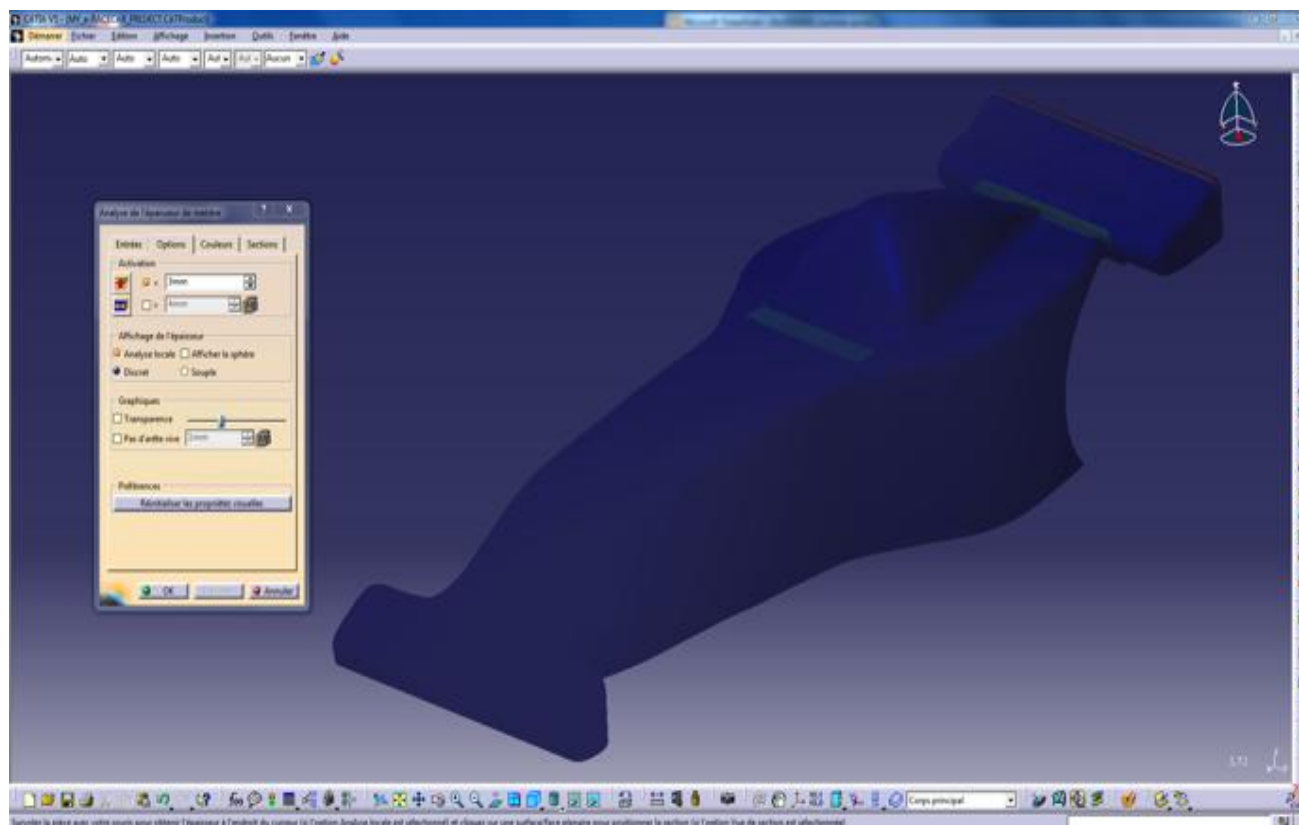
1. Double click on the product **MY\_e-RACECAR**.
2. Make sure you are in the Assembly Design workbench.
3. Click the icon **Inertial measurements**.
4. You can now control the mass of your assembly. You can also see on the visualization the location of the center of gravity as well as its coordinates with respect to the origin of the frame.



Center of gravity and axis of thrust

# Thickness analysis

We are going to perform a thickness analysis using CATIA. The specification states that no part of the car's body should be less than 3mm thick. The tool we are going to use is ideal for checking this.



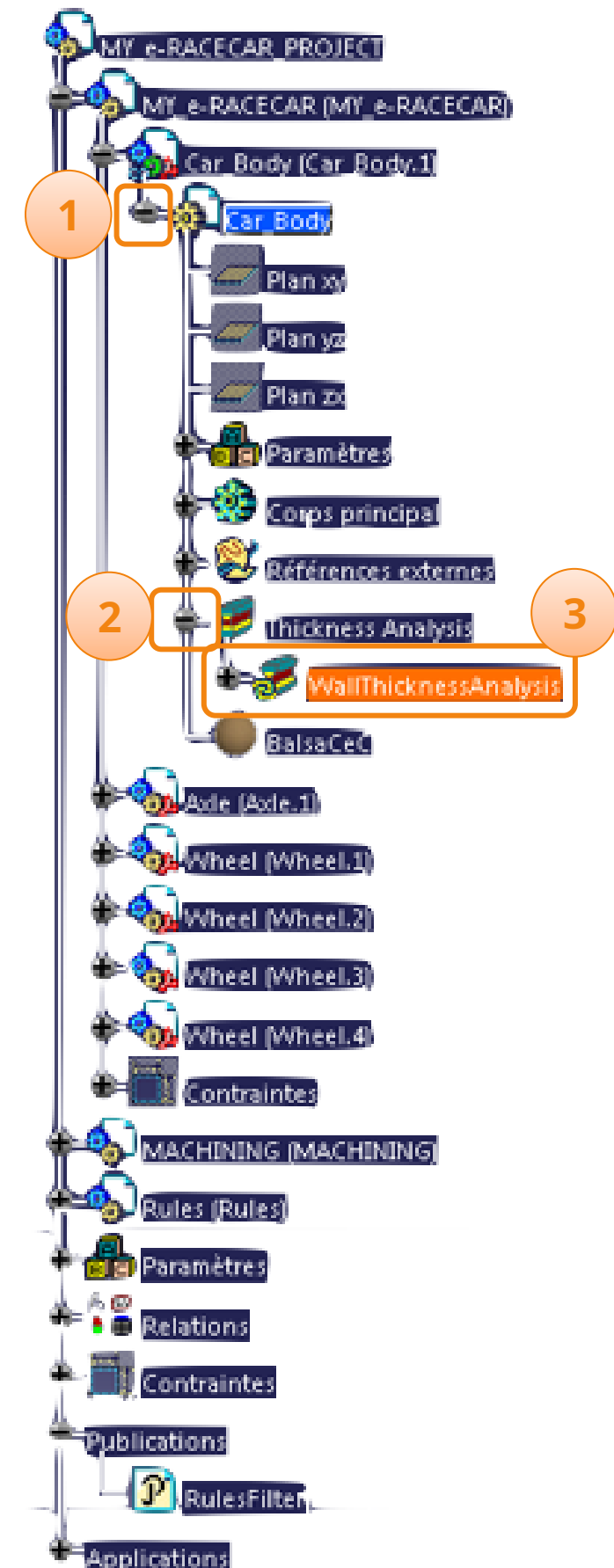
Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. *Your racing car.*
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. *Control and modify your car.*
    - i. Mass control.
    - ii. *Thickness analysis.*
  - h. Create realistic renderings.
  - i. Simulate machining.
4. Method sheets.

# Thickness analysis

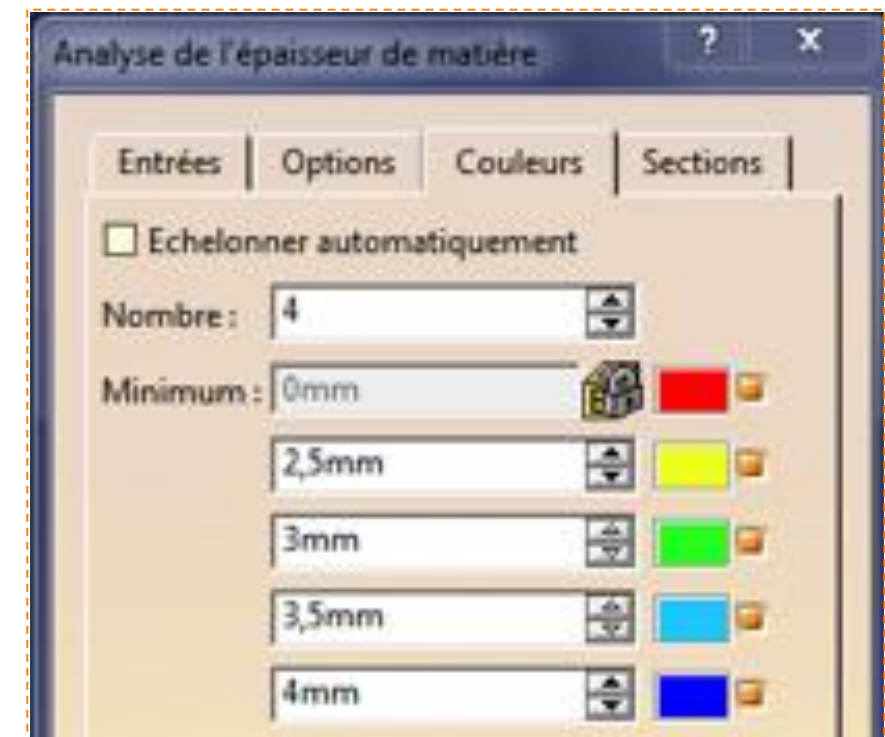
The thickness analysis is already introduced in the project tree, it is ready to be used. To use it:

1. Go to the specification tree, and open the branch **Car\_Body**.
2. Open the branch **Thickness Analysis**.
3. Double-click **WallThicknessAnalysis**.
4. The window called Wall Thickness Analysis below appears, click on **Execute**.



# Thickness analysis

- Exploitation of results :





# Create realistic renderings

In this part we will see how to achieve a realistic rendering of your assembly.



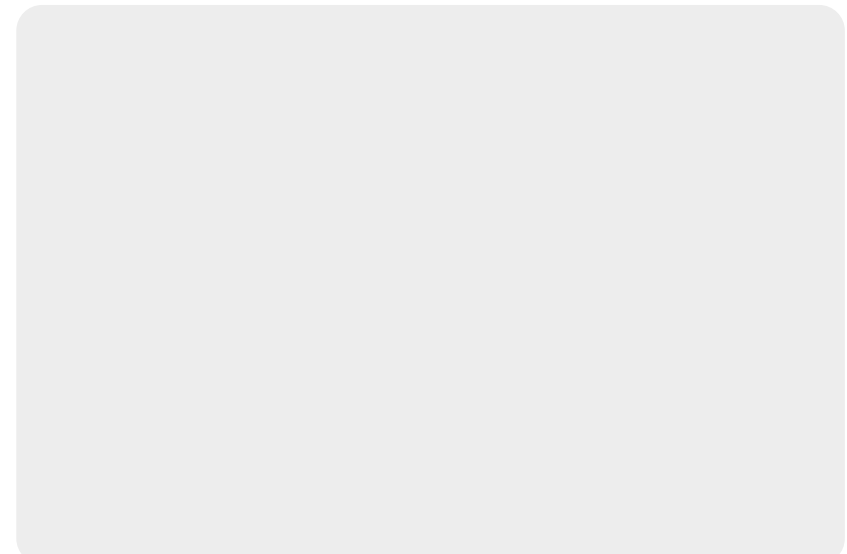
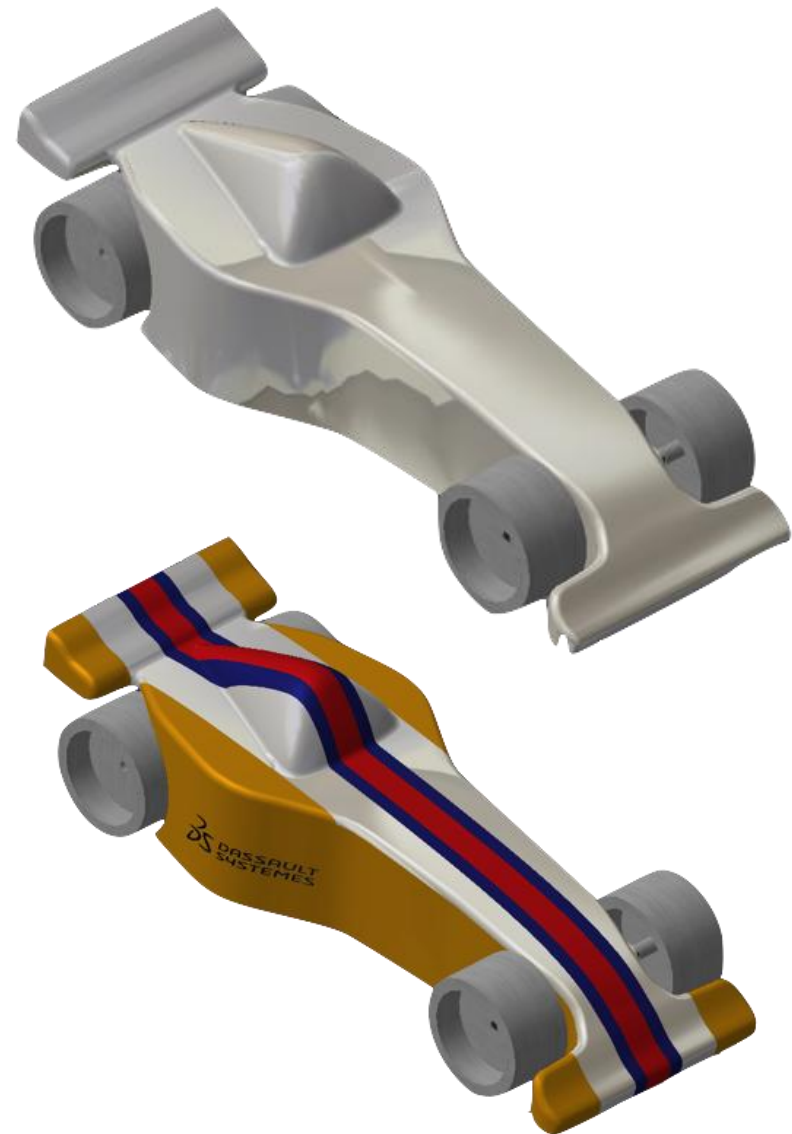
Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. *Your racing car.***
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.***
    - i. Introduction.***
      - ii. Painting.
      - iii. Decals.
      - iv. Returns.
    - i. Simulate machining.
4. Method sheets.

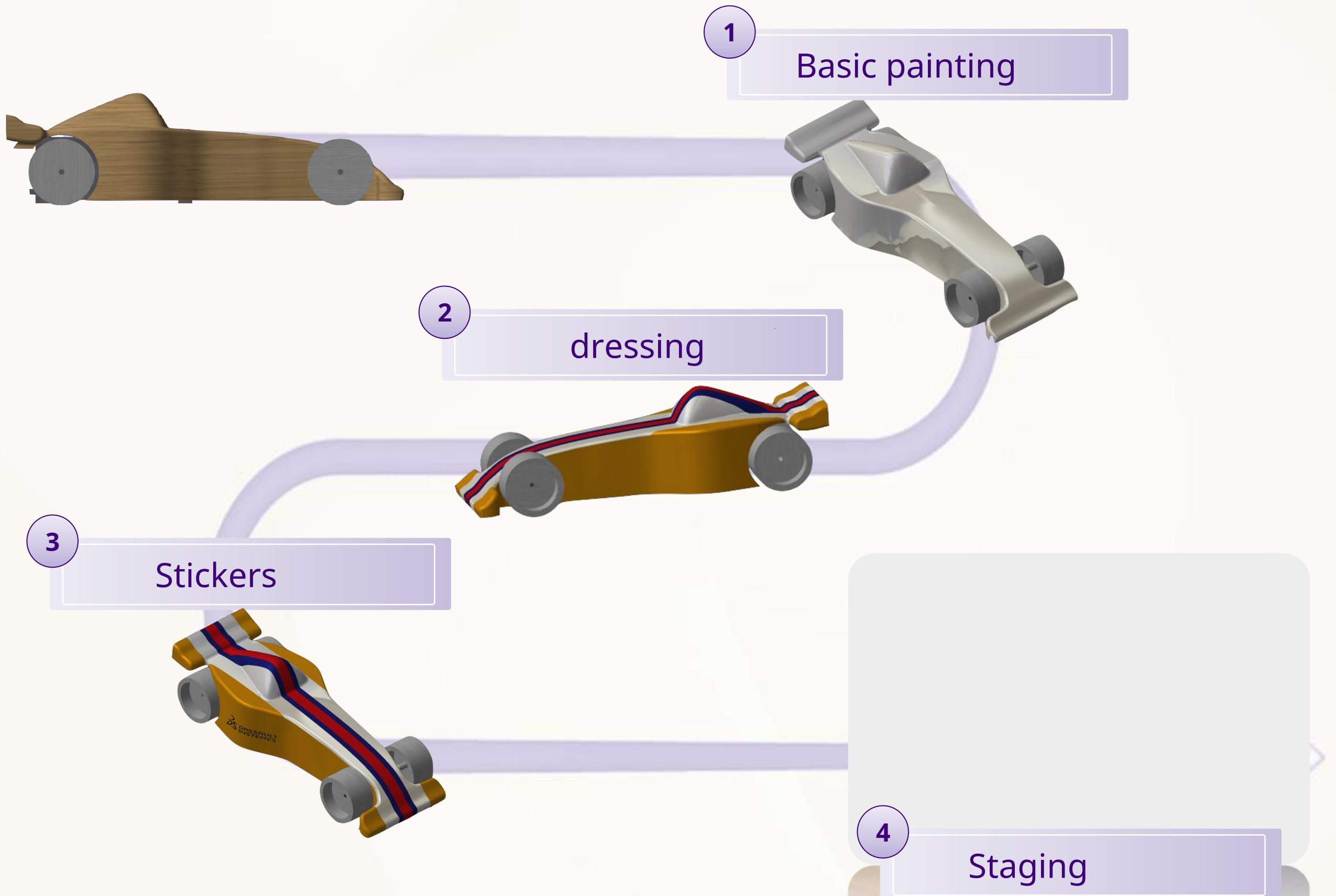
# Create Realistic Renderings - Introduction

Here is how the creation of a realistic rendering works:

- ▶ Application of a paint that will be the main color of your car.
- ▶ Addition of stickers to decorate and dress the car with the different logos of your sponsors.
- ▶ Setting up the rendering scene, involving setting the lighting, ground, and the environment.

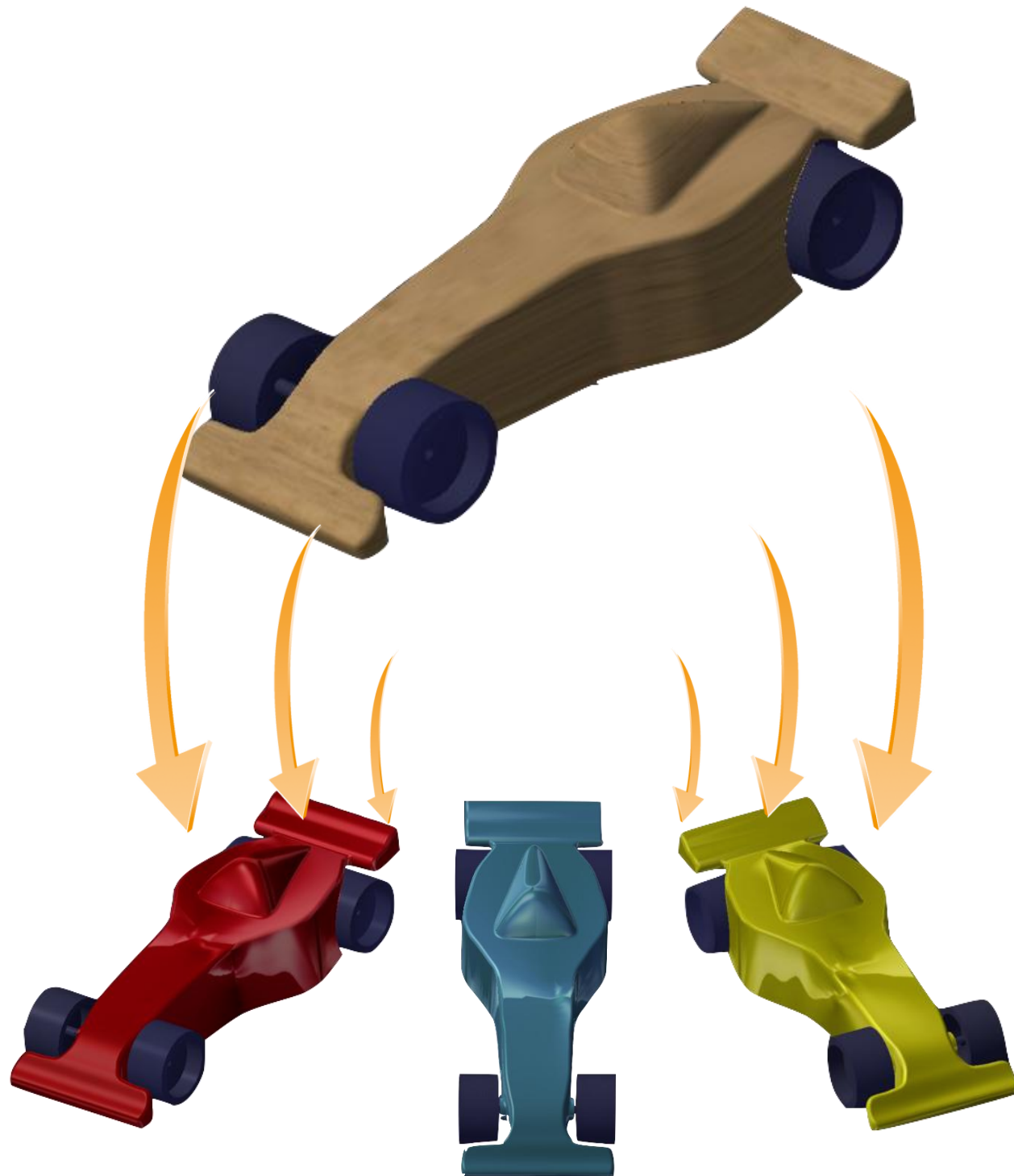


# Create Realistic Renderings - Process



# Create Realistic Renderings - Painting

In this part we will see how to achieve a realistic rendering of your car. We will start by defining the base paint of the body.



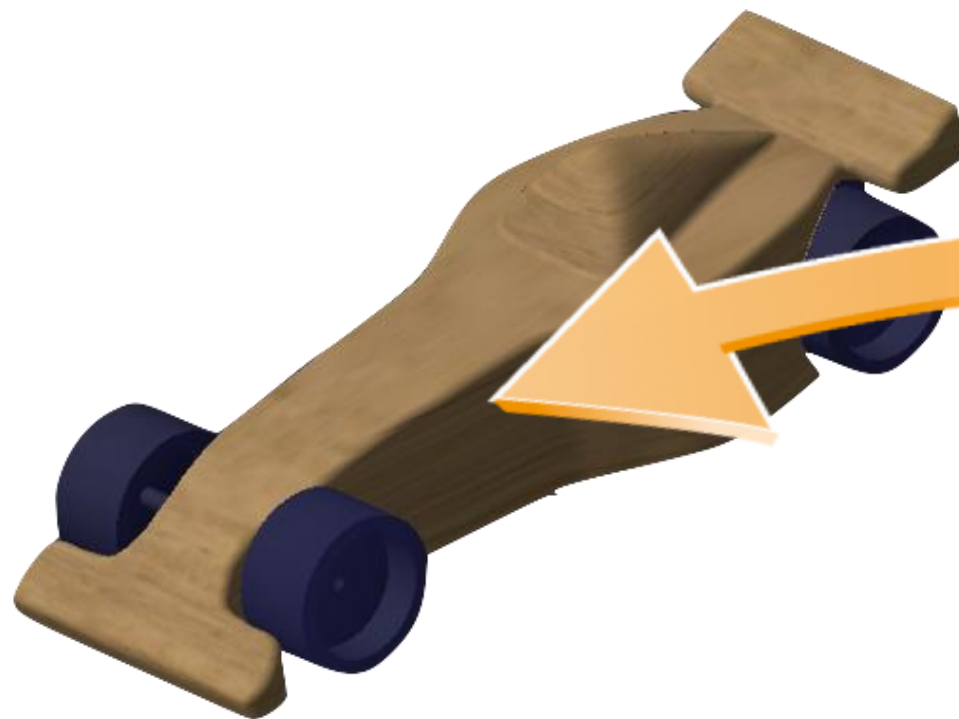
Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.**
    - i. Introduction.
    - ii. Painting.**
    - iii. Decals.
    - iv. Returns.
  - i. Simulate machining.
4. Method sheets.

# Painting

The application of a paint is carried out in the same way as the application of balsa. For that :

1. Click on **Apply materials**.
2. Click on **Opens a catalog of materials**.
3. Select the CATIA default catalog:  
`C:\Program Files\Dassault Systemes\B19\intel_a\startup\materials\Catalog.CATMaterial`
4. Click on the tab **painting**.
5. Drag and drop the paint you want on the body of the car.
- 6.
- 7.

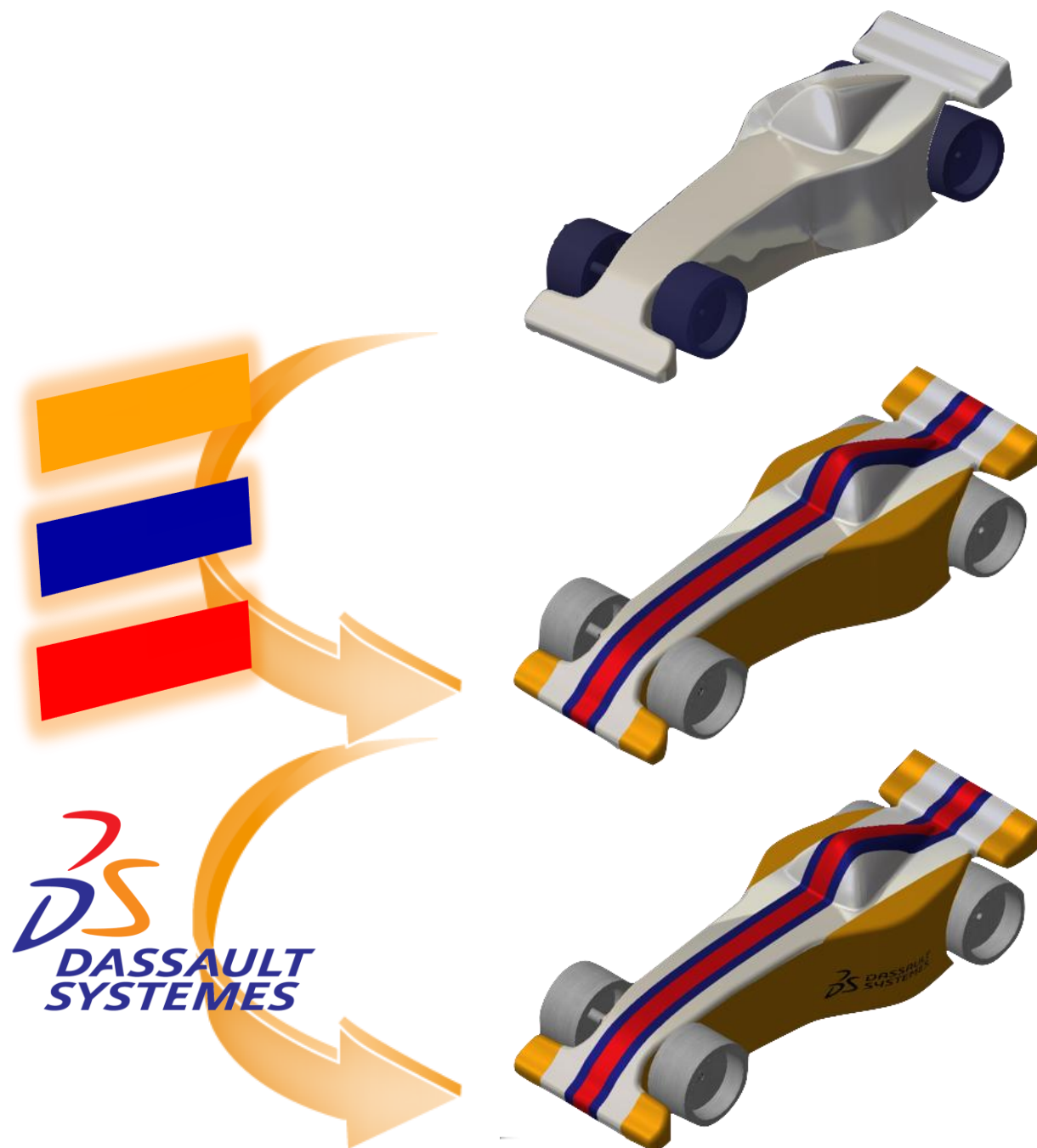


The paint should now be visible on your car provided you are in the correct, realistically rendered viewing mode.



# Realistic Rendering in CATIA – Stickers

In this part we will see how to apply stickers on the car, but also modify their properties. We will use two styles of stickers, those that will be used to color the car and the logos.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.**
    - i. Introduction.
    - ii. Painting.
    - iii. Decals.**
    - iv. Returns.
      - i. Simulate machining.
4. Method sheets.

## The PhotoStudio workshop

To apply stickers we will use the PhotoStudio workbench. This allows you to create:

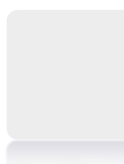
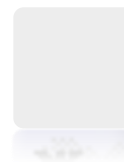
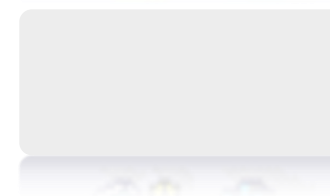
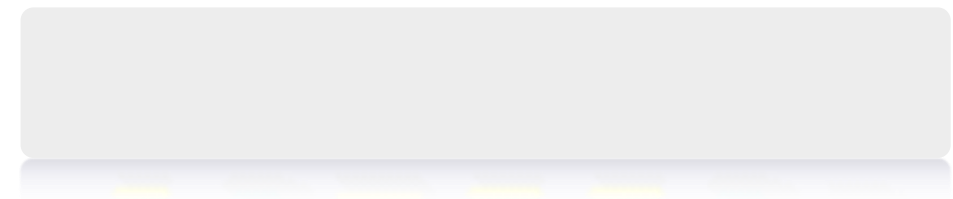
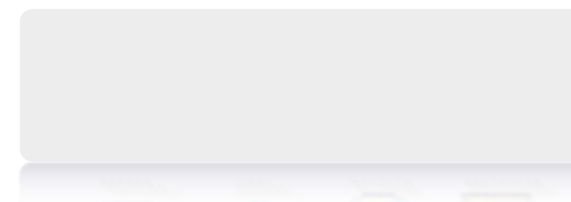
environments.

Various types of lighting.

A turntable animation.

Cameras.

Apply stickers.



## Sticker Tips

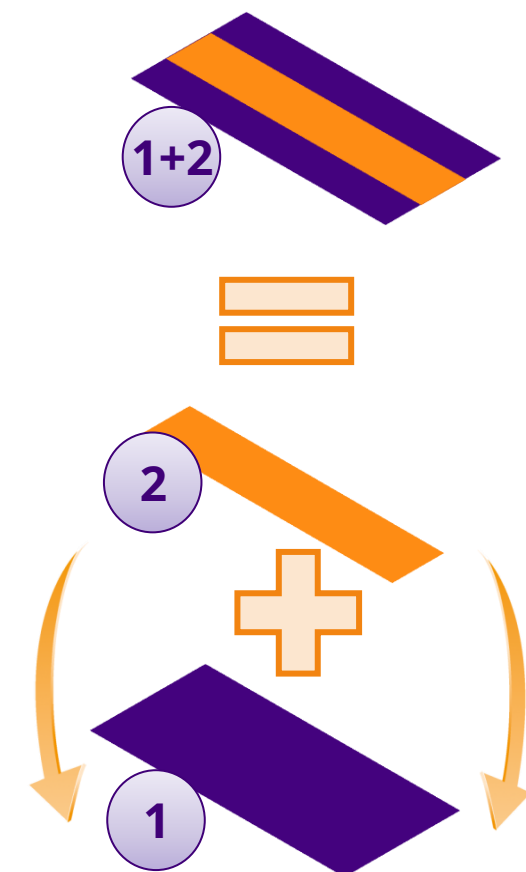
It is strongly advised to use high quality images. To do this, create files **images with high resolution** to avoid pixelation during renders.

You can use images in JPEG and BMP format but **prefer BMP format (bitmap)** which will give you a better quality for the renders.

**Please note that the GIF format cannot be used.**

You can use many stickers, feel free to create them separately. In this way, if an element no longer suits you, you can modify or delete it.

The stickers overlap as you add them. Thus, the last sticker added will pass below all the others. **Before you start, think about the order in which you will import them.**



## decals

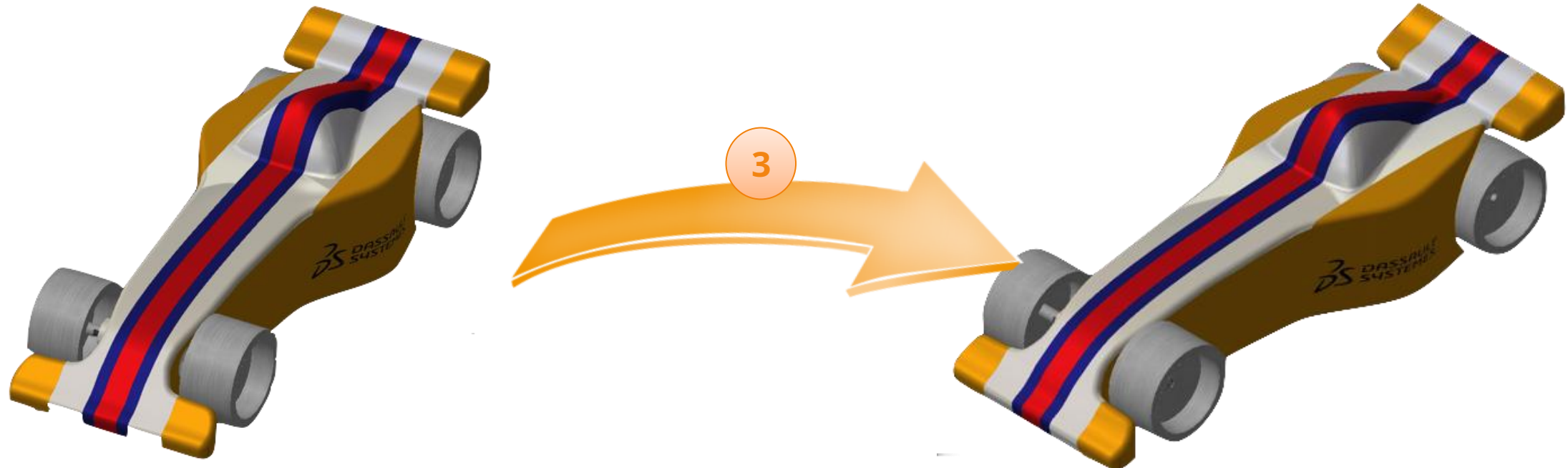
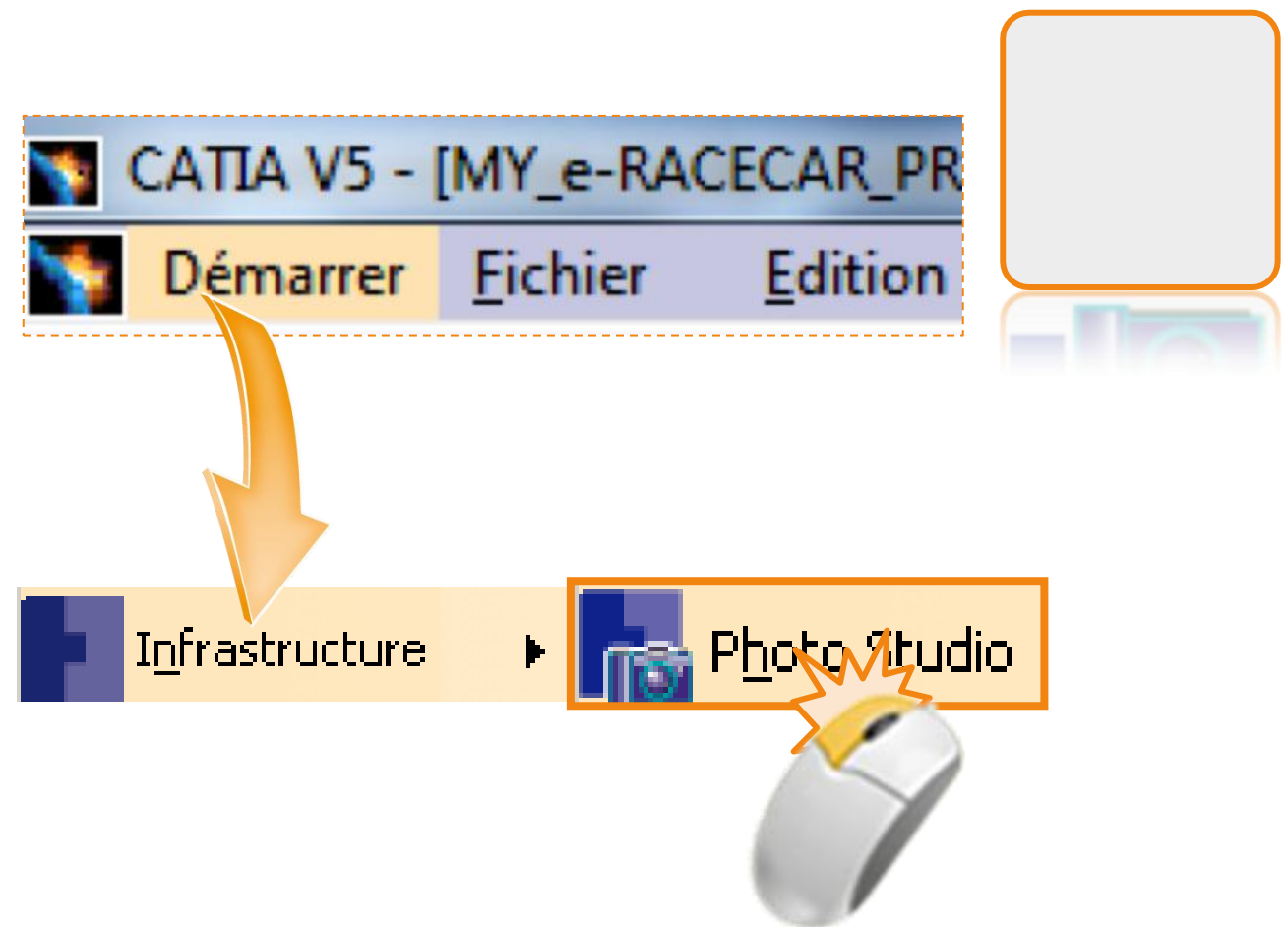
We must first go to the appropriate workshop. To do this, start by opening the Photo Studio workbench.

1. Check that the file **MY\_e-RACECAR\_PROJECT** is open because PhotoStudio can only be used in an assembly context.

2. Click on :**Start > PhotoStudio Infrastructure**.

As we want a realistic image use perspective display. For this

3. Select **View > Re Perspective Style**.



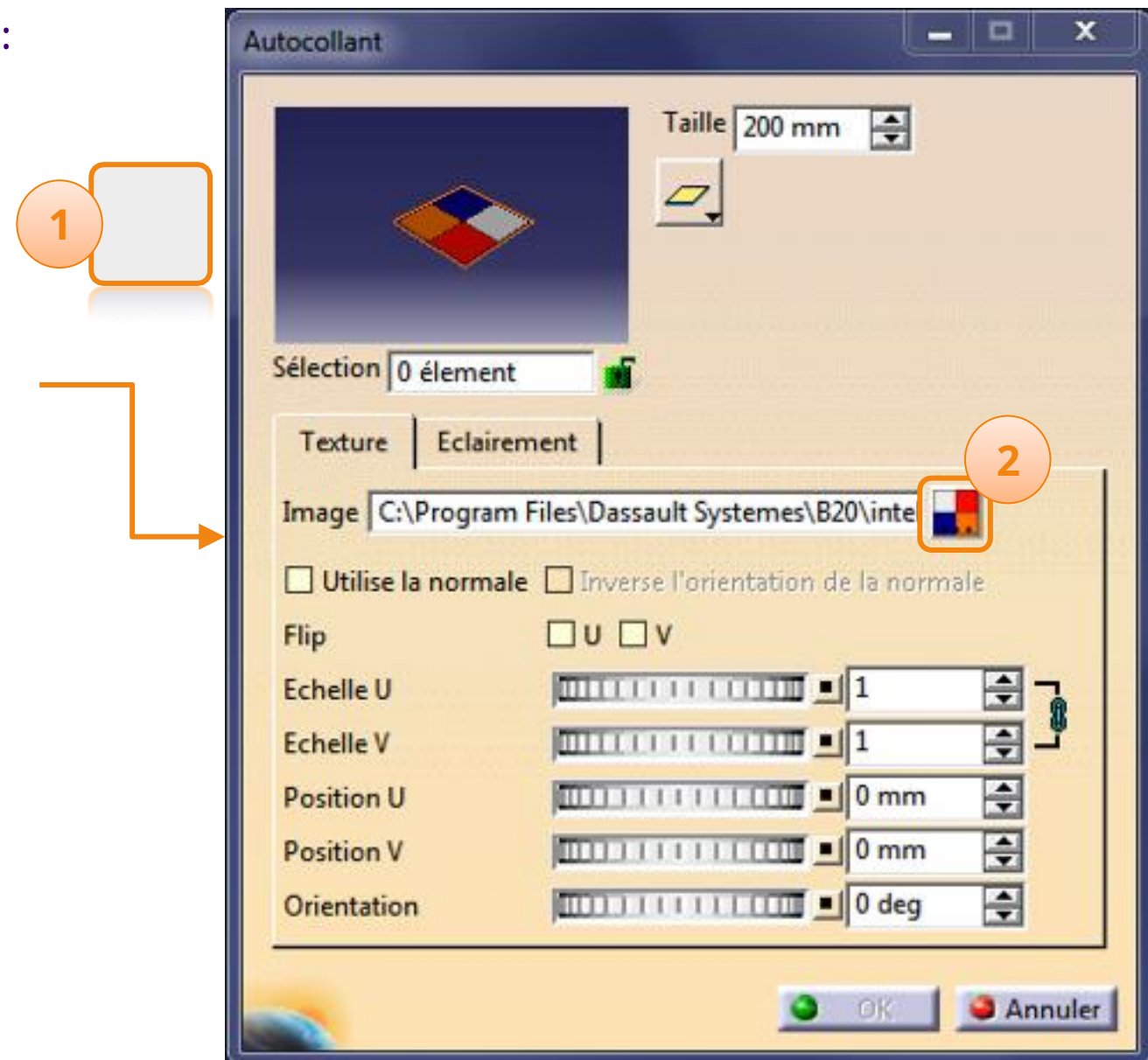
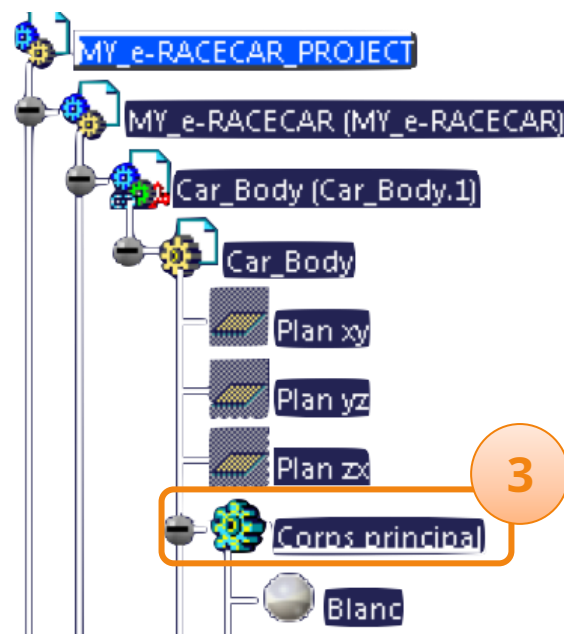
# decals

Now we are going to stick a sticker that will be used to color the car, like a colored band for example. For that :

1. Click the icon **Apply a sticker** who is located at the bottom of the screen.

The window for configuring the sticker opens.

2. Click on the sticker icon. This will open a window allowing you to choose it.
3. Open the file **Yellow Stripe** located in the directory **Mini F1 2010 – Start Package**.
4. To define on which parts to apply the sticker, click on **Main body** of the room **Car\_Body**.



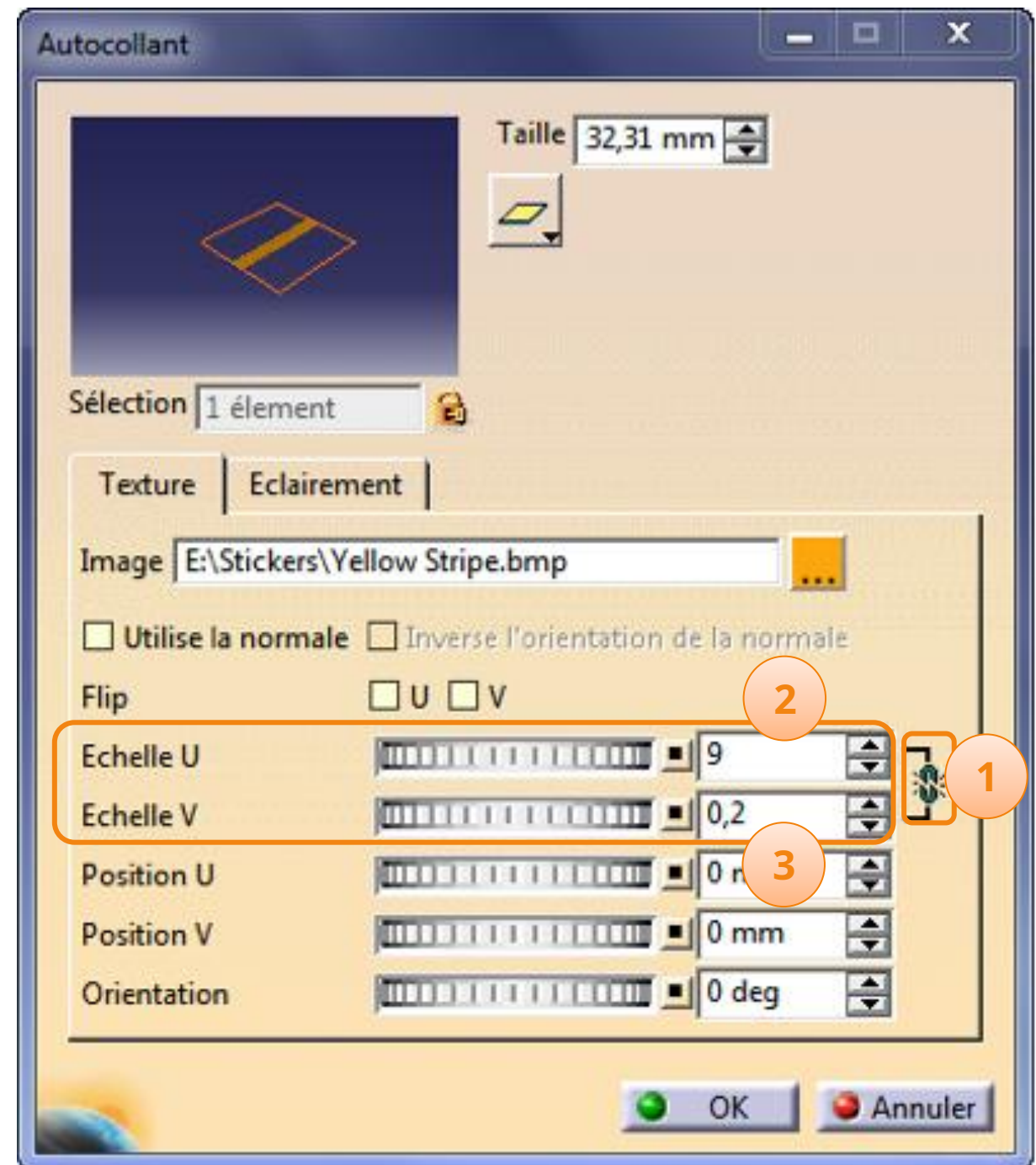
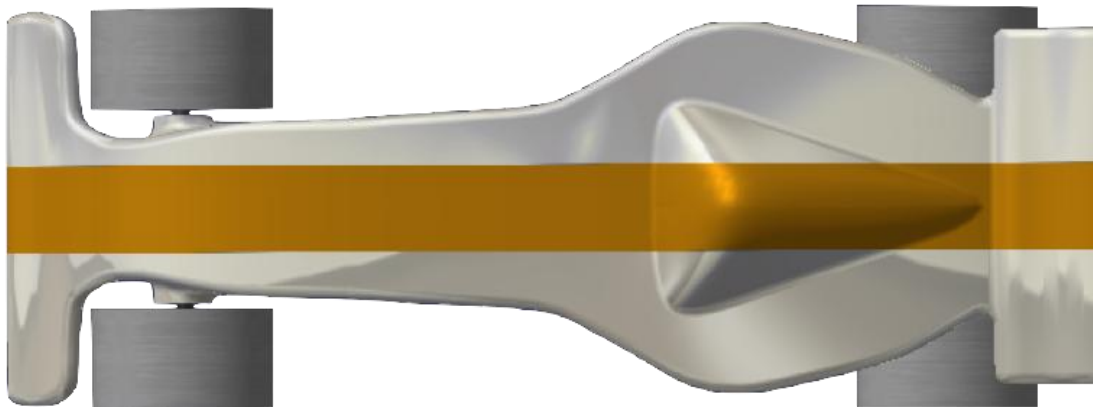


# decals

We want to make a strip of yellow color. To resize the sticker:

1. Click on **chain** to the right of the line **Scales**. This will allow you to edit the two dimensions separately.
2. Enter the value [9] in the line **U-scale**.
3. Enter the value [0,2] in the line **V-scale**.
4. Click in a row other than **V-scale** to update the display.

You should get the following result. If not, reverse the previous values.



# decals

To move this sticker wherever you want, you have two methods:

## First method:

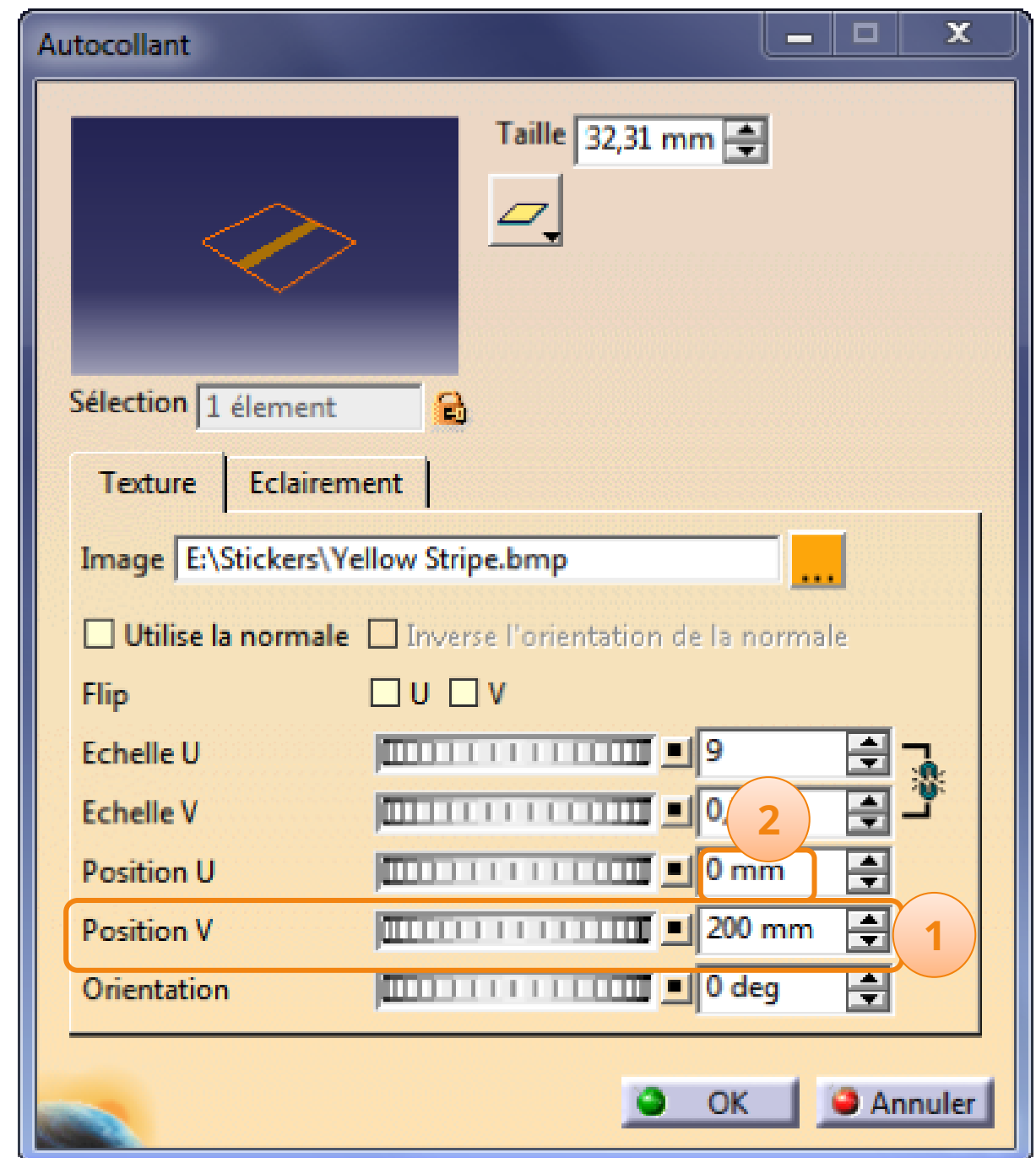
1. Enter a value [200 mm] for example, in the line **V-Position**.
2. Click in another row of the window to apply the modification.
3. Observe the result on the car and decide whether this value should be higher or lower depending on the desired result.

## Advantage :

- Placements are precise, which comes in handy when stickers need to be symmetrical. With positive values on one side and negative on the other.

## Inconvenience :

- The trial and error to find the right position values can be long.



# decals

## Second method:

1. Click on **okay** to close the sticker properties window.

At the bottom of the specification tree you find the category **Stickers**.

2. Click on **Sticker 1** to select it.

3. Observe the compass that is attached to the sticker.

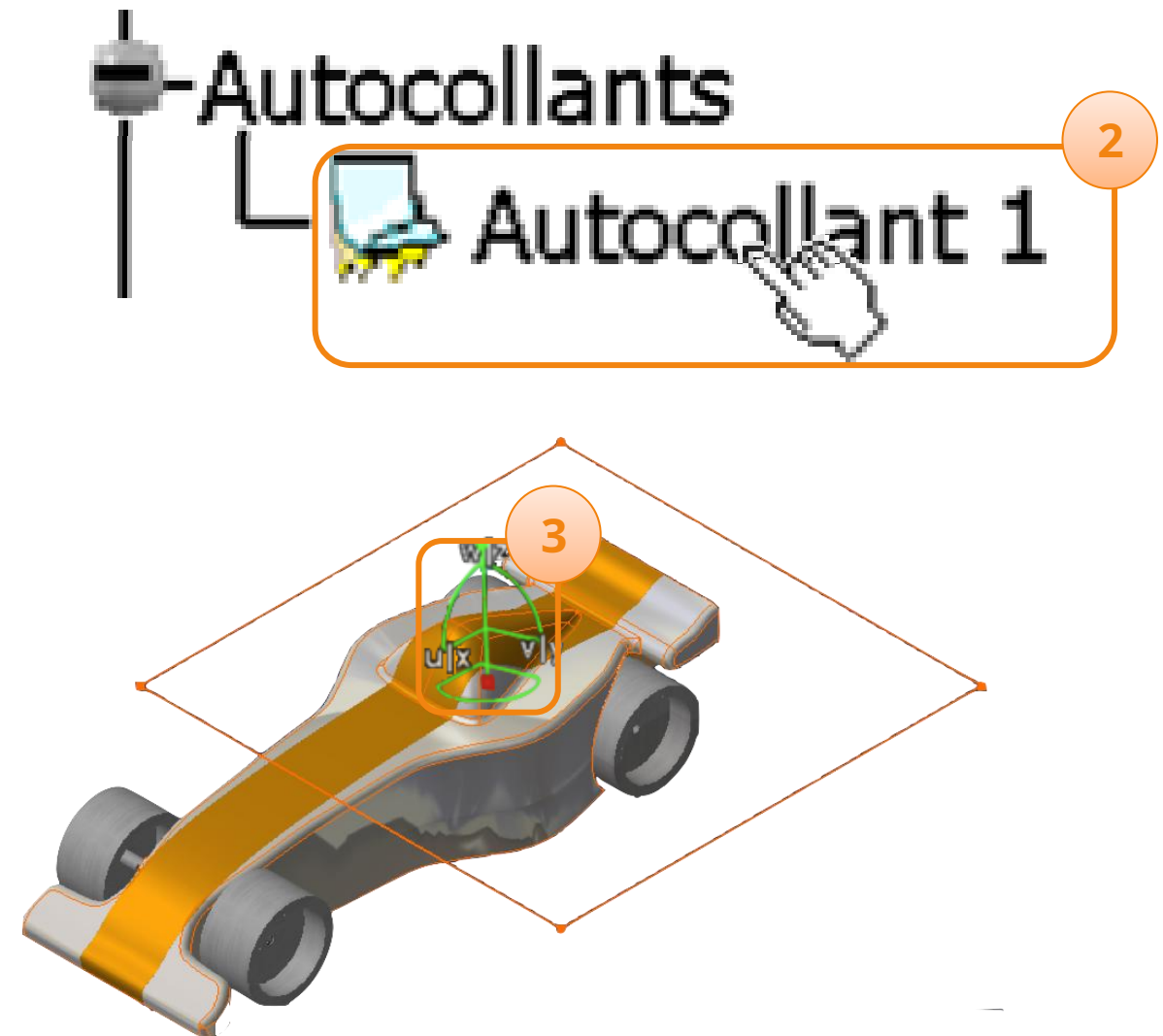
4. Using the compass you can now move the sticker along the axes that interest you.

## Advantage :

- Fast and intuitive sticker placement. Ideal for logos.

## Inconvenience :

- Placement is imprecise. Problematic for stickers which must be symmetrical.

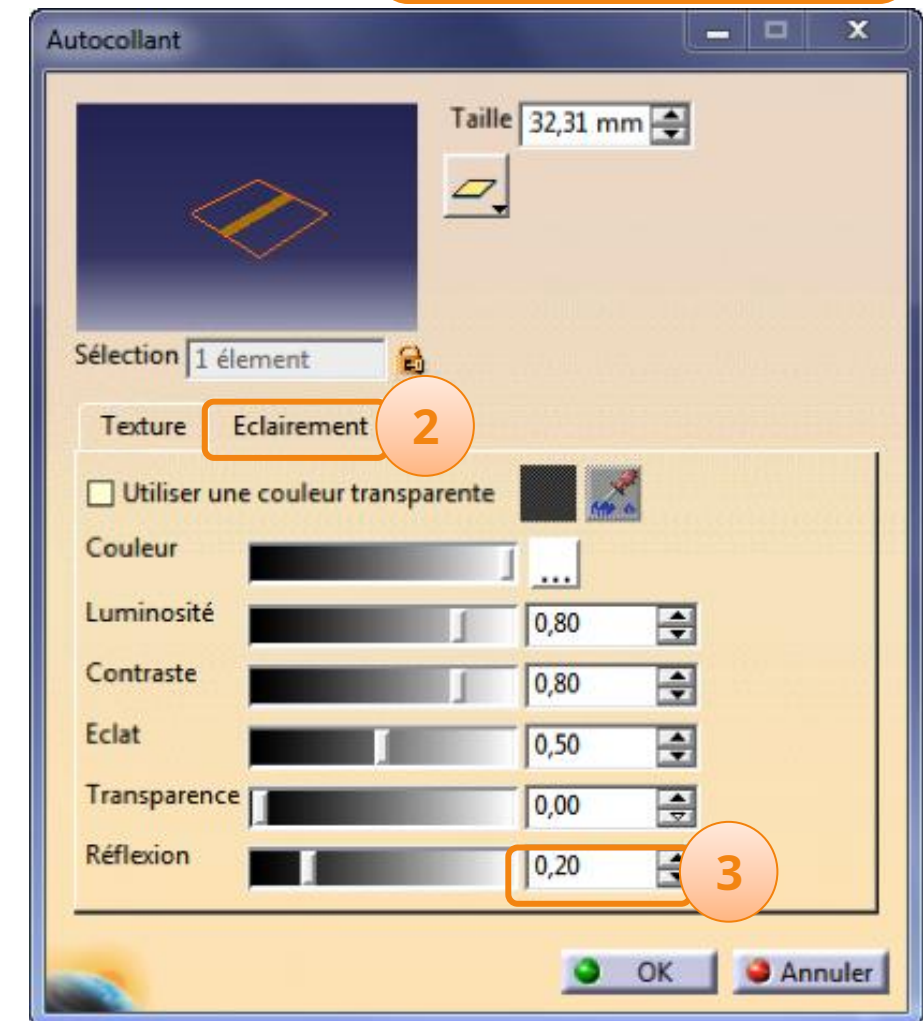
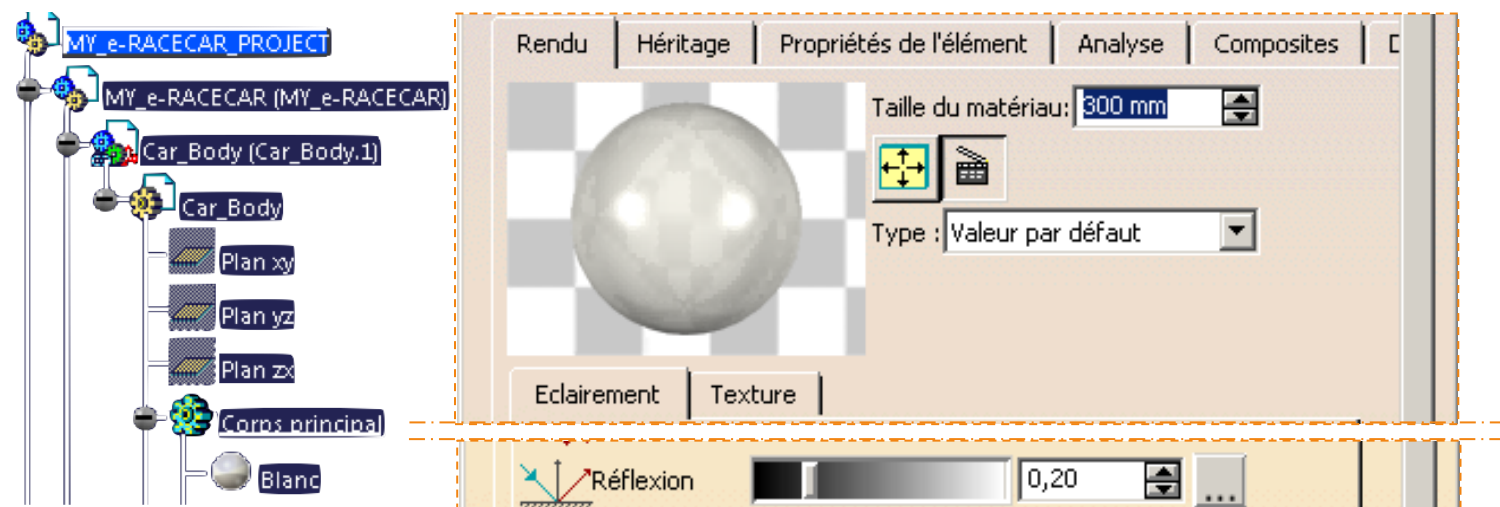


# decals

We're going to change a setting that will improve the look of your stickers by making them look brighter. For this you must:

1. Access the properties window by double-clicking on a sticker at the bottom of the specification tree.
2. Click on the tab **Illuminance**.
3. Increase Reflection coefficient to [0.20]. So the sticker will have the same reflection coefficient as your paint.

You can check that the coefficient of the paint used is the same by double clicking on the material you applied to the main body.





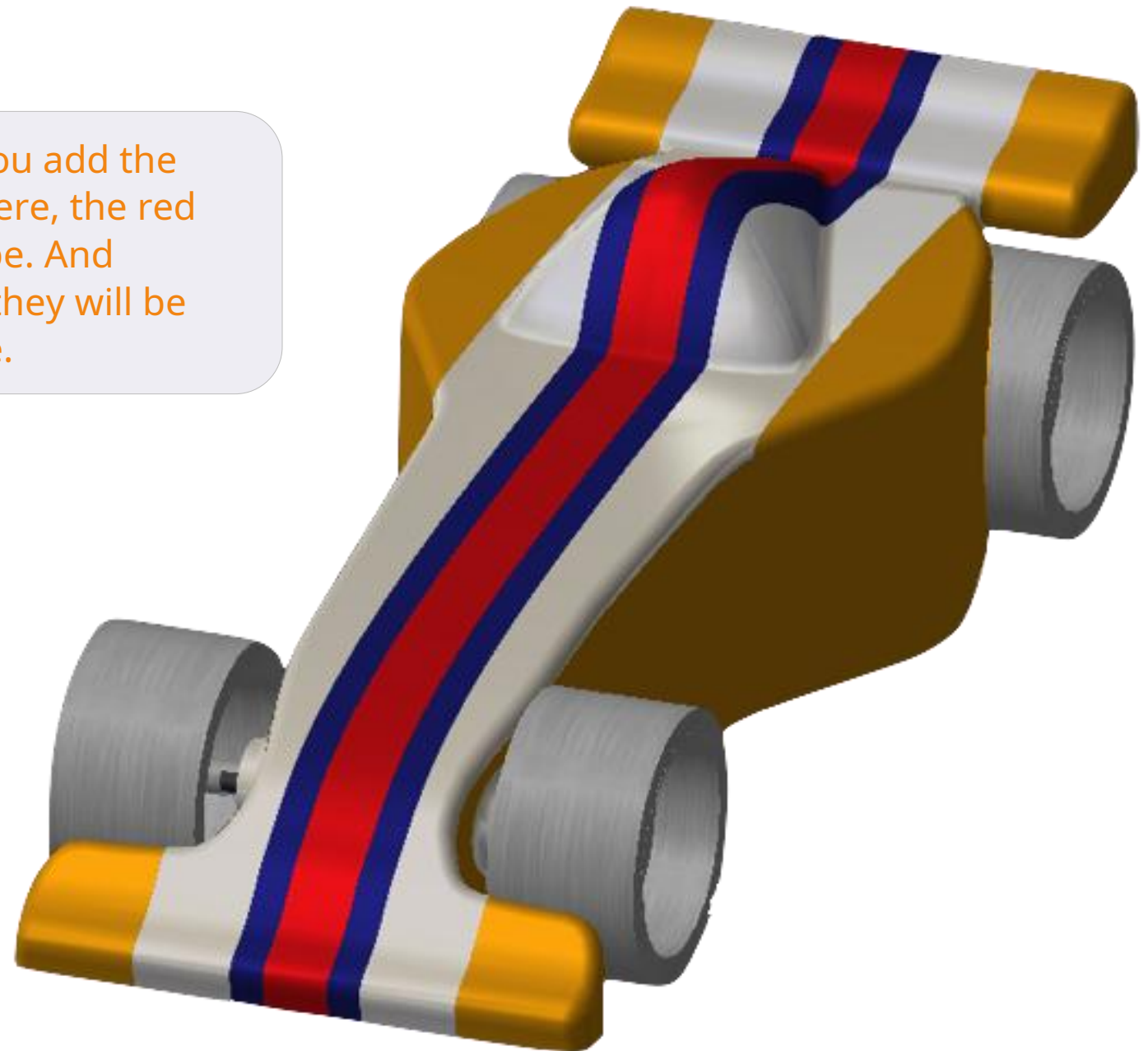
## decals

Now you know how to insert sticker, move sticker, edit sticker.

You can repeat the operation as many times as you wish to obtain, for example, against.



Remember that the order in which you add the stickers is important. For example here, the red stripe was added after the blue stripe. And remember to rename your stickers, they will be easier to spot in the specification tree.

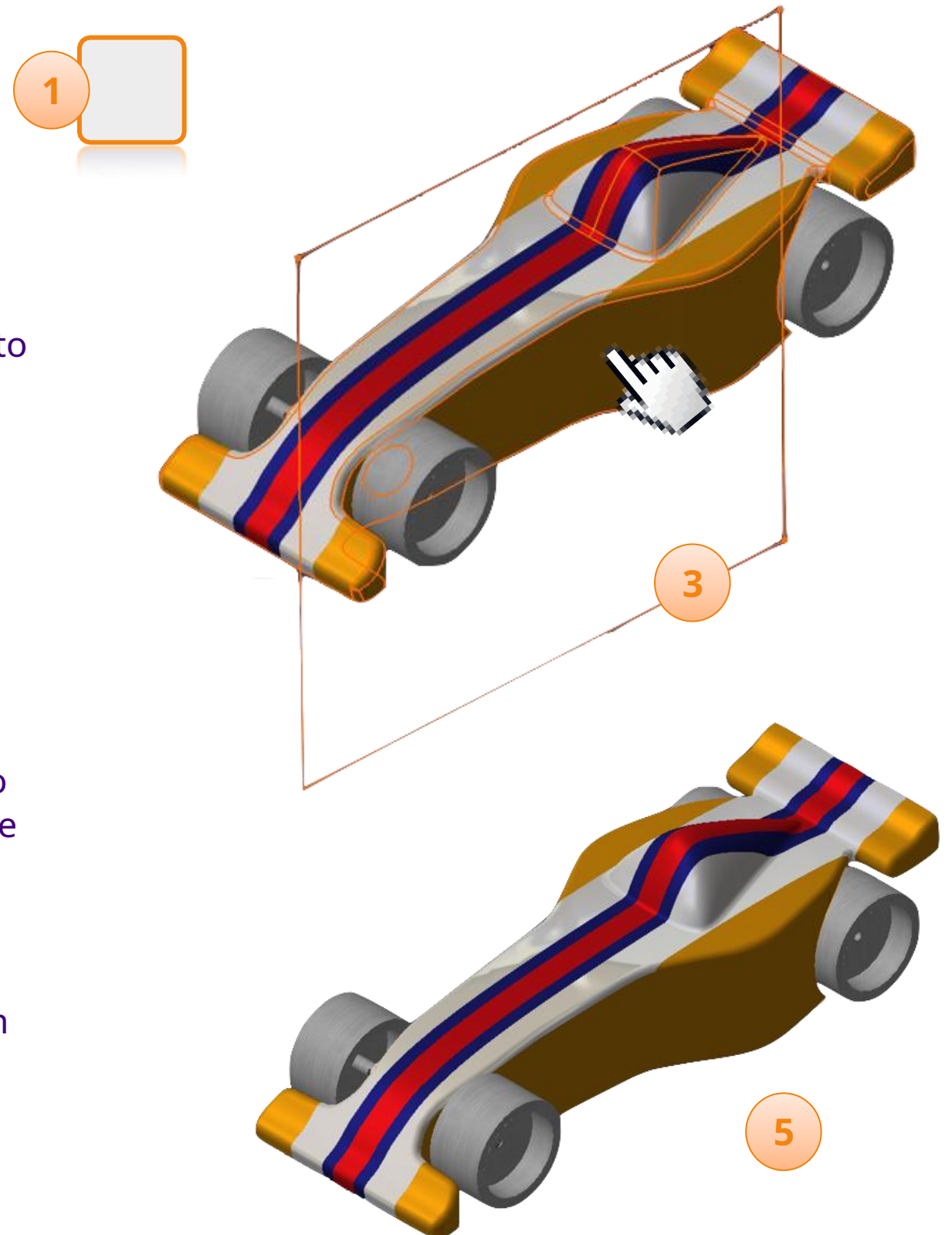




## decals

You can also apply stickers in the other direction. For that :

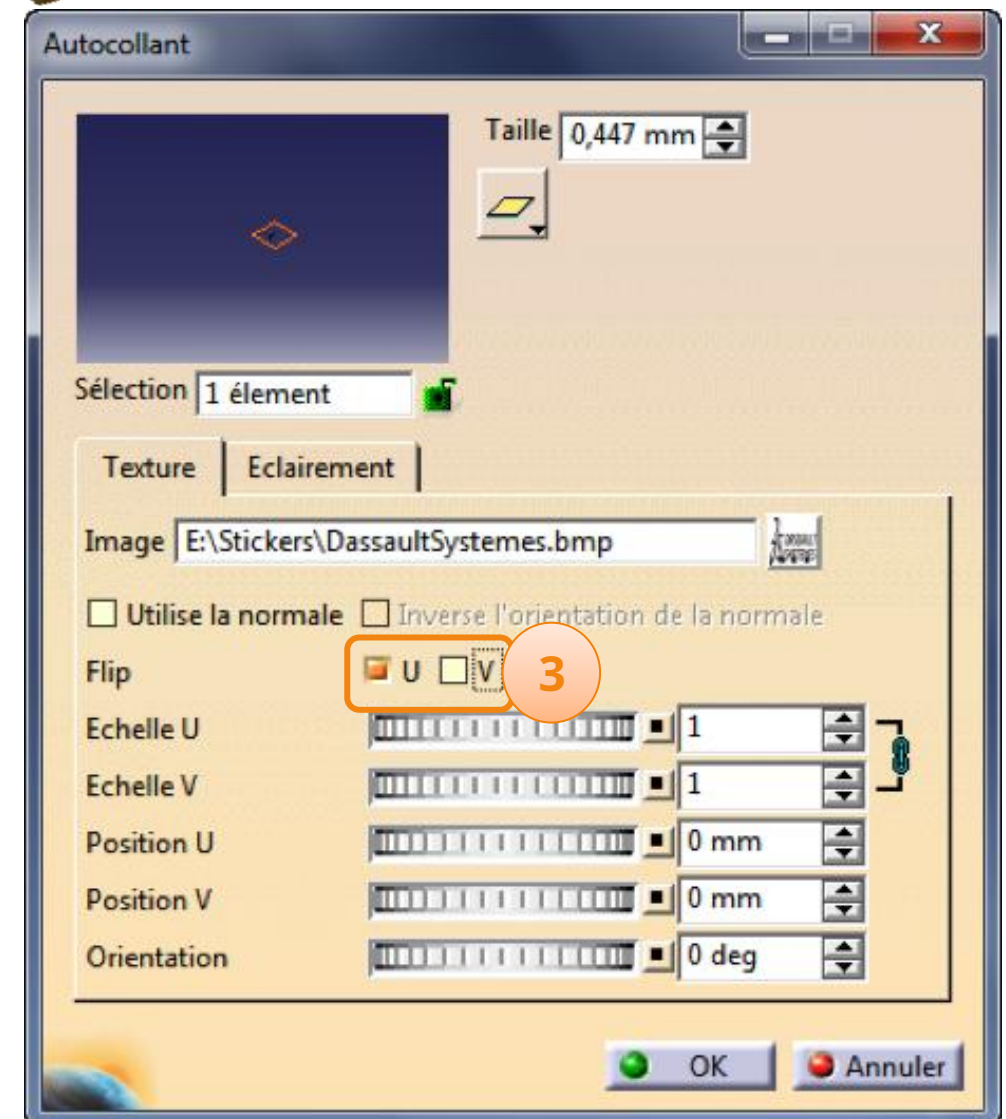
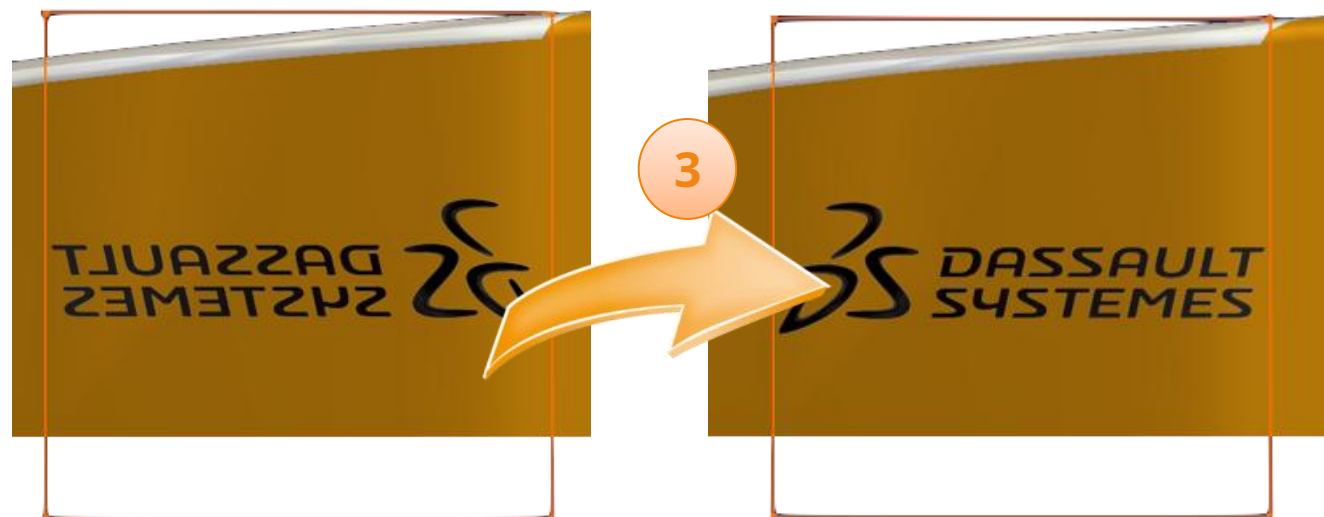
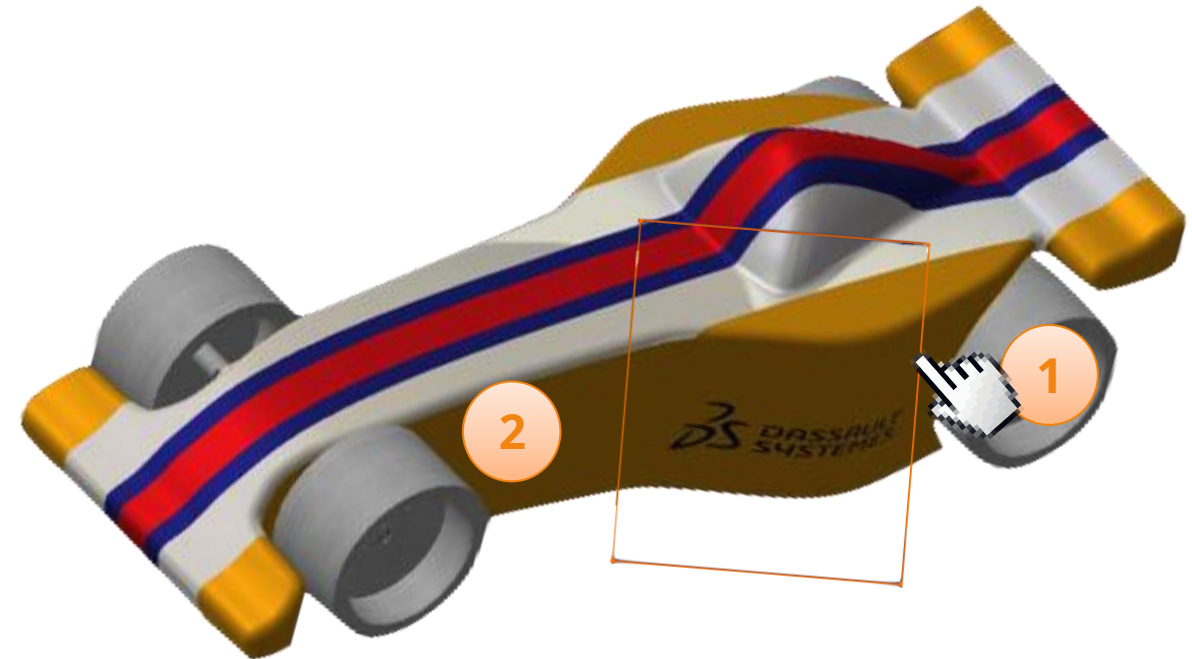
1. Click the icon **Apply a sticker**. The window for configuring the sticker opens.
2. Click on the sticker icon. This will open the window to choose it. Open the file **Yellow Stripe**.
3. Click on one side of the car. It is important that this face be **perpendicular to the ground**. Indeed in this way the application of the sticker will be vertical.
4. Now click on the **Main body** of the room **Car\_Body** to apply the sticker to the entire car, not just the face you just clicked.
5. All you have to do is play with the scale and position parameters to size and move your vertical face.



# decals

In the same way you can apply the logos of your sponsors and decorative logos. Here are some tips for applying your logos:

1. To change the size of a logo use the orange frame that surrounds it by clicking on its corners.
2. Prefer the face selection rather than the main body, otherwise your sticker will appear on two or more faces. The example opposite shows in orange the faces that have been selected for a logo.
3. The logo may be reversed. To correct this problem, use the options **Flip** opposite.





## decals

4. If your decal is on a can declare this color co For this:

has. In the property window double-clicking the autoc specs, click **Illuminance**.

b. Check the option **Use transparent u**.

vs. Click on the icon of the **P**

d. In the small window which color which should be tra the image.

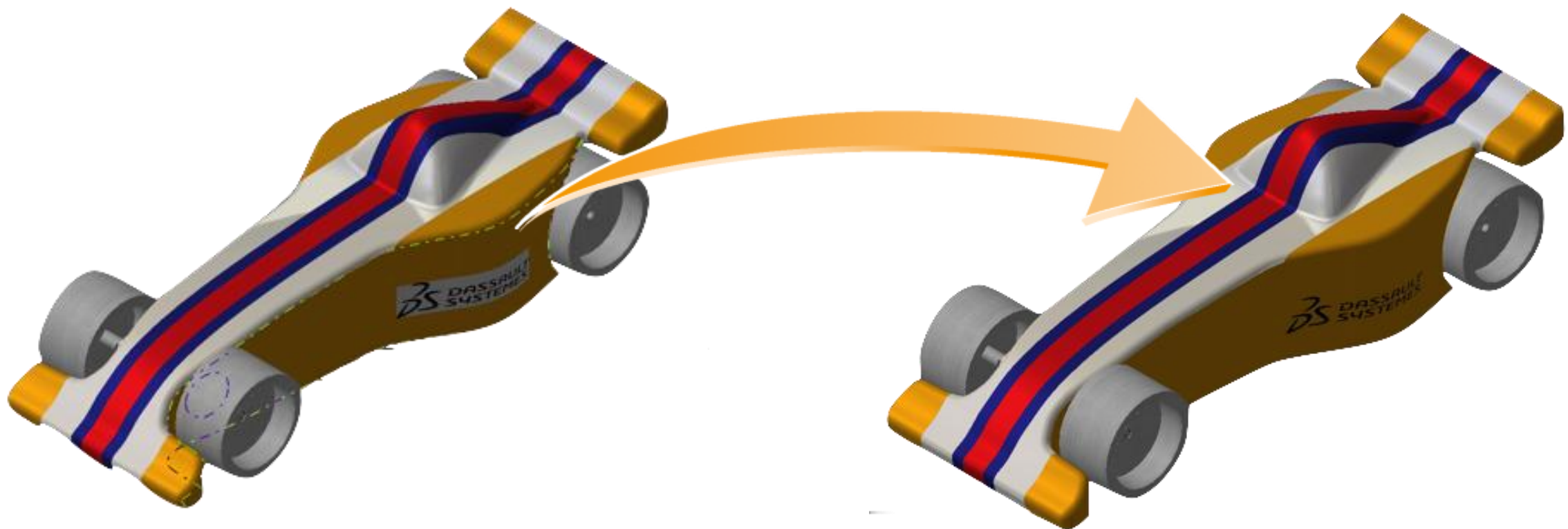
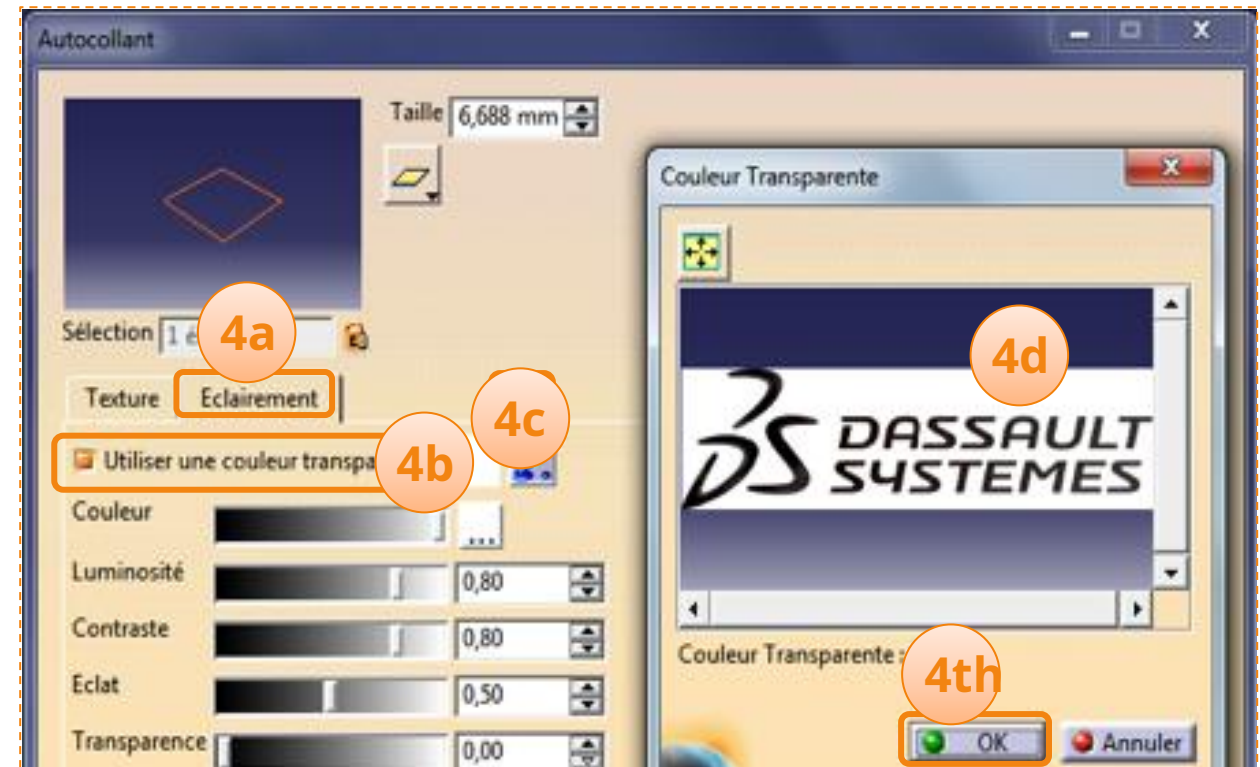
e. Click on **okay**.

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# Realistic Rendering in CATIA – Renderings

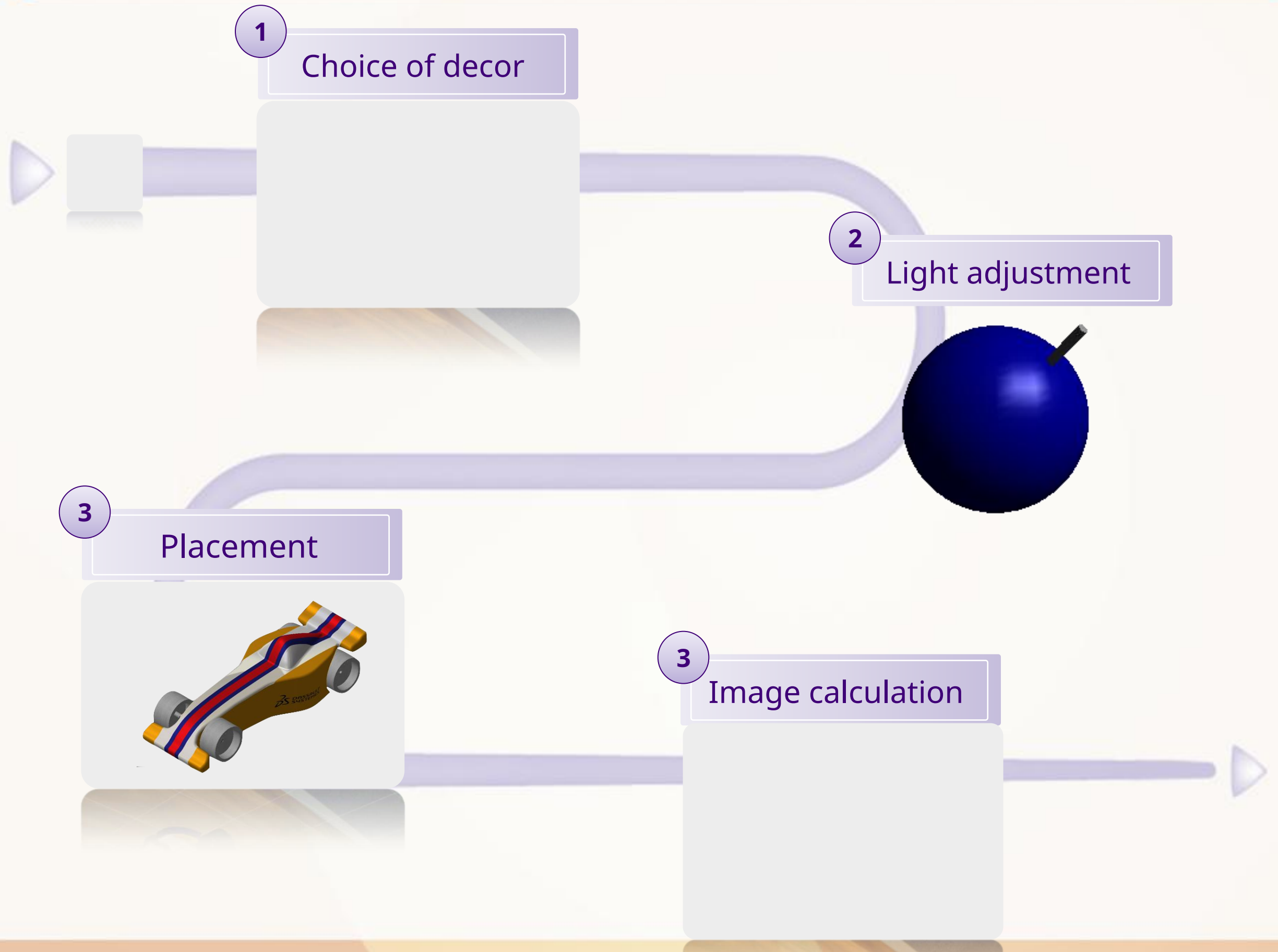
Finally we will see how to generate the realistic image of your car.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. Your racing car.**
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.**
    - i. Introduction.
    - ii. Painting.
    - iii. Decals.
    - iv. Returns.**
      - i. Simulate machining
4. Method sheets.

## Renders





## PhotoStudio Easy Tool

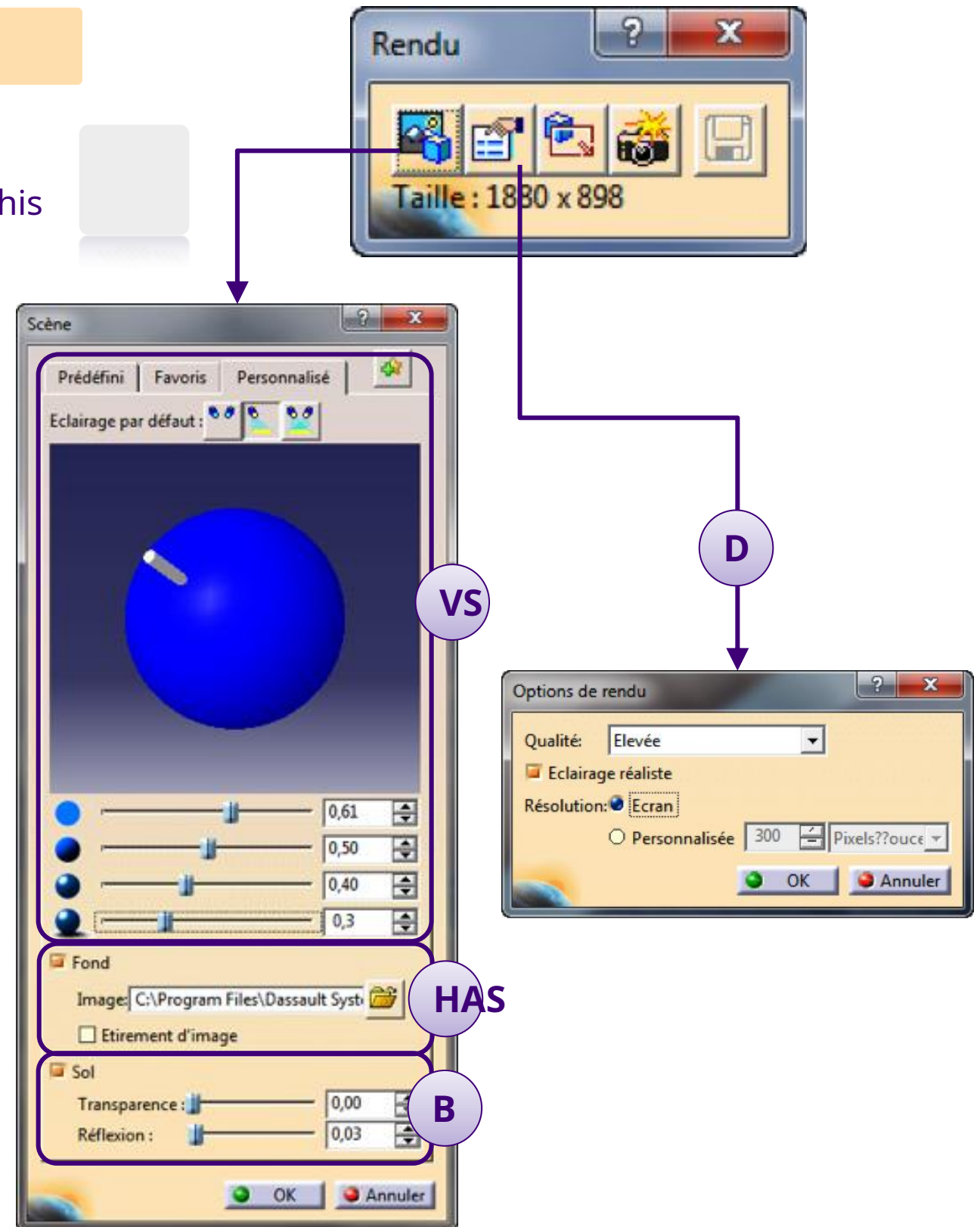
To generate the image we will use the rendering method offered by the Photo Studio Easy Tool. This allows you:

**HAS** To add decor.

**B** To manage the reflection or the transparency of the ground.

**VS** Easily control lighting, and realistic shadows.

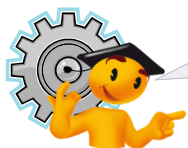
**D** Configure the rendering quality.



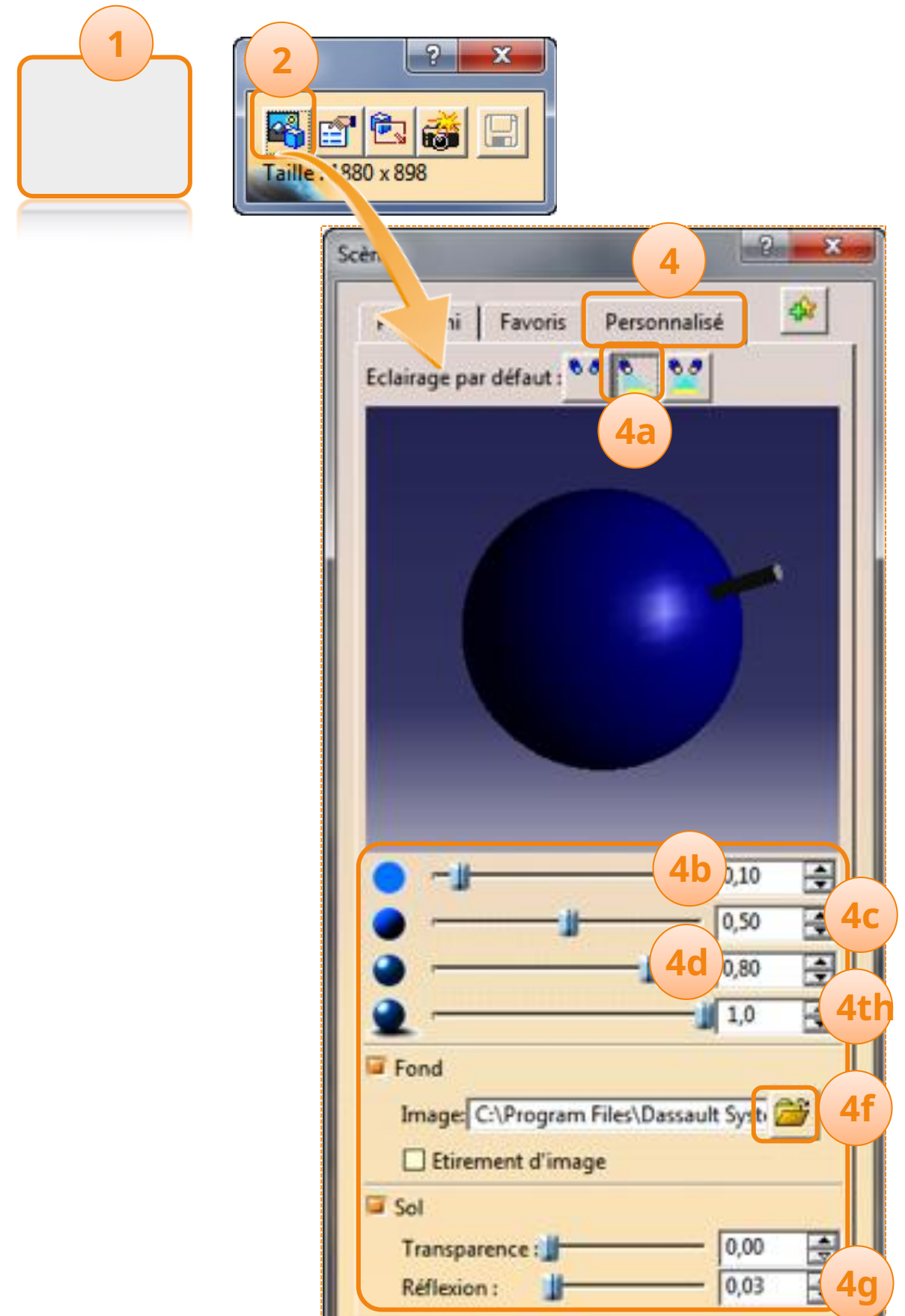
# Renders

Let's start by setting up the scene. For that :

1. Click the icon **Photo Studio Easy Tools**.
2. Click the icon **Select scene**.
3. Select the image of your choice or if none suits you, go to the next step.
4. Click on the tab **Custom** and set the following parameters:
  - has. Default lighting: **a light**.
  - b. Ambient: [0.10].
  - vs. Diffuse: [0.50].
  - d. Specular: [0.80].
  - e. Shading quality: [1.0].
  - f. If you want to use a custom image, click the icon **browse image**.
  - g. Activate it **Ground**, and the reflection at [0.06].



Characteristics of a light source.



## Renders

If the default lighting does not match your image (orientation, colors) you can configure it this way:

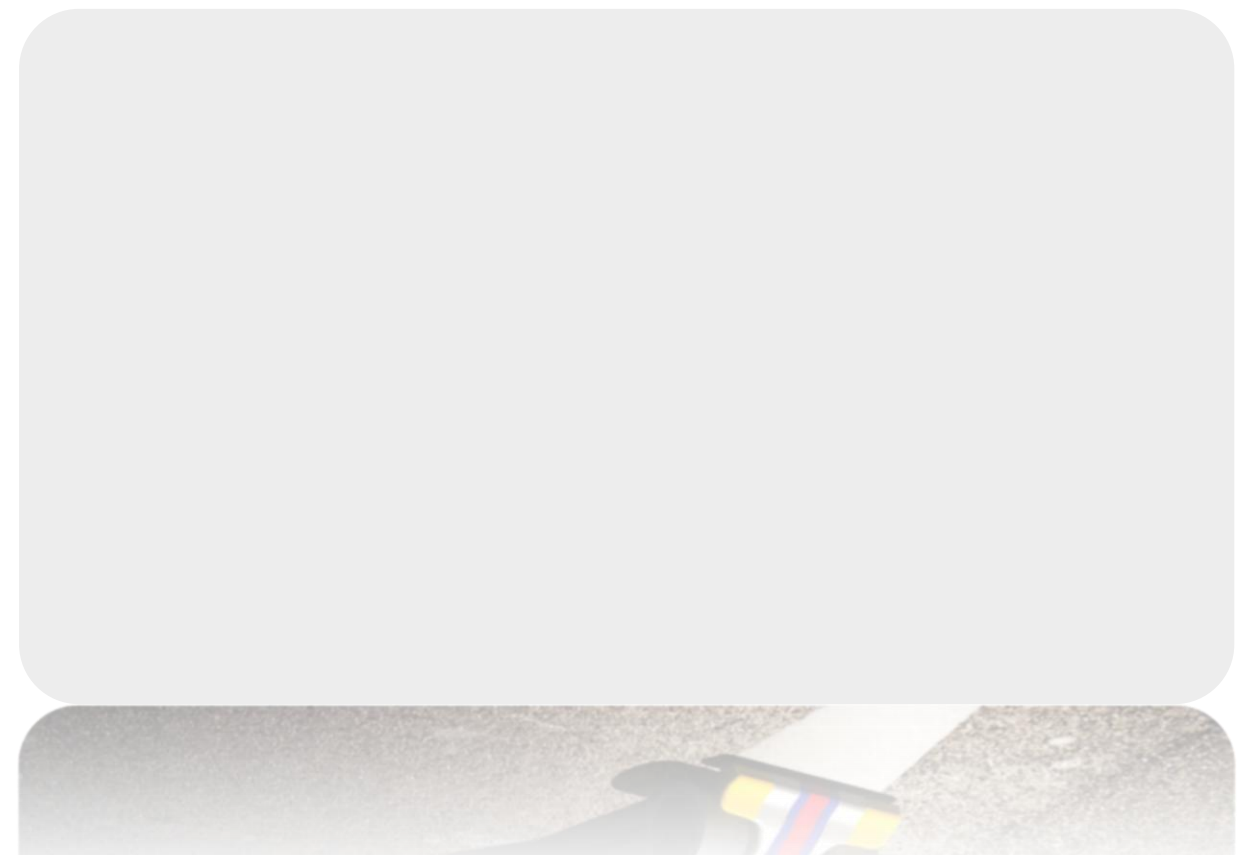
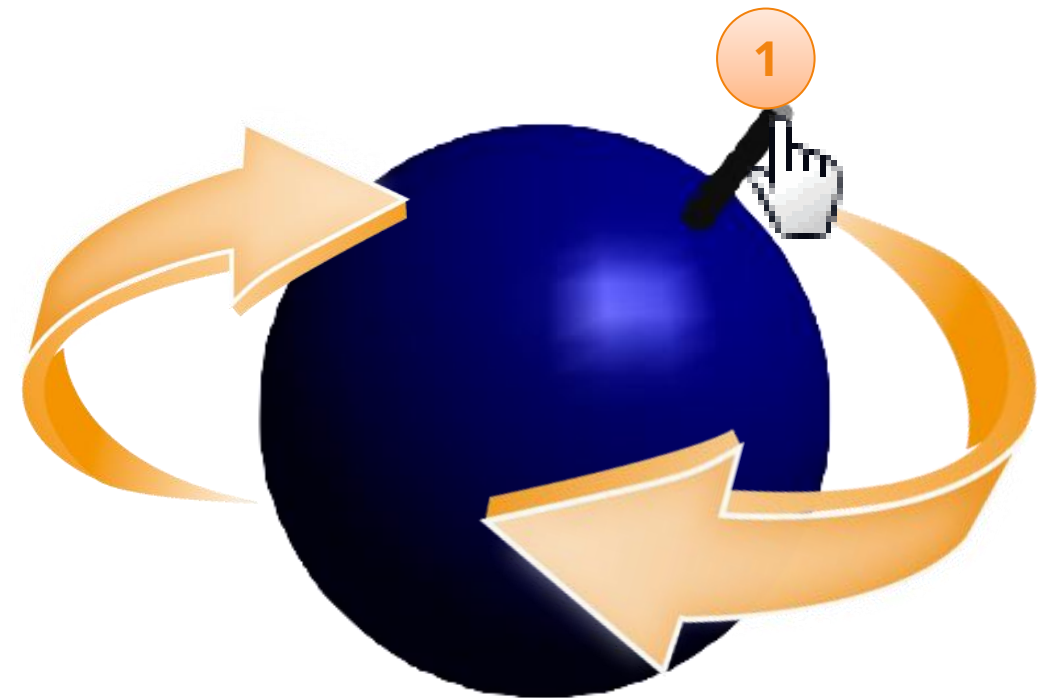
1. Double click on the lamp to be able to change the color. For natural lighting, use white.

Here are some settings depending on the circumstances:

**has.**Sunsets: [R=255];[G=205];[B=105].

**b.**Backlight:      Ambient: [0]  
                         Diffuse: [0.95]  
                         Specular: [1]

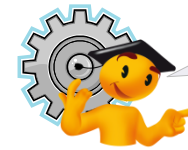
2. By clicking and holding the click on the lamp you can change its orientation by moving the mouse. You can see the changes in real time on the car.
3. When the lighting suits you, you can click on **okay** to validate the lighting configuration.



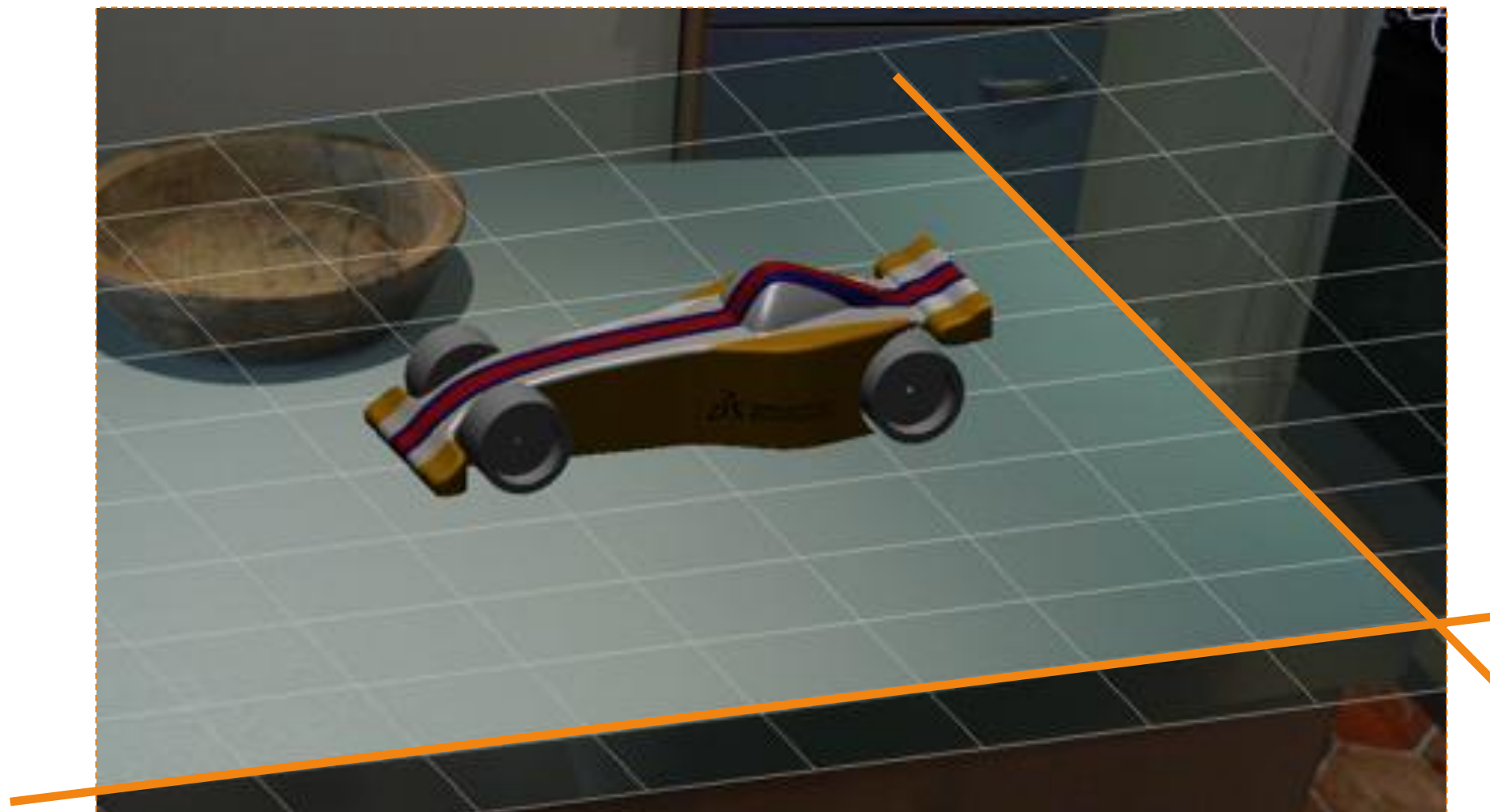
# Renders

Now try to place your car in your environment. To help you:

1. The grid represents the ground, the shadow and the reflection will apply on it.
2. Try to match the gridlines to the perspective of your photo. The car should look like it fits into your decor.



Perspective effect, and point leak.

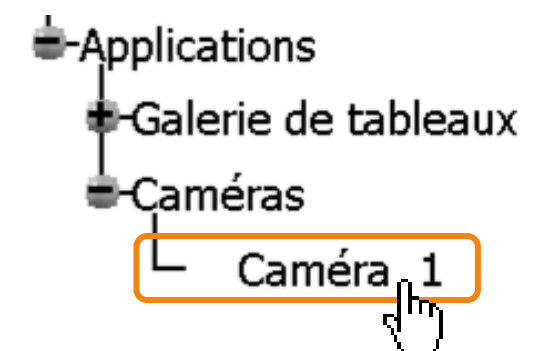
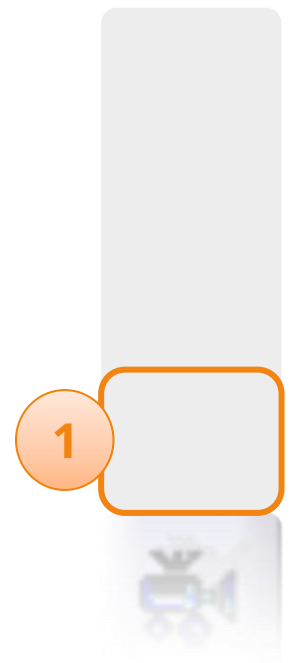
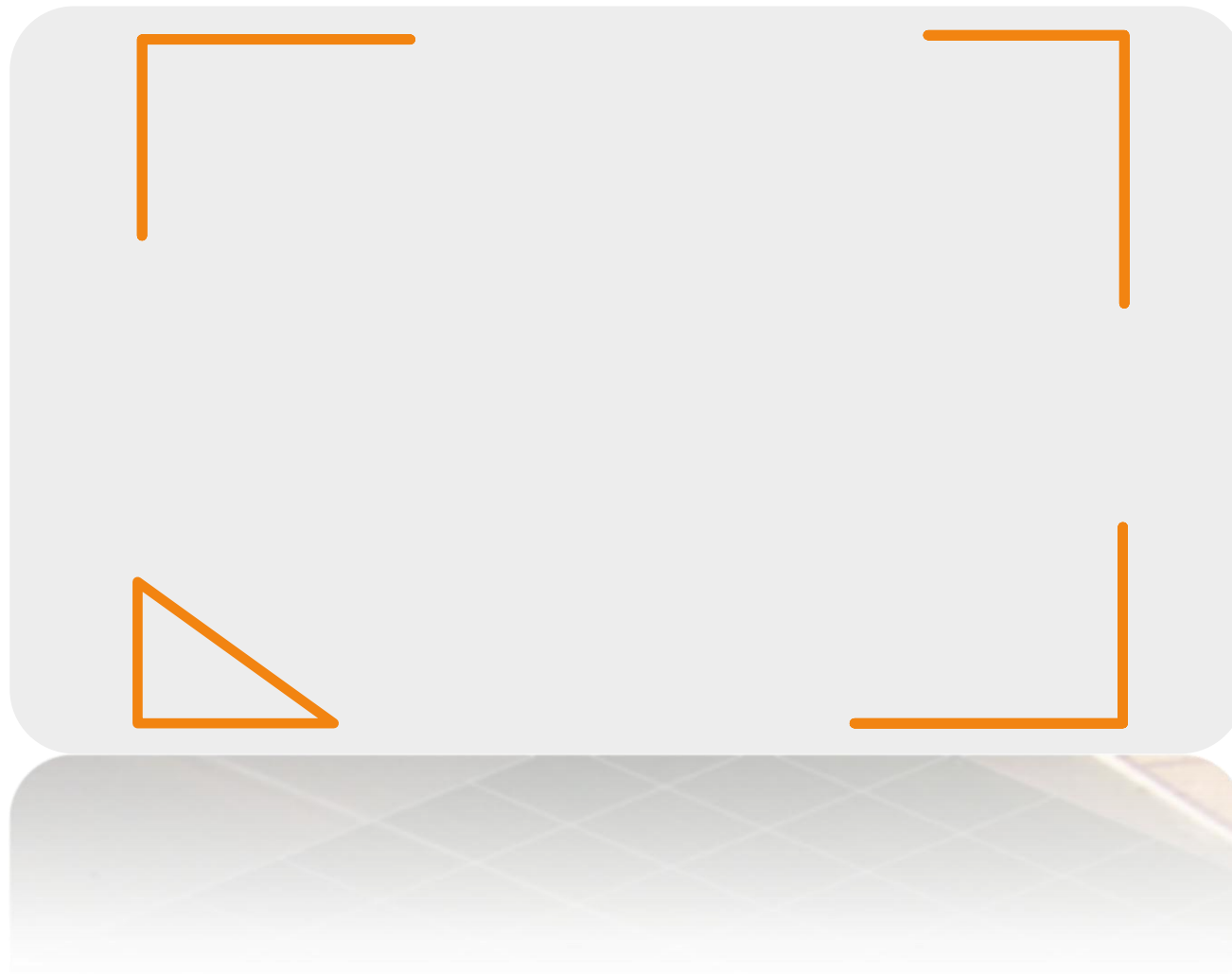


## Renders

When the view suits you, you can save it by creating a camera. For that :

1. Click the icon **Create a Camera**.

You will notice that a frame has appeared, and a camera is now available in the specification tree.





## Renders

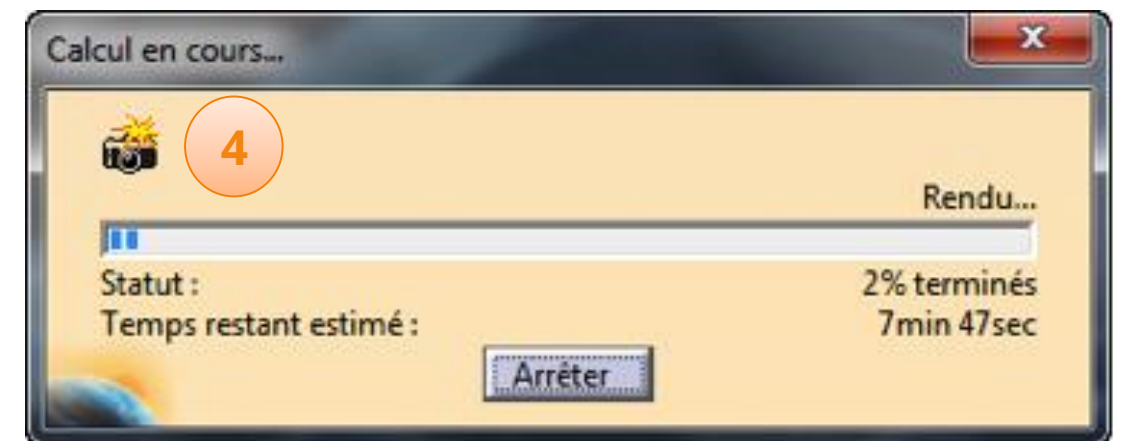
To avoid having to do a long calculation, you can designate a calculation area that will allow you to check the correct settings for your scene. For that :

1. Click on **Define rendered area**.
2. Mark out a small area that allows you to control the lighting on the 3D model.
3. Click the icon **Render**.
4. Wait...

You can now analyze the computed image plot to choose whether or not you can start computing the full image.

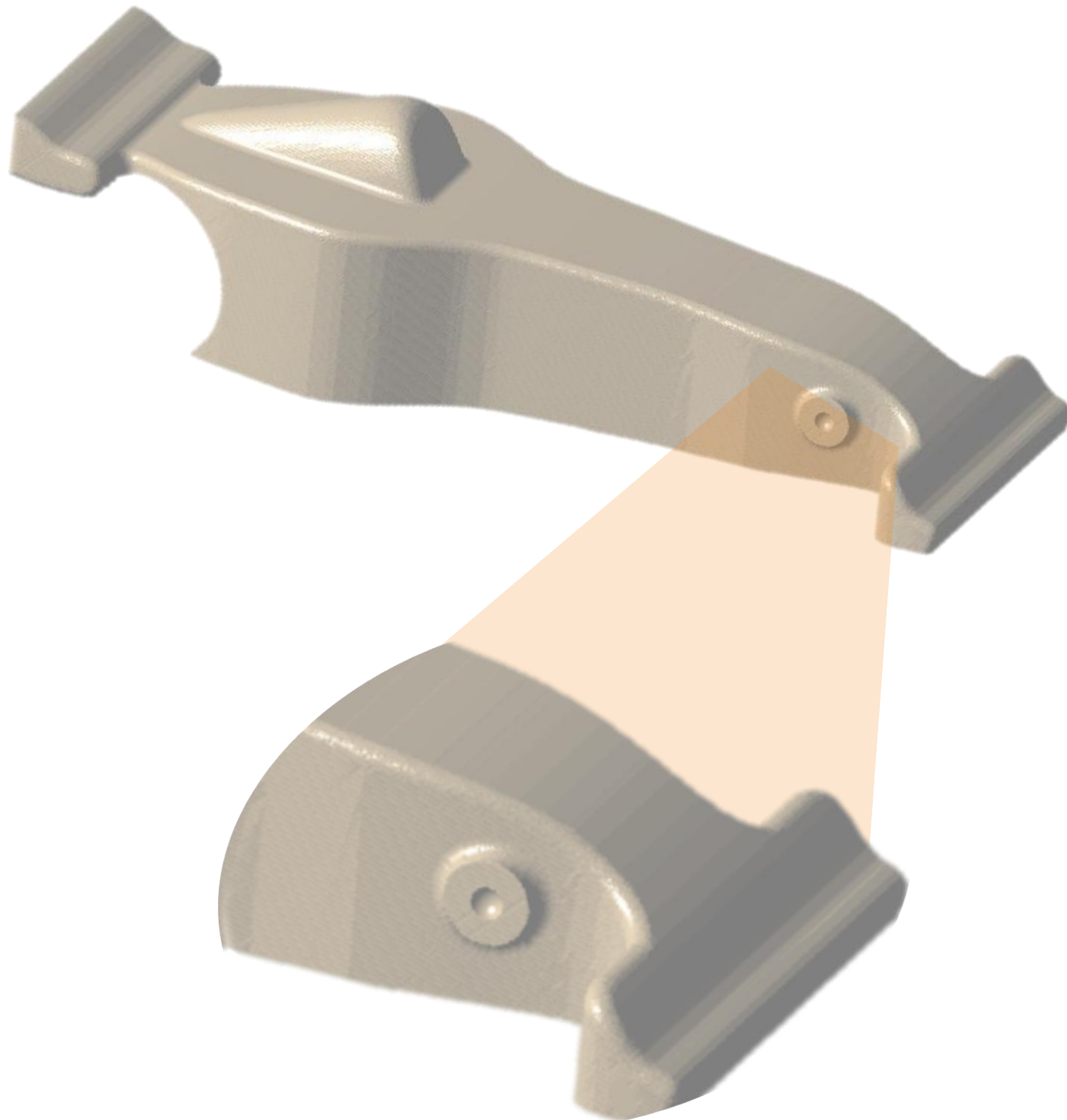
Here is the final step, you will launch the calculation of the image.  
For that :

5. Click the icon **Render**.
6. Wait... the time to calculate an image varies depending on the capabilities of your machine and the details of the image (reflection, transparency, lighting,...).



# Simulate machining

In this part you will learn how to carry out the virtual machining of your car.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
- 3. *Your racing car.***
  - has. Draw your car.
  - b. Open your project.
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  - d. Model your car.
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  - g. Control and modify your car.
  - h. Create realistic renderings.
  - i. Simulate machining.***
    - i. Machining process.
    - ii. Generate an APT file.
    - iii. Generate an ISO file.
4. Method sheets.

## Simulate machining

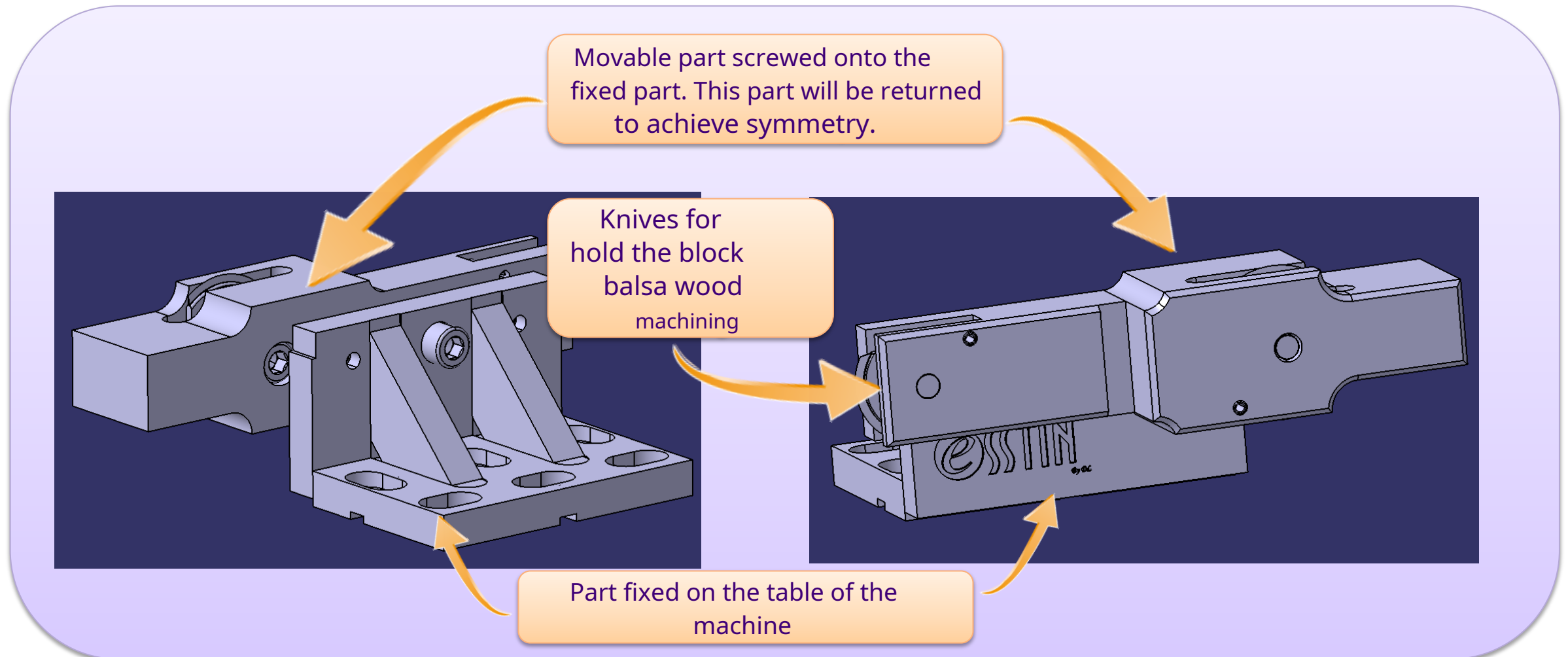
This tutorial will introduce you to the basics of CAD/CAM (Computer Aided Manufacturing Design).



### Important :

The parameter settings given in this tutorial are designed so that the machining is carried out at the ESSTIN with the Stratoconcept machine and a very specific set-up. These settings will not be the same in the case of machining with another machine of the Charlyrobot 4U type.

- Presentation of the ESSTIN assembly:



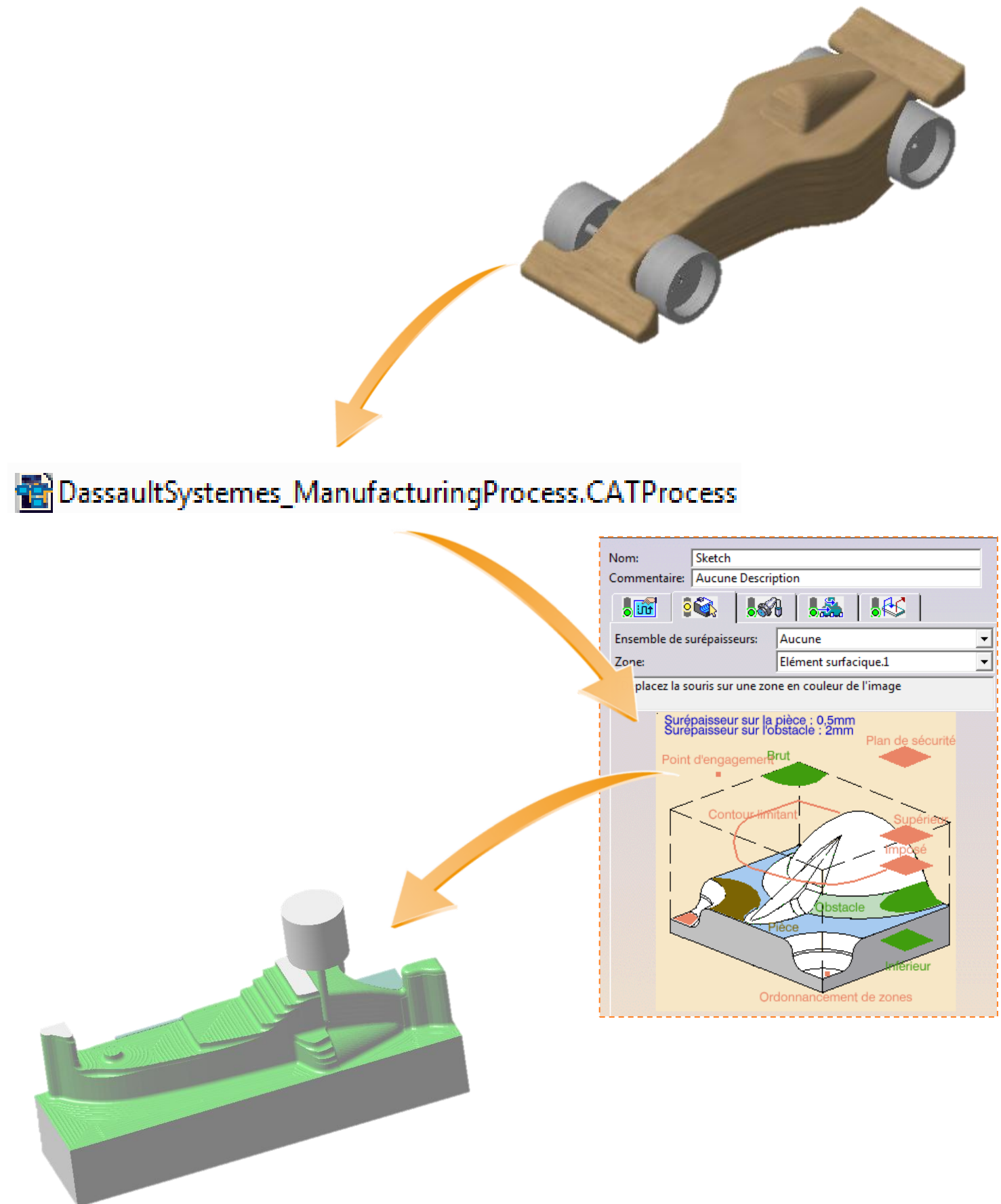
# Introduction to Computer Aided Manufacturing (CAM)

- ▶ The manufacturing process of your car consists of several stages. This chapter will show you how to create a manufacturing process to drive a 3-axis machine with fixture.

- ▶ Advantage: as the process is integrated with CATIA, you can modify your design and update your process without having to start all over again.

- ▶ You will go through the following steps to create the CAM process:

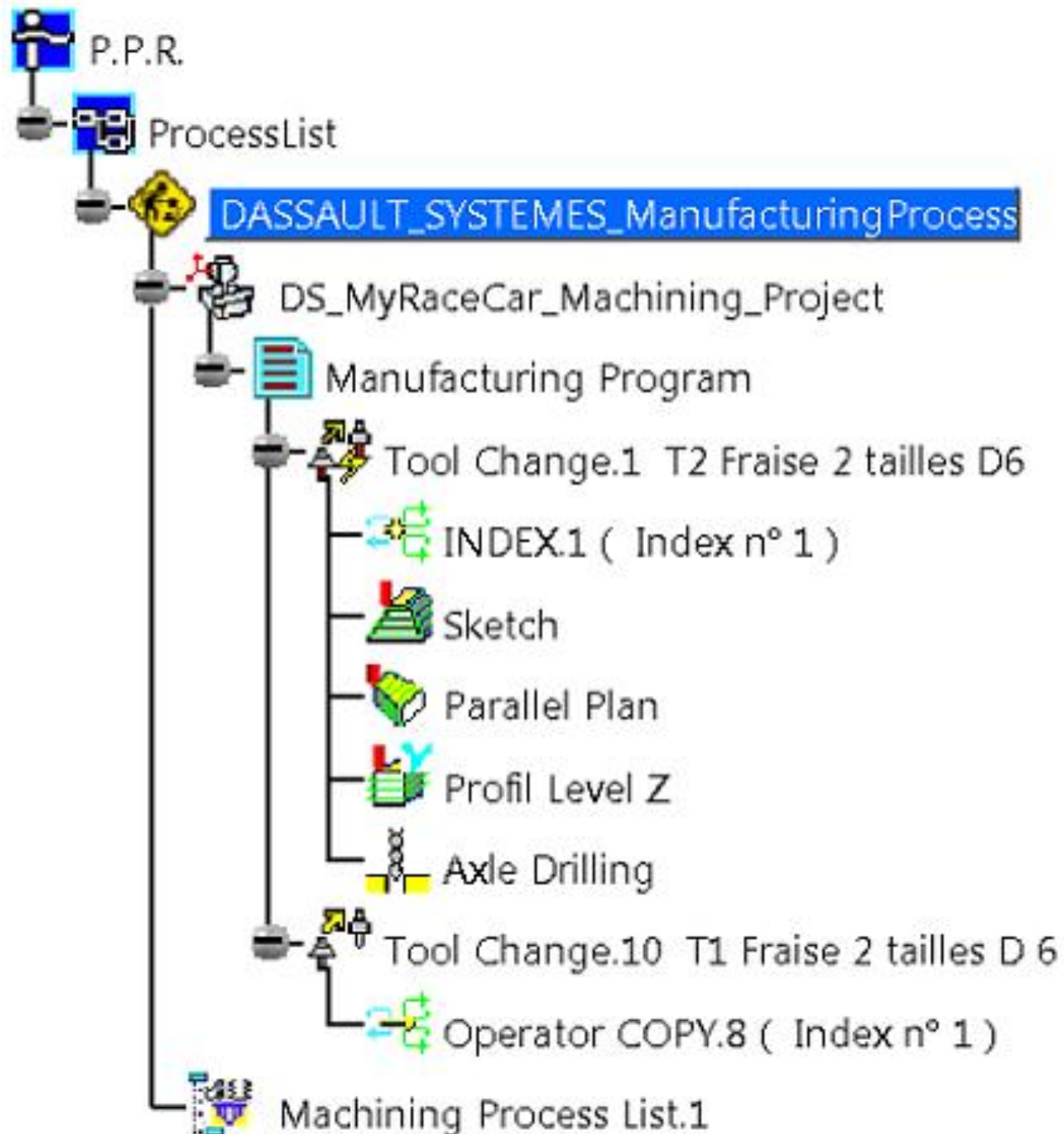
- Open a predefined machining process in CATIA V5.
- Define the geometries to be machined.
- Simulate and check the smooth running of the machining.





# Machining process

In this part you will learn how to carry out the virtual machining of your car.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
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  - has. Draw your car.
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  - f. Create a drawing.
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  - h. Create realistic renderings.
  - i. *Simulate machining.*
    - i. *Machining process.*
      - ii. Generate an APT file.
      - iii. Generate an ISO file.
4. Method sheets.

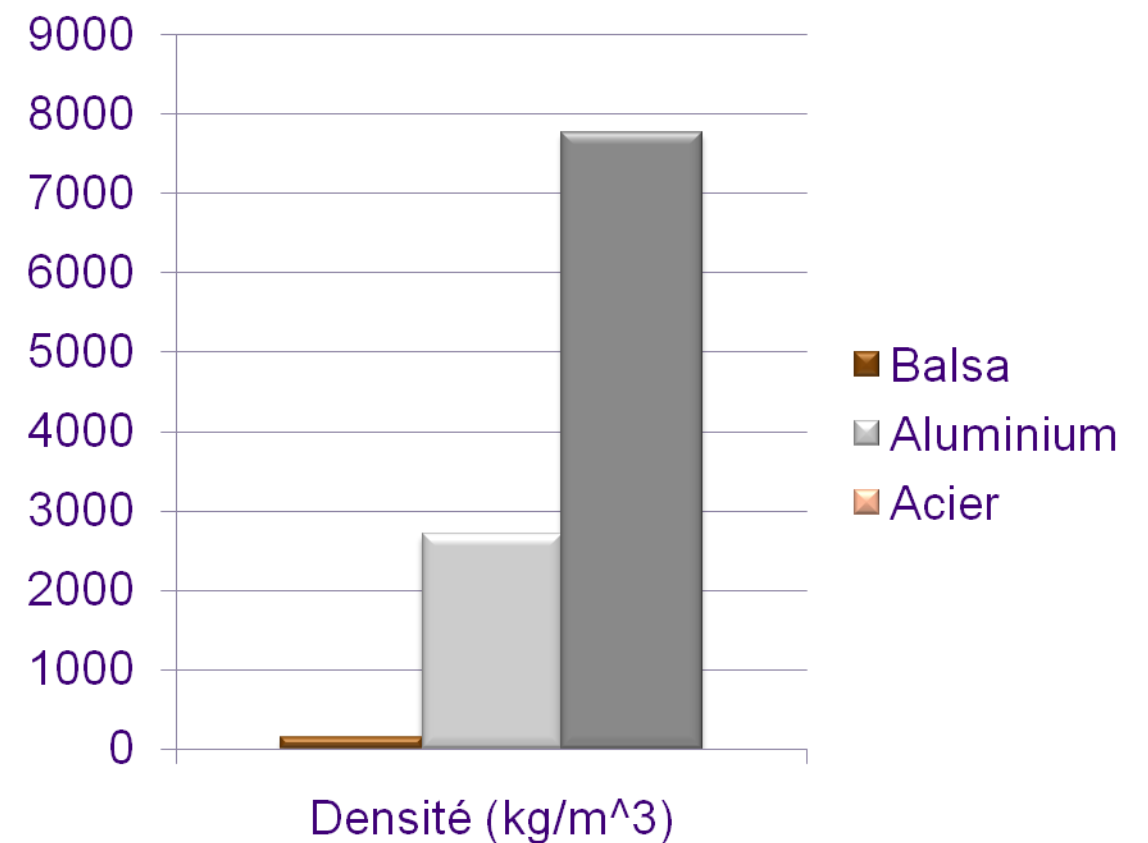
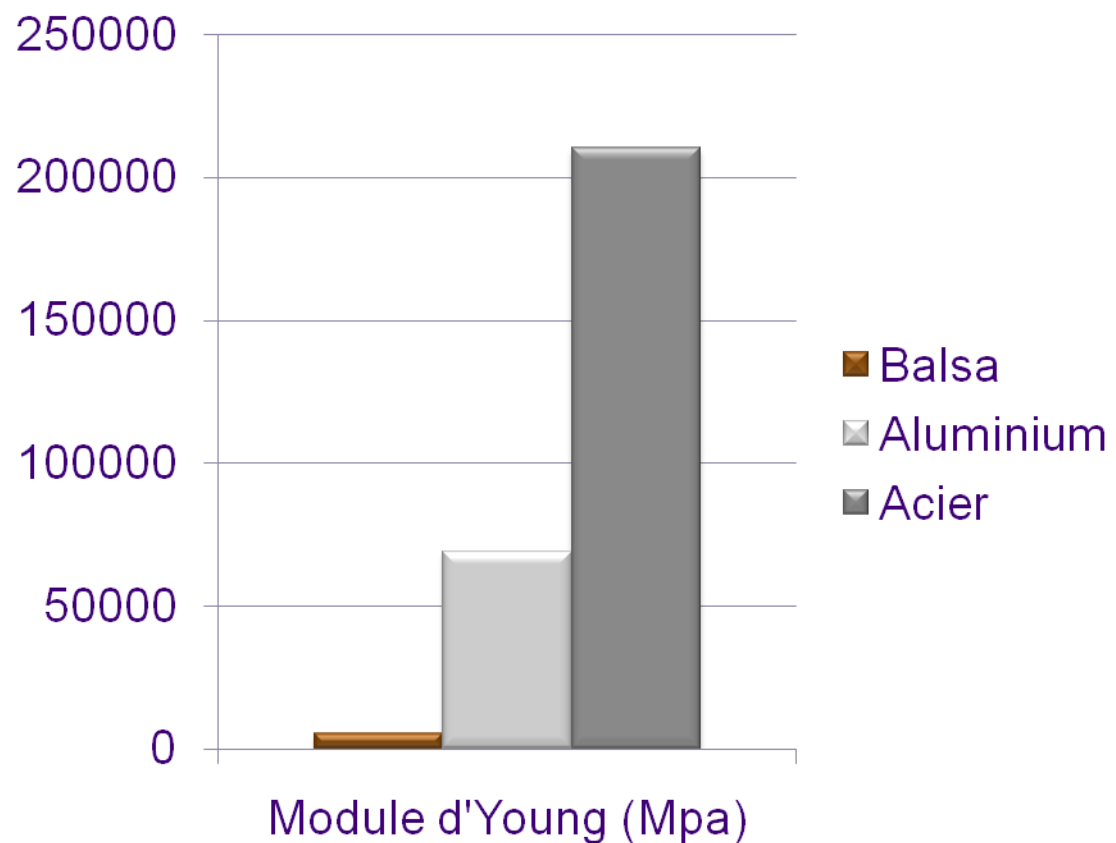


# Machining process

You will now complete a predefined machining process for CATIA. In CATIA you can machine using different methods, and many of them could have been used to machine your car.

Below are the different characteristics of the predefined process.

► **The mechanical characteristics explain why it is easy to machine balsa compared to metals like aluminum or steel.**



► 95% of the parameters specific to the machining of the car are predefined.

# Machining process

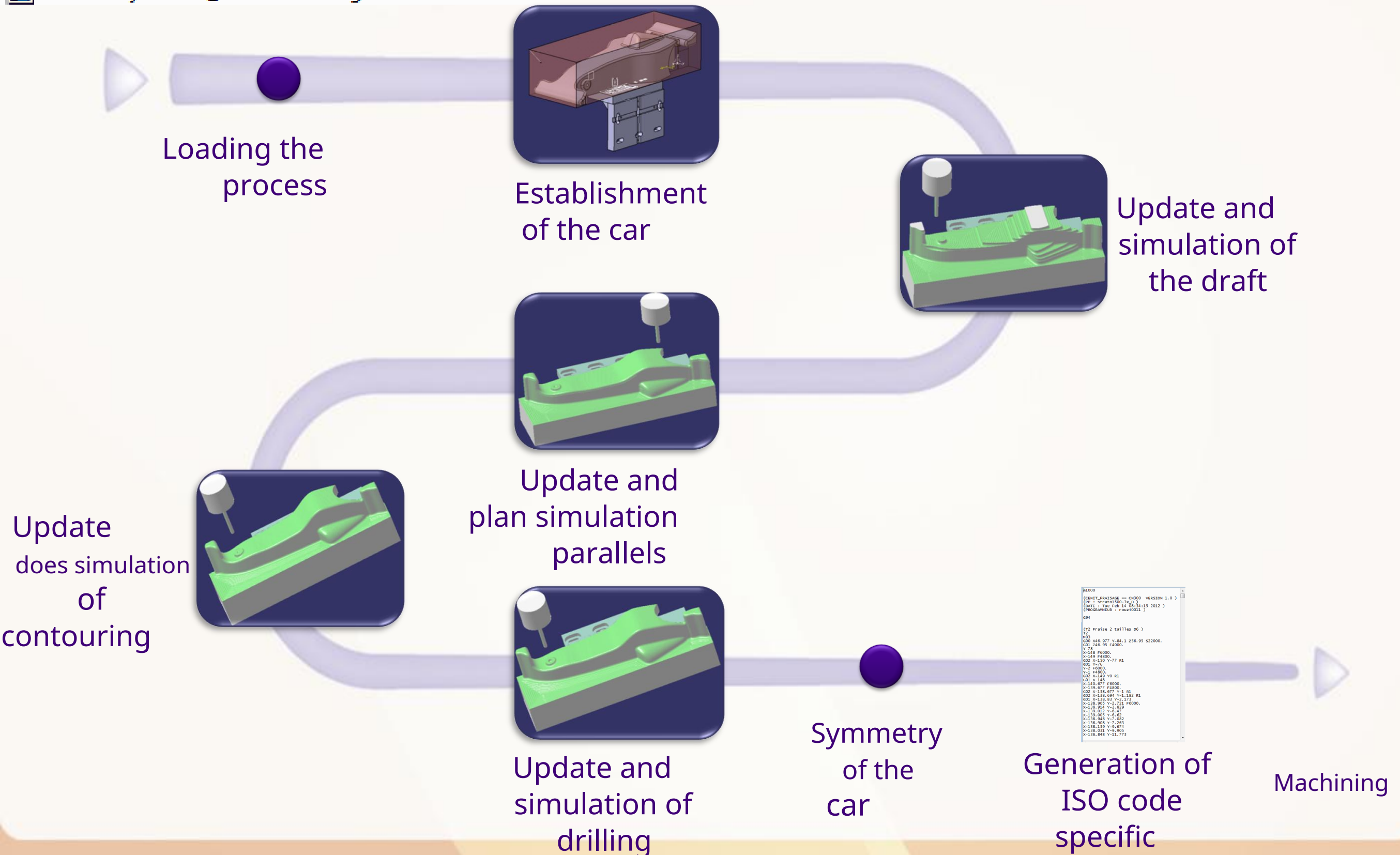
- ▶ All the parameters specific to a 3-axis machine with a specific assembly are predefined.
- ▶ If you want to use a 3-axis machine without specific assembly, you can find the corresponding chapter in the Method Sheets.



Machine Charly Robot "Stratoconcept"

# Machining process

DassaultSystemes\_ManufacturingProcess.CATProcess



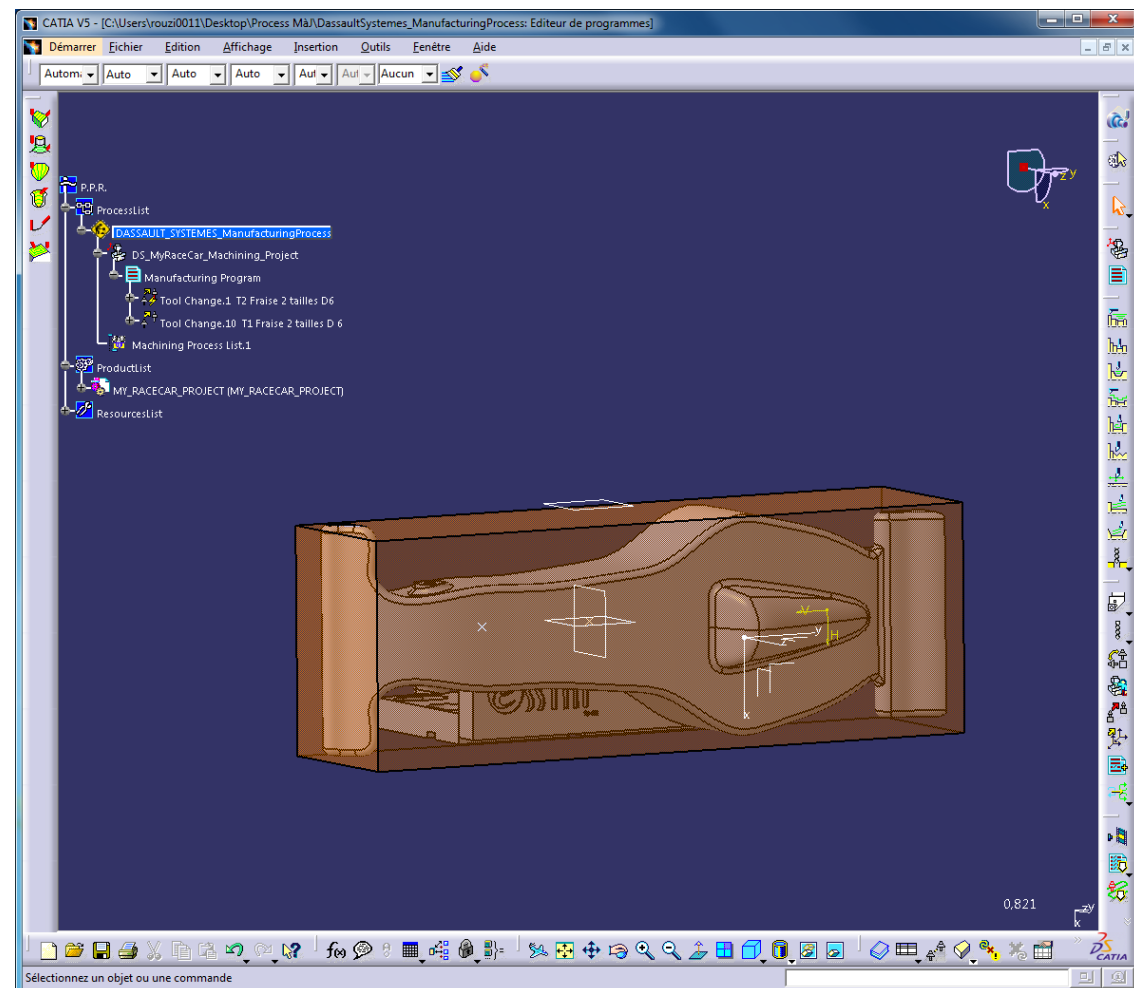
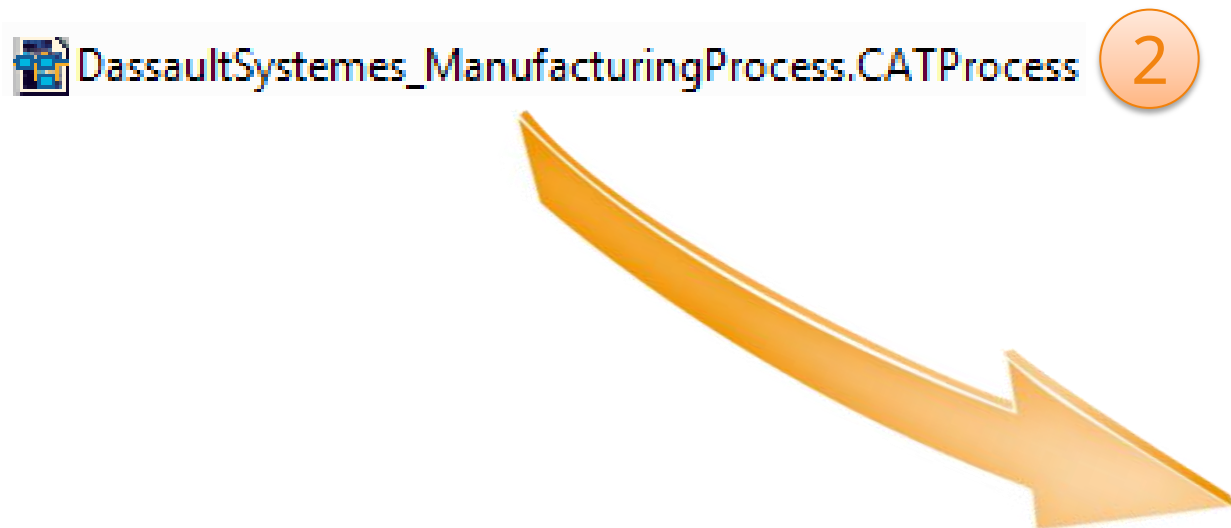
# Machining process

First, you will open the manufacturing process file. For that :

1. Save and close all open documents.
2. Click on **File > Open** and select the file named:  
**DassaultSystemes\_ManufacturingProcess.CATProcess**

As you can see the official car is you will have to do is replace it with your v

on the machining fixture. So the first thing that





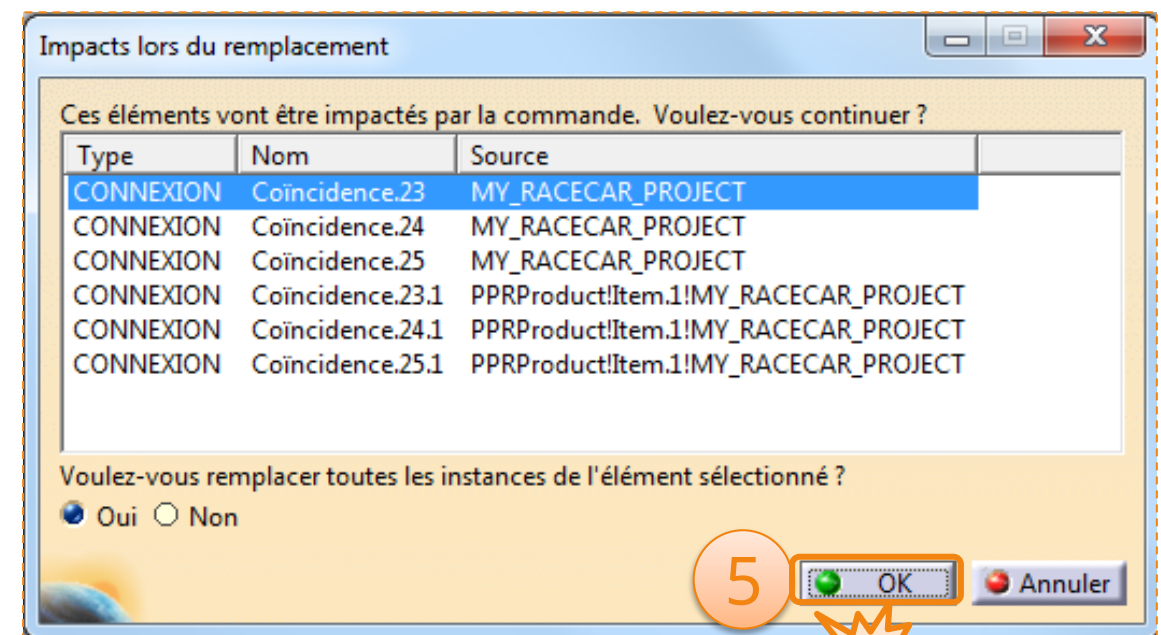
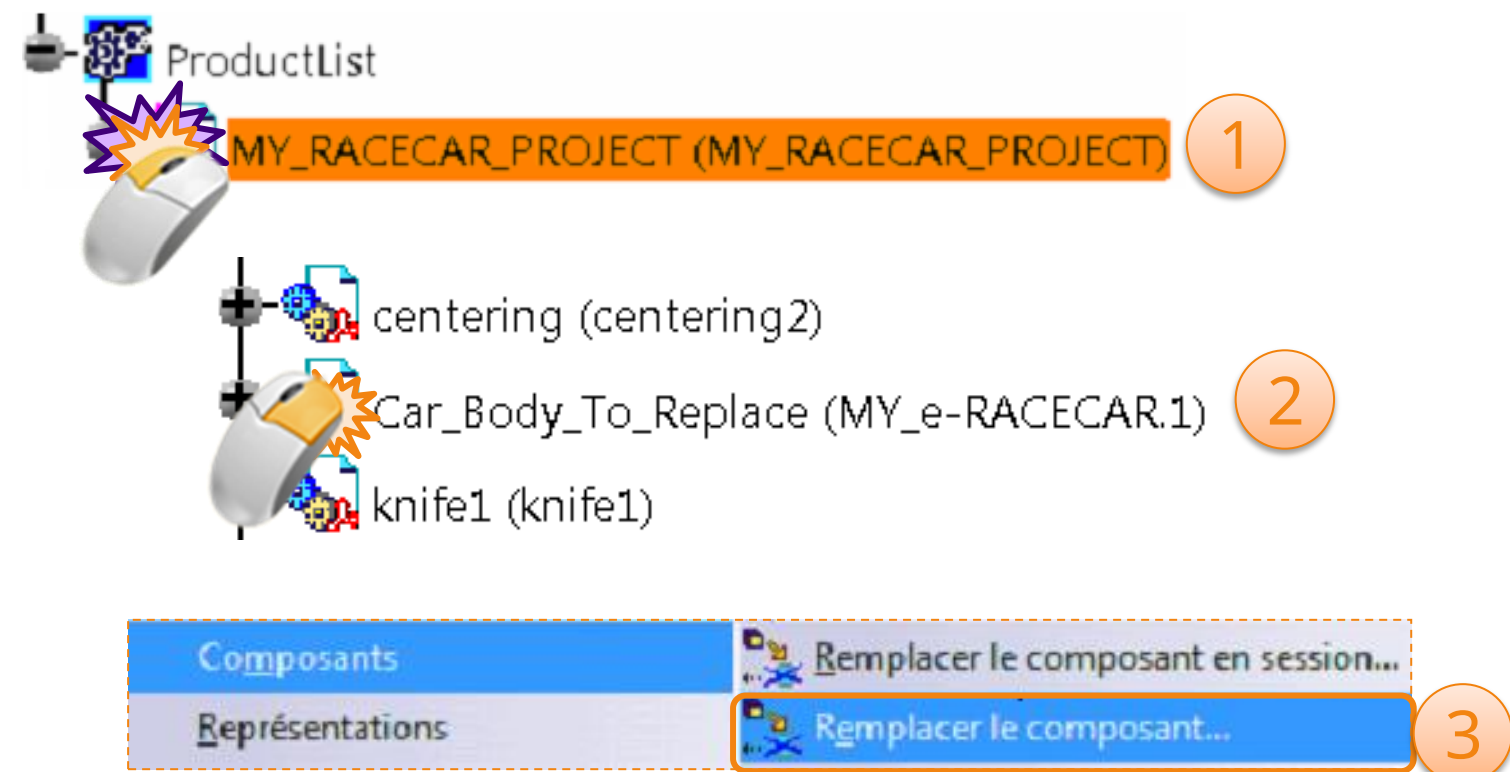
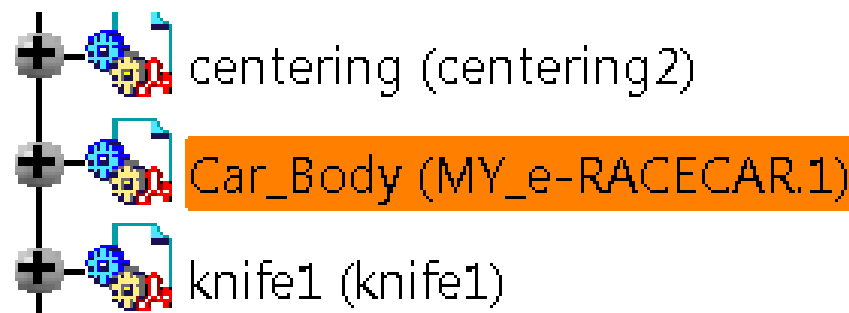
# Machining process

To replace the car:

1. Double click on the product **MY\_RACECAR\_PROJECT**.
2. Right click on the part **Car\_Body\_To\_Replace**.
3. Select **Components > Replace Component**.
4. Choose your file **Car\_Body.CATPart**.

The window opposite appears. It contains the different elements that will be impacted by this replacement.

5. Click on **okay** to validate. You should get the result below.



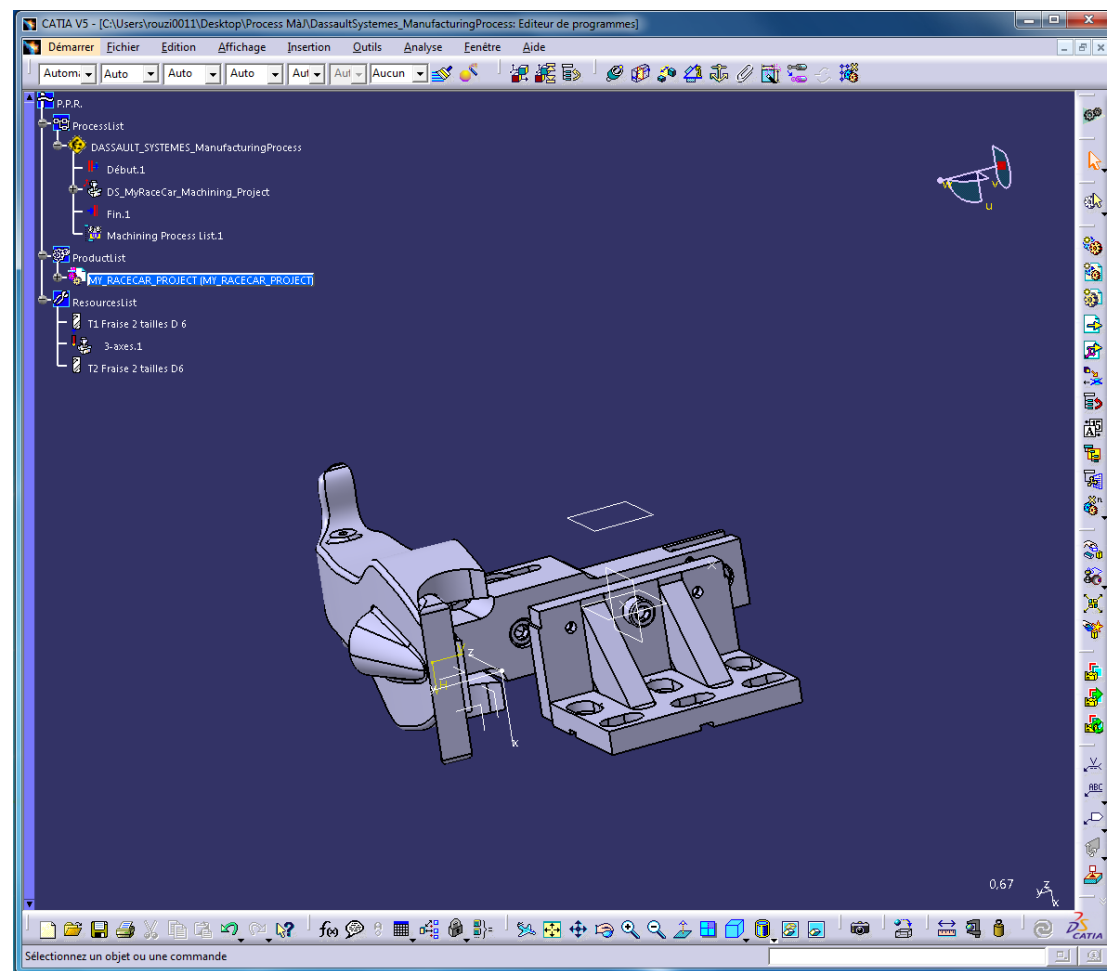


# Machining process

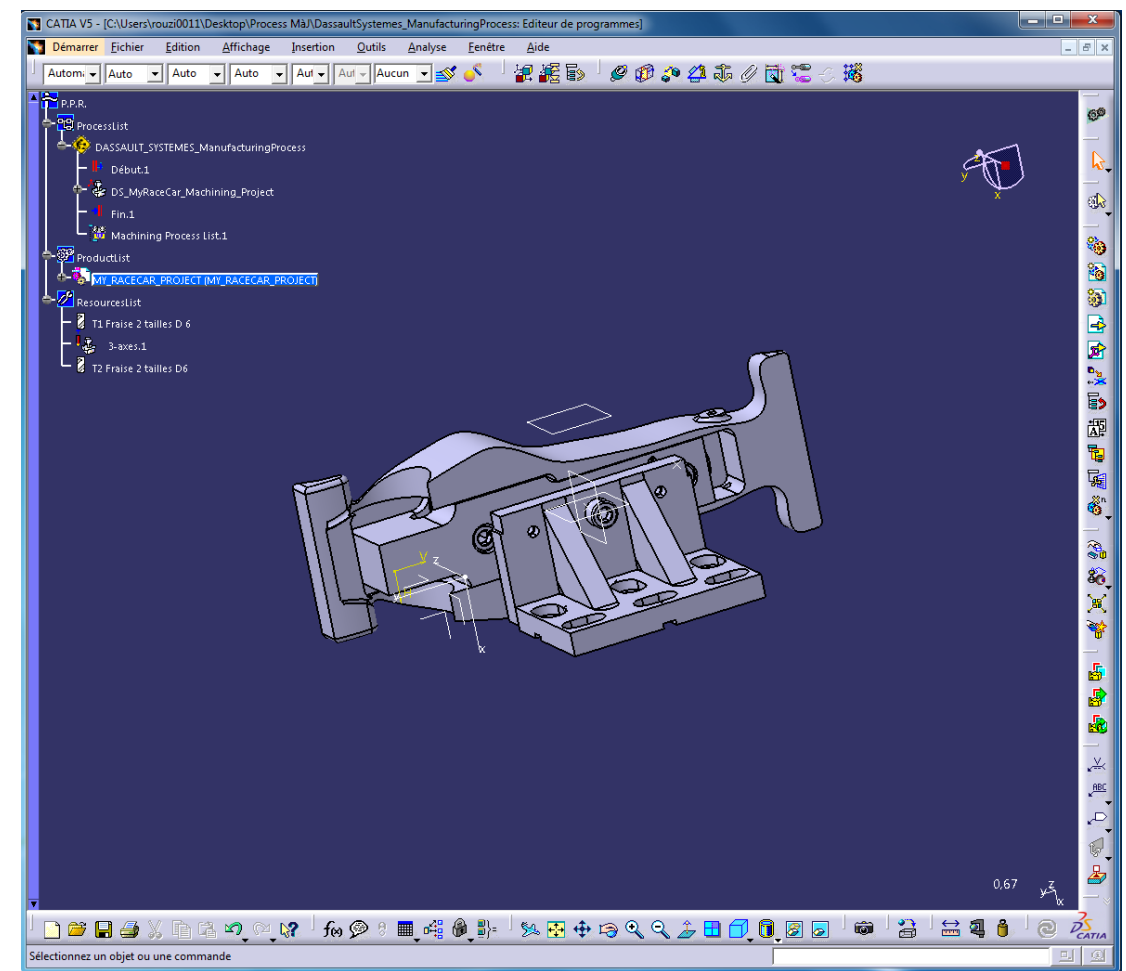
Our car must be in place on the assembly

1. Check that the car is well centered and that the assembly is in place.

If you notice that your car is not well placed on the movable part of the assembly, you will have to reposition it.



A poorly fitted car.

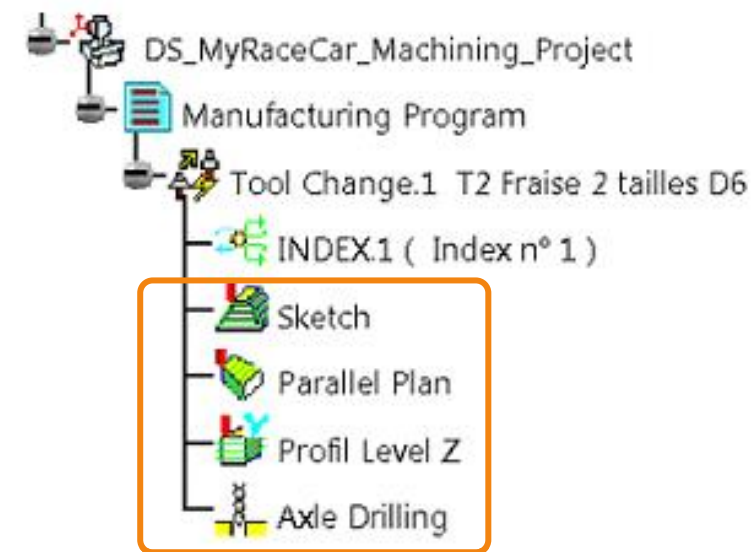


A car puts correctly on the assembly.

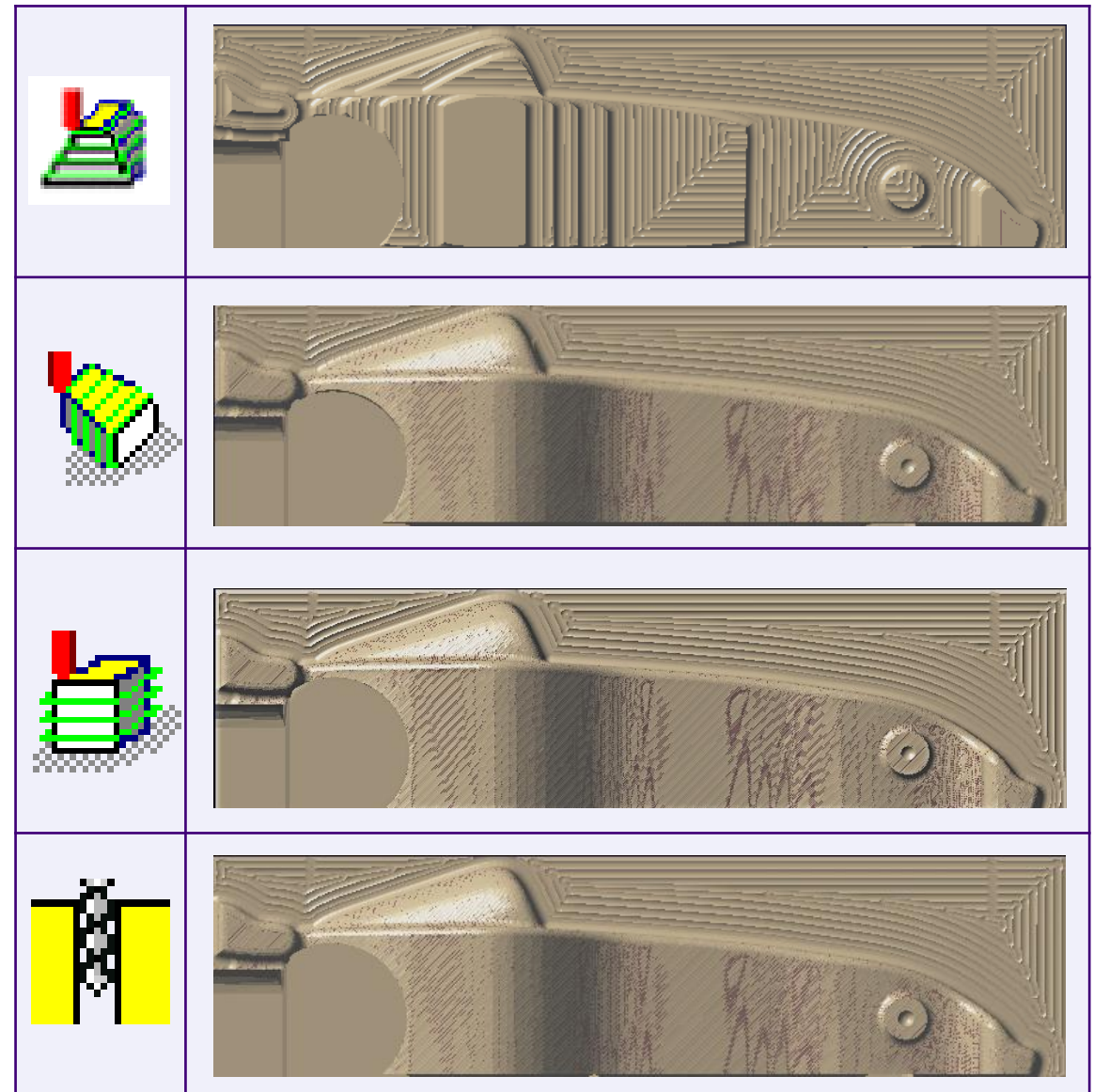
# Machining process

Now you are going to edit the operations present in the machining program. Below are the main operations.

You need to edit a total of 4 operations for the first side, and a mirror operation for the second.



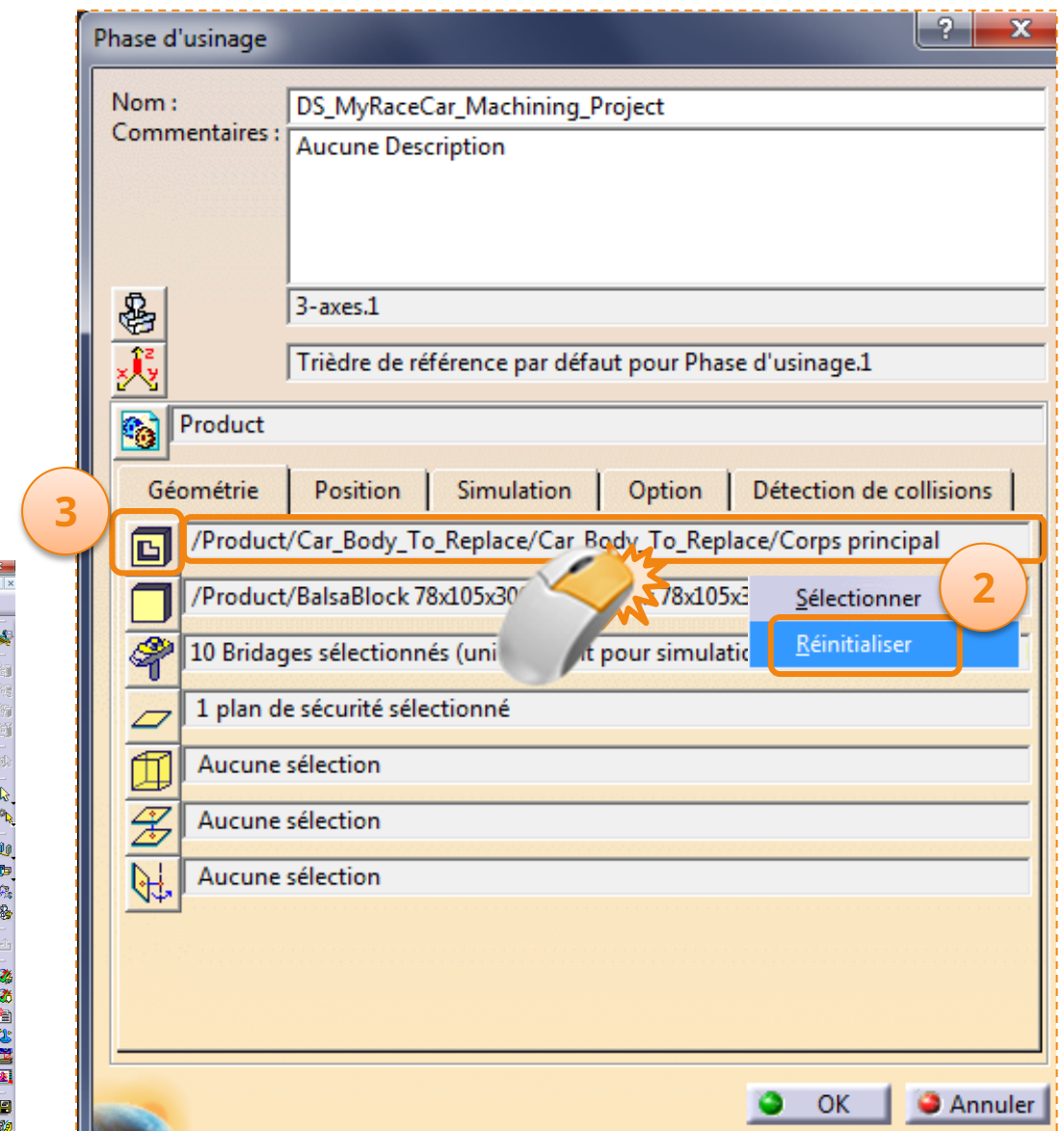
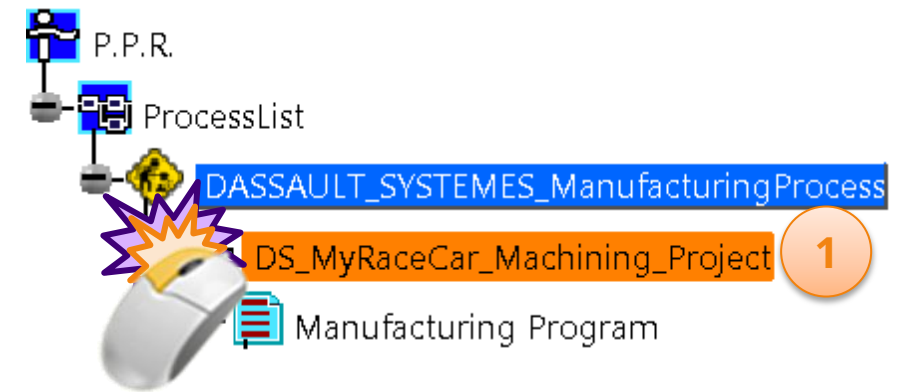
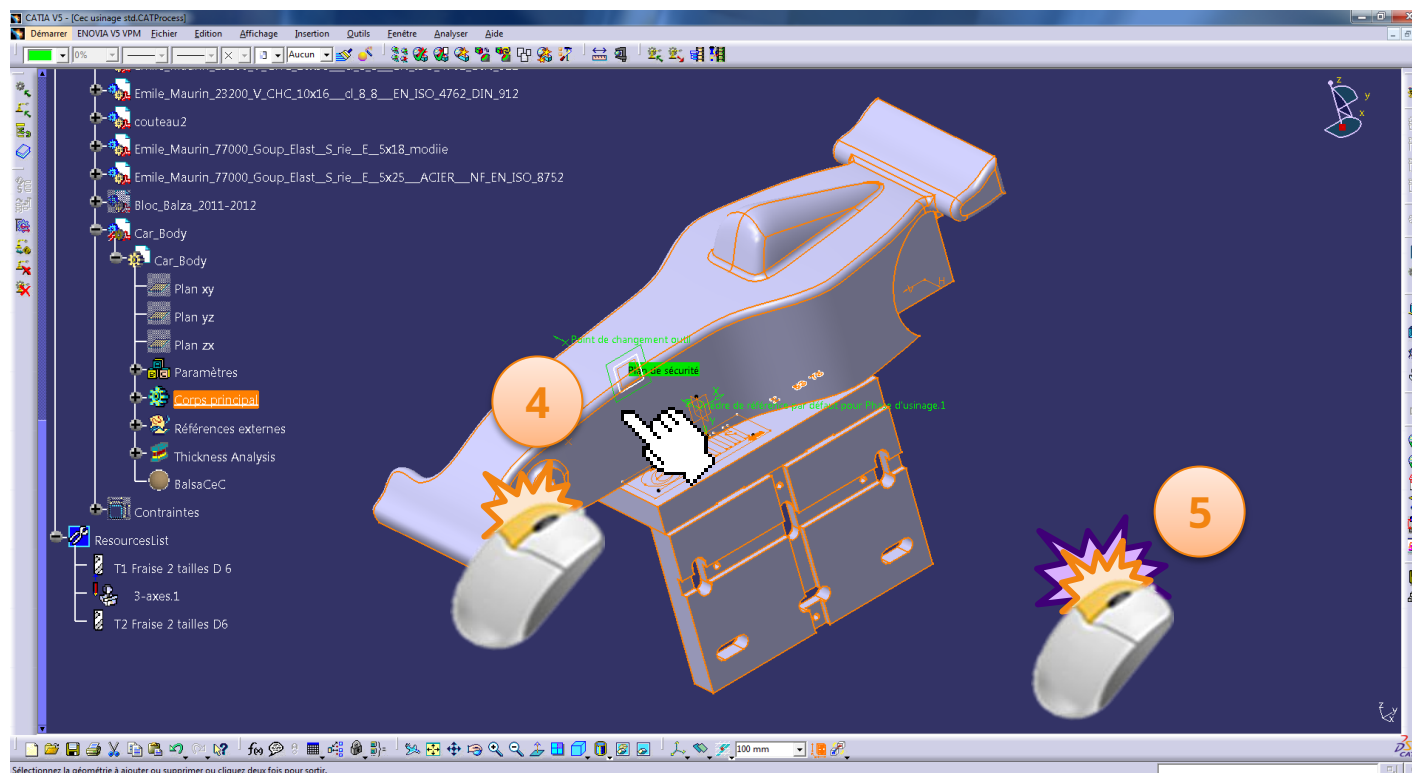
1. The roughing first removes as much material as possible from the balsa block in a minimum of time. After a sketch, we get a car made of strata.
2. An operation of parallel planes makes it possible to refine the horizontal walls.
3. Contouring makes it possible to refine the vertical walls.
4. The drilling operation will not make it possible to drill the block over its entire width to make the axis of the wheels, but to mark the location of the latter to finally drill it using a drill.



# Machining process

Now that your car is in place, you will designate the part to calculate the part to be machined.

1. Double click 2 TIMES (the first to activate the process, then to open the properties window) on **DS\_MyRaceCar\_Machining\_Project**.
2. In the window that opens, right-click and reset the first field. You should then read: *No design parts selected* »
3. Click the icon **Part drawing for simulation**. This will hide the window.
4. Then click on the **main body** in 3D space or in the construction tree.
5. Double-click in an empty space to confirm selection and display the dialog box again.

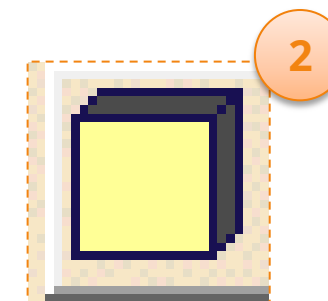
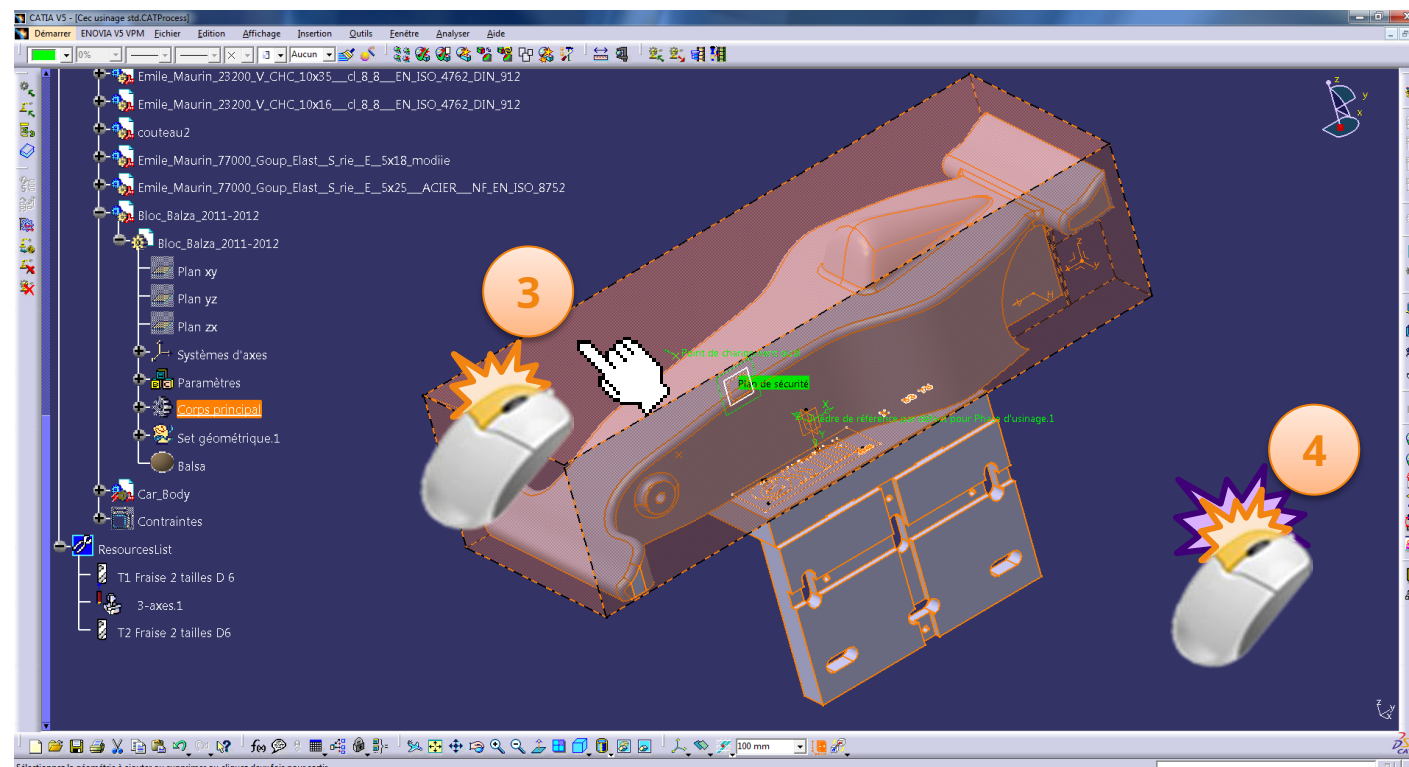
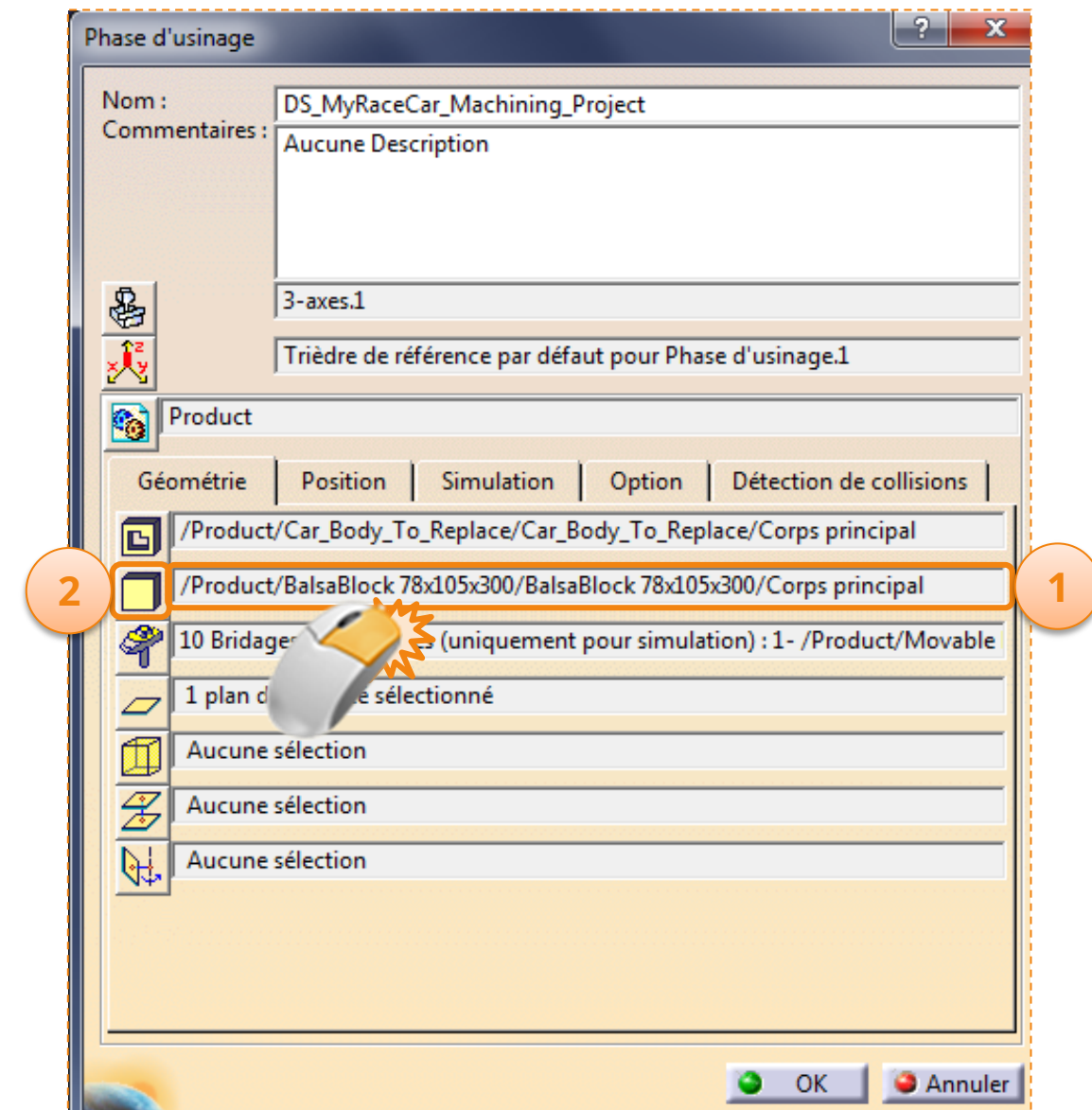




# Machining process

Still in the same window, we will perform the same steps but this time for the balsa block.

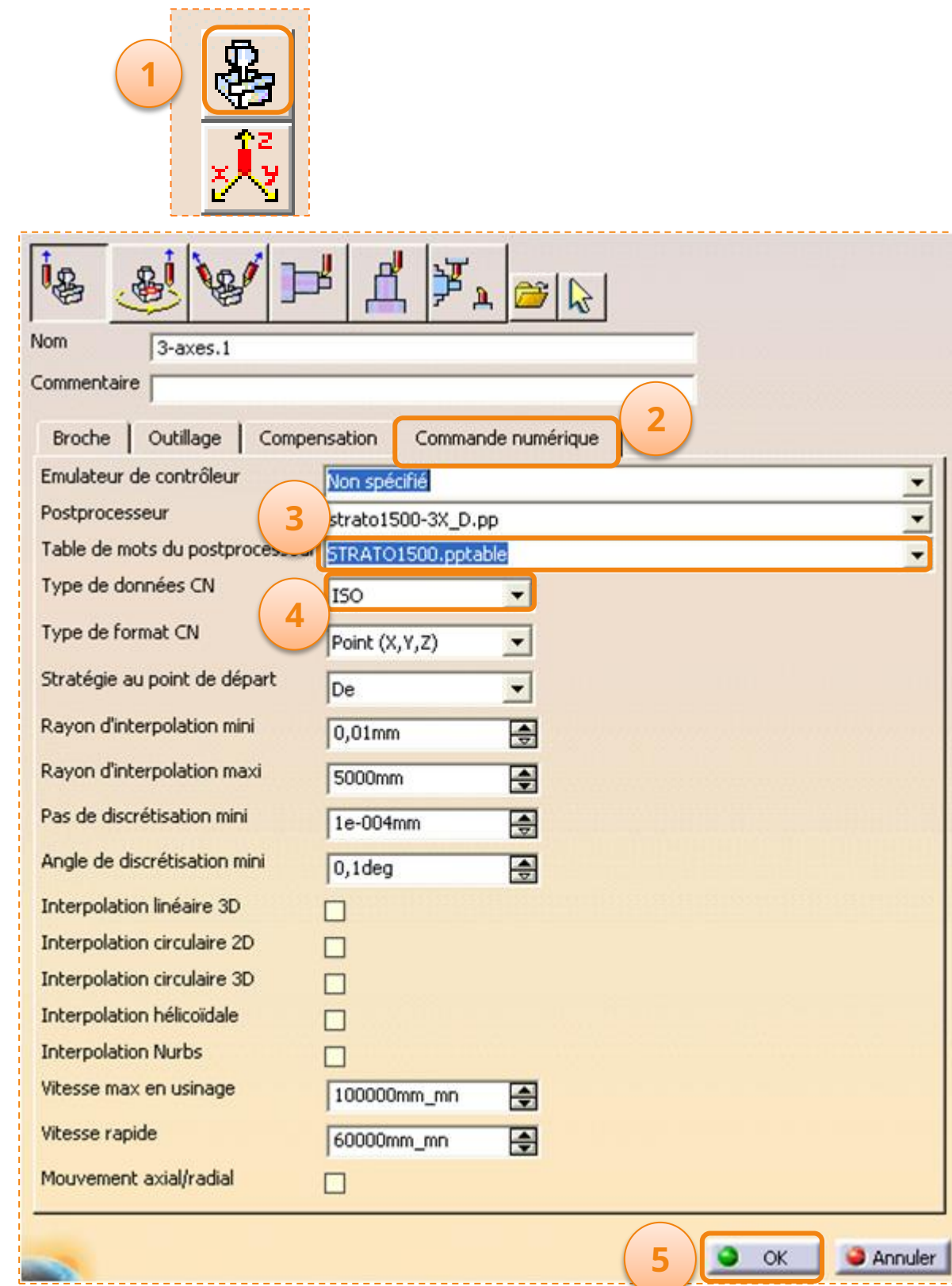
1. Right click and reset the second field.
2. You will see "*No selection*" in the fields ". We will now define our balsa block. To do this, click on the icon **Raw for simulation**. This will hide the window.
3. Then click on the balsa block in 3D space or in the construction tree on the **main body** balsa block.
4. Double-click in an empty space to confirm selection and display the dialog box again.
5. DO NOT CLICK IMMEDIATELY ON **okay**.



# Machining process

Now that the raw product and the finished product are set up, you need to check a few other settings.

1. In the window **Machining phase** open click icon **Machine**.
2. Go to the tab **Numerical commands**.
3. Check that the **postprocessor word table** either set to *STRATO1500.pptable*.
4. Also check that the **NC data type** be set to *ISO*.
5. Click on **okay**.

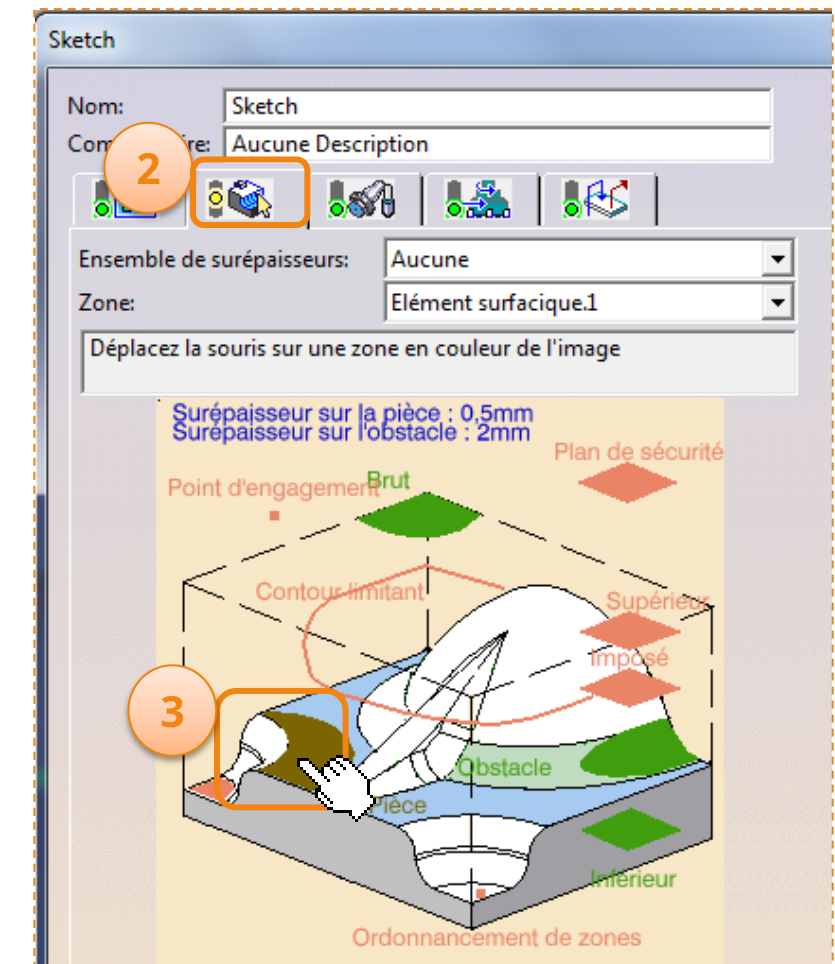
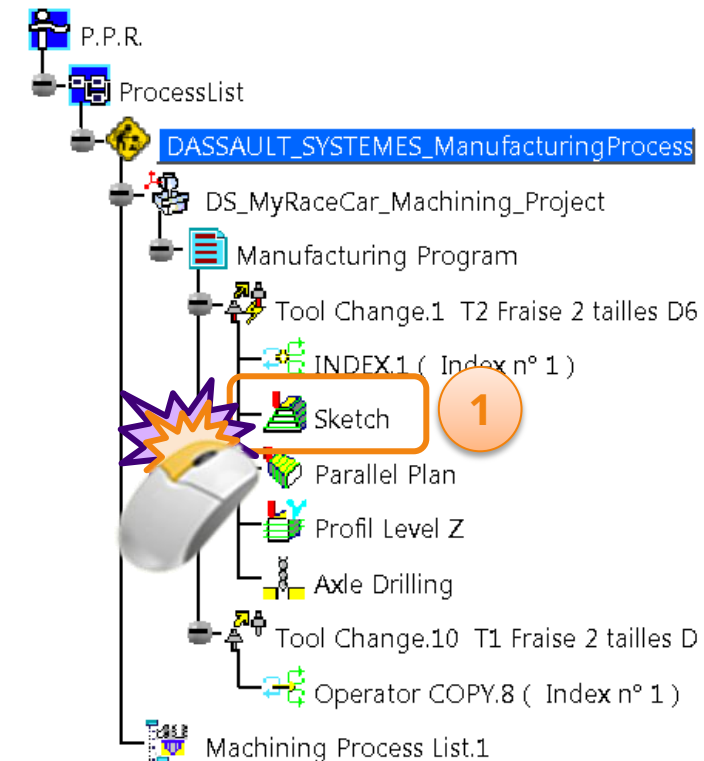




# Machining process

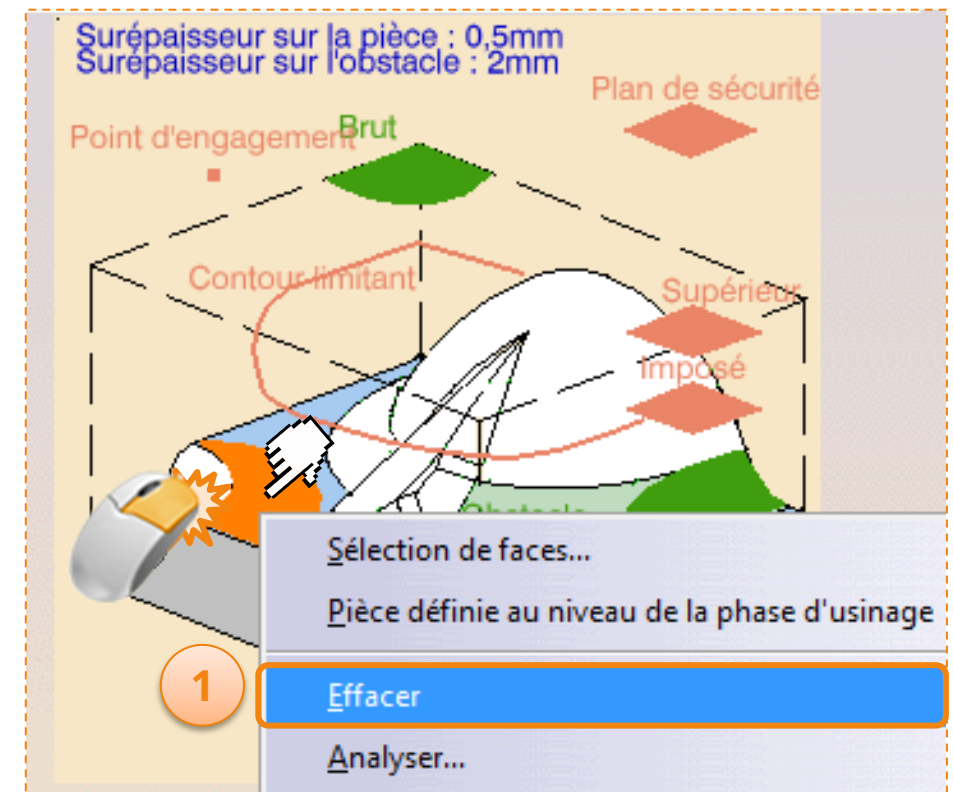
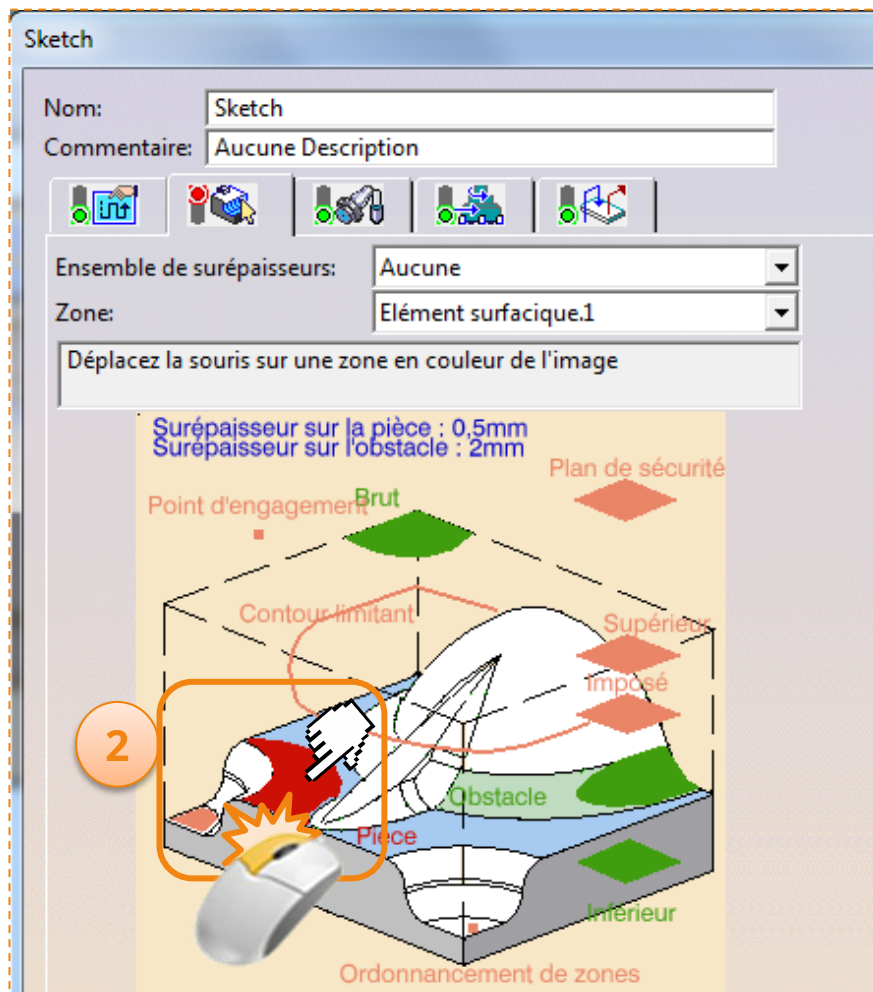
To continue, unfold the tree as in the image opposite to display the different operations that you are going to edit.

1. Double click on the operation **Sketch**.
2. The window below appears. The traffic light in the **geometric components** tab is yellow, which means you must select one or more items to complete the operation.
3. You will find that the surface **Pièce** is colored brown. It is therefore necessary to re-edit this part of the operation.



# Machining process

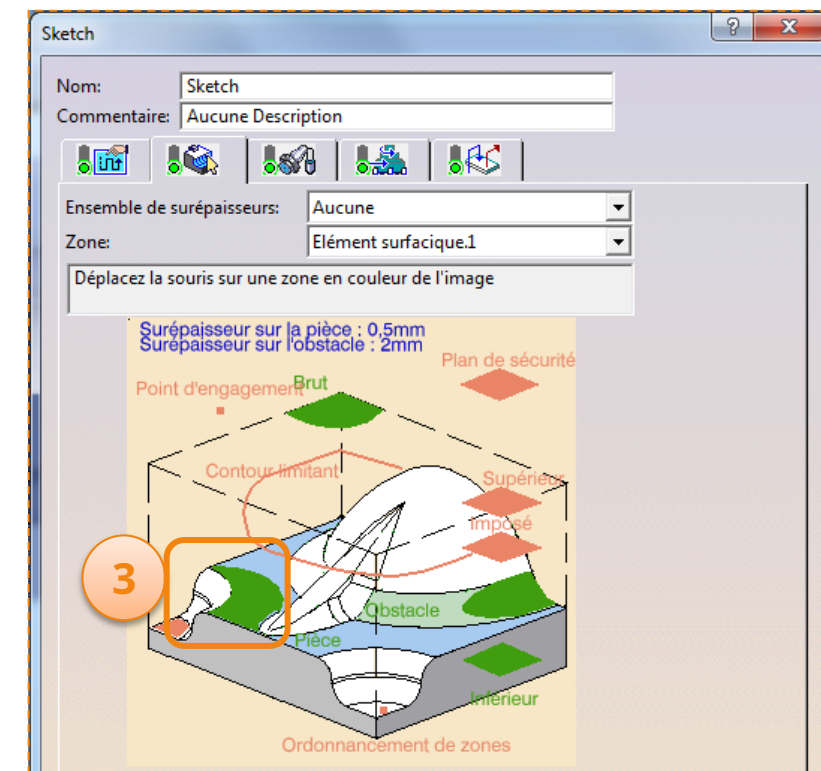
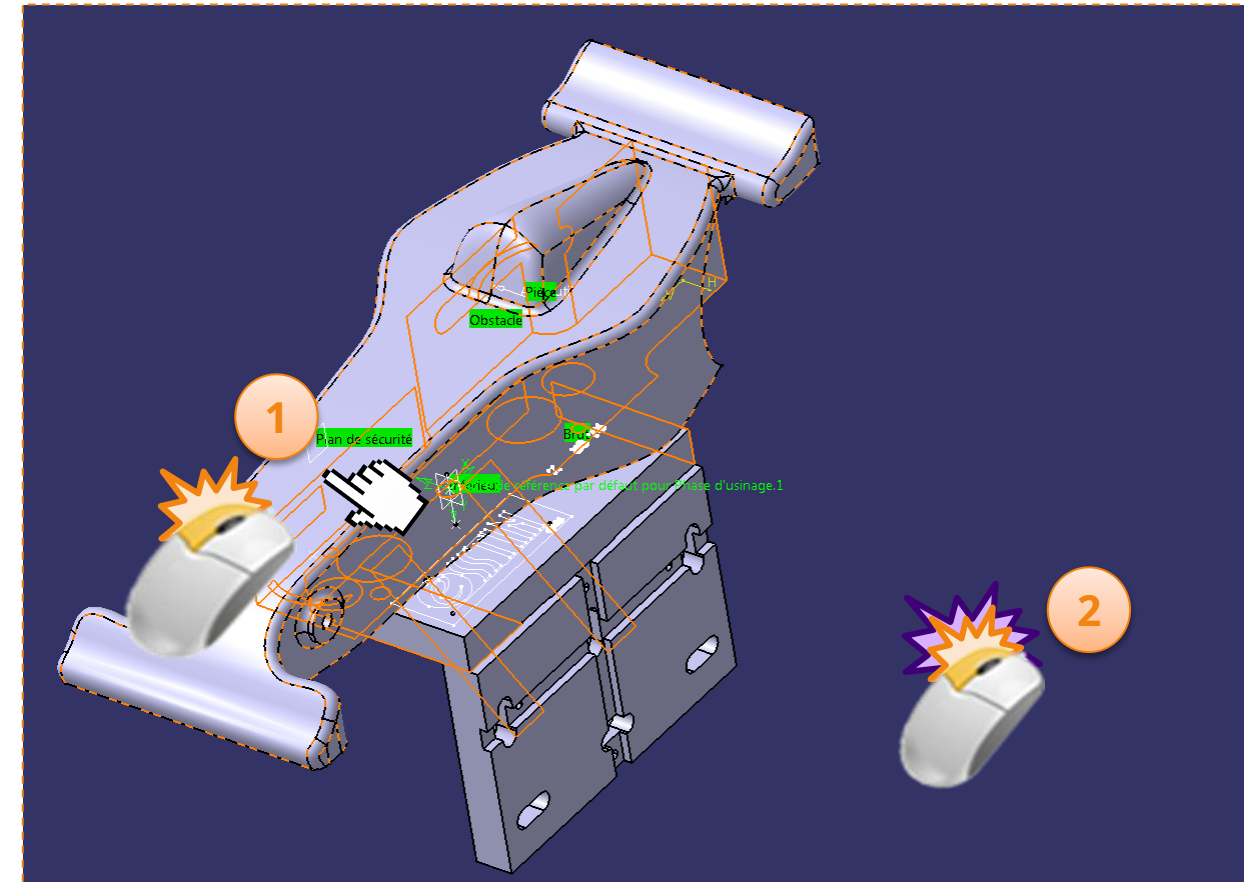
1. Right-click on the surface corresponding to the part, and select **Wipe off**.  
The map then appears in red.
2. Click on the **red part** which represents the part. The window disappears to allow you to select items.



## Machining process

The operation **Sketch** will allow us to remove a maximum of material from the balsa block in a minimum of time.

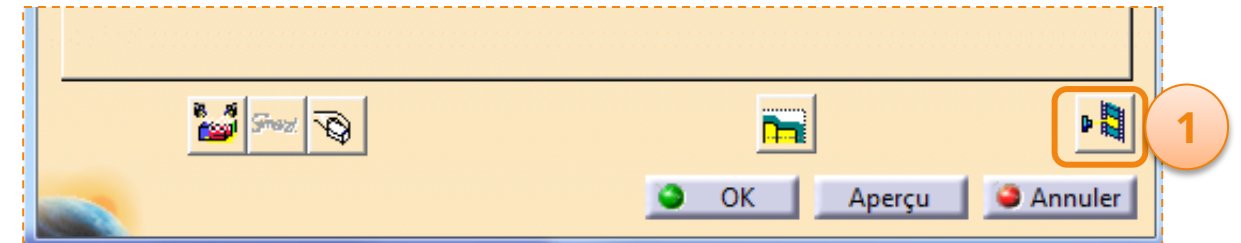
1. In the 3D environment, click on the **car body** as shown opposite.
2. Double-click in an empty space to confirm the selection and display the dialog box to new.
3. The red area is now green indicating that the geometries have been selected.
4. **Do not click yet okay.**



# Machining process

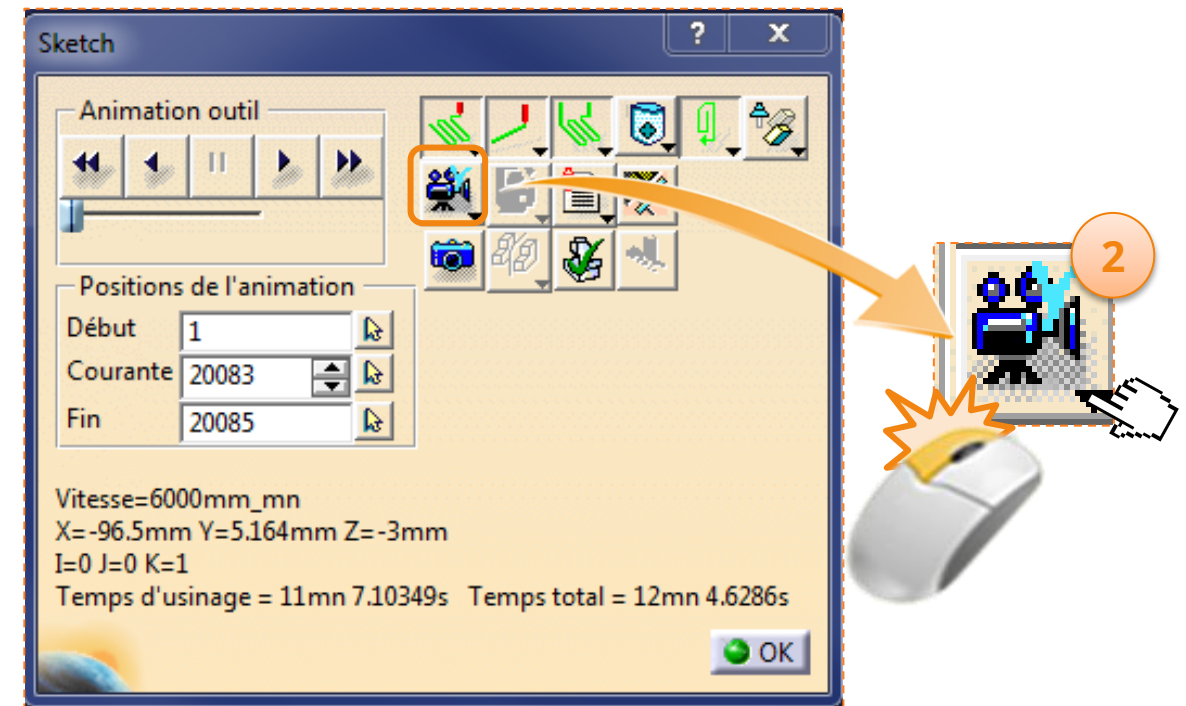
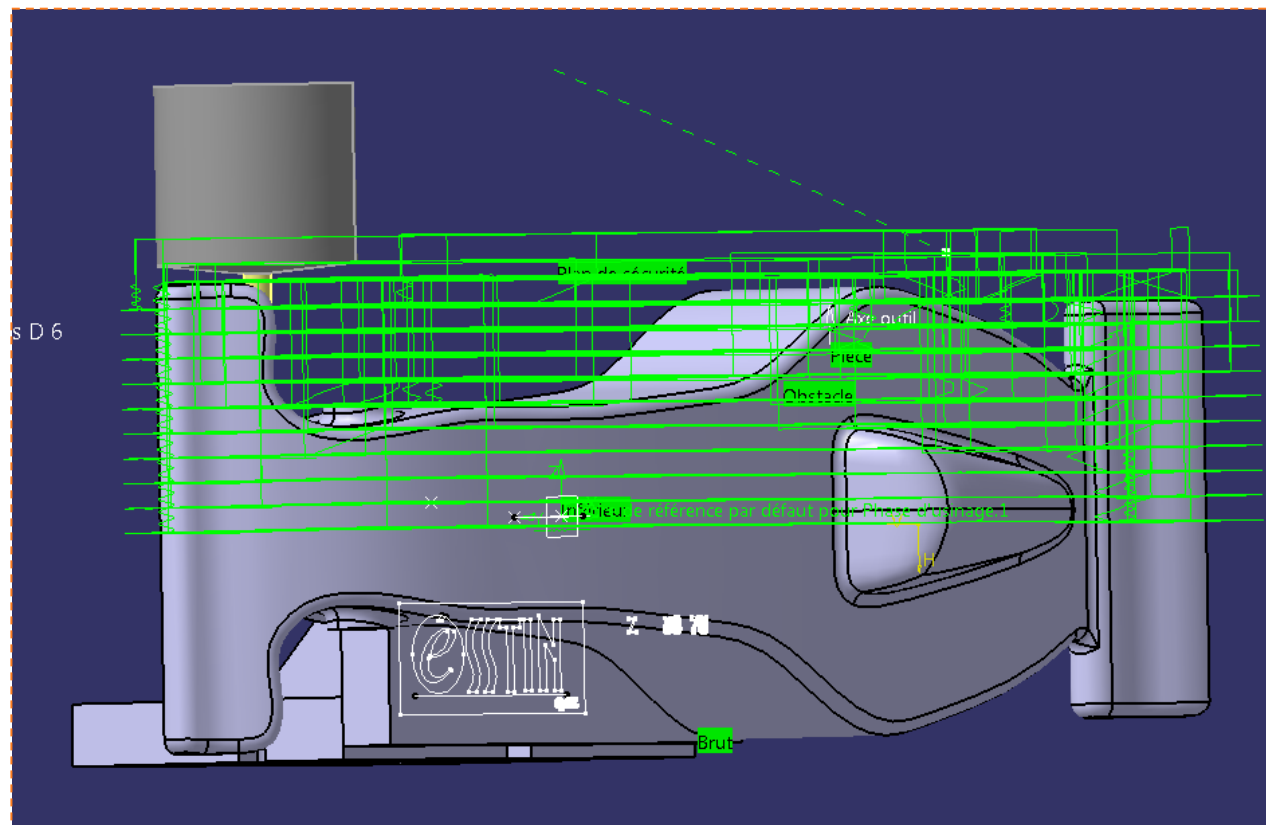
We are now going to simulate this operation in order to check that it is working properly.

1. Click the icon **Tool path animation** at the bottom right of the dialog box **Sketch**. The CATIA software will then calculate the new tool path.



To switch to simulation mode, the green lines represent the tool paths for this machining operation.

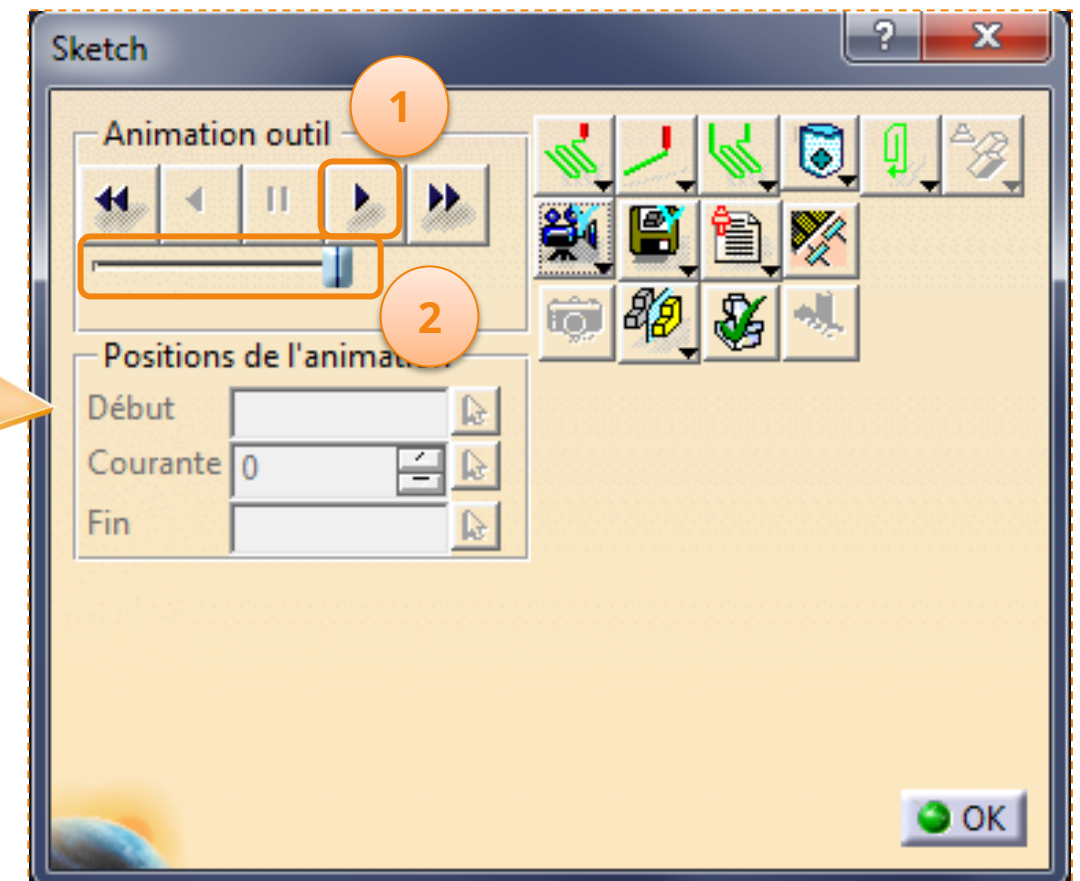
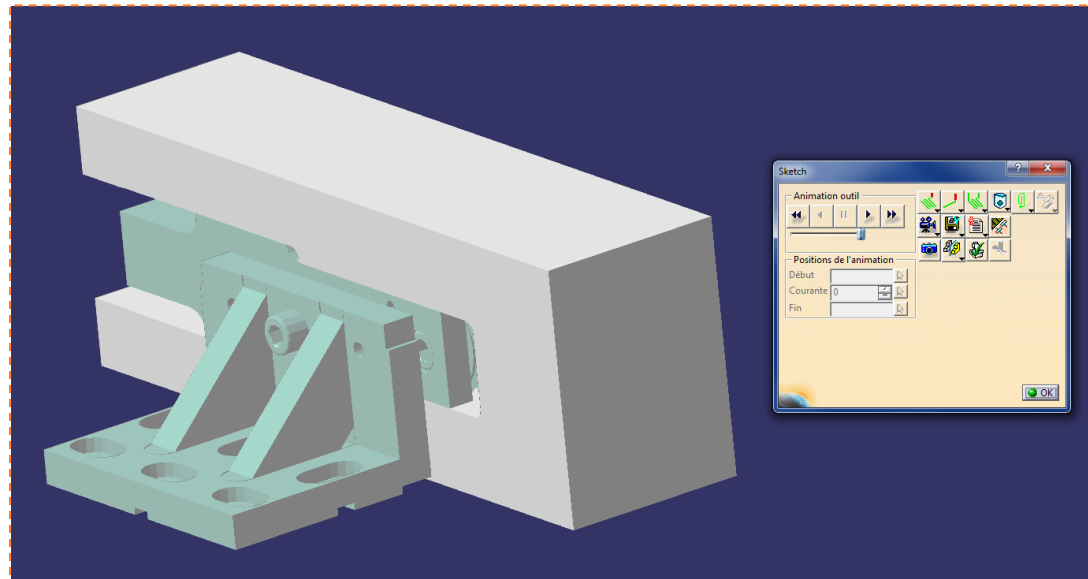
2. Click the icon **Video since last saved result** in the form of a camera.





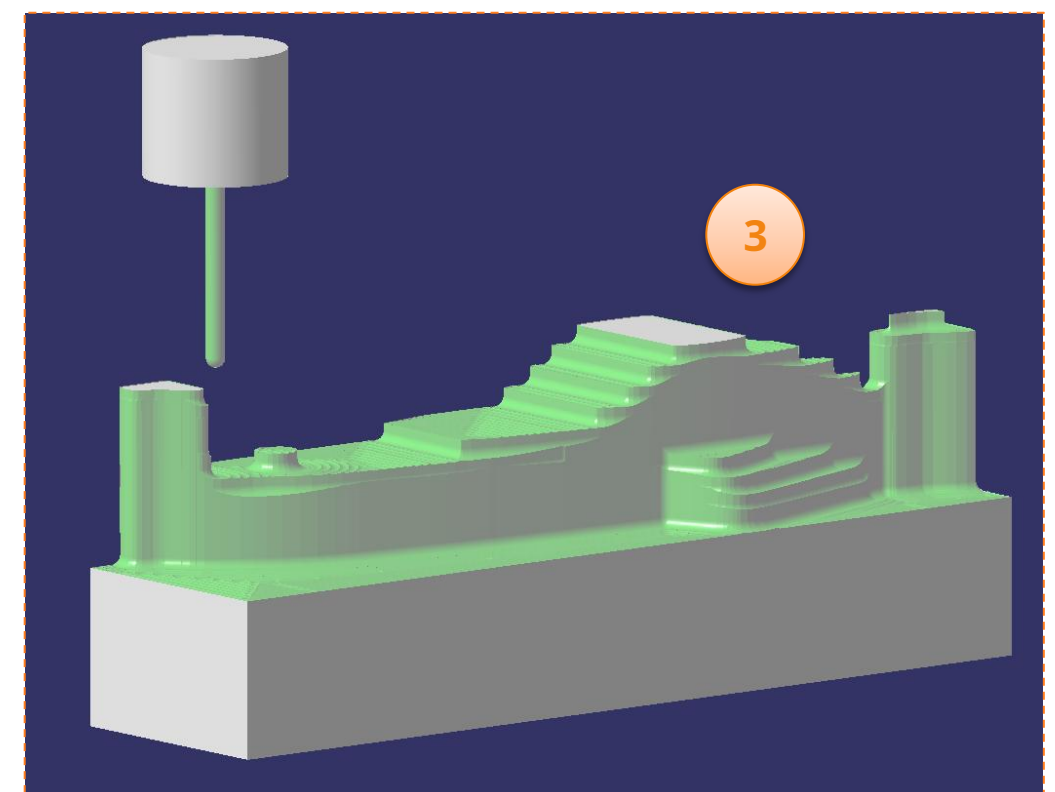
# Machining process

You should have this screen:



1. Press the play button (**Tool path progress in the forward direction**).
2. If the tool advances too much or not fast enough, you can always modify the execution speed with the horizontal scroll bar below the visualization controls.
3. Once you've completed the draft, you should get a result similar to the one shown here.

**Make sure that the tool does not come into contact with the assembly during the simulation. To do this, observe the result from all angles.**



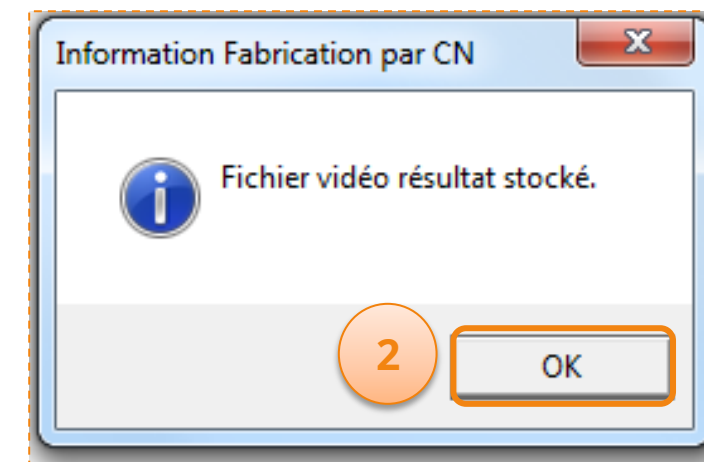


# Machining process

We will now save our result.

1. Click on the small arrow on the icon with a floppy disk to display the drop-down menu.
2. Selected **Associate the video result to the operation**.  
Your simulation of the draft is now saved, so in future simulations we can start from this position.
3. Click on **okay** to validate the recording.
4. Click on **okay** to exit the simulation.
5. Click on **okay** to exit the machining operation **Sketch**.

Now the tool paths of the operation **Sketch** are calculated ("Solved" appears after the name of the operation in the Process).

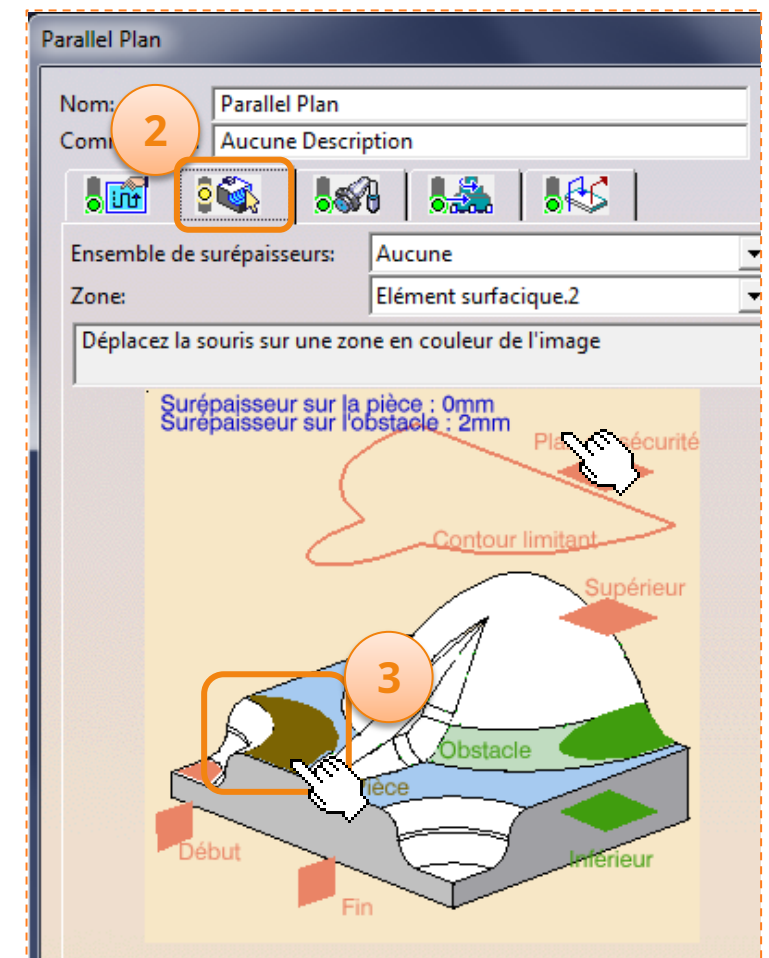
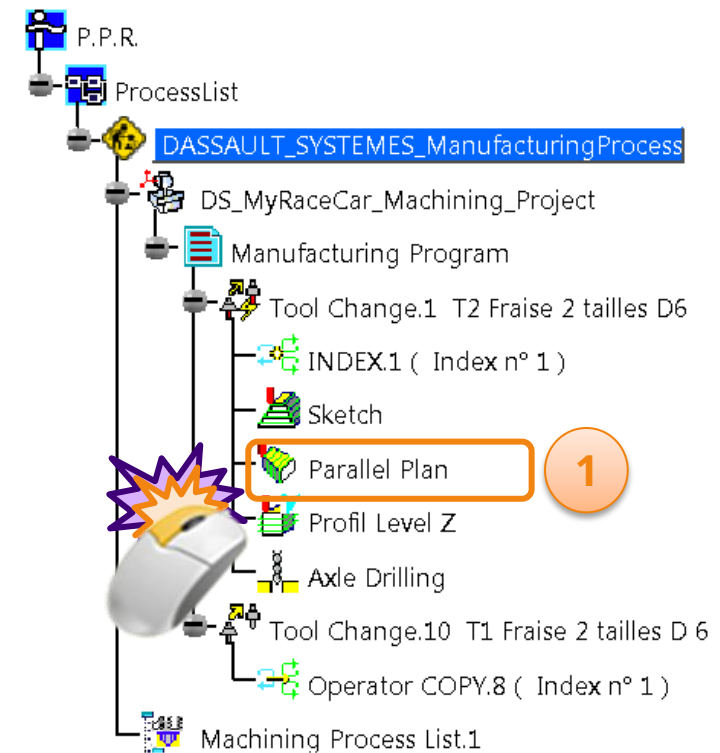


Be sure after each simulation to record the video result. Otherwise, the simulator will restart from the start of machining.

# Machining process

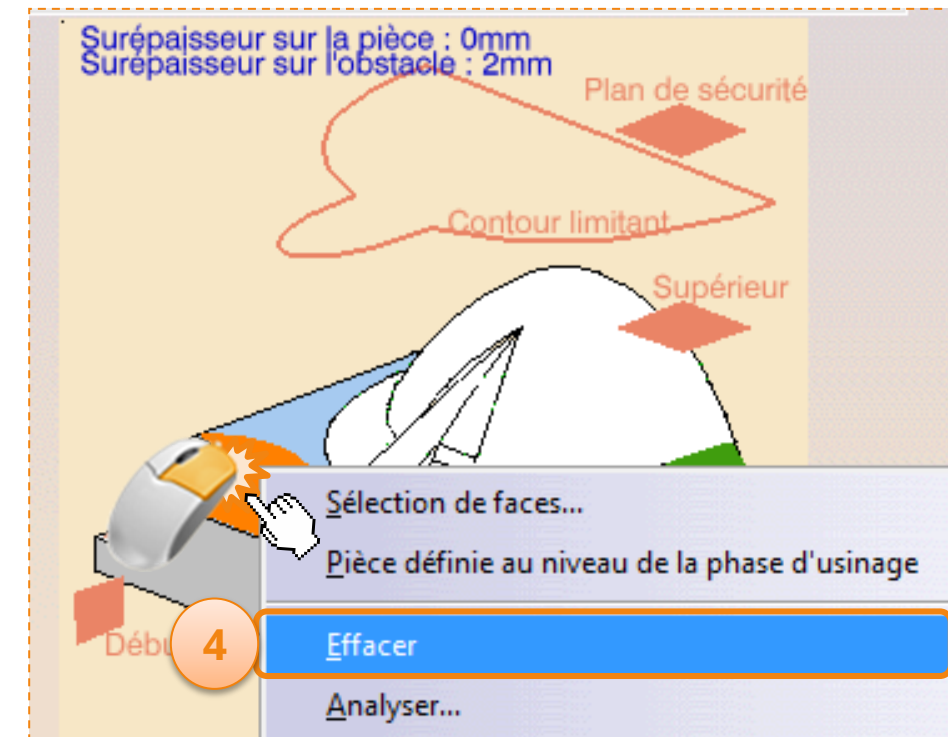
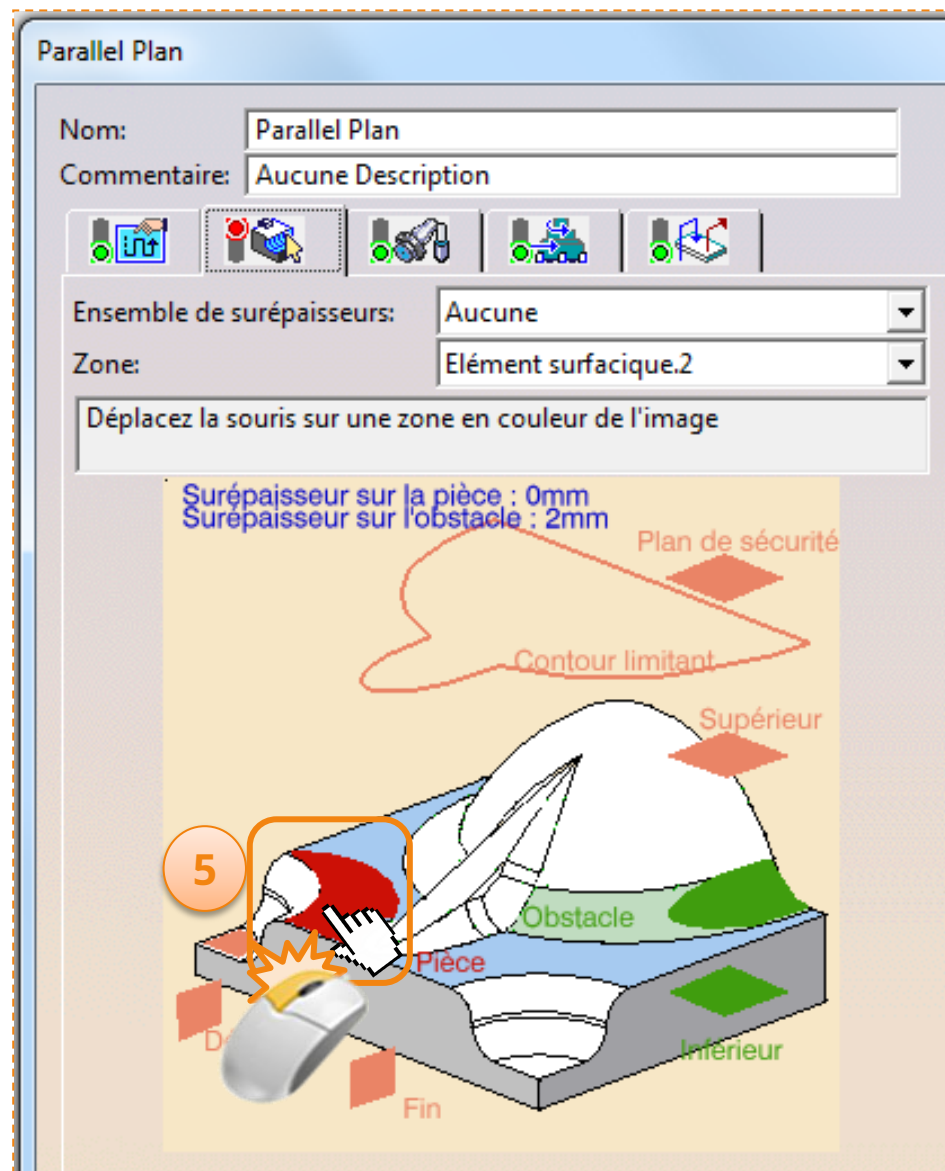
After the operation **Sketch**, you will edit the operation **Parallel Plan**.

1. Double click on the operation **Parallel Plan** in the construction tree.
2. The window below appears. The traffic light in the **geometric components** tab is yellow. This means you must select one or more items to complete the operation.
3. You will find that the surface **Piece** is colored brown, so this part of the operation.



# Machining process

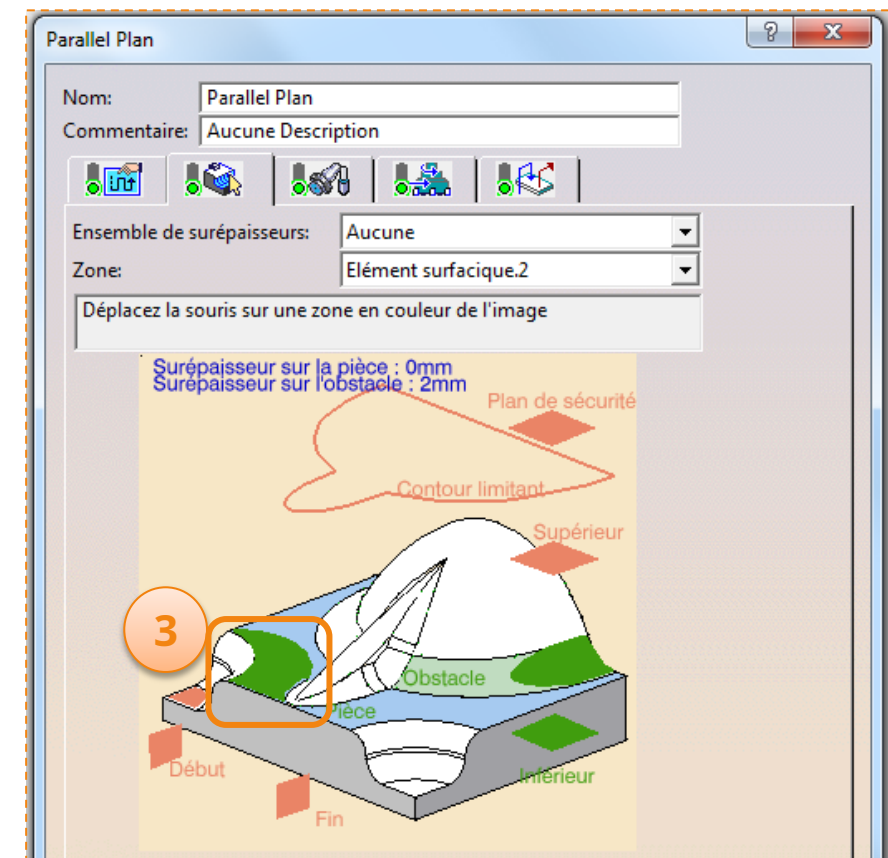
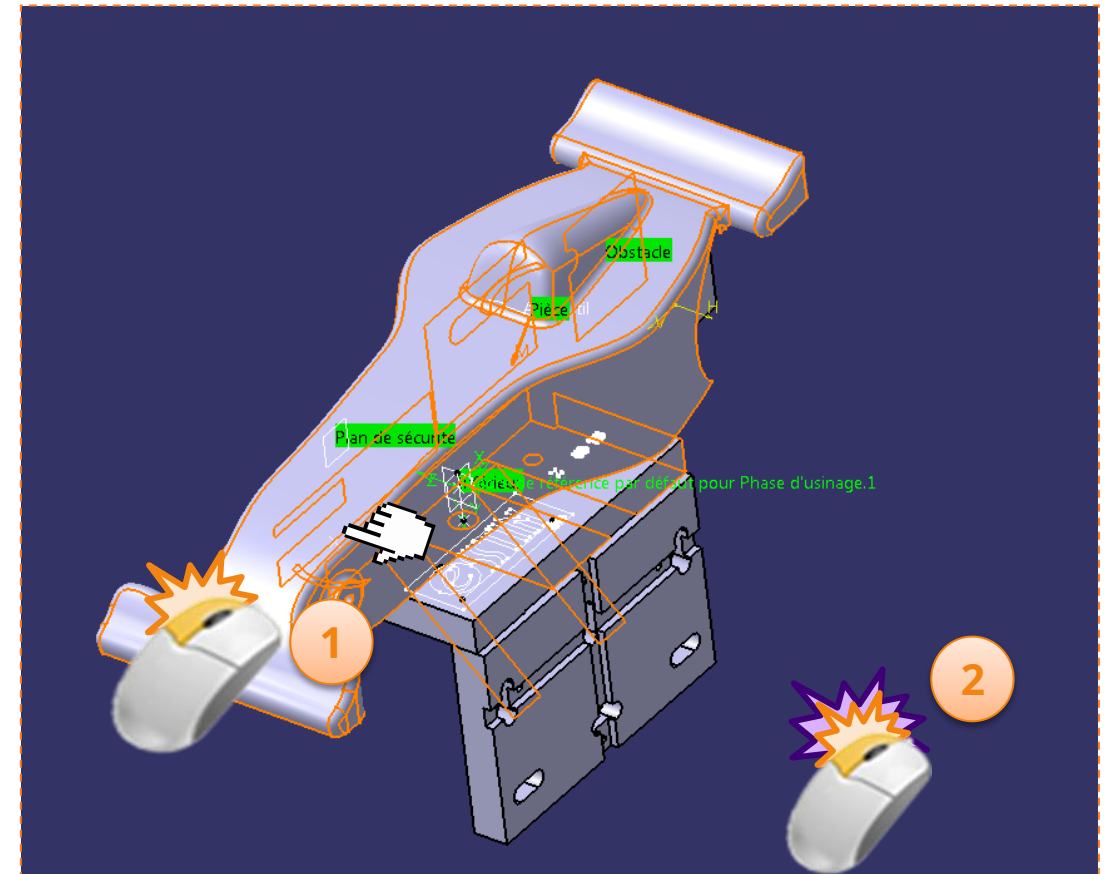
4. Right-click on the surface corresponding to the part, and select **Wipe off**. The map then appears in red.
5. Click on the **red part** which represents the part. The window disappears to allow you to select items.



# Machining process

The operation **Parallel Plan** will allow us to refine the horizontal walls.

1. In the 3D environment, click on the **car body** as shown below.
2. Double-click in an empty space to confirm the selection and display the dialog box to new.
3. The red area is now green indicating that the geometries have been selected.
4. Do not click yet **okay**.



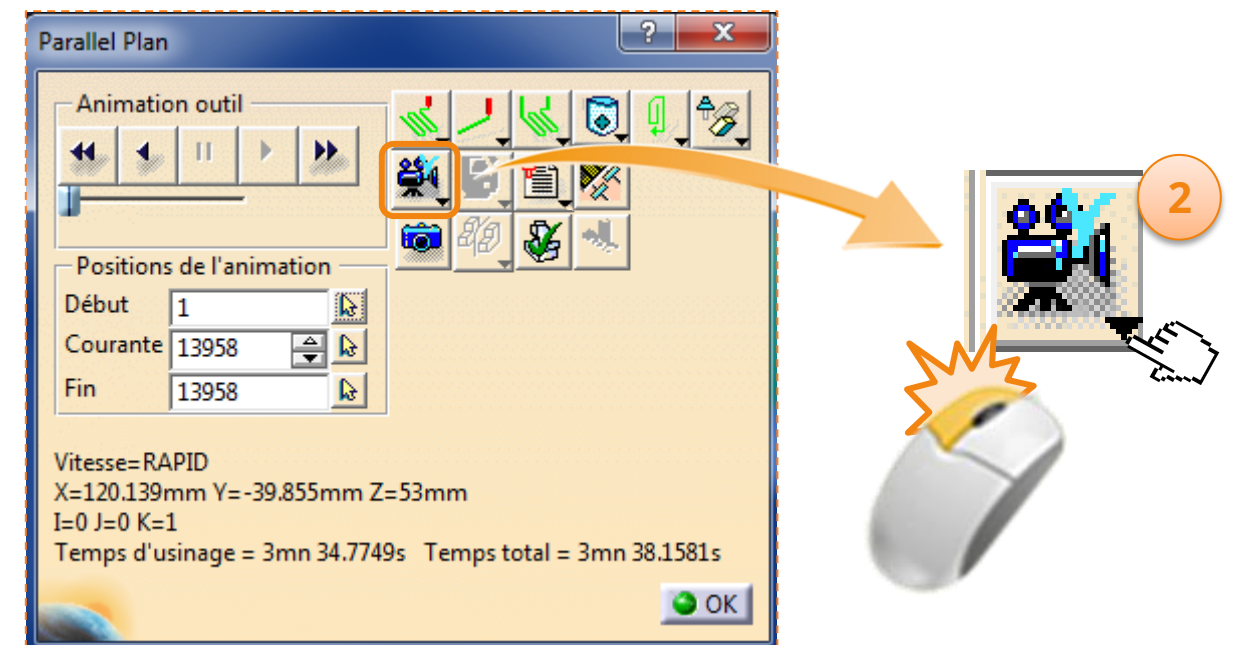
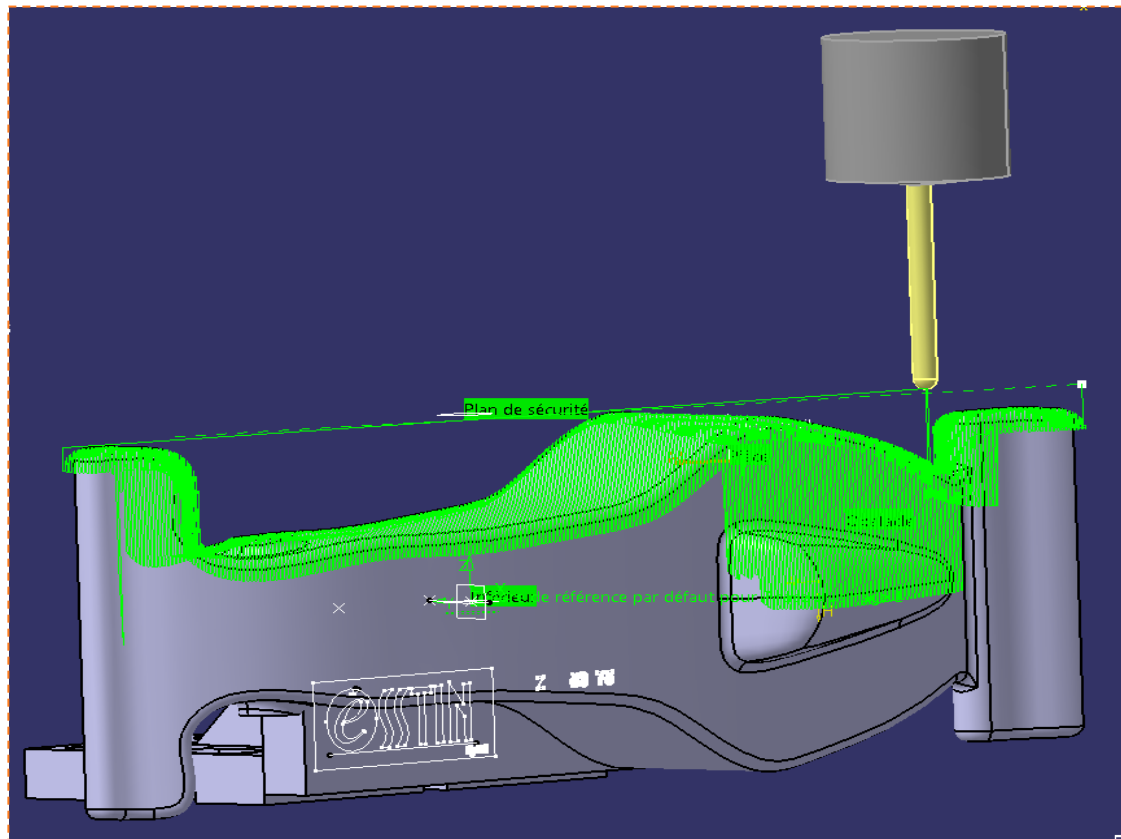
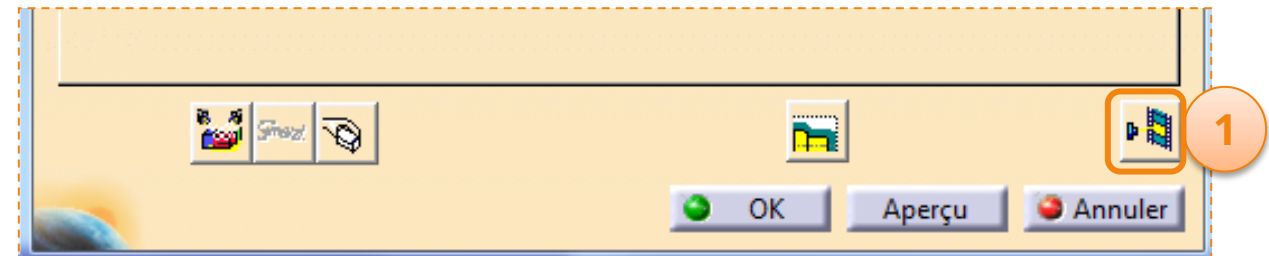
# Machining process

We are now going to simulate this operation in order to check that it works correctly.

1. Click the icon **Tool path animation** at the bottom right of the dialog box **Parallel Plan**.

You will switch to simulation mode, the green lines represent the tool paths for this machining operation.

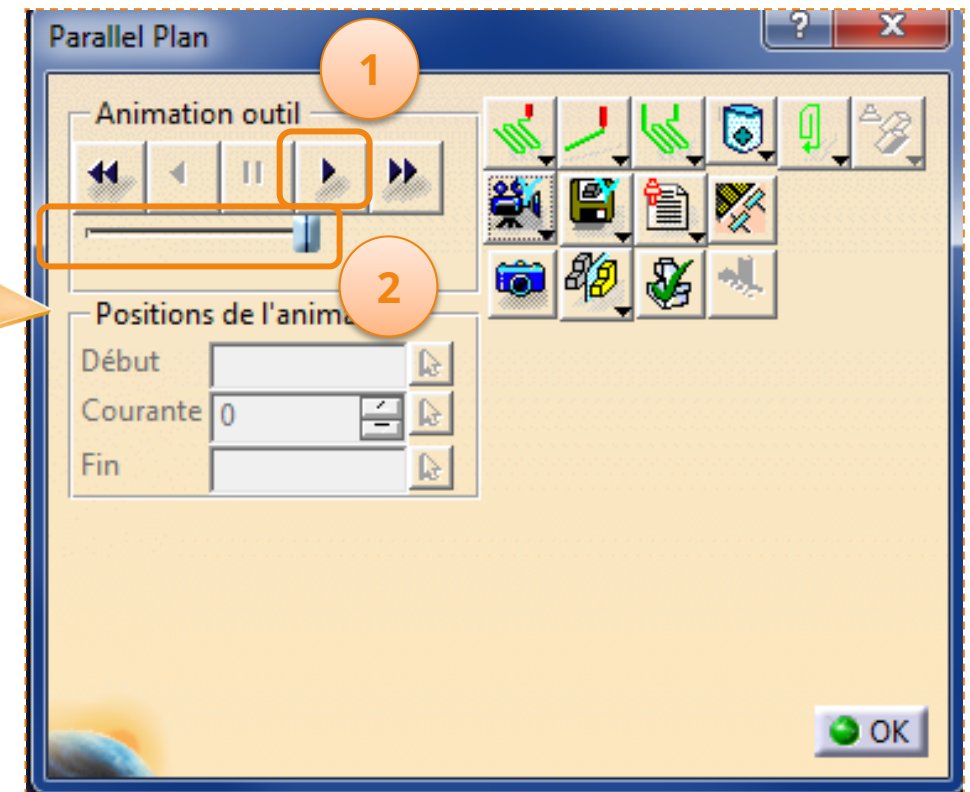
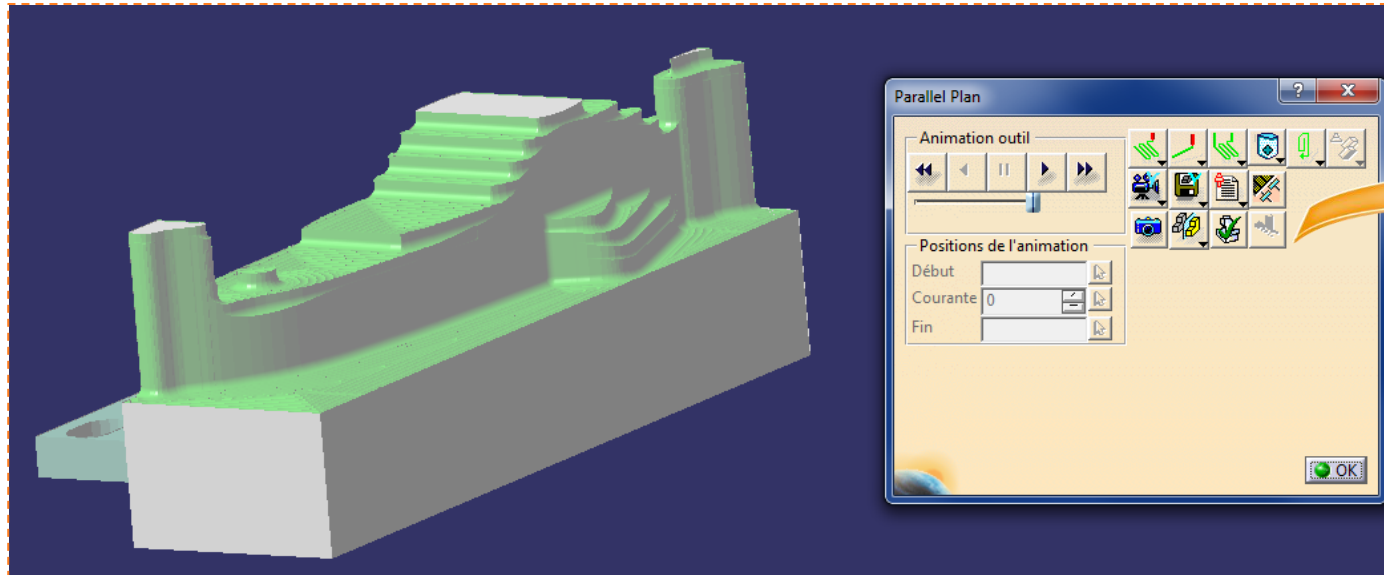
2. Click on the small arrow of the camera icon and choose **Video since last saved result**. In this way the simulation resumes from the last recorded result.





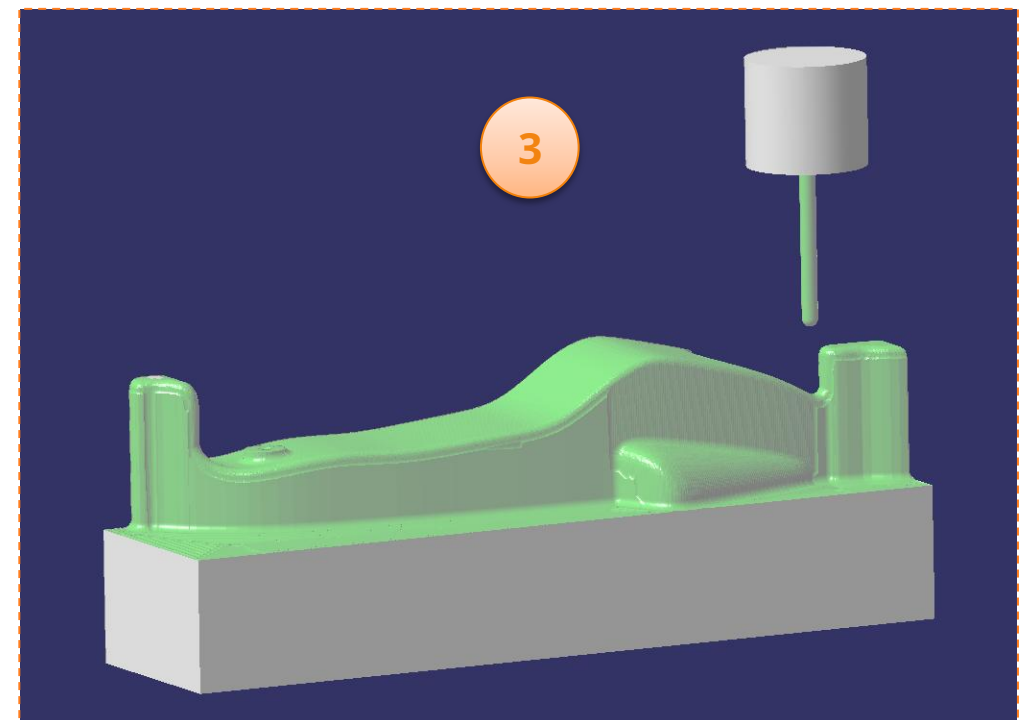
# Machining process

You should then have this screen:



1. Press the “play” button (**progress of the tool path in the forward direction**).
2. If the tool advances too much or not fast enough, you can always modify the execution speed with the horizontal scroll bar below the visualization controls.
3. Once you have finished refining the horizontal walls, you should obtain a result similar to the one shown opposite.

**Make sure that the tool does not come into contact with the assembly during the simulation. To do this, observe the result from all angles.**

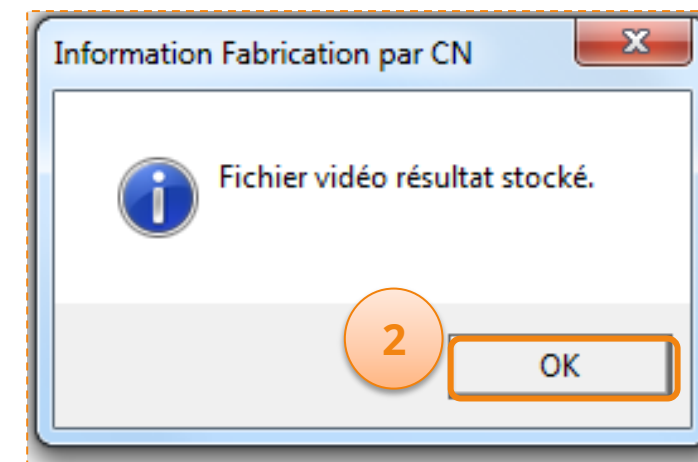
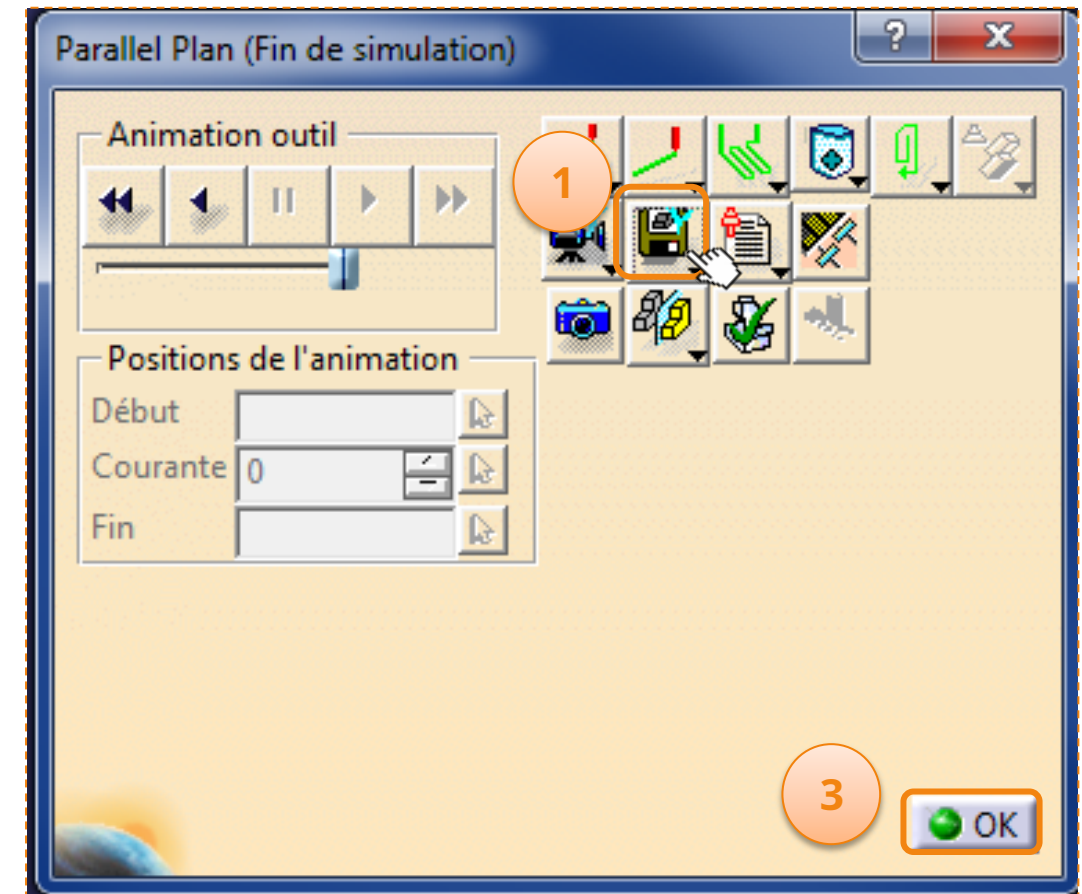


# Machining process

We will now save our result.

1. Click on the small arrow on the icon with a floppy disk to display the drop-down menu.
2. Select **Associate the video result to the operation**.  
Your simulation of the thinning of the horizontal walls is now saved, so during the next simulation we can start from this position.
3. Click on **okay** to validate the recording.
4. Click on **okay** to exit the simulation
5. Click on **okay** to exit the machining operation **Parallel Plan**.

Now the tool paths of the operation **Sketch** are calculated ("Solved" appears after the name of the operation in the Process).

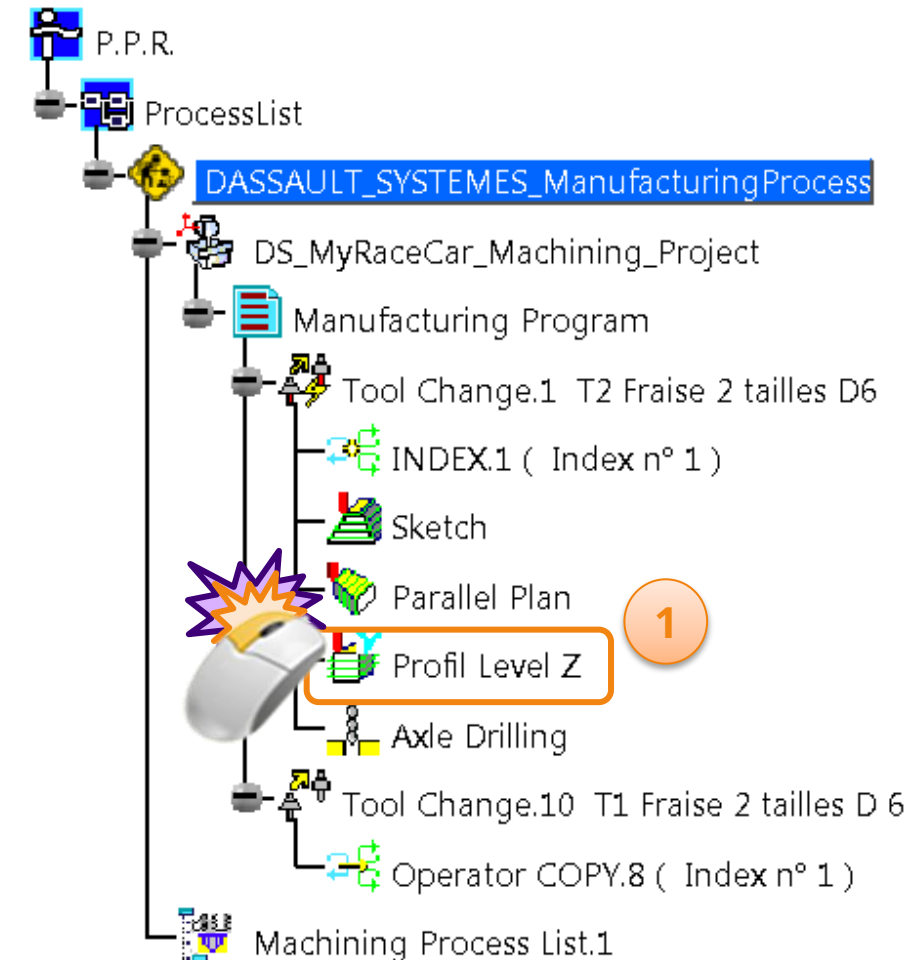
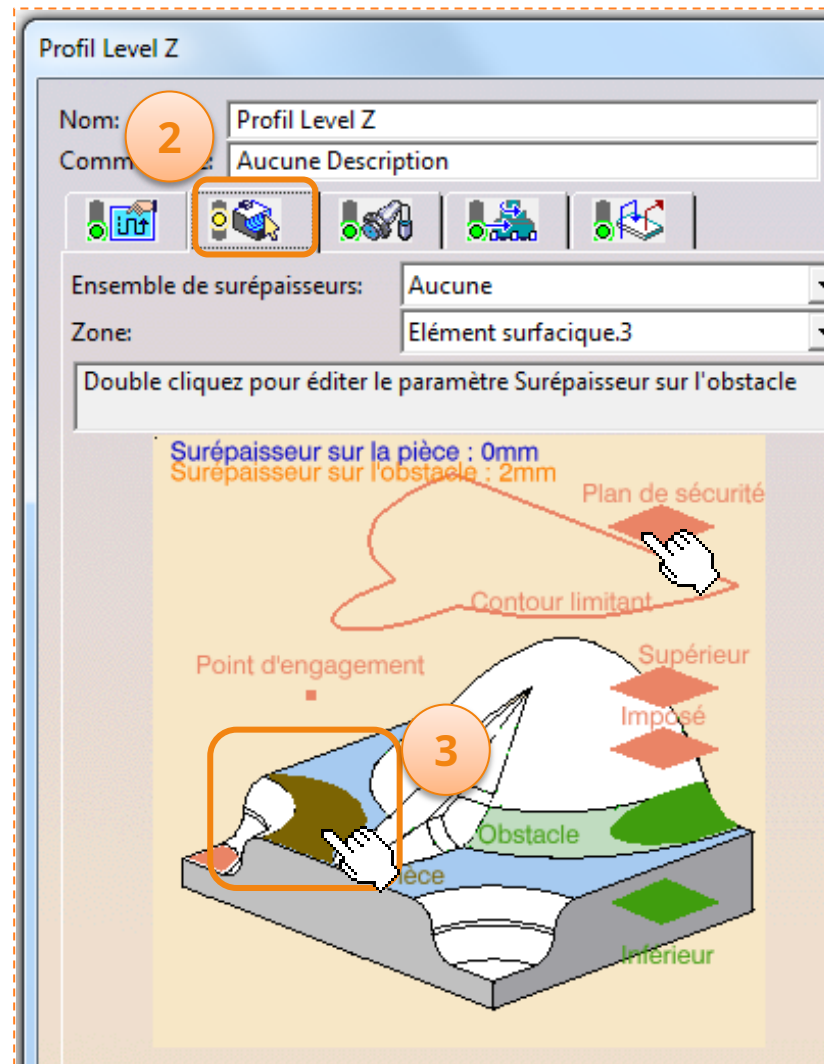


Be sure after each simulation to record the video result. Otherwise, the simulator will restart from the start of machining.

# Machining process

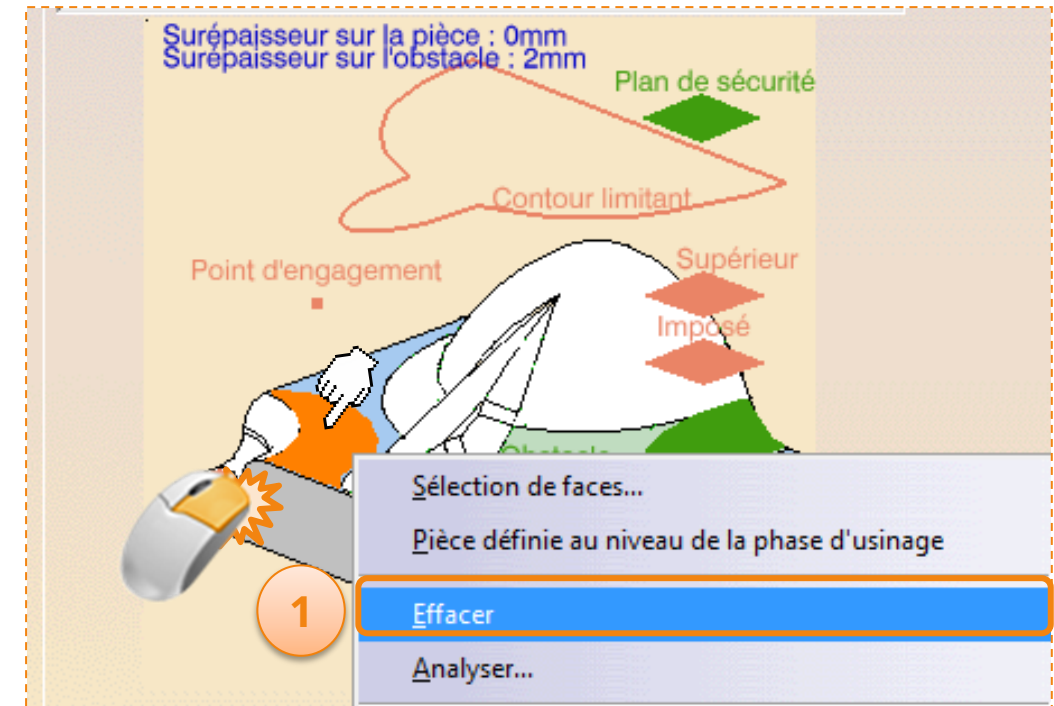
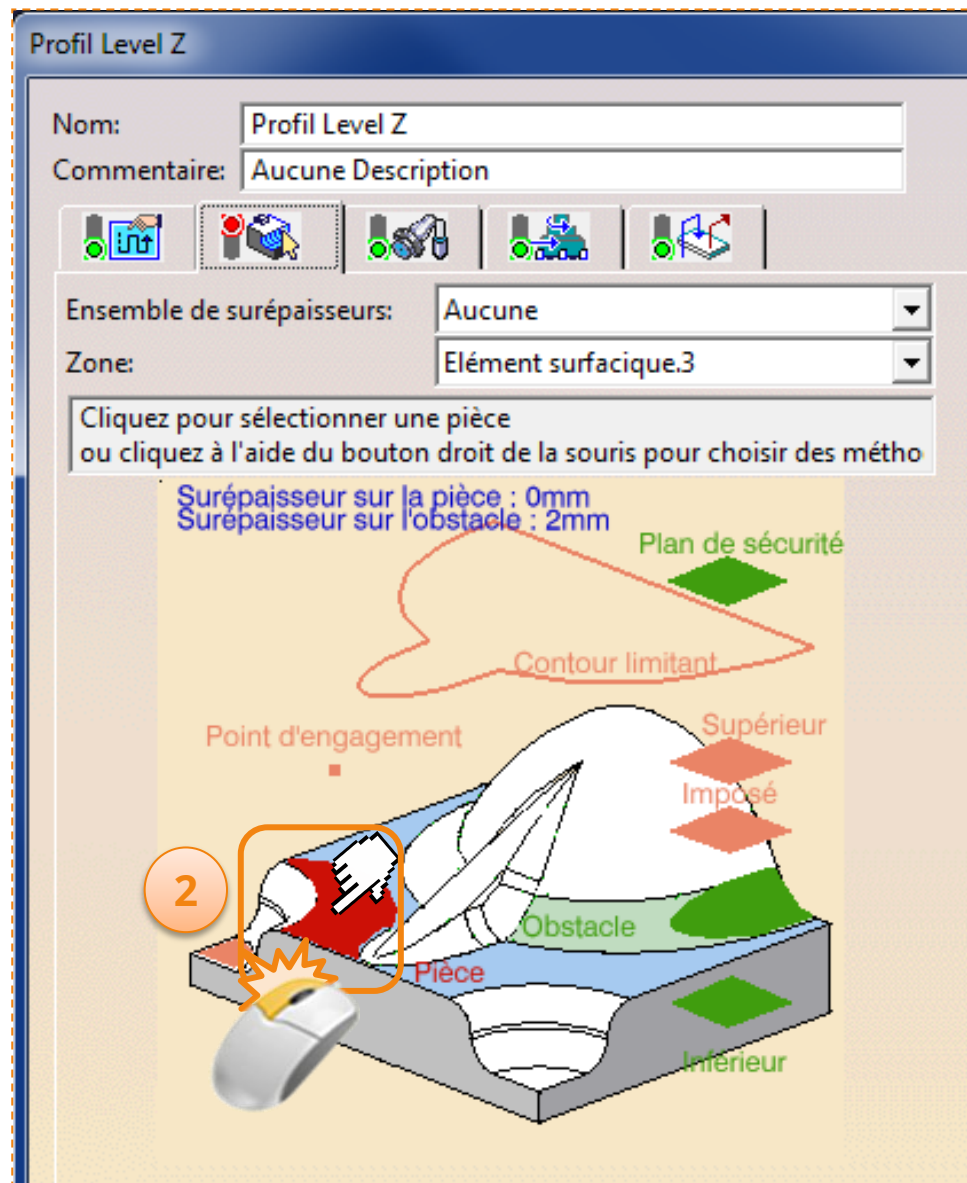
After editing the operation **Parallel Plan**, you will edit the operation **Profile Level Z**.

1. Double click on the operation **Profile Level Z** in the construction tree.
2. The window below appears. The traffic light in the **geometric components** tab is yellow. This means you must select one or more items to complete the operation.
3. You will find that the surface **Pièce** is colored brown. It is therefore necessary to re-edit this part of the operation.



# Machining process

1. Right-click on the surface corresponding to the part, and select **Wipe off**. The map then appears in red.
2. Click on the **red part** which represents the part. The window disappears to allow you to select items.

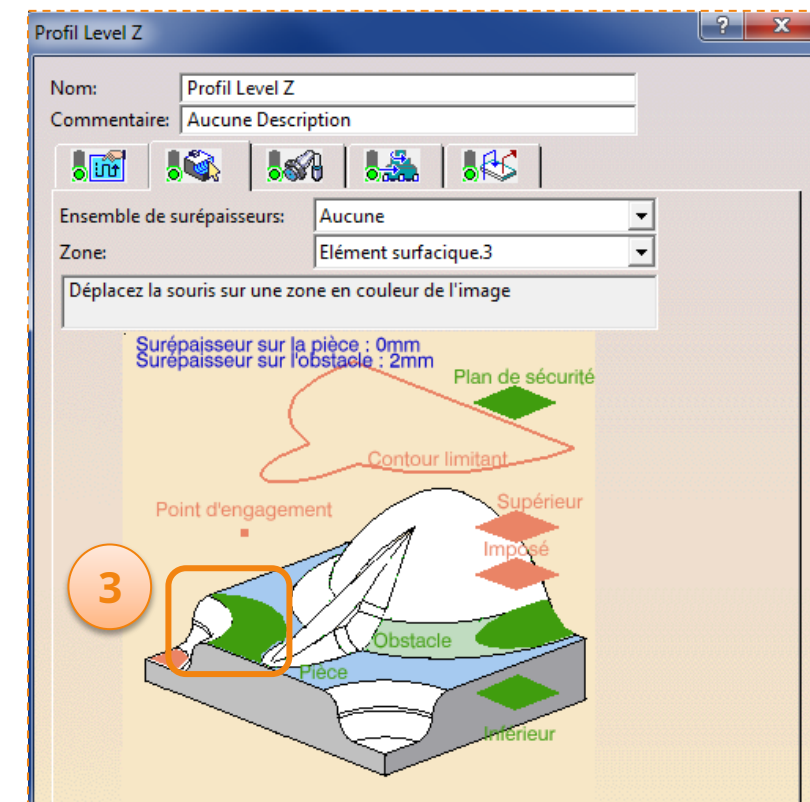
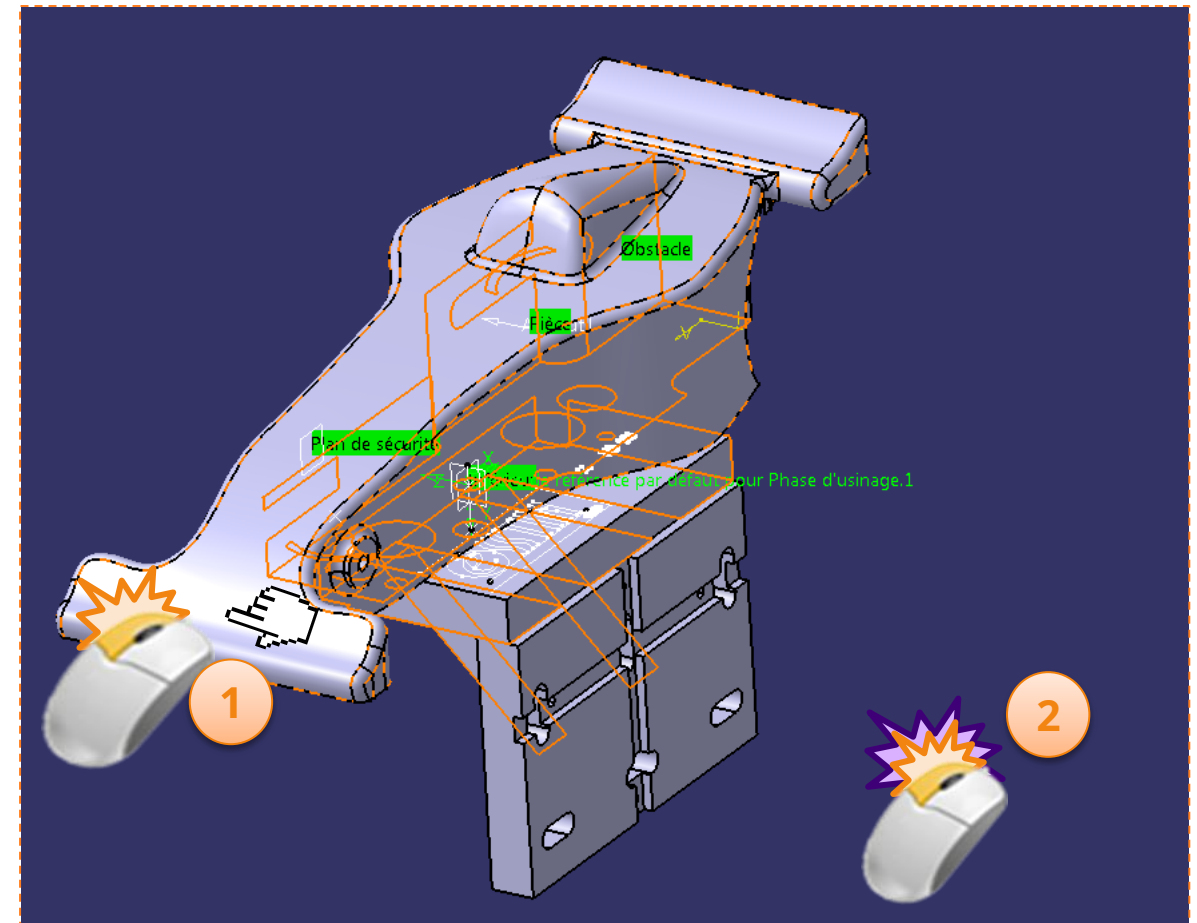




# Machining process

The operation **Profile Level Z** will allow us to refine the horizontal walls.

1. In the 3D environment, click on the **car body** as shown below.
2. Double-click in an empty space to confirm the selection and display the dialog box to new.
3. The red area is now green indicating that the geometries have been selected.
4. Do not click yet **ok**.





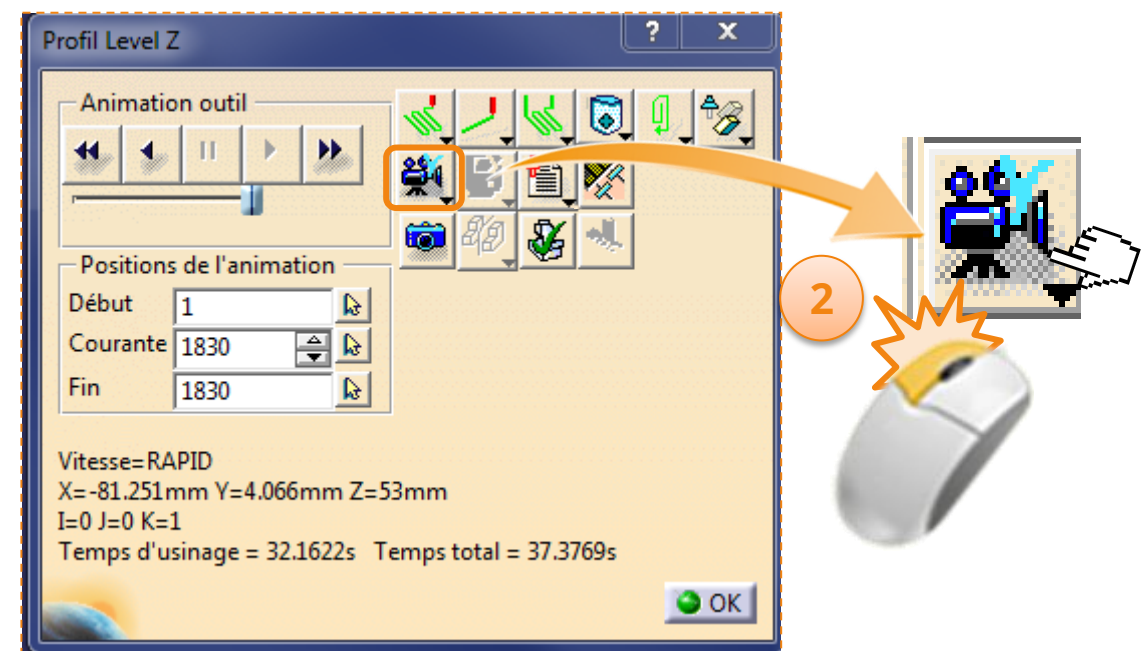
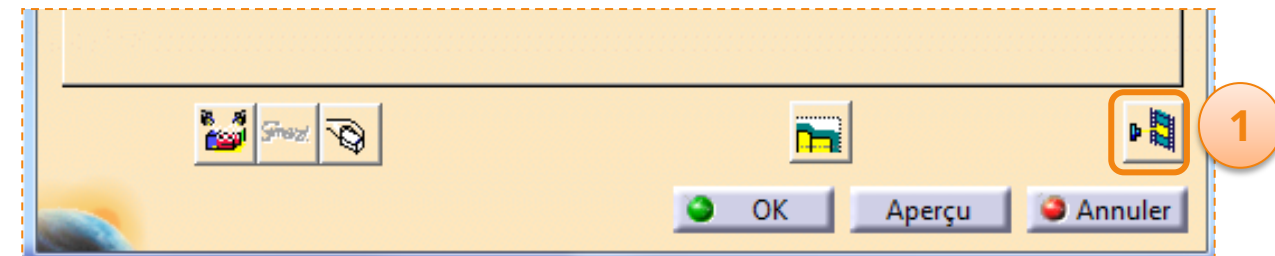
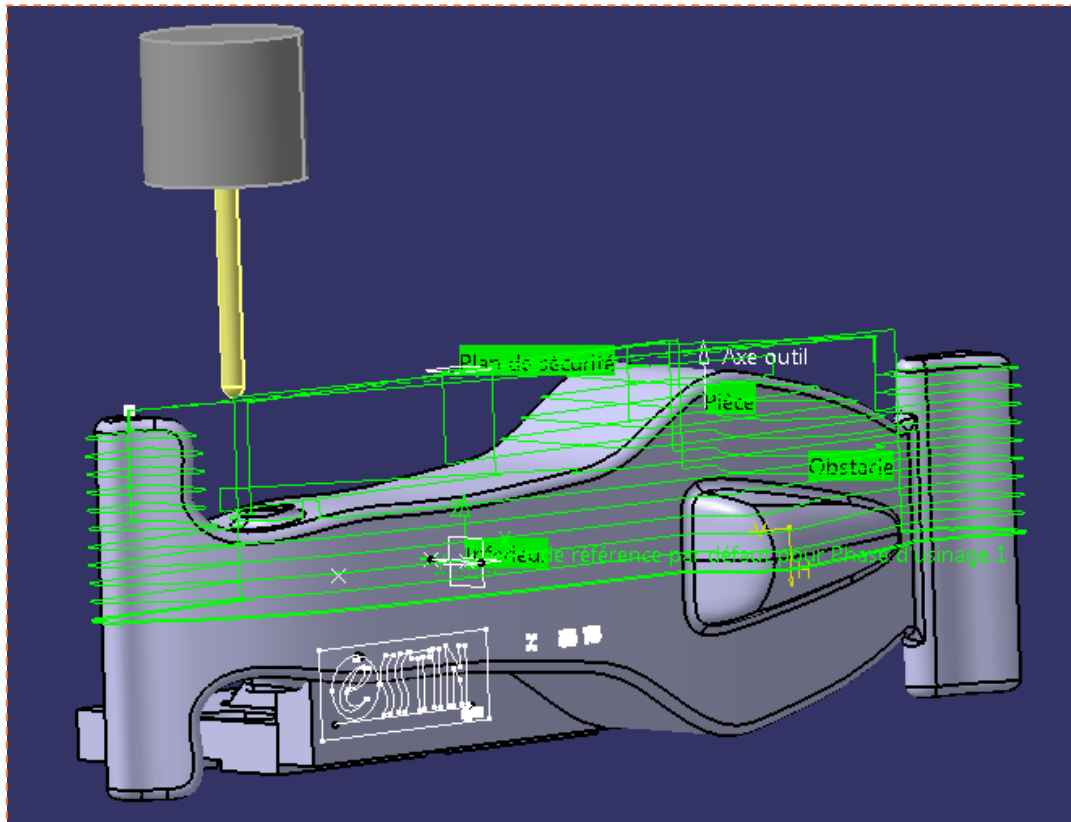
# Machining process

We are now going to simulate this operation in order to check that it works correctly.

1. Click the icon **Tool path animation** at the bottom right of the dialog box **Profile Level Z**.

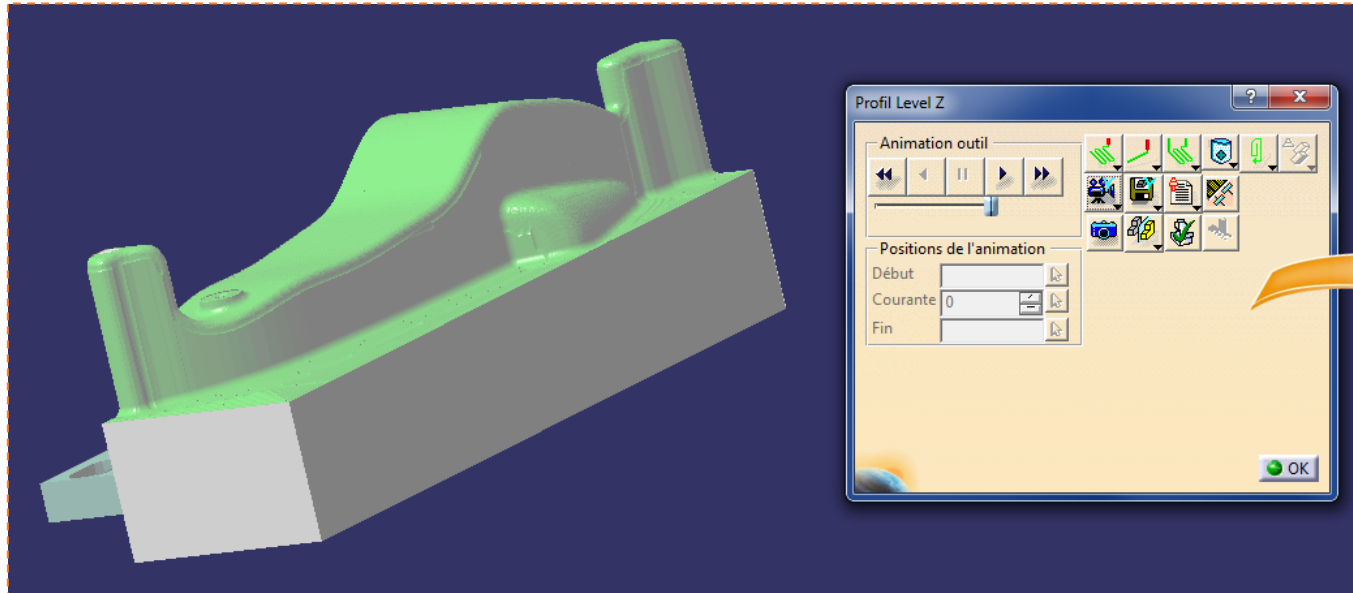
You will switch to simulation mode, the green lines represent the tool paths for this machining operation.

2. Click on the small arrow of the camera icon and choose..... In this way the simulation resumes from the last recorded result.



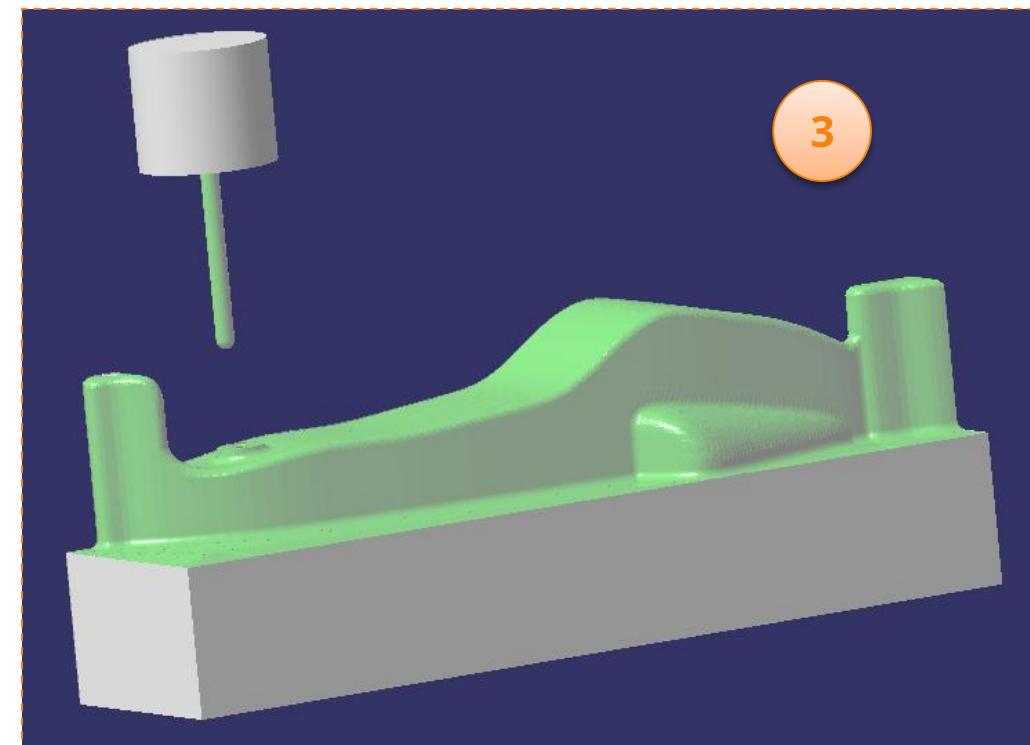
# Machining process

You should then have this screen:



1. Press the play button (**progress of the tool path in the forward direction**).
2. If the tool advances too much or not fast enough, you can always modify the execution speed with the horizontal scroll bar below the visualization controls.
3. Once the bypass is complete, you should get a result similar to the one shown here.

**Make sure that the tool does not come into contact with the assembly during the simulation. To do this, observe the result from all angles.**

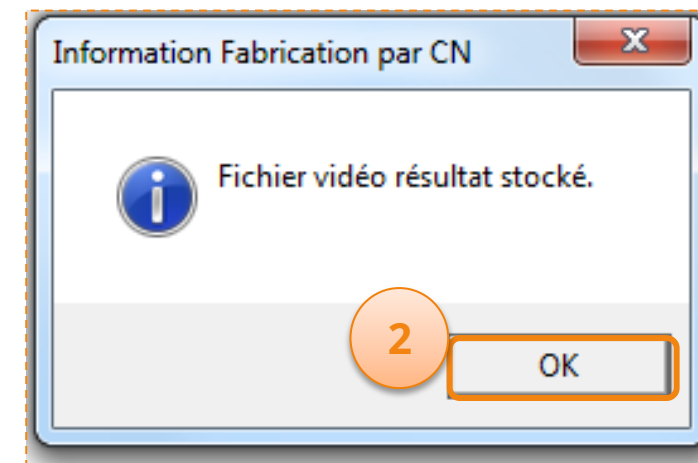


# Machining process

We will now save our result.

1. Click on the small arrow on the icon with a floppy disk to display the drop-down menu.
2. Select **Associate the video result to the operation**.  
Your bypass simulation is now saved, so in future simulations we can start from this position.
3. Click on **okay** to validate the recording.
4. Click on **okay** to exit the simulation.
5. Click on **okay** to exit the machining operation **Profile Level Z**.

Now the tool paths of the operation **Sketch** are calculated ("Solved" appears after the name of the operation in the Process).

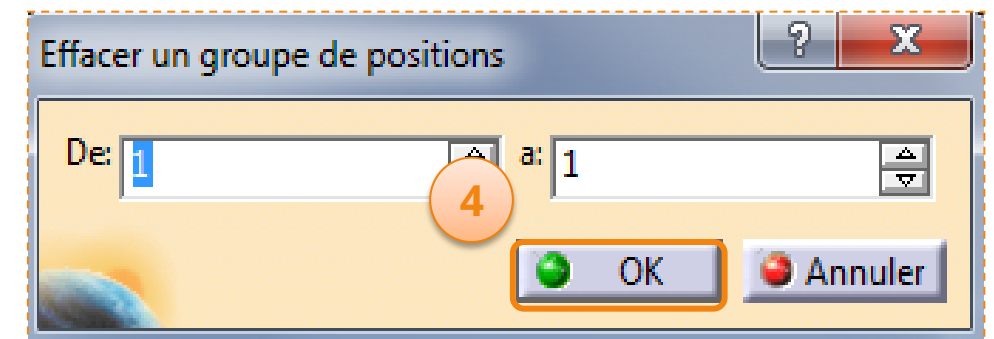
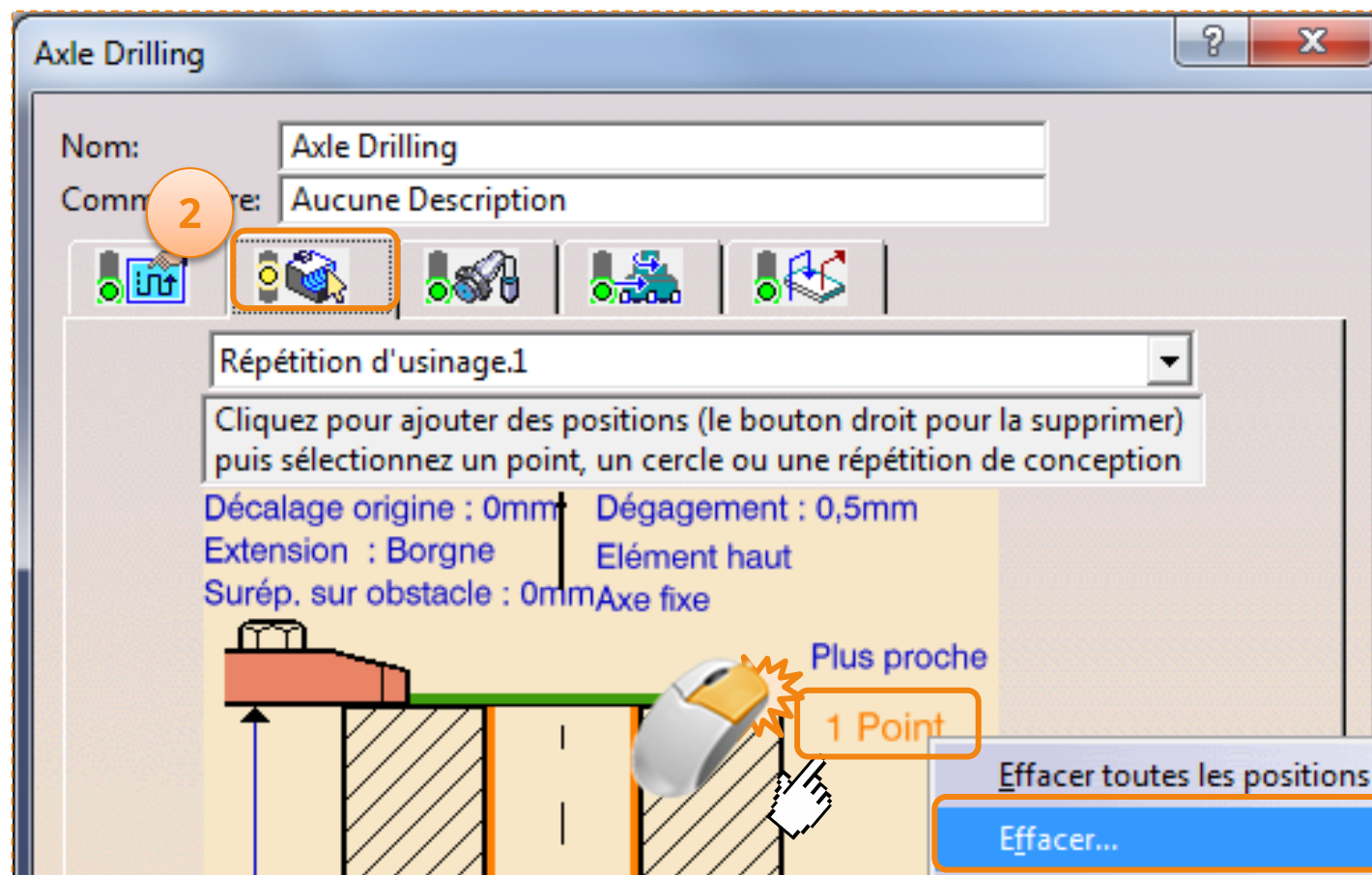
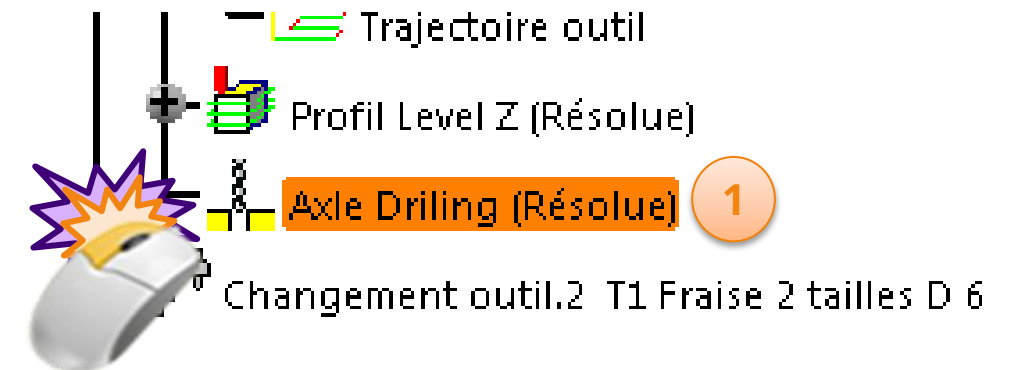


Be sure after each simulation to record the video result. Otherwise, the simulator will restart from the start of machining.

# Machining process

We are now going to update the operation of drilling the front end of the car.

1. Double click on the operation **Axle Drilling**.
2. The window below appears. The traffic light in the **geometric components** tab is in yellow. This means that you must modify the selection of an element to update the operation.
3. Right click on **1 point** and select **Wipe off**.
4. Click on **okay** to confirm deletion of the point.

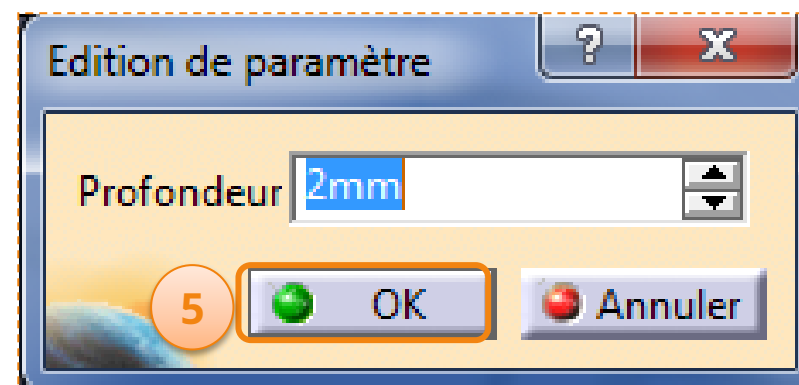
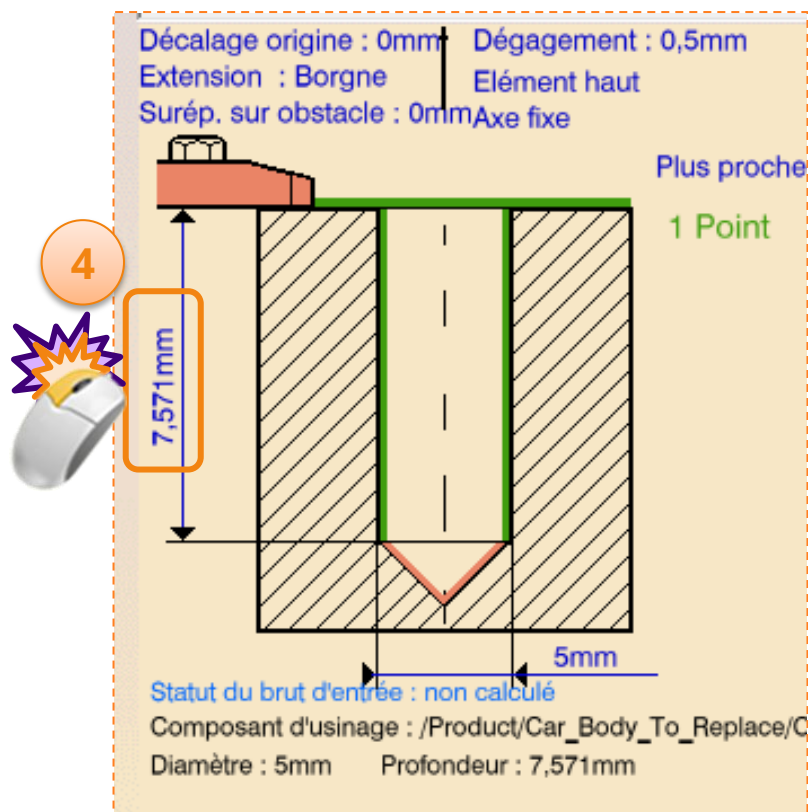
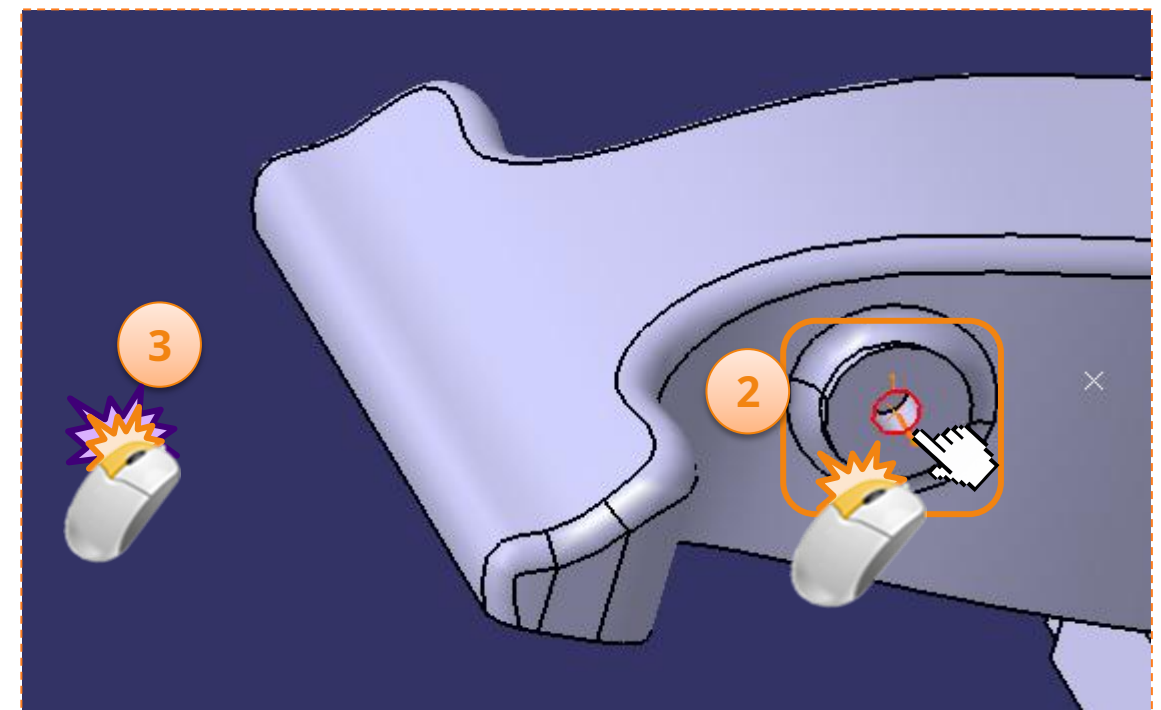
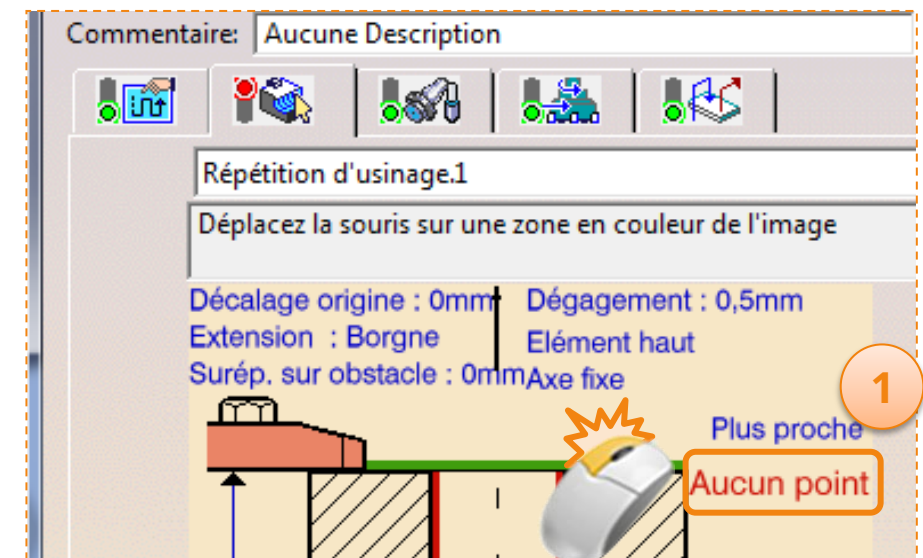




# Machining process

We will now define the center of our hole.

1. Click on **no points**.
2. The screen switches to the car. In the 3D environment, click **the circular edge of the front hole**.
3. Double-click in an empty space to confirm the selection and display the dialog box to new.
4. Modify the depth of the hole by double clicking on the depth value and fill in [2mm].
5. Validate by clicking on **okay**. In this way the drill will mark the center, then we can do the drilling manually on a drill press.





# Machining process

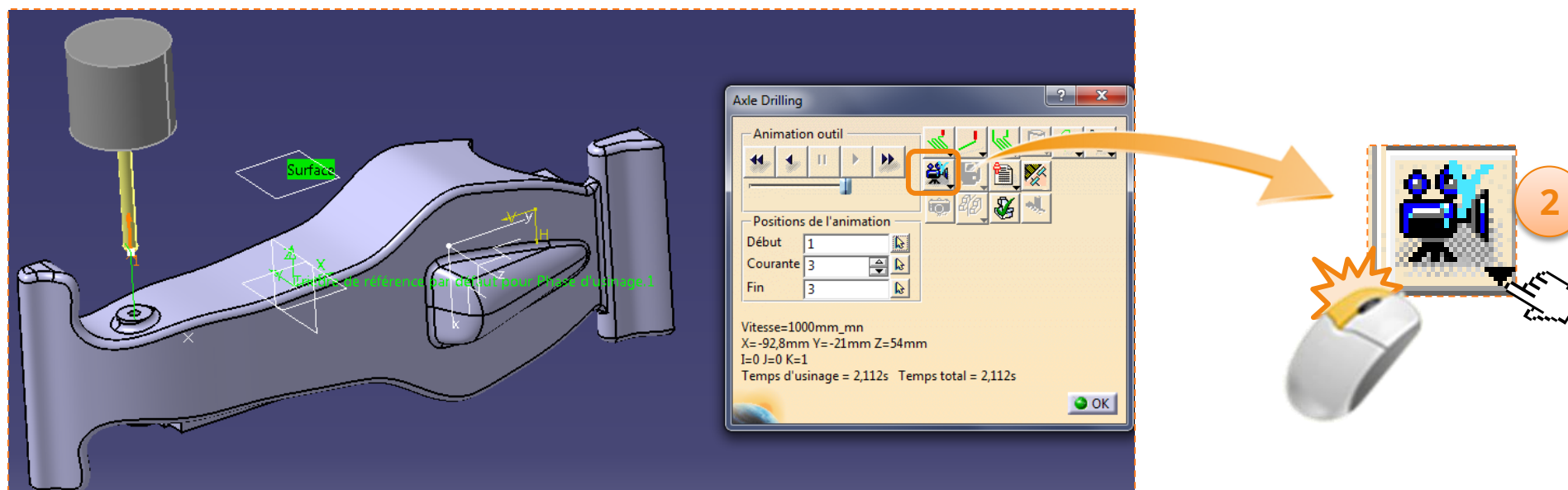
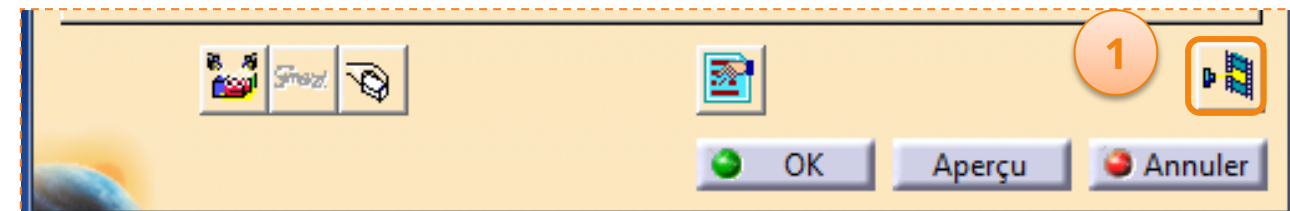
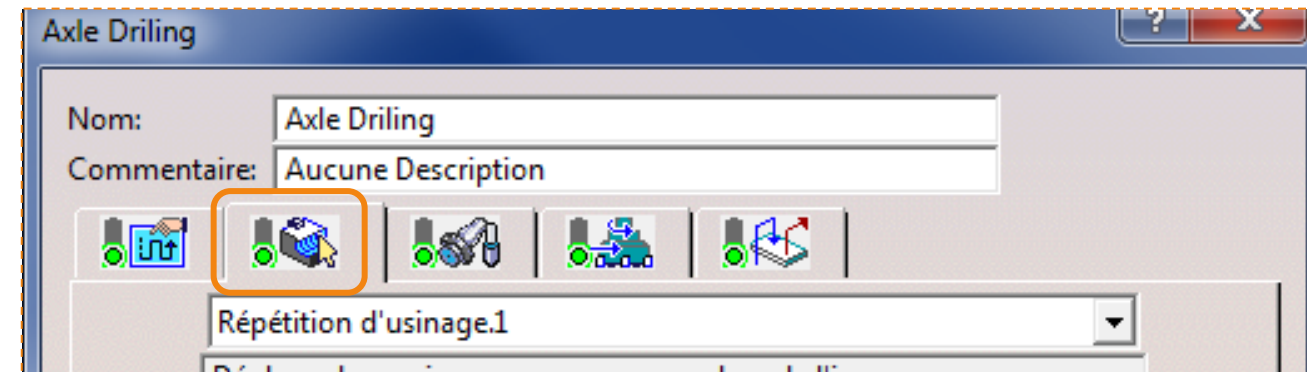
The light has turned green, so the geometries have been selected. **Do not click yet okay.**

We are now going to simulate this operation in order to check that it works correctly.

1. Click the icon **Tool path animation** at the bottom right of the dialog box **Axle Driling**.

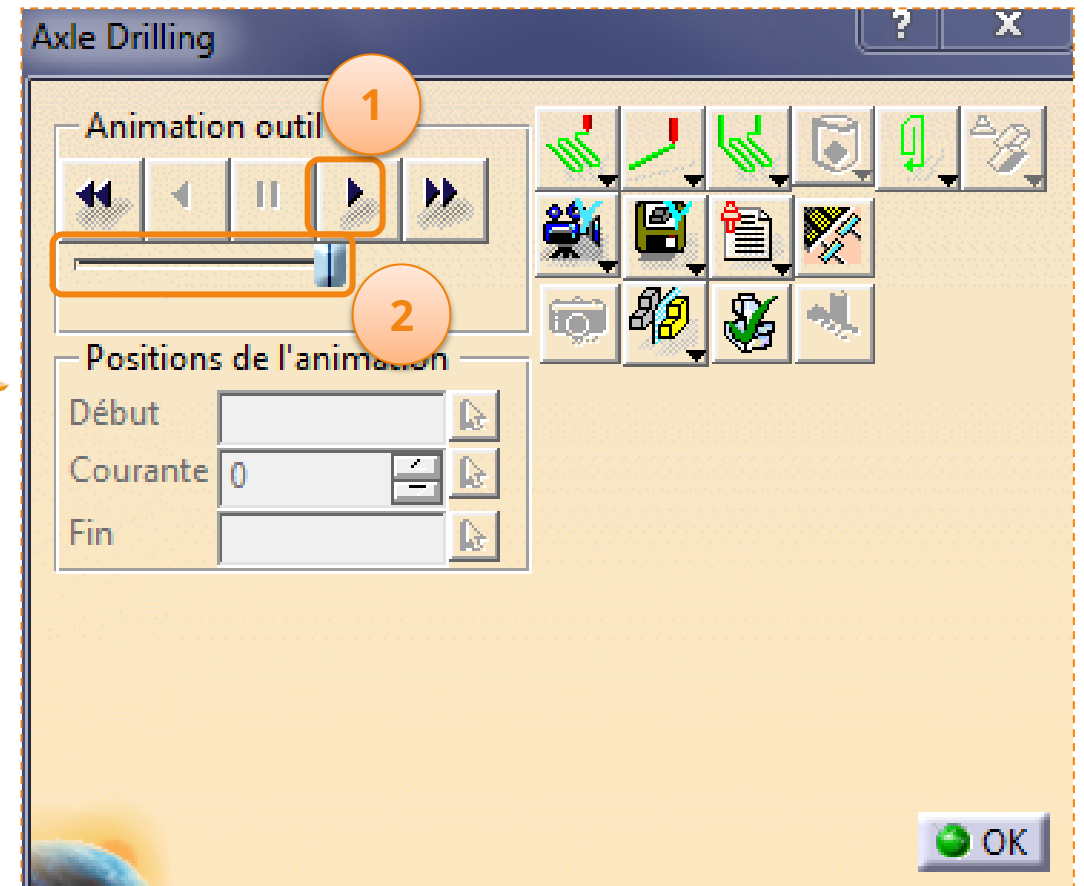
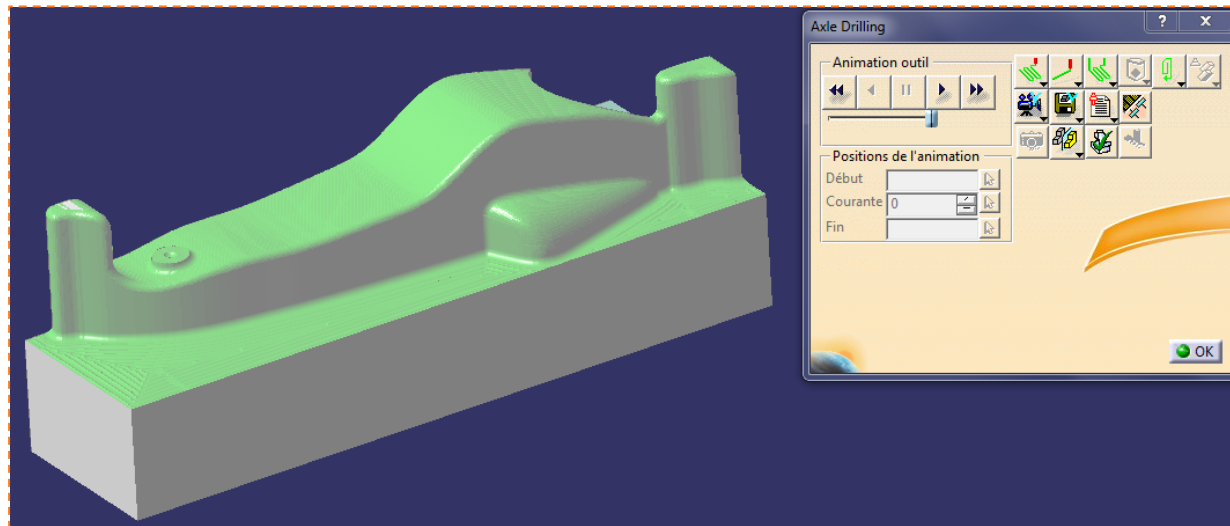
You will switch to simulation mode, the green lines represent the tool paths for this machining operation.

2. Click on the small arrow of the camera icon and choose **Video since last saved result**. In this way the simulation resumes from the last recorded result.



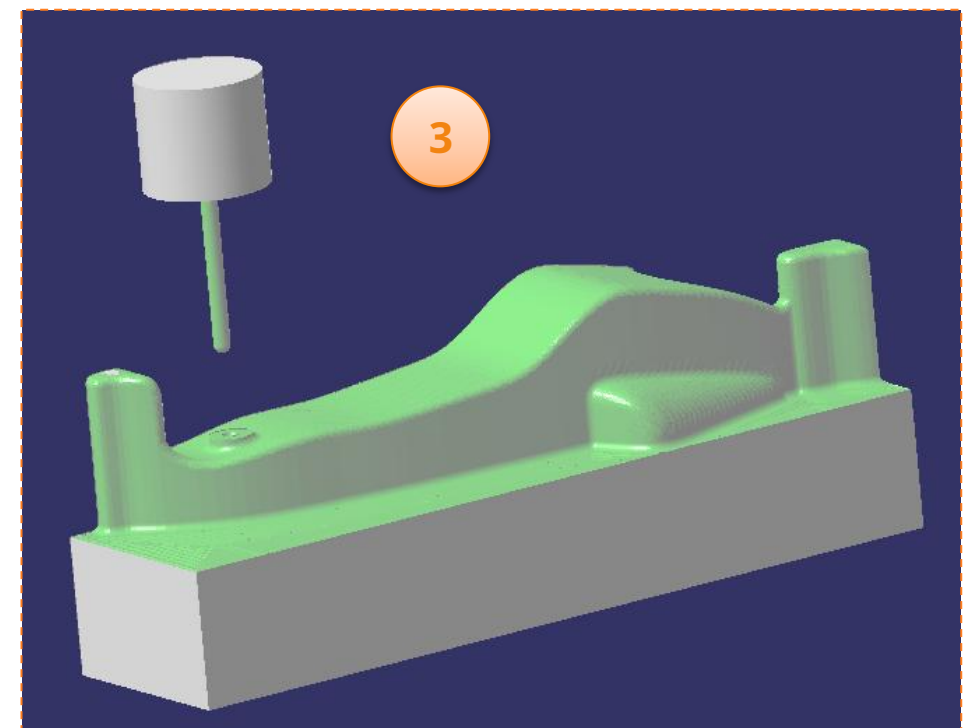
# Machining process

You should then have this screen:



1. Press the play button (**progress of the tool path in the forward direction**).
2. If the tool advances too much or not fast enough, you can always modify the execution speed with the horizontal scroll bar below the visualization controls.
3. Once the drilling is complete, you should obtain a result similar to that shown opposite.

**Make sure that the tool does not come into contact with the assembly during the simulation. To do this, observe the result from all angles.**



# Machining process

We will now save our result.

1. Click on the small arrow on the icon with a floppy disk to display the drop-down menu.
2. Select **Associate the video result to the operation**.  
Your drilling simulation of the front axle is now saved, so during the next simulation we can start from this position.
3. Click on **okay** to validate the recording.
4. Click on **okay** to exit the simulation.
5. Click on **okay** to exit the machining operation **Axle Driling**.

Now the tool paths of the operation **Sketch** are calculated ("Solved" appears after the name of the operation in the Process).

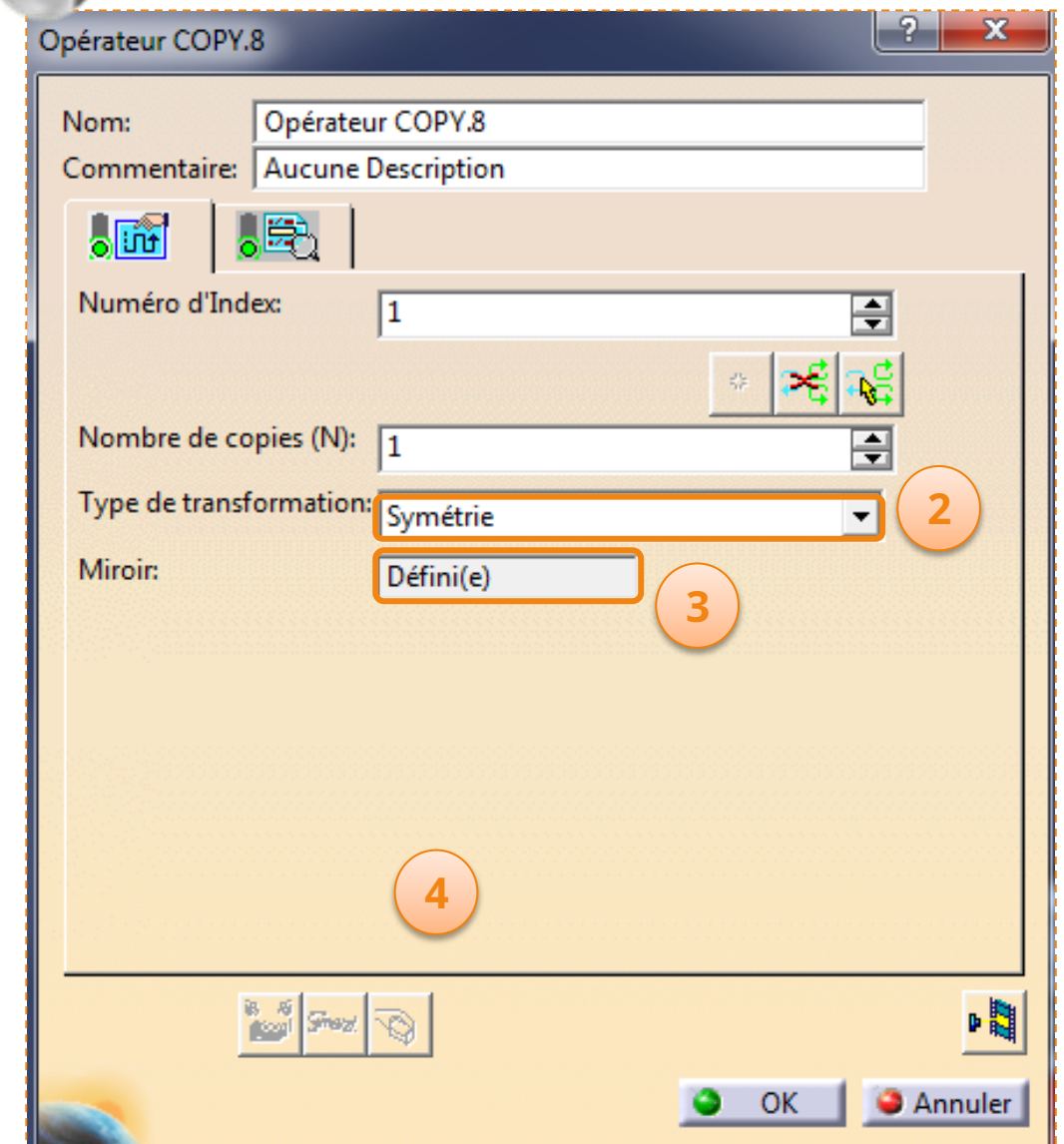
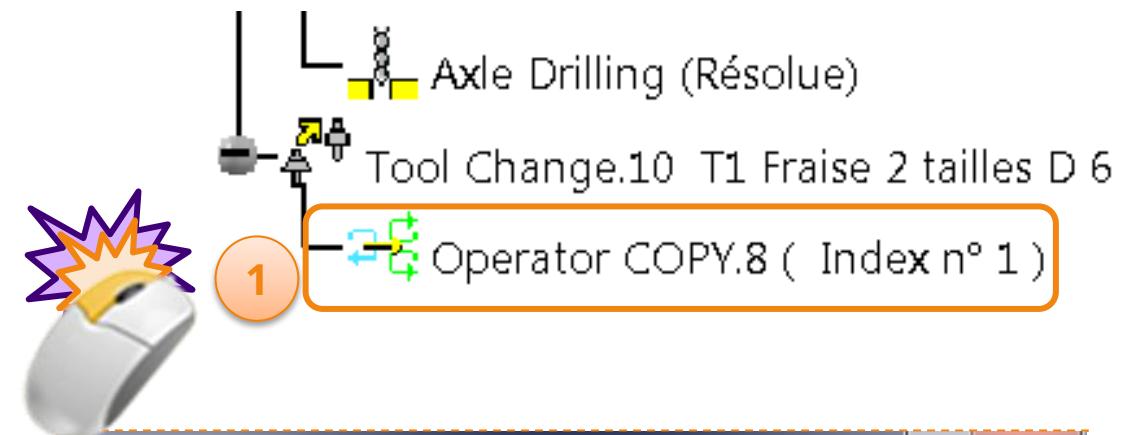
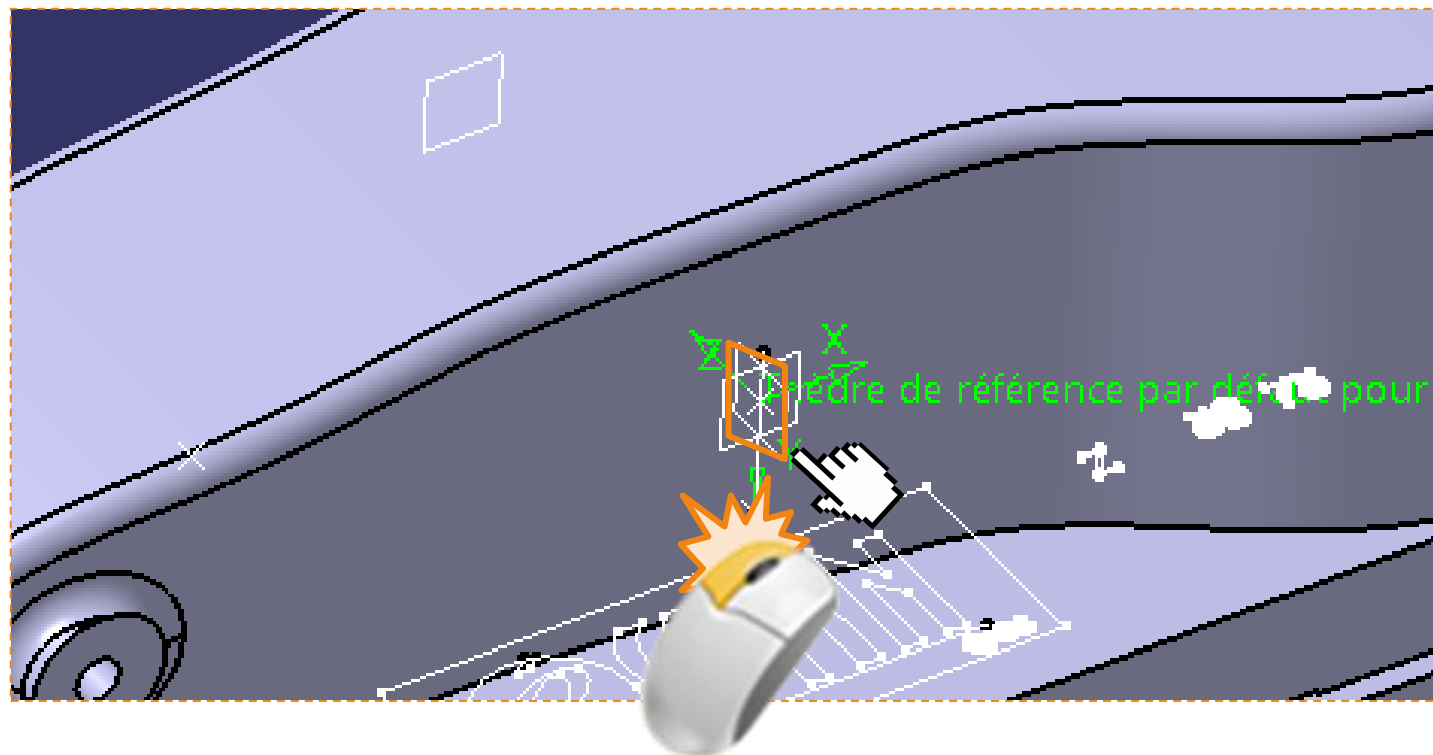


Be sure after each simulation to record the video result. Otherwise, the simulator will restart from the start of machining.

# Machining process

Now that the machining operations are parameterized for one face, it remains to parameterize a symmetry operation to perform the other face.

1. Double click **operator COPY.8 ( Index n°1)**.
2. In *type of transformation*, make sure that is selected **symmetry**.
3. Then click on the field **Mirror**. The window disappears to allow you to select the plane of symmetry.
4. Select the **plane (ZY)** of the car (the plane separating the front and rear part of the car). In this way we will be able to carry out a symmetry of the trajectories.



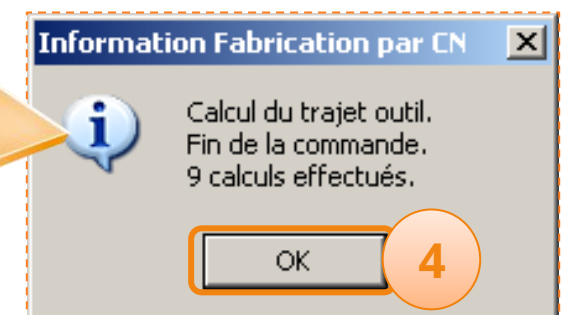
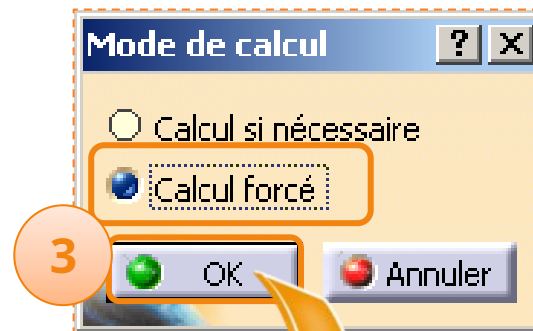
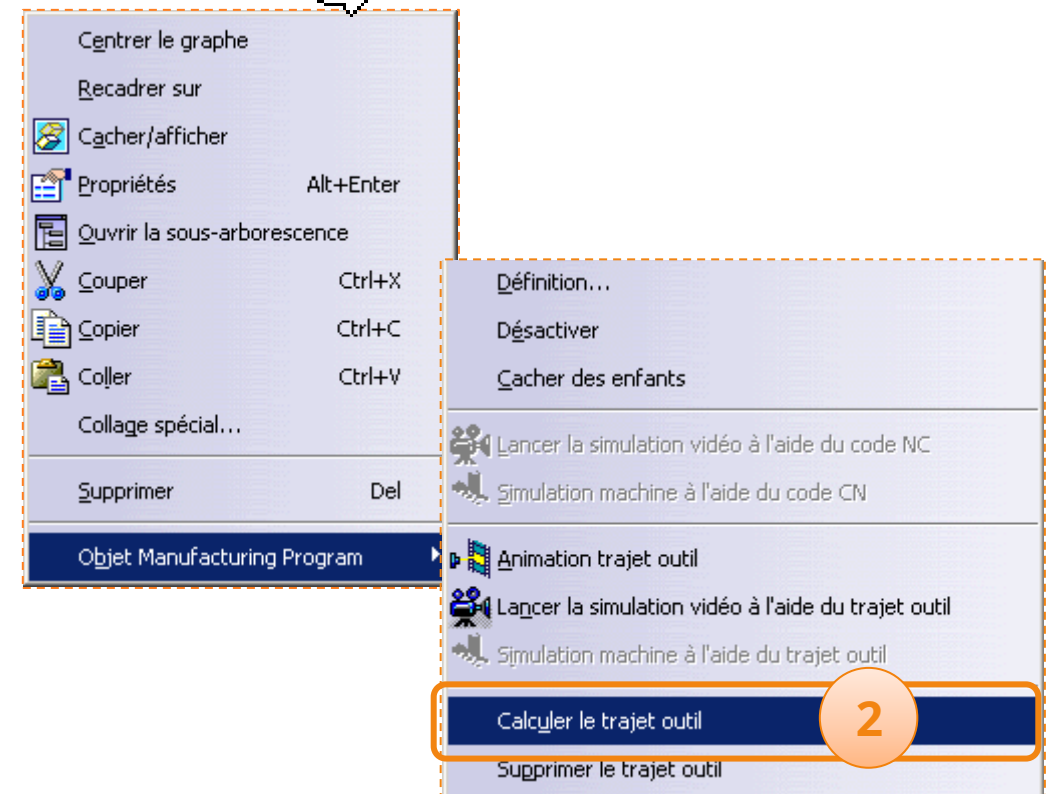
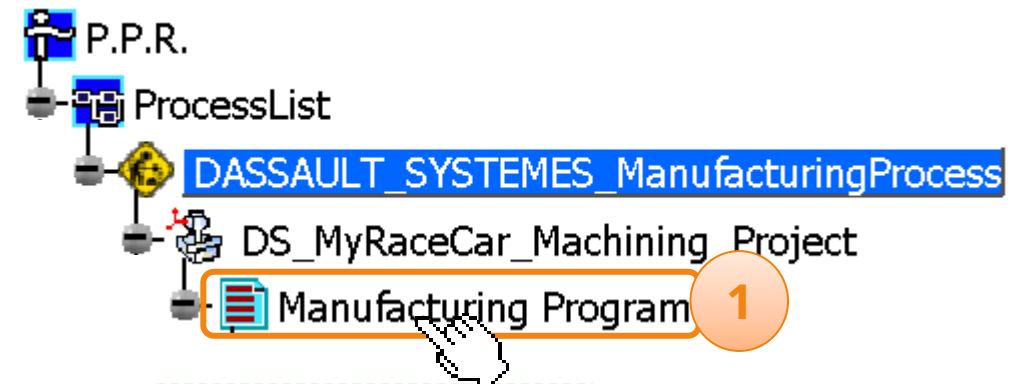
# Machining process

Now that you have completed the process for the second side, you can calculate the tool paths. For that:

1. Right click on **Manufacturing Program**.
2. Select **Manufacturing Program Object > Calculate Toolpath**.
3. Choose **Forced calculation** and click on **okay**.
4. A progress bar is displayed. Wait for the calculation to complete, then click **okay** in the information message.



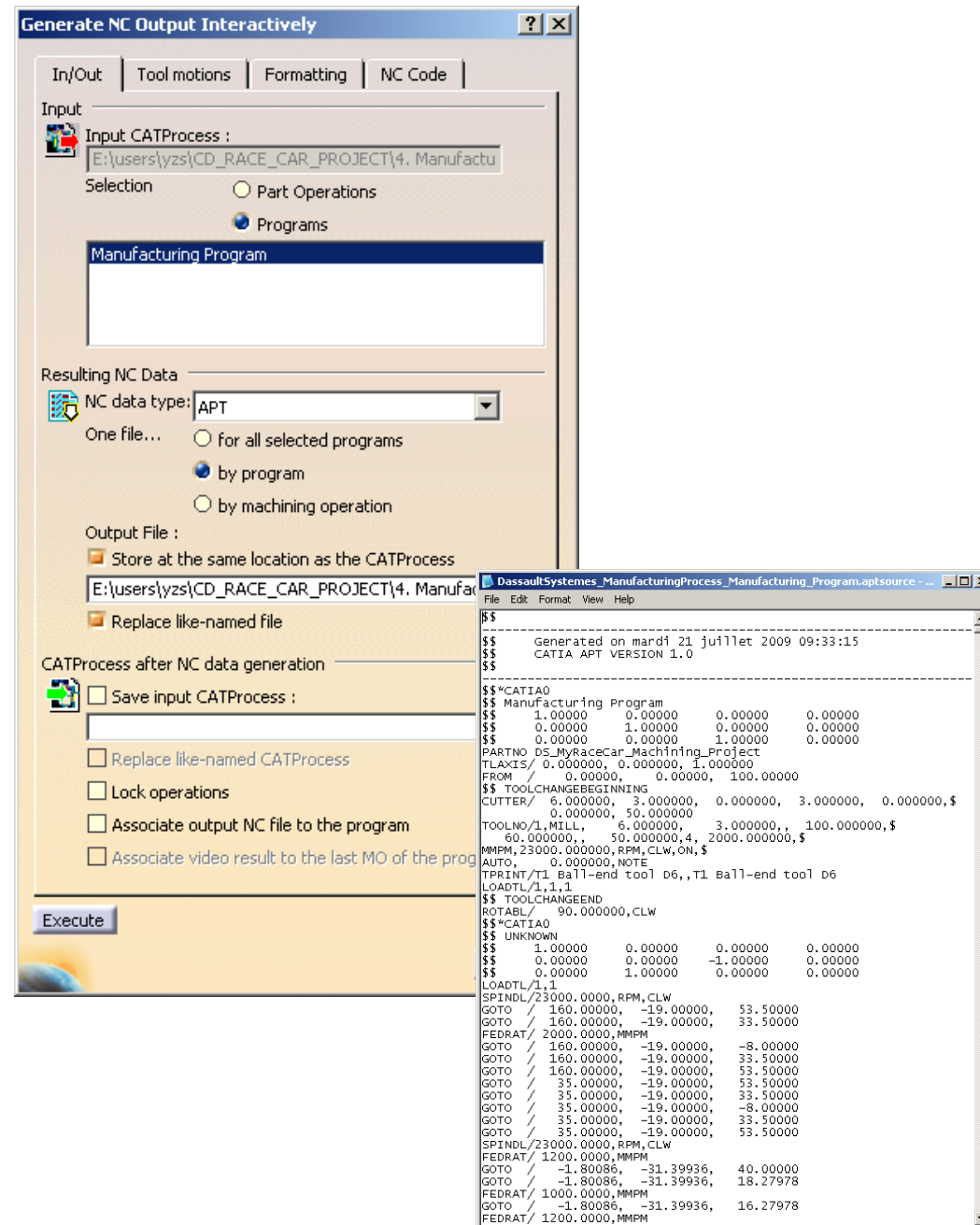
When the calculation is finished, each operation of the tree attach(**solved**).





# Generate an APT file

You will learn how to generate an APT file.



Here are the steps to follow:

1. Introduction.

2. CATIA V5 QuickStart.

**3. Your racing car.**

has. Draw your car.

b. Open your project.

vs. Create an immersive sketch.

d. Model your car.

e. Assemble your car.

f. Create a drawing.

g. Control and modify your car.

h. Create realistic renderings.

**i. Simulate machining.**

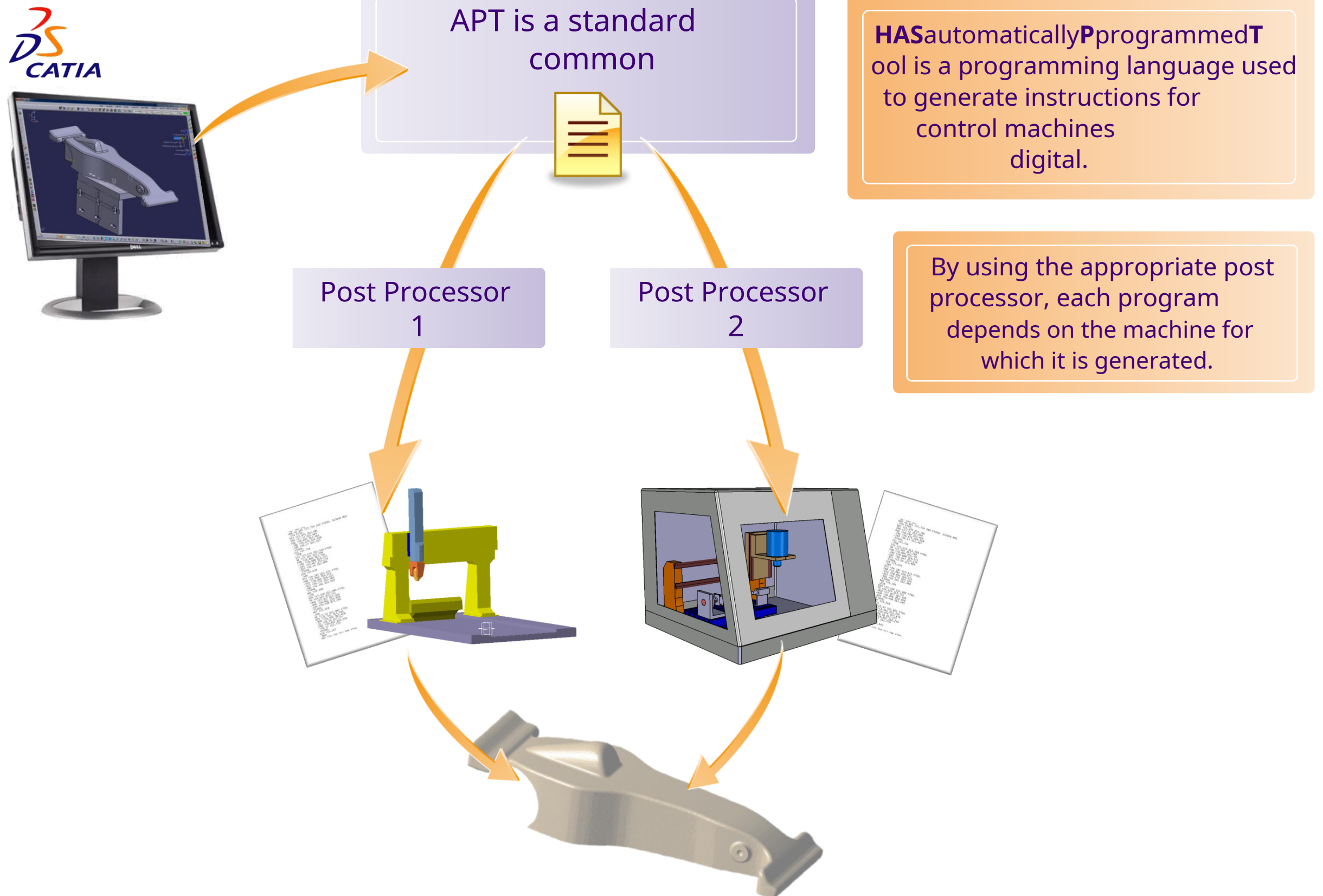
i. Machining process.

**ii. Generate an APT file.**

iii. Generate an ISO file.

4. Method sheets.

# Generate an APT file



# Generate an APT file

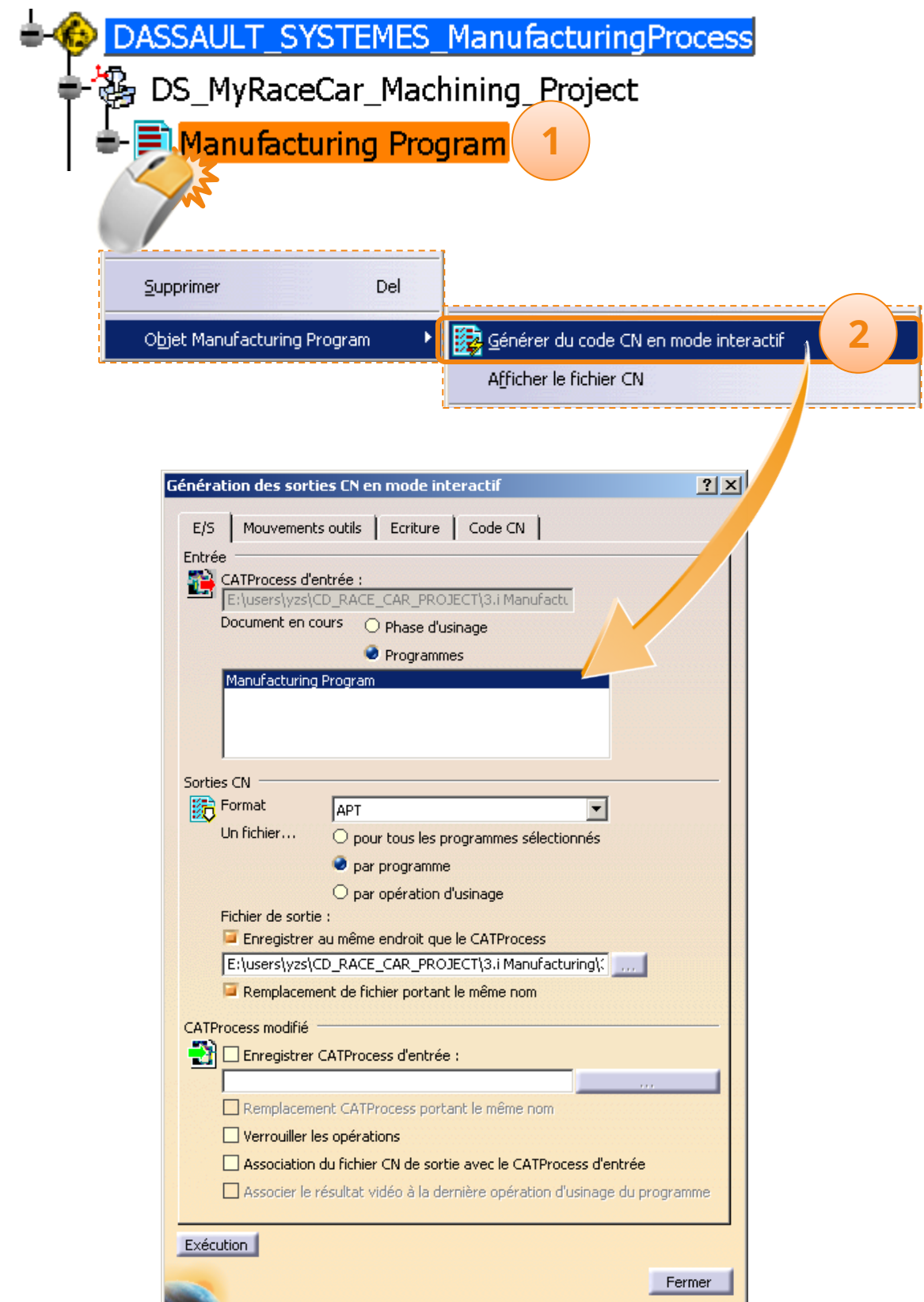
You will learn how to generate the APT file of the program you edited. But before that make sure you have backed up all your documents. For that :

1. Select **File > Save All**.

2. To generate the APT file, right click on **Manufacturing Program** in the specification tree.

3. Select **Object Manufacturing Program > Generate NC code in interactive mode**.

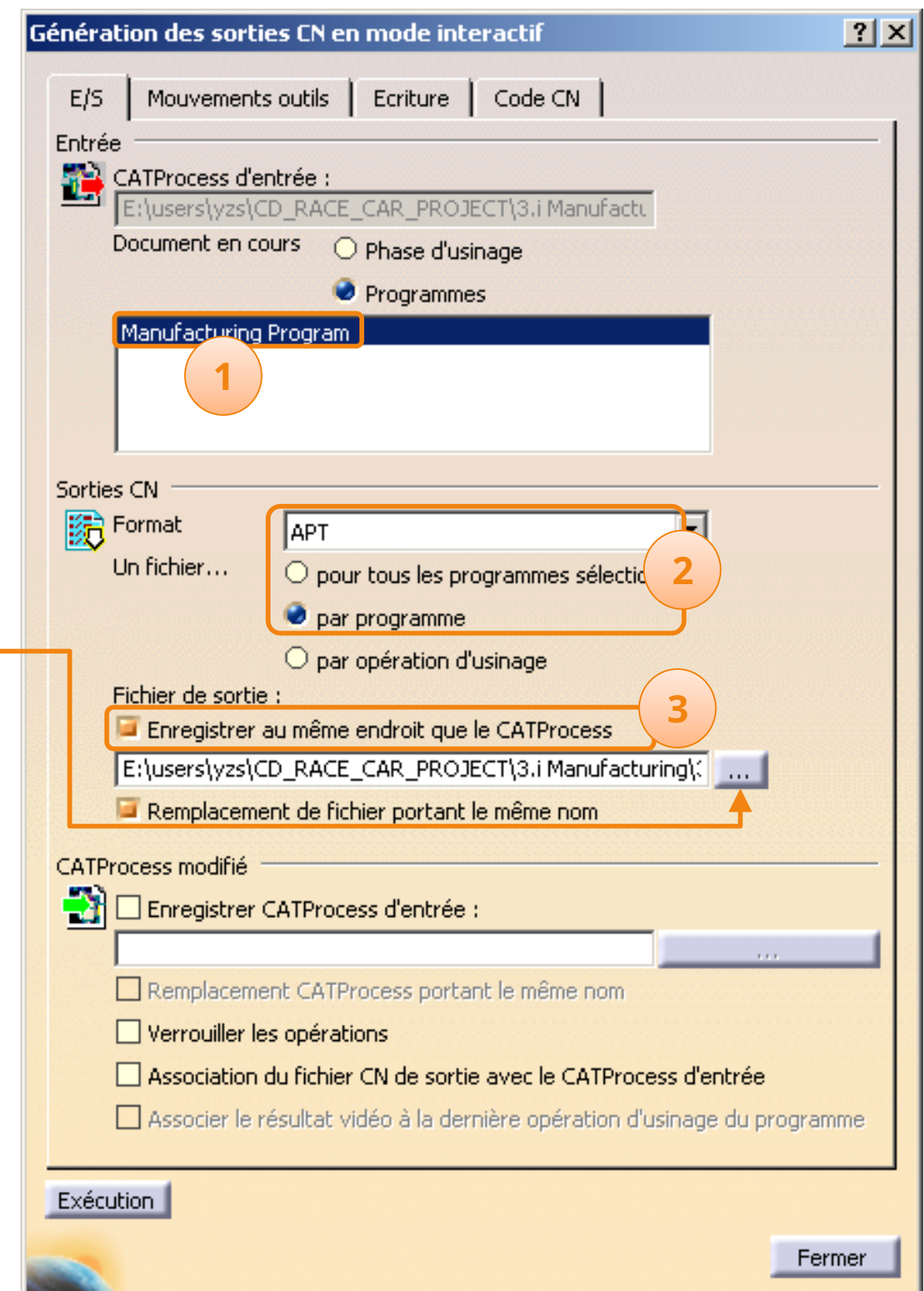
The following control panel appears, it contains four tabs.



# Generate an APT file

In this window, make sure to set the parameters correctly.

1. Check that **Manufacturing Program** is selected.
2. Select **APT** in the drop-down list, and check **programs** so that all machining operations are integrated into the APT file.
3. You can select the option **Save in the same place as the CATProcess**. Or specify the directory you want by clicking on the "... »".





# Generate an APT file

1. Now check in each tab that you have the same settings.

The image shows three overlapping screenshots of the 'Génération des sorties CN en mode interactif' dialog box, illustrating the settings across different tabs.

**Leftmost screenshot (Mouvements outils tab):**

- Stratégie au point de départ : D'après machine
- ☐ Inclure l'ordre GOTO lors des changements d'outils
- ☐ Sortir les syntaxes CYCLES
- ☐ Ne pas placer d'ordre GOTO avant les cycles
- ☐ Traiter les opérations de COPY et de TRACUT
- ☐ Retirer les points doubles après les commandes
- ☐ Retirer les points alignés
- Avances
  - ☒ Utiliser la valeur de l'avance rapide plutôt que la
  - ☐ Démarrer chaque opération en avance rapide
- Interpolation circulaire
  - Interpolation circulaire : Aucun
  - Rayons Max-Min : D'après machine
  - Rayon Max : 1000,000000
  - Rayon Min : 0,100000
  - Type des cercles : 3000 - 5000
  - Interpolation hélicoïdale : D'après machine

**Middle screenshot (Ecriture tab):**

- Commentaires
  - Déplacement outil : D'après machine
  - Commentaires : Aucun
  - Commentaires de la phase d'usinage : Aucun
  - Commentaires des opérations : Aucun
- Format des coordonnées des points (X,Y,Z)
  - Nombre de chiffres : 11
  - Nombre de décimales : 5
  - Réinitialiser
- Format des composantes axiales (I,J,K)
  - Nombre de chiffres : 9
  - Nombre de décimales : 6
  - Réinitialiser

**Rightmost screenshot (Code CN tab):**

- Fichier post-processeur IMS : Non spécifié ?



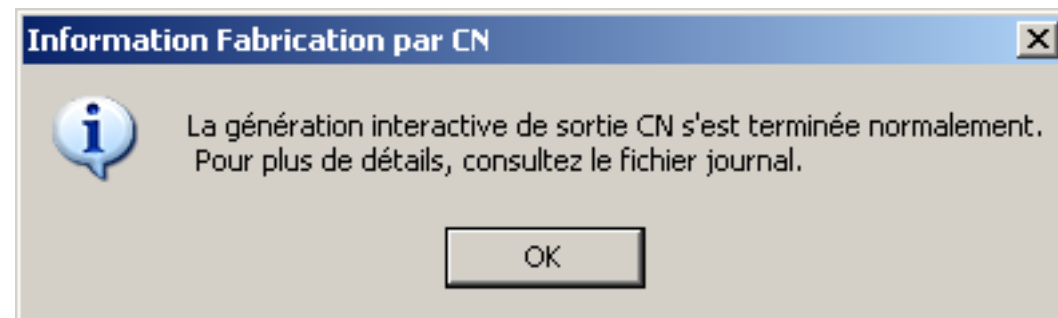
Notice that you don't need select a Post Processor for generate an APT file.



# Generate an APT file

You can now generate the APT file:

1. Click on **Execution** to create the APT file. The following dialog box appears:



2. Click on **okay**.

3. The APT file is now available in the directory you designated. It can be read with any text editor.



```
DassaultSystemes_ManufacturingProcess_Manufacturing_Program.aptsource ...
File Edit Format View Help
--
$ $
$ $ Generated on mardi 21 juillet 2009 09:33:15
$ $ CATIA APT VERSION 1.0
$ $
$ $*CATIAO
$ $ Manufacturing Program
$ $ 1.00000 0.00000 0.00000 0.00000
$ $ 0.00000 1.00000 0.00000 0.00000
$ $ 0.00000 0.00000 1.00000 0.00000
PARTNO DS_MyRacecar_Machining_Project
TLAXIS/ 0.000000, 0.000000, 1.000000
FROM 0.00000, 0.00000, 100.00000
$ $ TOOLCHANGEBEGINNING
CUTTER/ 6.000000, 3.000000, 0.000000, 3.000000, 0.000000,$
0.000000, 50.000000
TOOLNO/1,MILL, 6.000000, 3.000000, 100.000000,$
60.000000, 50.000000, 2000.000000,$
MMPM,23000.000000,RPM,CLW,ON,$
AUTO, 0.000000,NOTE
TPRINT/T1 Ball-end tool D6,,T1 Ball-end tool D6
LOADTL/1,1,1
$ $ TOOLCHANGEND
ROTABL/ 90.000000,CLW
$ $*CATIAO
$ $ UNKNOWN
$ $ 1.00000 0.00000 0.00000 0.00000
$ $ 0.00000 0.00000 -1.00000 0.00000
$ $ 0.00000 1.00000 0.00000 0.00000
LOADTL/1,1,1
SPINDL/23000.0000,RPM,CLW
GOTO / 160.00000, -19.00000, 53.50000
GOTO / 160.00000, -19.00000, 33.50000
FEDRAT/ 2000.0000,MMPM
GOTO / 160.0000, -19.0000, -8.00000
GOTO / 160.0000, -19.0000, 33.50000
GOTO / 160.0000, -19.0000, 53.50000
GOTO / 35.00000, -19.0000, 53.50000
GOTO / 35.0000, -19.0000, 33.50000
GOTO / 35.0000, -19.0000, -8.00000
GOTO / 35.0000, -19.0000, 33.50000
GOTO / 35.0000, -19.0000, 53.50000
SPINDL/23000.0000,RPM,CLW
FEDRAT/ 1200.0000,MMPM
GOTO / -1.80086, -31.39936, 40.00000
GOTO / -1.80086, -31.39936, 18.27978
FEDRAT/ 1000.0000,MMPM
GOTO / -1.80086, -31.39936, 16.27978
FEDRAT/ 1200.0000,MMPM
```

**Génération des sorties CN en mode interactif**

E/S | Mouvements outils | Ecriture | Code CN

Entrée

CATProcess d'entrée :  
E:\users\yzs\CD\_RACE\_CAR\_PROJECT\3.i Manufactu

Document en cours ☐ Phase d'usinage  
☒ Programmes

Manufacturing Program

Sorties CN

Format

Un fichier... ☐ pour tous les programmes sélectionnés  
☒ par programme  
☐ par opération d'usinage

Fichier de sortie :  
☒ Enregistrer au même endroit que le CATProcess  
E:\users\yzs\CD\_RACE\_CAR\_PROJECT\3.i Manufacturing\...  
☒ Remplacement de fichier portant le même nom

CATProcess modifié

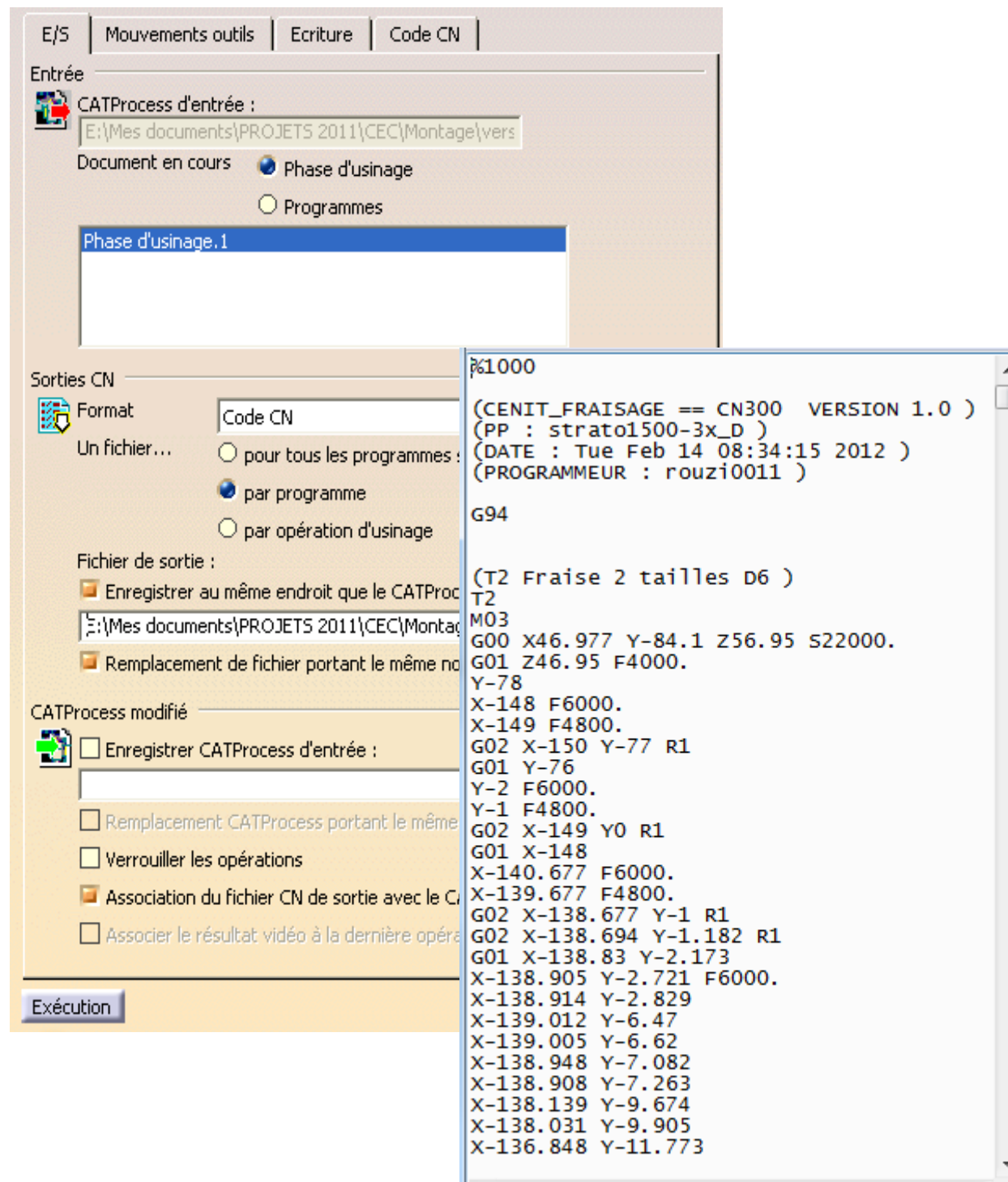
☒ Enregistrer CATProcess d'entrée :  
...  
☐ Remplacement CATProcess portant le même nom  
☐ Verrouiller les opérations  
☐ Association du fichier CN de sortie avec le CATProcess d'entrée  
☐ Associer le résultat vidéo à la dernière opération d'usinage du programme

1 Exécution

Fermer

# Generate ISO file

You will learn how to generate an ISO file.



Here are the steps to follow:

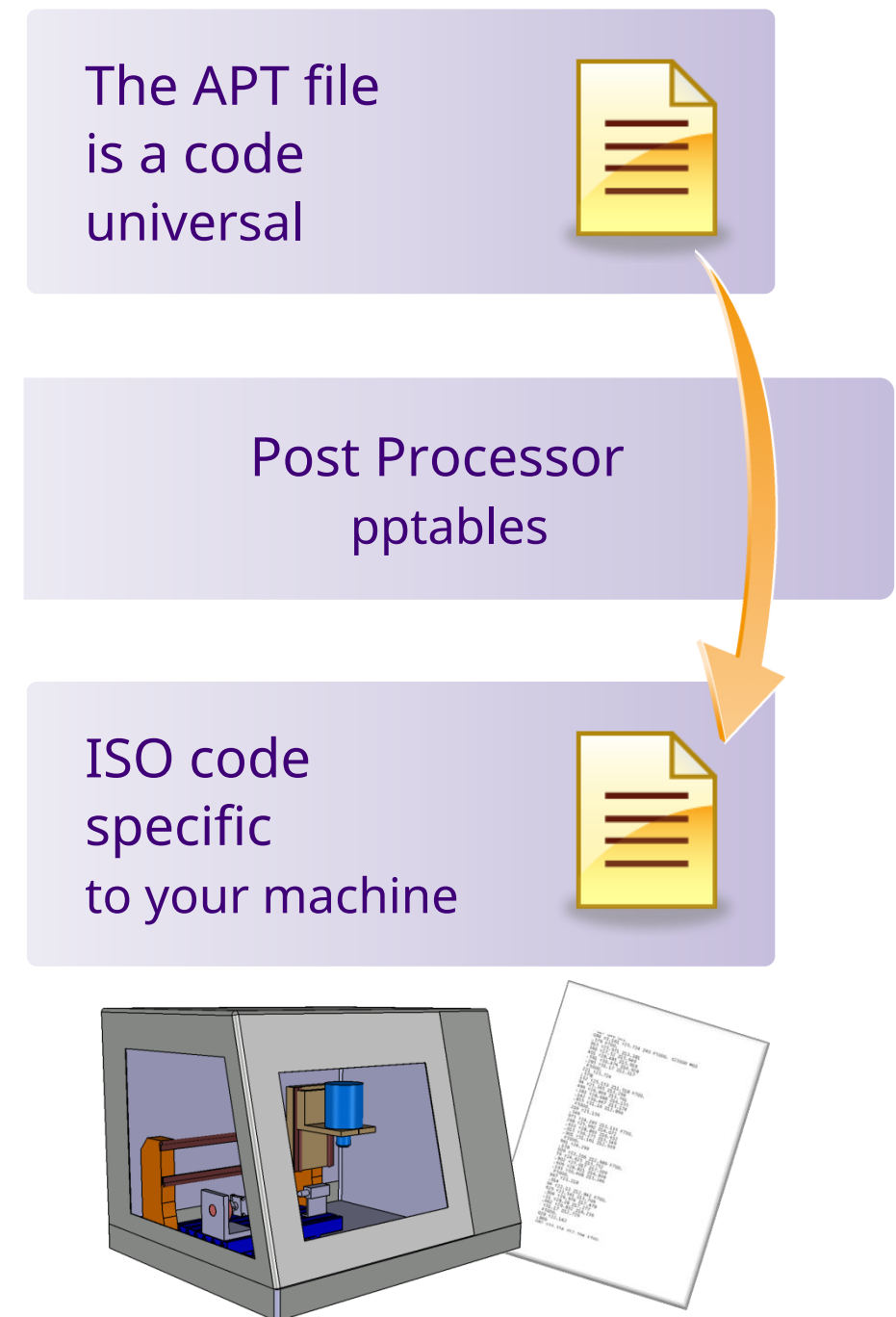
1. Introduction.
2. CATIA V5 QuickStart.
3. *Your racing car.*
  - has. Draw your car.
  - b. Open your project.
  - vs. Create an immersive sketch.
  - d. Model your car.
  - e. Assemble your car.
  - f. Create a drawing.
  - g. Control and modify your car.
  - h. Create realistic renderings.
- i. Simulate machining.*
  - i. Machining process.
  - ii. Generate an APT file.
- iii. Generate an ISO file.*
4. Method sheets.

# Generate ISO code

In this part you will learn how to generate an ISO code to communicate with CNC machines.

The post processor is a program that relies on pptables to convert a common standard APT file to a machine-specific ISO file. We can assimilate the pptables to a kind of library containing an equivalence between the instructions of the APT file and the instructions of the ISO file.

There are different post processor manufacturers. pptables are thus specific to post processors.

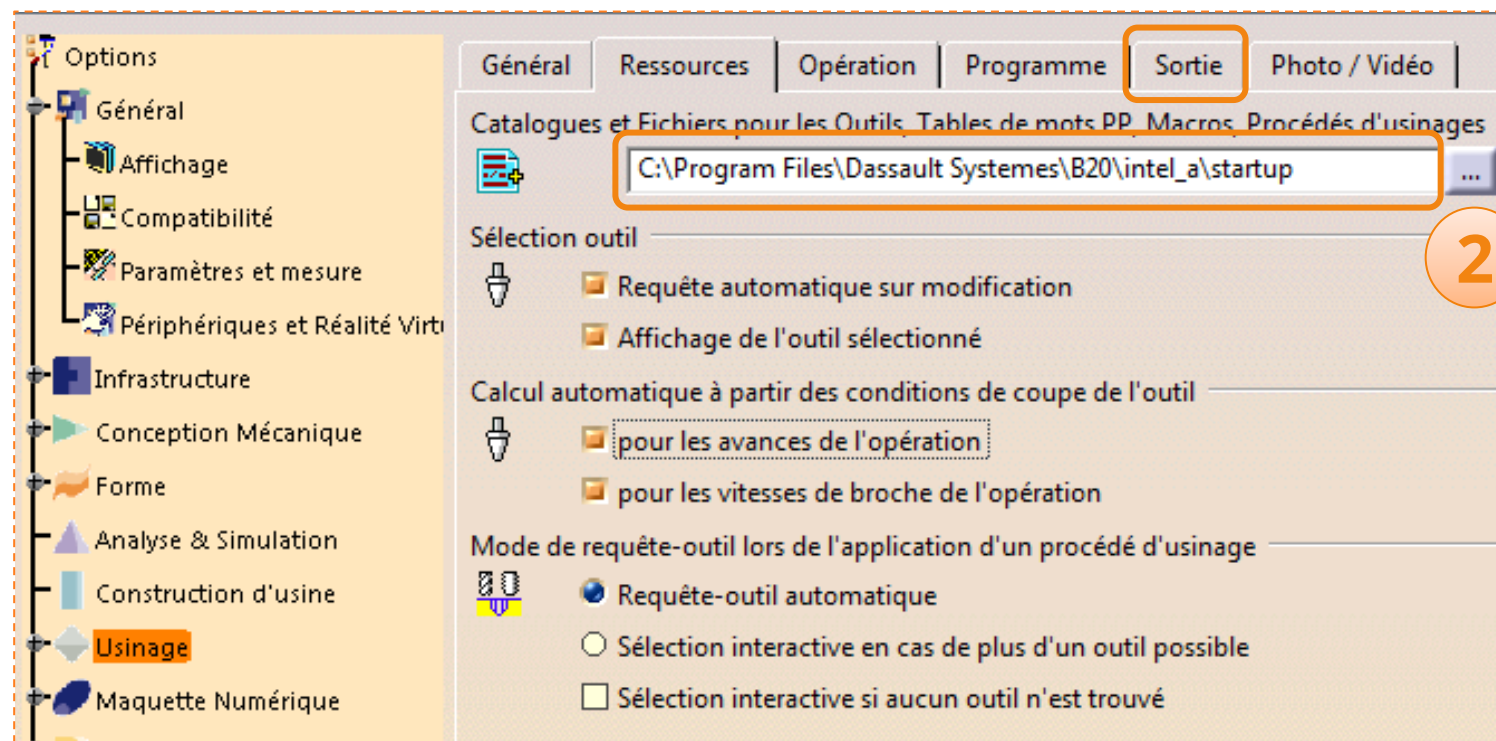
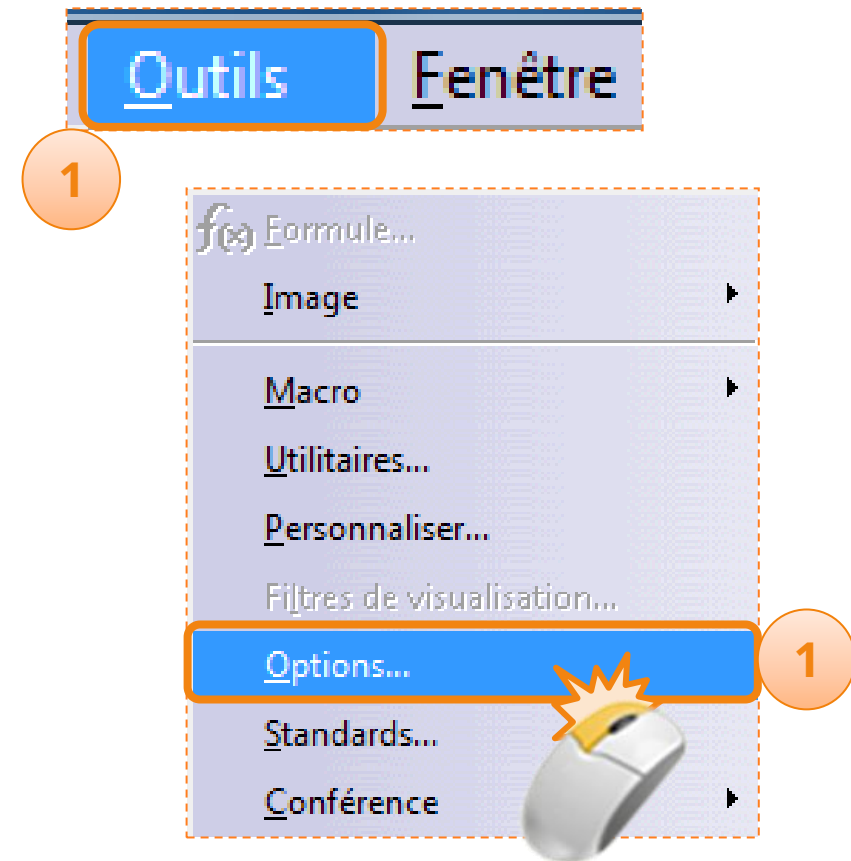


# Generate ISO file

You are now going to learn how to generate an ISO file that can be interpreted by Stratoconcept and Charly4U type machines from CharlyRobot, present in most colleges, high schools and resource centers.

At prior, make sure that machining that is correctly configured:

1. Go into **Tools > Options... > Machining**.
2. Inspect the tab **Resources**: make sure the path to catalogs and files ends with *Istartup* and that the window is configured as follows:

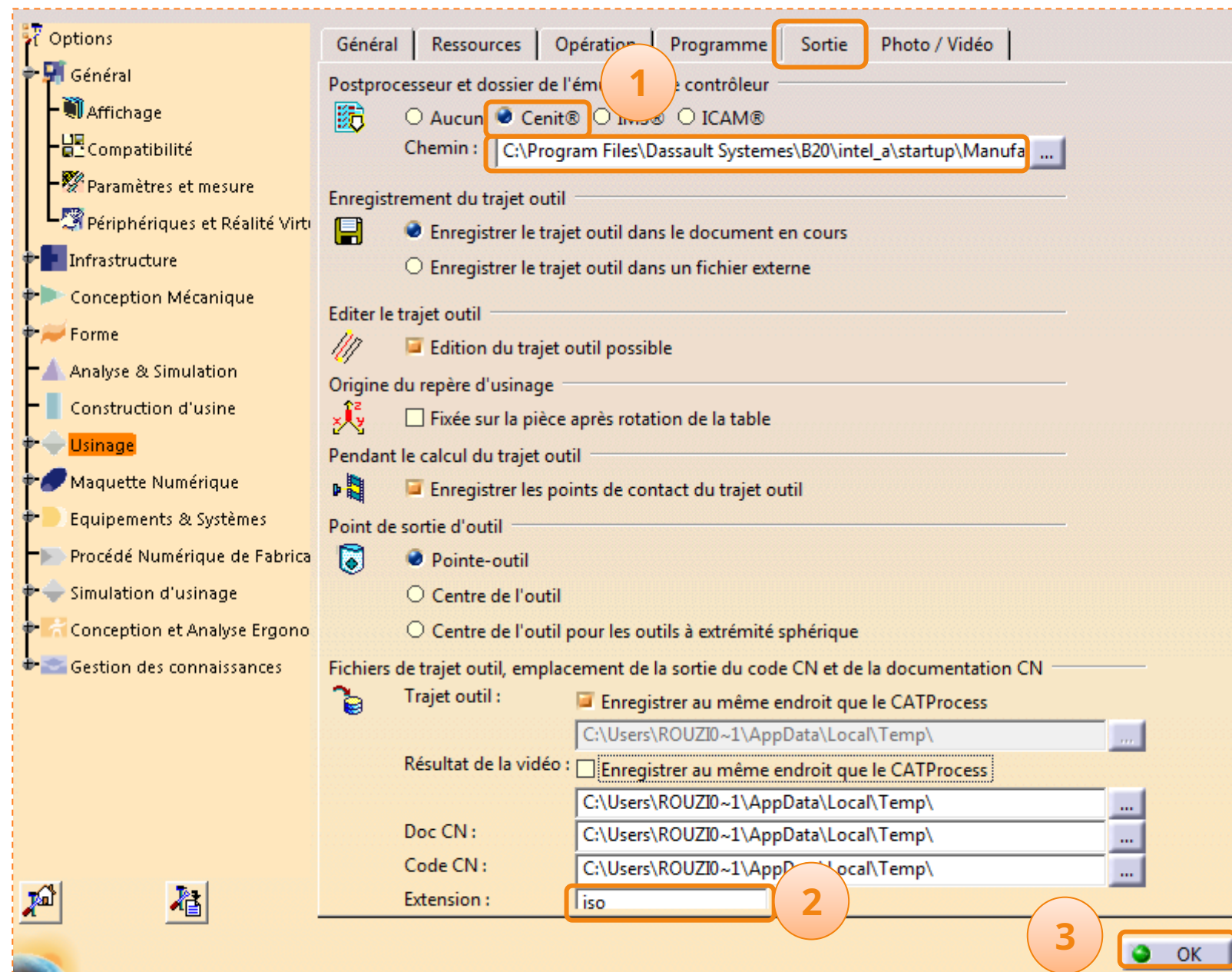




# Generate ISO file

1. Then inspect the tab **Exit**: check again that the path is of the type **!start-up\Manufacturing** and the options are set as follows:
2. Finally, to generate the ISO code at the end, scroll down the list of options and in the "extension" field, enter [ISO].

3. Finally, click **okay**.





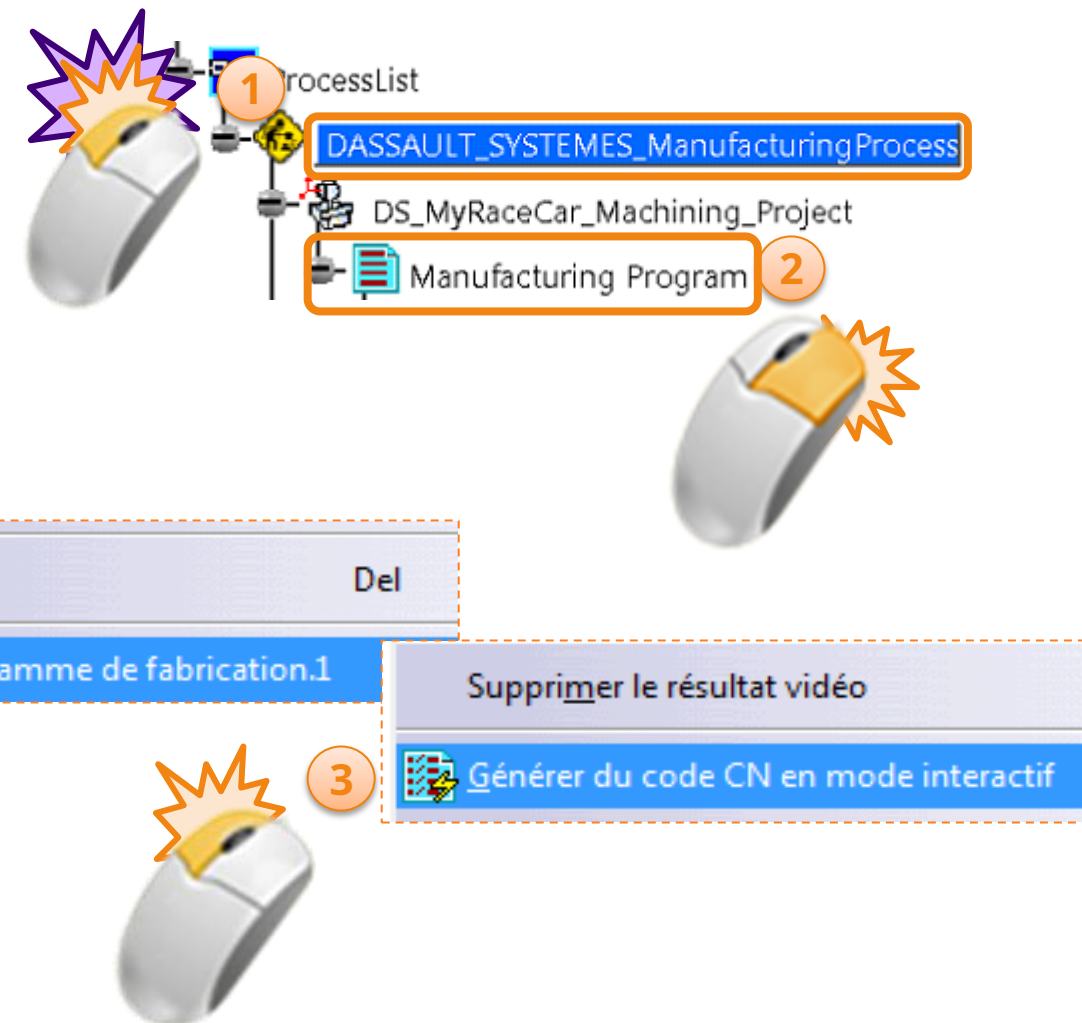
# Generate ISO file

Once the options are checked, all you have to do is generate the ISO code.

1. Double click **DASSAULT\_SYSTEMES\_ManufacturingProcess** in the tree.

2. Right click on **Manufacturing Program**.

3. In **Object Manufacturing program.1**, select **generate NC code in interactive mode.1**.



# Generate ISO file

A window opens and you must configure it.

1. Be sure that *Document in progress* be sure **Machining phase**.
2. Adjust the *NC output* on **NC code** and tick **by program**.
3. Check the box **Save in the same place as the CATProcess**. You will then find your program in the same directory as your process file and it will have the same name as your CATProcess file.
4. Finally, generate your ISO code by clicking on **Execution**.
5. When the code will be edited without errors, you will have the window below. Then click on **okay**.

E/S | Mouvements outils | Ecriture | Code CN

Entrée

CATProcess d'entrée : C:\Users\rouzi0011\Desktop\Process Mà\Da... **1**

Document en cours : ☒ Phase d'usinage ☐ Programmes

DS\_MyRaceCar\_Machining\_Project

Sorties CN

Format : Code CN

Un fichier... ☐ pour tous les programmes sélectionnés ☒ par programme **2** ☐ par opération d'usinage

Fichier de sortie : ☒ Enregistrer au même endroit que le CATProcess **3**

C:\Users\rouzi0011\Desktop\Process Mà\DassaultSystemes

☒ Remplacement de fichier portant le même nom

CATProcess modifié

☐ Enregistrer CATProcess d'entrée : ...

☐ Remplacement CATProcess portant le même nom

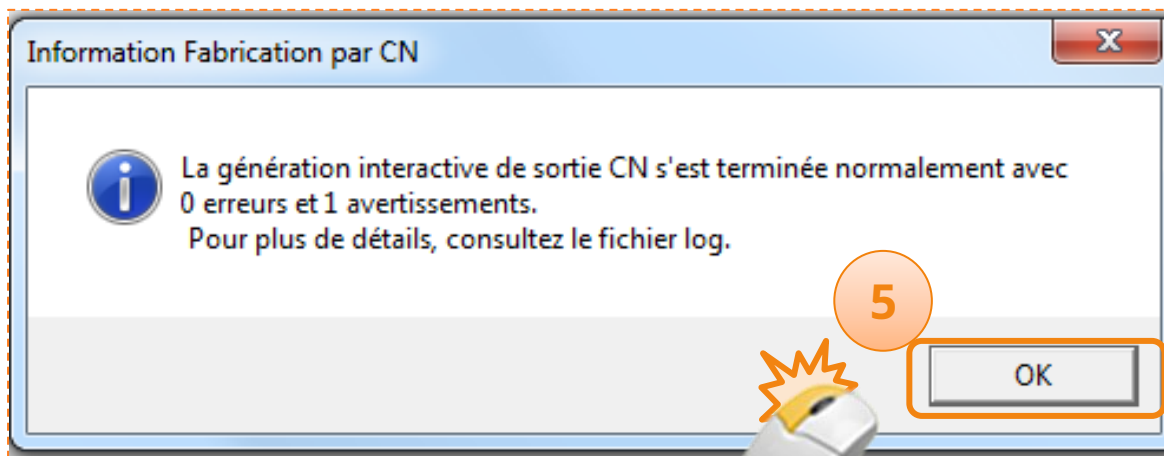
☐ Verrouiller les opérations

☐ Association du fichier CN de sortie avec le CATProcess d'entrée

☐ Associer le résultat vidéo à la dernière opération d'usinage du programme

**4** Exécution

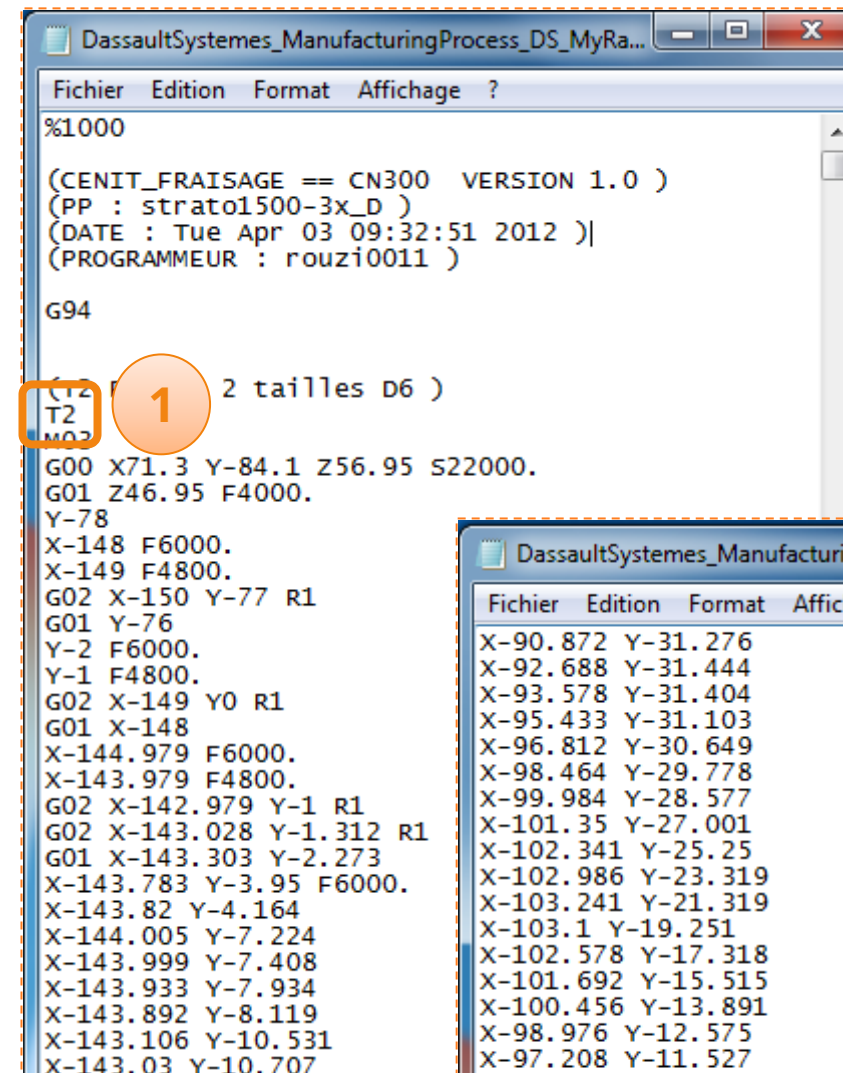
Fermer



# Generate ISO file

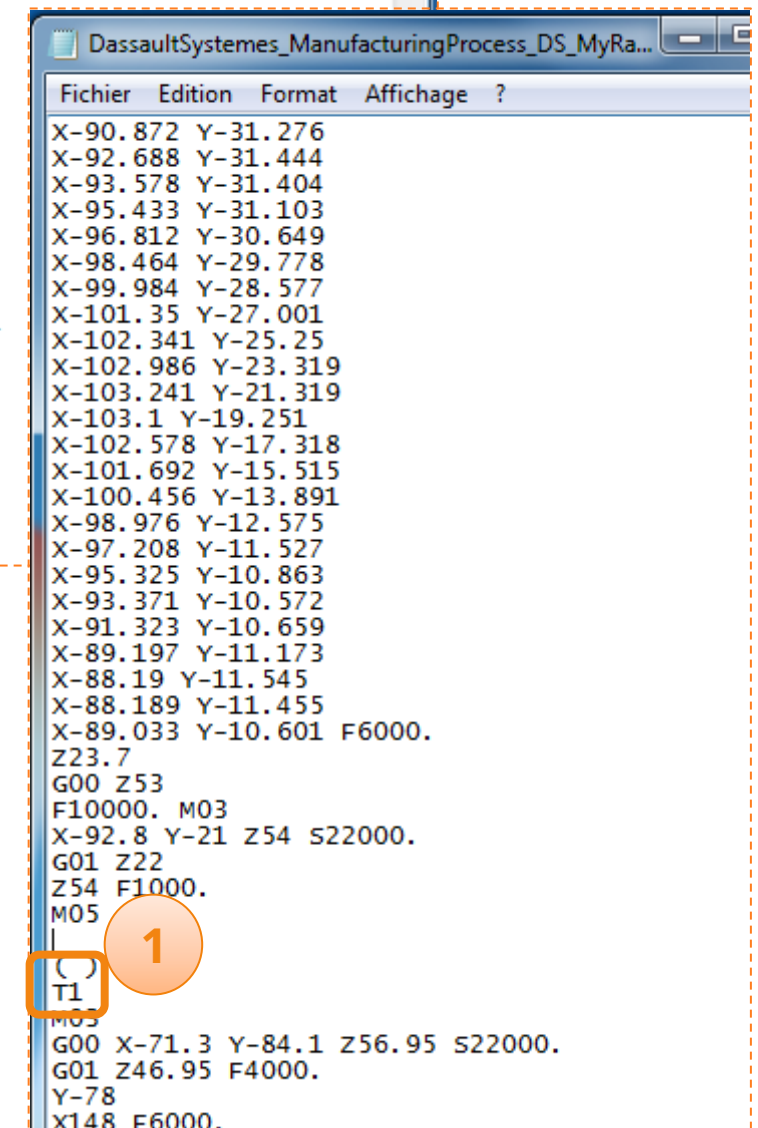
The ISO code is generated. To ensure that it is complete, open it using a text editor.

1. Check that the instructions **T1** and **T2** are well edited.  
These instructions correspond to tool changes at the beginning and in the middle of machining. Thanks to these instructions, the machining will be interrupted so that the symmetry of the car can be achieved.
2. If these instructions are present, the code should be generated correctly. All that remains is to transfer it to a digital machine and then to machine.



```
DassaultSystemes_ManufacturingProcess_DS_MyRa...
Fichier Edition Format Affichage ?
%1000
(CENIT_FRAISAGE == CN300 VERSION 1.0 )
(PP : strato1500-3x_D )
( DATE : Tue Apr 03 09:32:51 2012 )|
( PROGRAMMEUR : rouzi0011 )

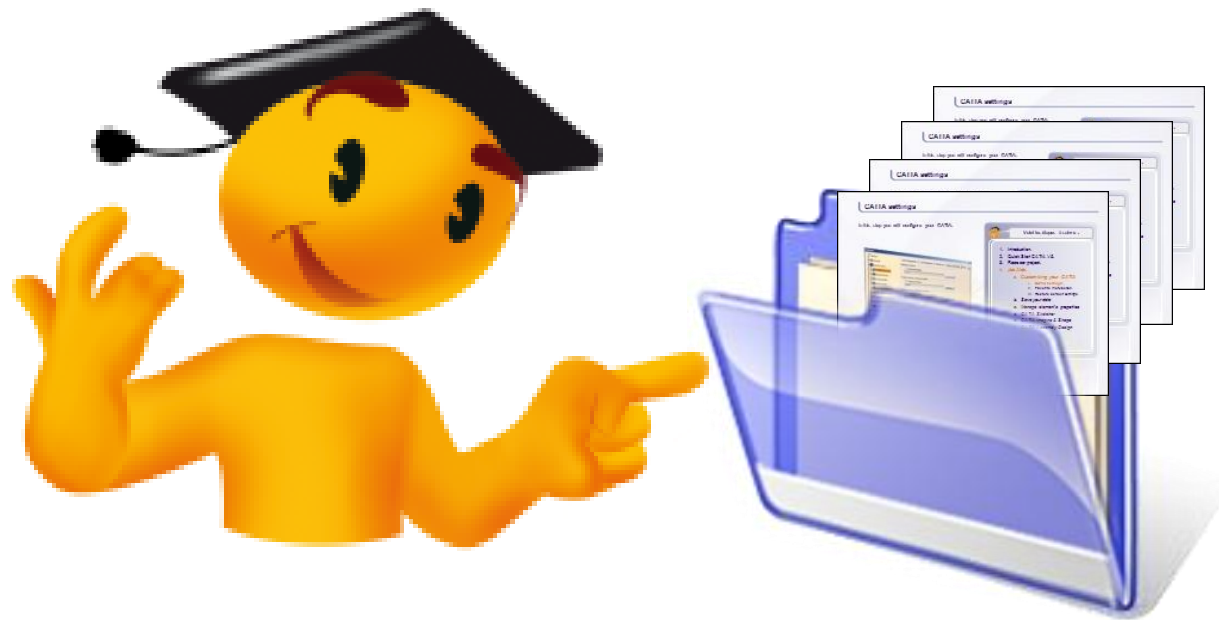
G94
(T2 1 2 tailles D6 )
T2
M03
G00 X71.3 Y-84.1 Z56.95 S22000.
G01 Z46.95 F4000.
Y-78
X-148 F6000.
X-149 F4800.
G02 X-150 Y-77 R1
G01 Y-76
Y-2 F6000.
Y-1 F4800.
G02 X-149 Y0 R1
G01 X-148
X-144.979 F6000.
X-143.979 F4800.
G02 X-142.979 Y-1 R1
G02 X-143.028 Y-1.312 R1
G01 X-143.303 Y-2.273
X-143.783 Y-3.95 F6000.
X-143.82 Y-4.164
X-144.005 Y-7.224
X-143.999 Y-7.408
X-143.933 Y-7.934
X-143.892 Y-8.119
X-143.106 Y-10.531
X-143.03 Y-10.707
```



```
DassaultSystemes_ManufacturingProcess_DS_MyRa...
Fichier Edition Format Affichage ?
X-90.872 Y-31.276
X-92.688 Y-31.444
X-93.578 Y-31.404
X-95.433 Y-31.103
X-96.812 Y-30.649
X-98.464 Y-29.778
X-99.984 Y-28.577
X-101.35 Y-27.001
X-102.341 Y-25.25
X-102.986 Y-23.319
X-103.241 Y-21.319
X-103.1 Y-19.251
X-102.578 Y-17.318
X-101.692 Y-15.515
X-100.456 Y-13.891
X-98.976 Y-12.575
X-97.208 Y-11.527
X-95.325 Y-10.863
X-93.371 Y-10.572
X-91.323 Y-10.659
X-89.197 Y-11.173
X-88.19 Y-11.545
X-88.189 Y-11.455
X-89.033 Y-10.601 F6000.
Z23.7
G00 Z53
F10000. M03
X-92.8 Y-21 Z54 S22000.
G01 Z22
Z54 F1000.
M05
(T1 1 )
T1
M03
G00 X-71.3 Y-84.1 Z56.95 S22000.
G01 Z46.95 F4000.
Y-78
X148 F6000.
```

# Method sheets

These files will allow you to improve your knowledge of some key functions of CATIA V5.



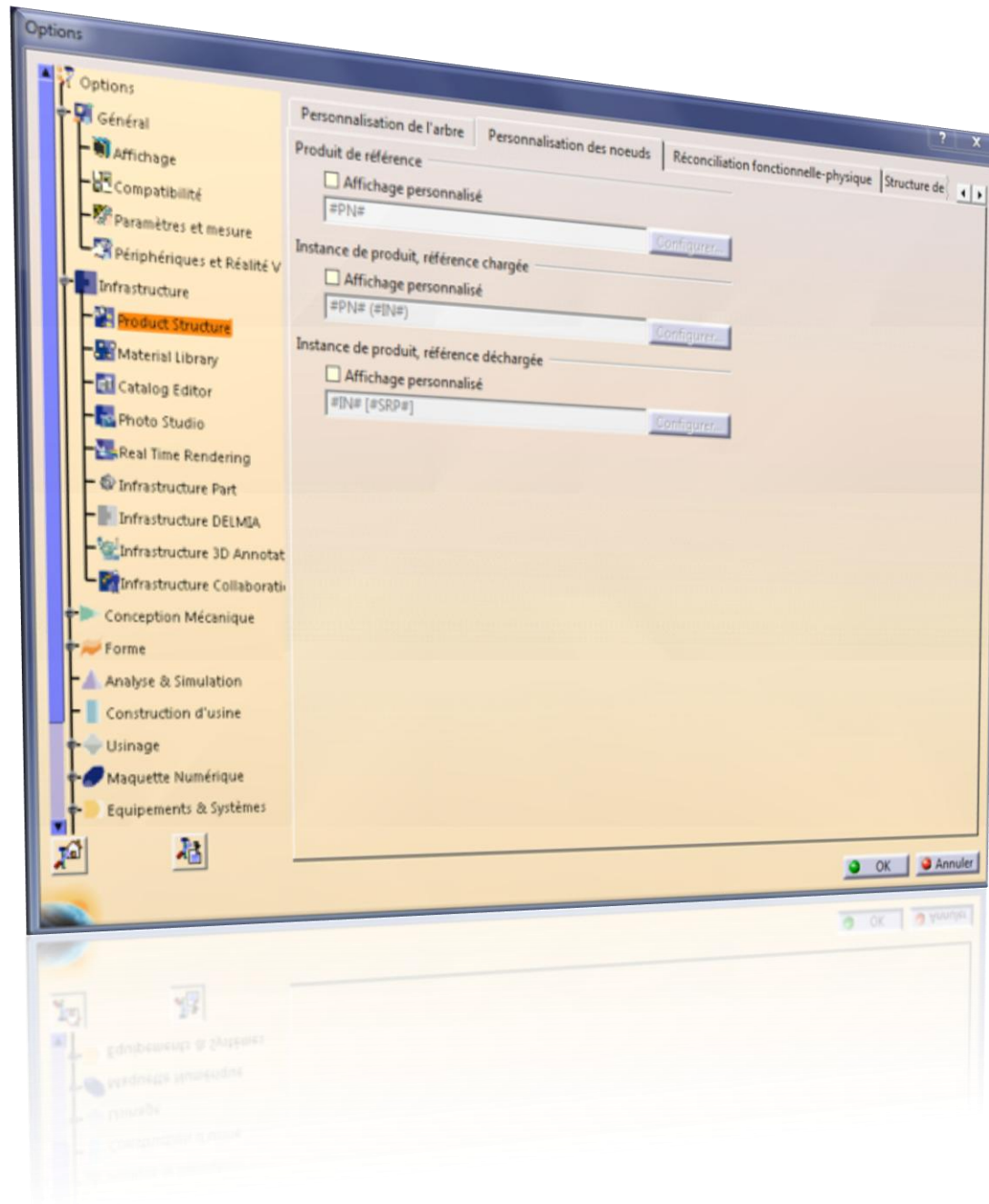
Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. Your racing car.
- 4. Method sheets.**
  - has. Configure CATIA.
  - b. Manage your data.
  - vs. Modify element properties.
  - d. Specification tree.
  - e. CATIA Sketcher.
  - f. CATIA Imagine & Shape.
  - g. CATIA Assembly Design.



# Configure CATIA

In this step you will configure CATIA for the purposes of Running.



Here are the steps to follow:

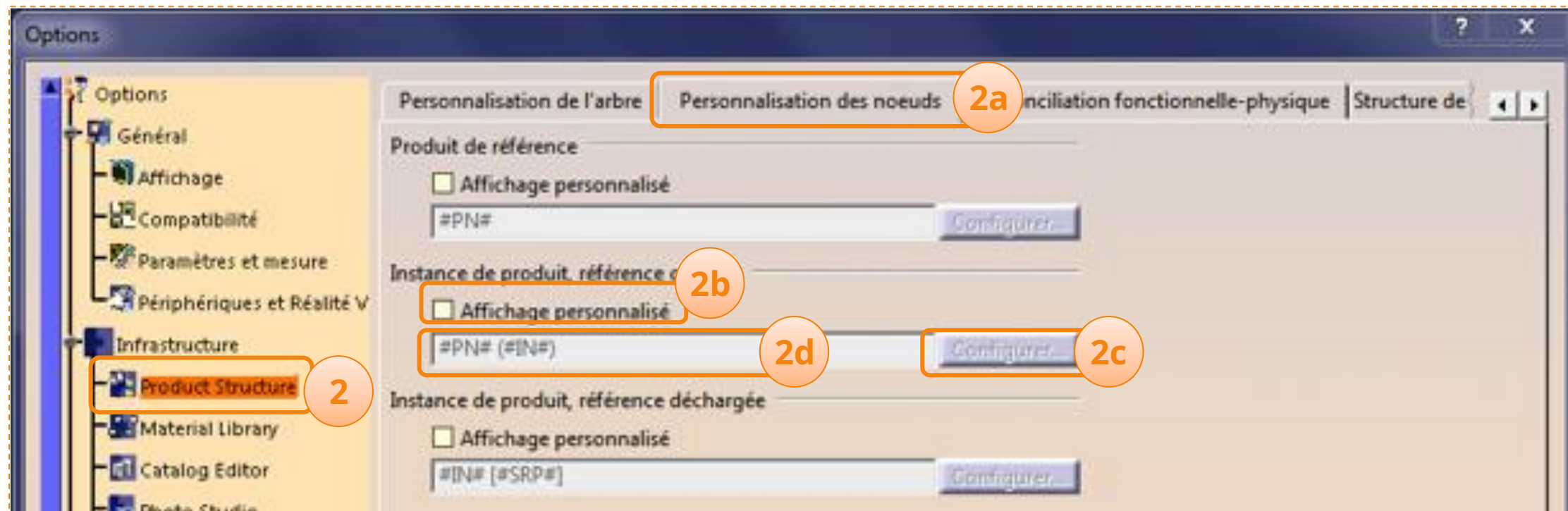
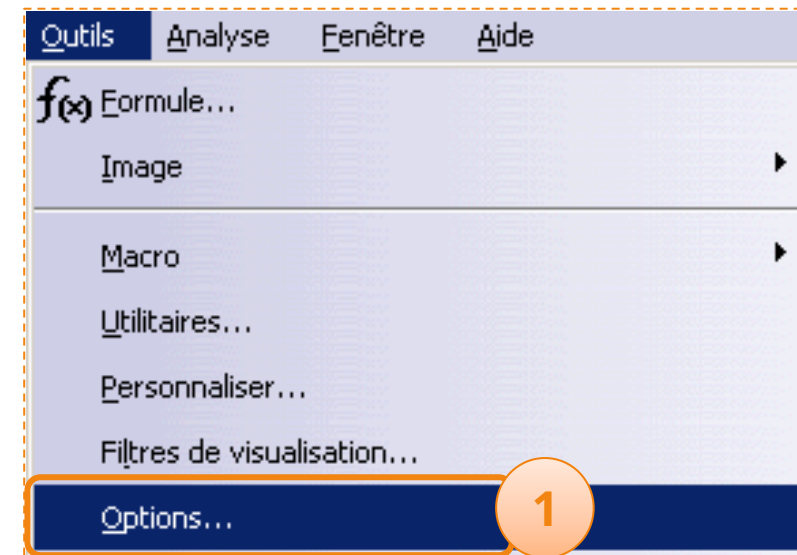
1. Introduction.
2. CATIA V5 QuickStart.
3. Your racing car.
4. *Method sheets.*
  - has. *Configure CATIA.*
    - i. *CATIA V5 settings.*
      - ii. Favorite workshops.
      - iii. Restore the display.
    - b. Manage your data.
    - vs. Modify element properties.
  - d. Specification tree.
  - e. CATIA Sketcher.
  - f. CATIA Imagine & Shape.
  - g. CATIA Assembly Design.



# CATIA V5 Settings

You are about to change the software settings.

1. Click on **Tools > Options**.
2. Then, in **Infrastructure, Product Structure**:
  - has. Select tab **Customizing nodes**.
  - b. Tick **Custom display** for product instance, reference loaded as below.
- vs. Click on **Configure**.
- d. Enter [#PN#].

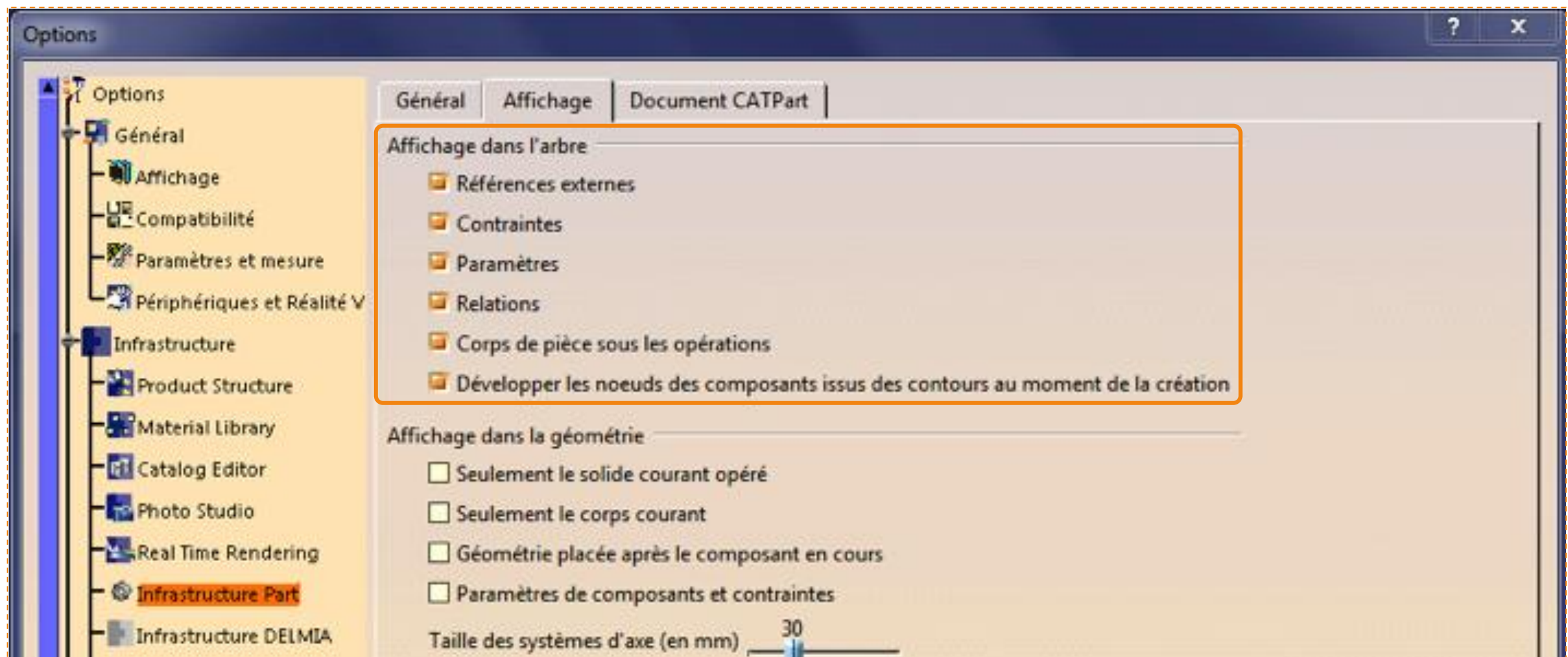


# CATIA Settings

Go into **Infrastructure > Infrastructure Part > tab Display:**

1. Check the following options:

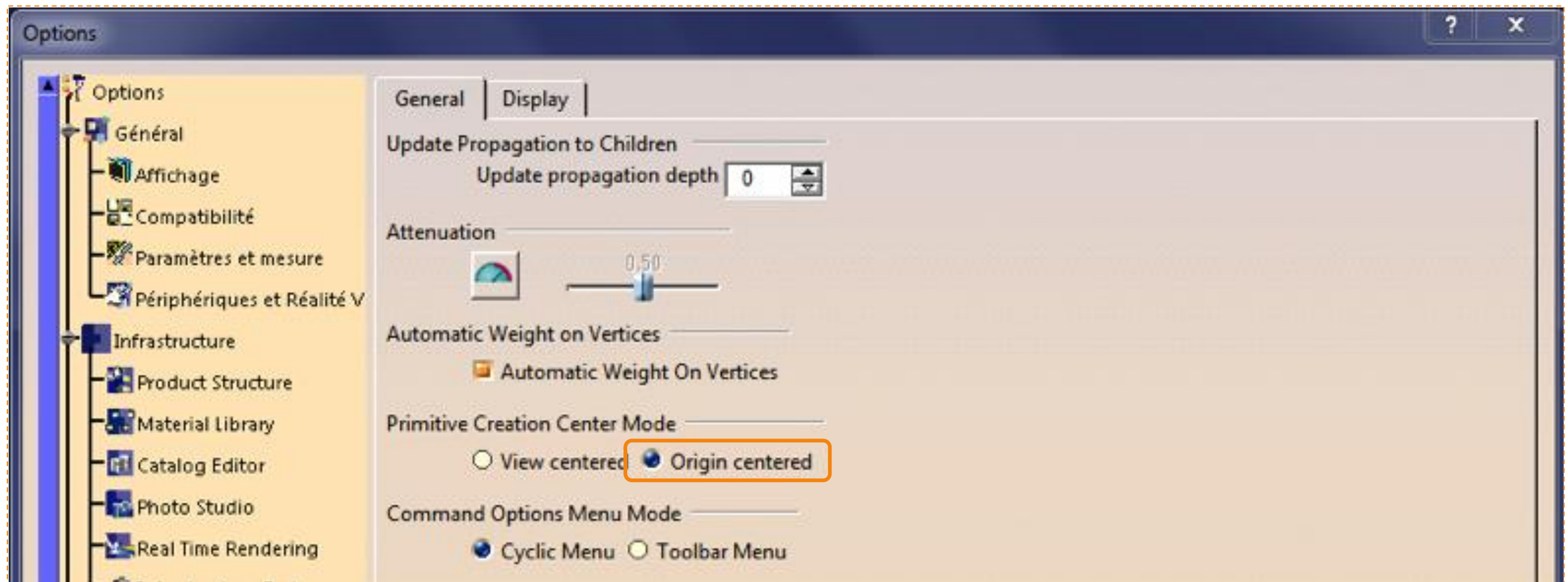
- has. External references.
- b. Constraints.
- vs. Settings.
- d. Relationships.
- e. Part body under operations.
- f. Expand component nodes from outlines at design time.



# CATIA Settings

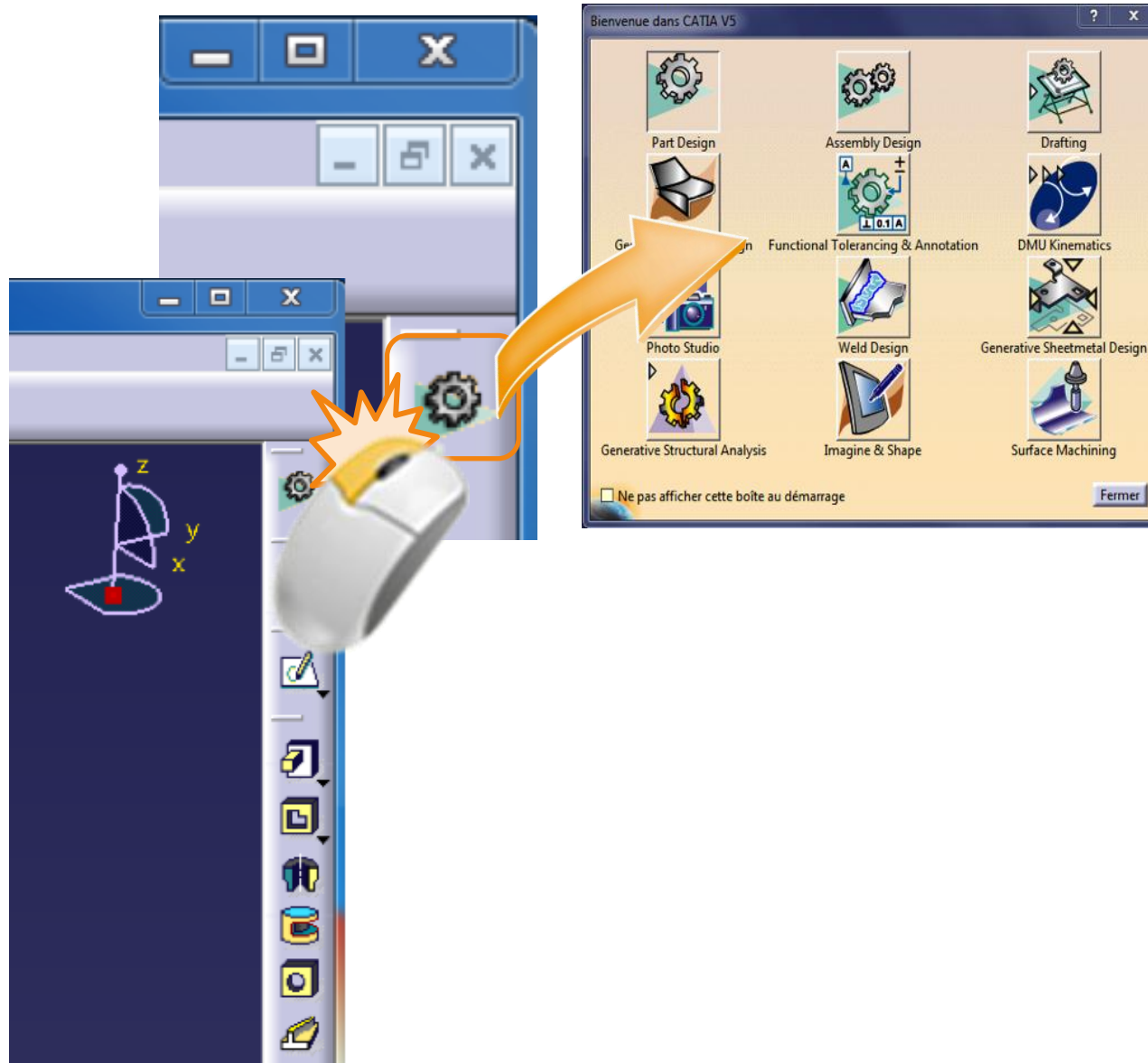
Now in **Shape > Imagine & Shape > tab General**:

1. Check the option **Origin-centered** to **Primitive Creation Center Fashion**.



## Favorite Workshops

CATIA's interface is customizable. You can configure shortcuts to access your favorite workshops by clicking on the active workshop icon.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. Your racing car.

### **4. Method Sheets.**

**has. Configure CATIA.**

i. CATIA V5 settings.

ii. **Favorite Workshops.**

iii. Restore the Display.

b. Manage your Data.

vs. Modify Element Properties.

d. Specification Tree.

e. CATIA Sketcher.

f. CATIA Imagine & Shape.

g. CATIA Assembly Design.

h. CATIA Surface Machining.



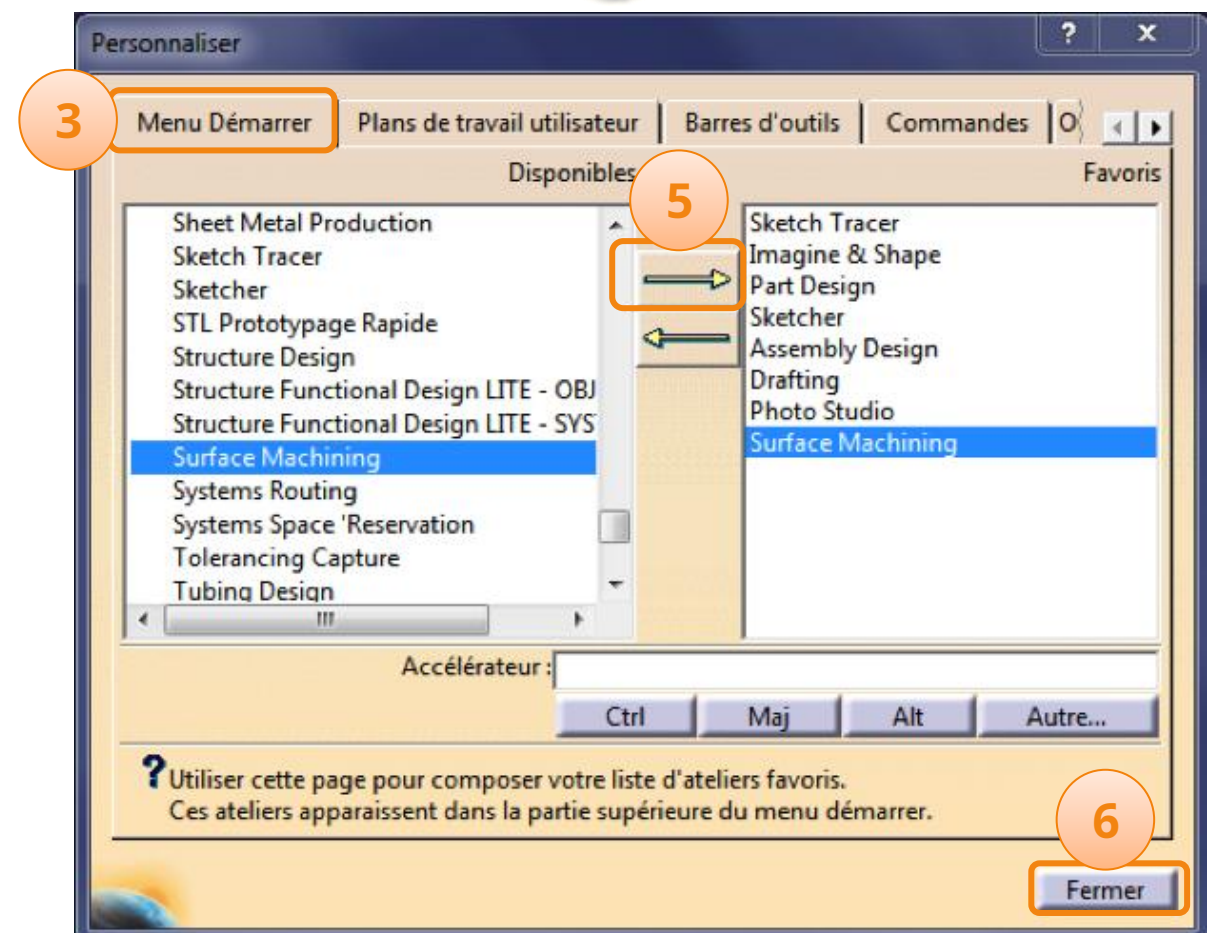
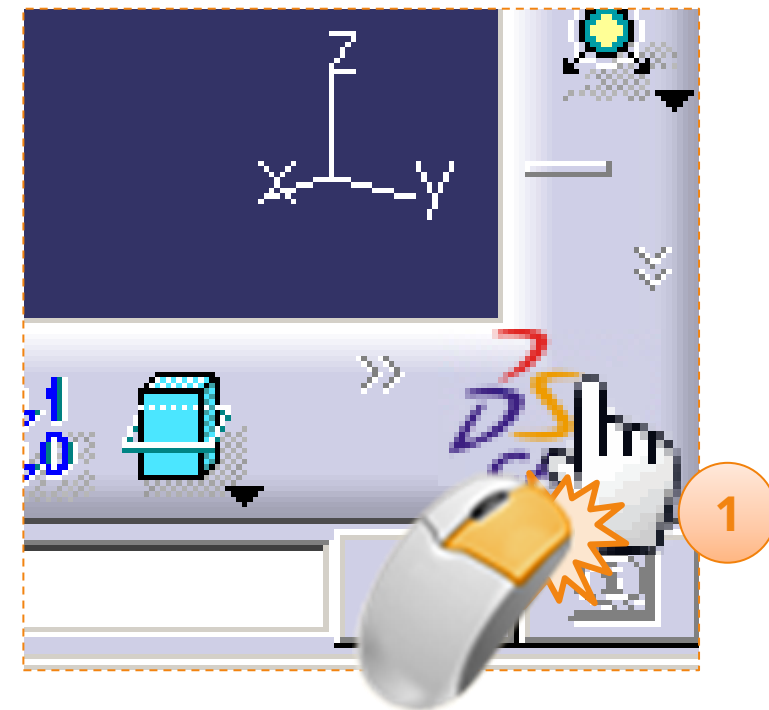
## Favorite Workshops

We are going to define as favorites the 8 workshops that we will need for the realization of the project. For that :

1. Right click on the logo **CATIA 3DS** located at the bottom right of the screen.
2. Select **Personalize**.
3. Click on the tab **Start menu**.
4. Select the workshops from the list on the left:
  - has. Sketch Tracer
  - b. Imagine & Shape
  - vs. Part Design
  - d. Sketcher
  - e. Assembly Design
  - f. Drafting
  - g. Photo Studio
  - h. Surface Machining

5. Click on **the arrow** to put them in the favorites list.

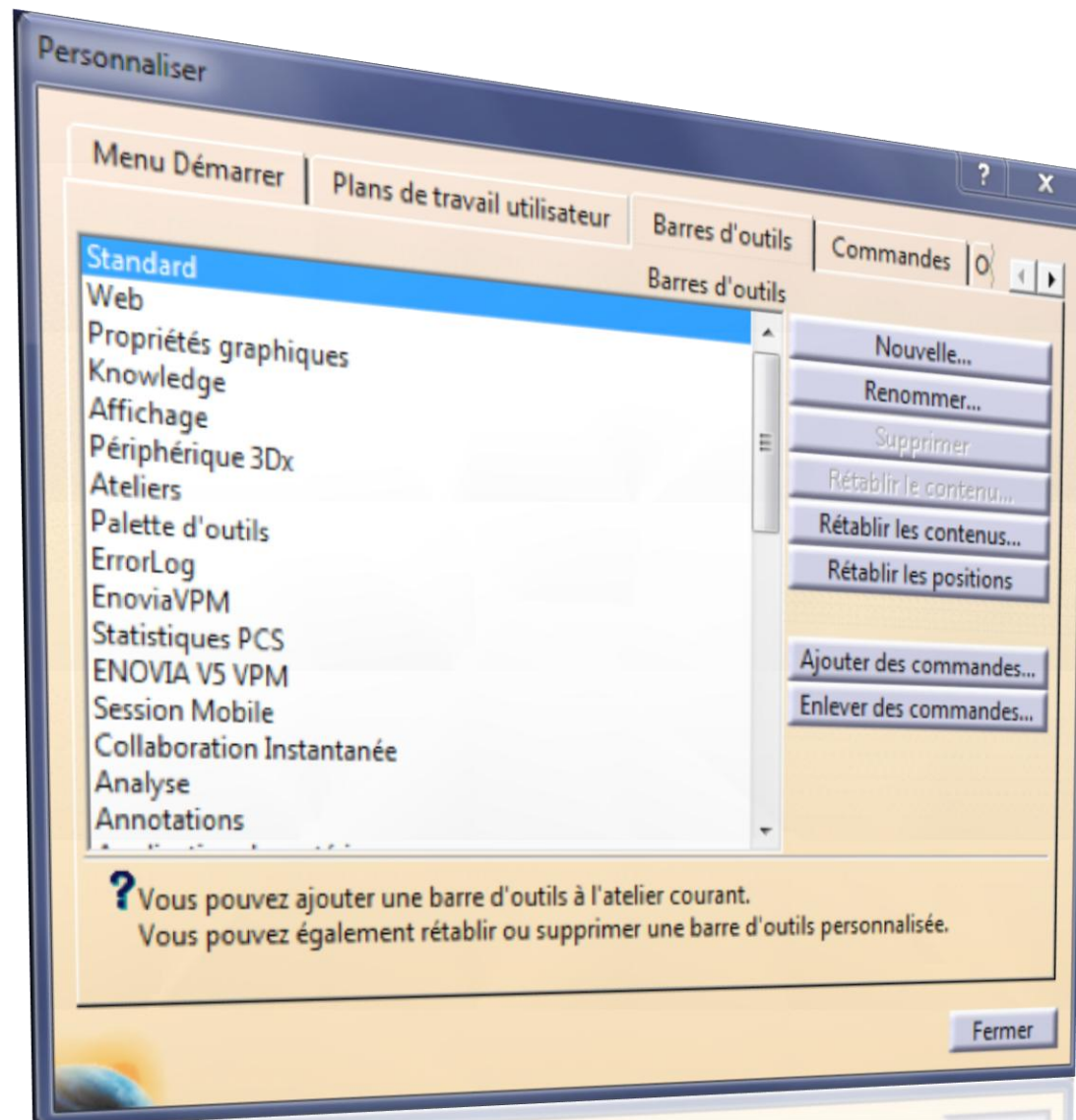
6. Click on **To close**.





# Restore display

In this step you will learn how to reset the position of toolbars.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. Your racing car.

#### **4. Method Sheets.**

**has. Configure CATIA.**

- i. CATIA settings.
- ii. Favorite workshops.

#### **iii. Restore the display.**

- b. Manage your data.
- vs. Manage item properties.
- d. Specification tree.
- e. CATIA Sketcher.
- f. CATIA Imagine & Shape.
- g. CATIA Assembly Design.
- h. CATIA Surface Machining.

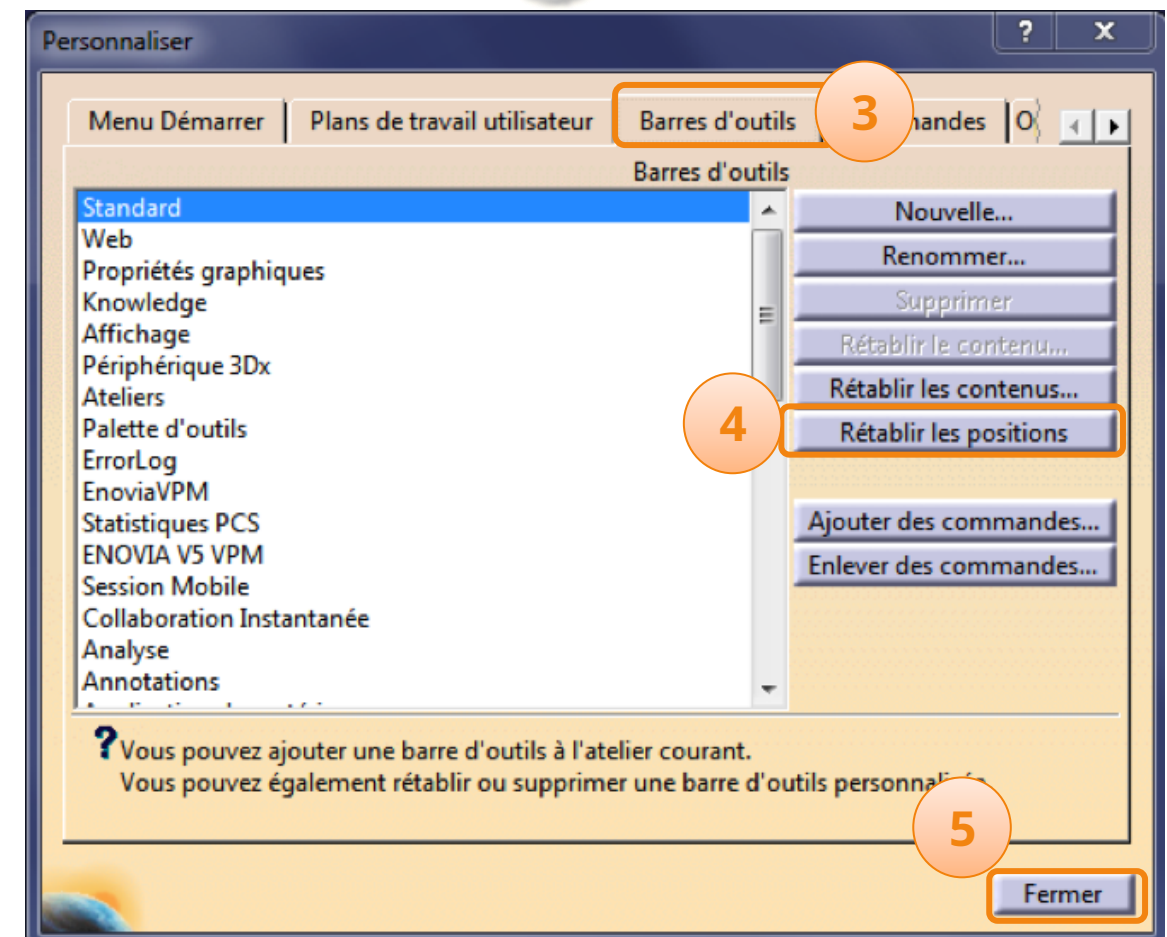
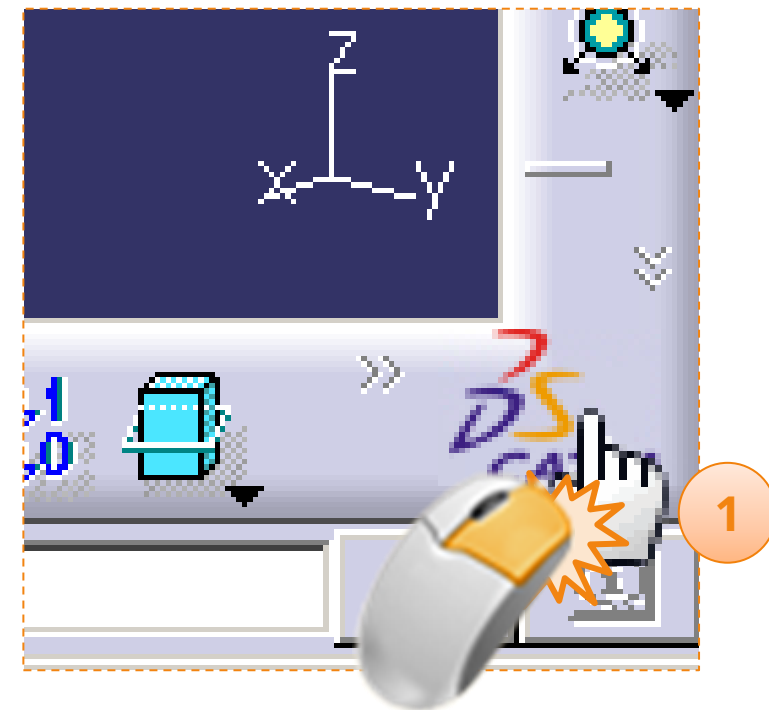
# Restore display

It is possible that some toolbars disappear or are moved following multiple manipulations.

You can reset toolbar positions as follows:

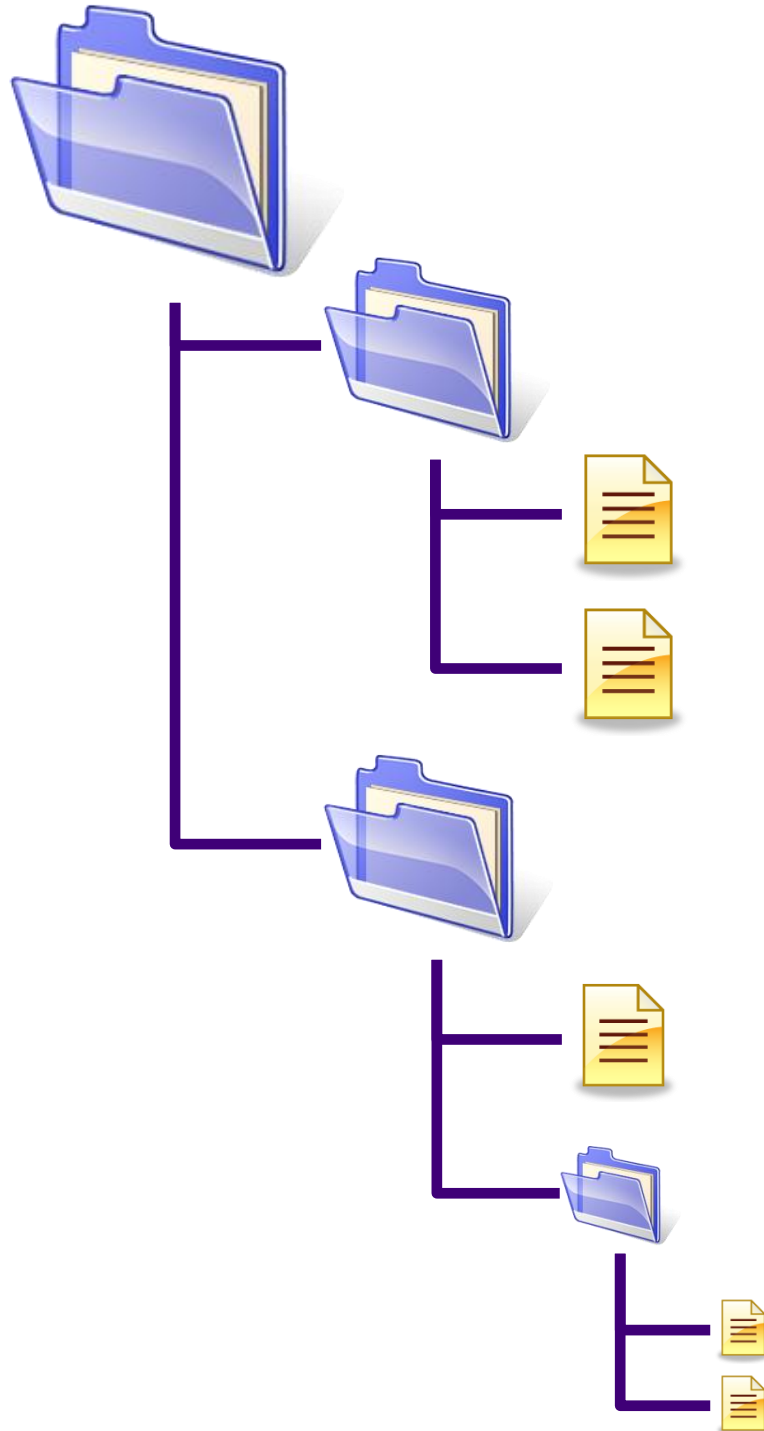
1. Right click on the logo **CATIA 3DS** located at the bottom right of the screen.
2. Select **Personalize**.
3. Click on the tab **Toolbar**.
4. Then click **Restore positions**.
5. Click on **To close**.

Note that this operation only affects toolbars in the active workbench.



# Manage your data

In this step you will learn how to manage the structure of CATIA V5 documents.

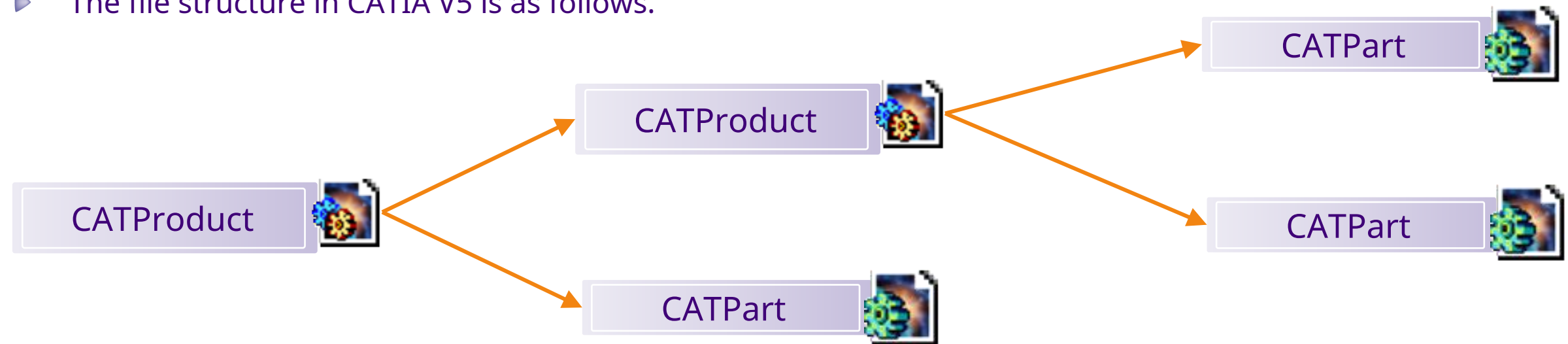


Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. Your racing car.
- 4. Method sheets.**
  - has. Configure CATIA.
  - b. Manage your data.**
    - i. File structure.
    - ii. Back up your data.
  - vs. Manage item properties.
  - d. Specification tree.
  - e. CATIA Sketcher.
  - f. CATIA Imagine & Shape.
  - g. CATIA Assembly Design.
  - h. CATIA Surface Machining.

# File structure

- The file structure in CATIA V5 is as follows.



## CATPart file:

- Similar to a room.
- Contains the 3D geometric representation of an element.

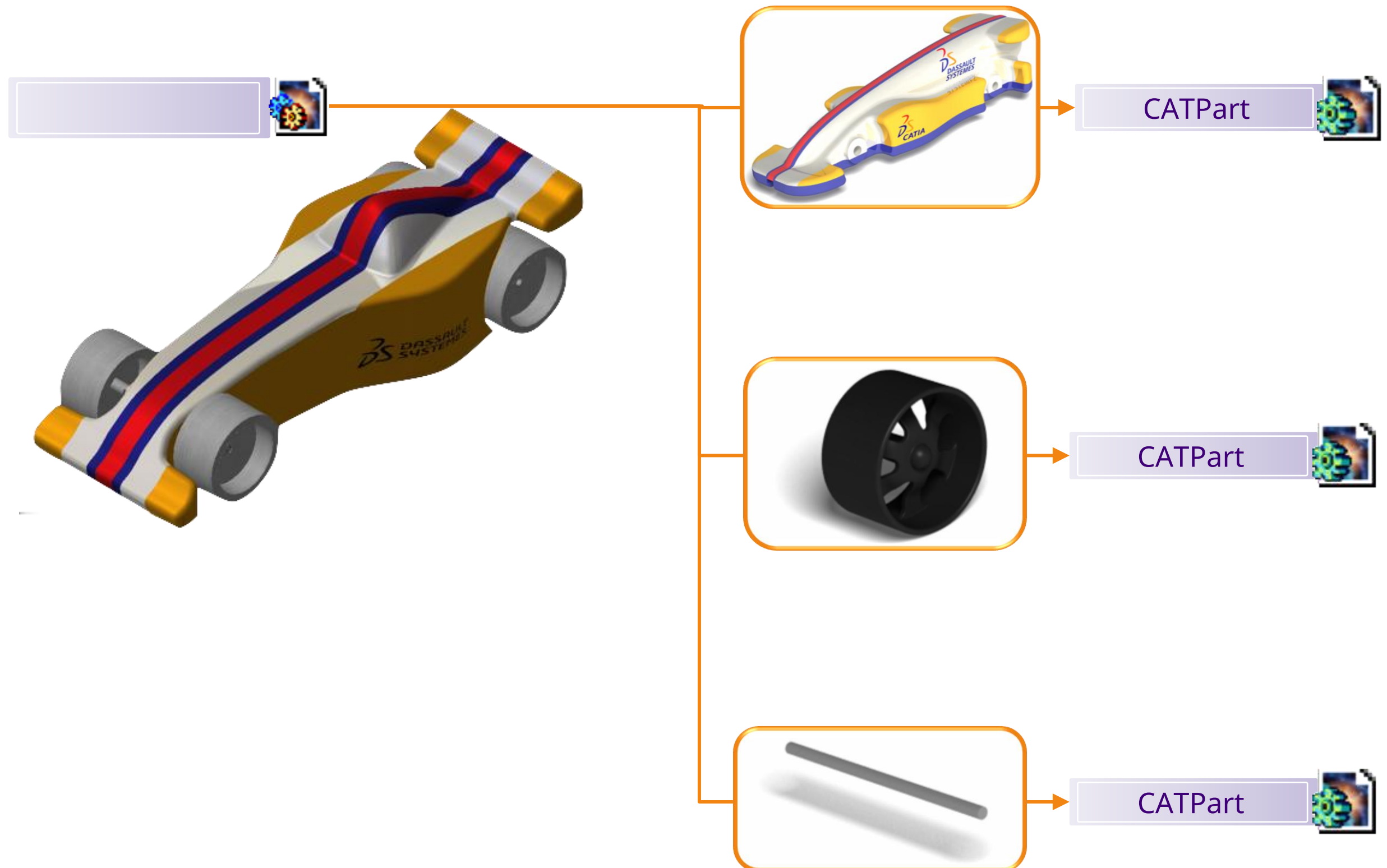


## CATProduct file:

- Similar to an assembly.
- Contains no 3D geometry.
- Contains all the information needed for an assembly:
  - Links to the elements that make up the assembly (parts or sub-assemblies).
  - Positions of elements in 3D space.

# File structure

- Here is the file structure you will use in this project:



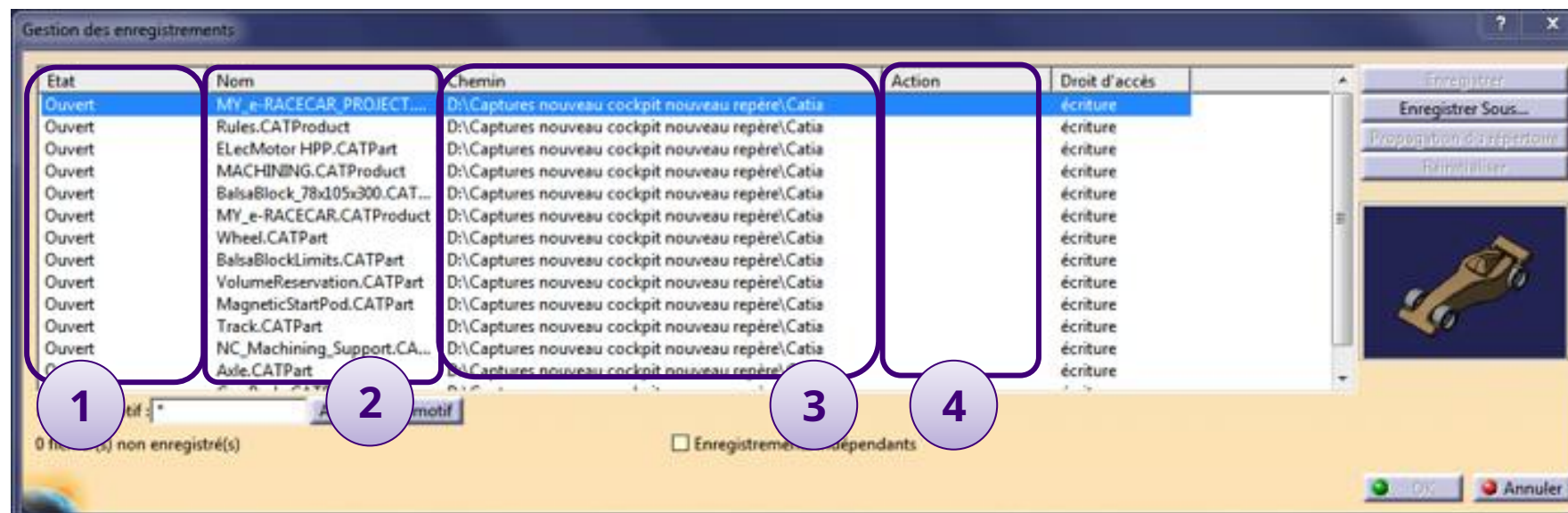


## Records management

To save your data, it is best to go through the registration manager.

To access it, click on **File > Manage Records...**

- 1 **File status:** Open, Modified, New...
- 2 The name and type of each open file.
- 3 The path of each file.
- 4 The action that the handler will perform.



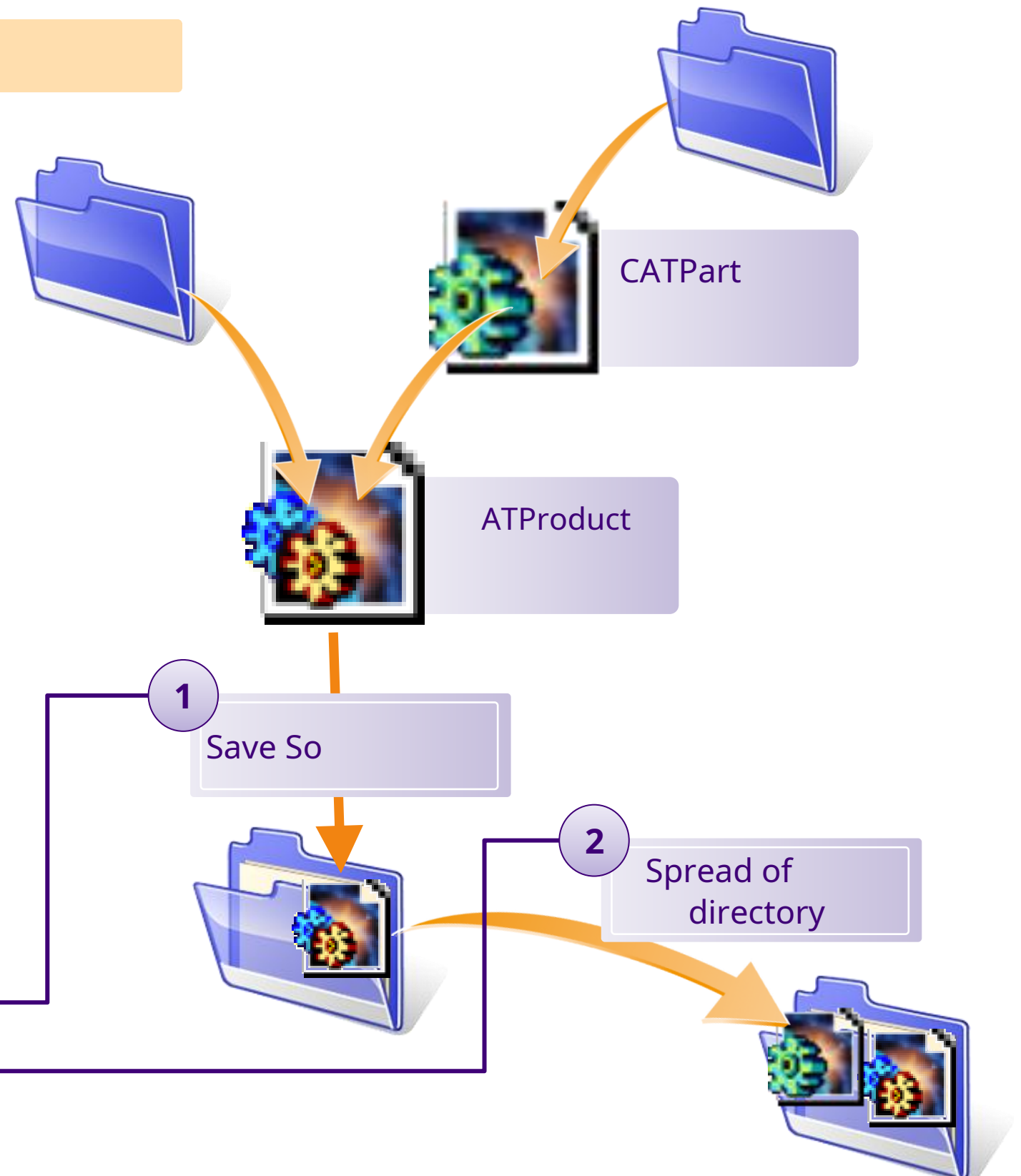
## Records management

When you save a product that is an assembly of different parts:

Save it in the desired folder using **Save as...** This will only save your product to this folder.

Then the function **Directory propagation** will allow you to save the parts (CATPart) associated with this product (CATProduct) in the same folder.

Click on **okay** to validate the recording.

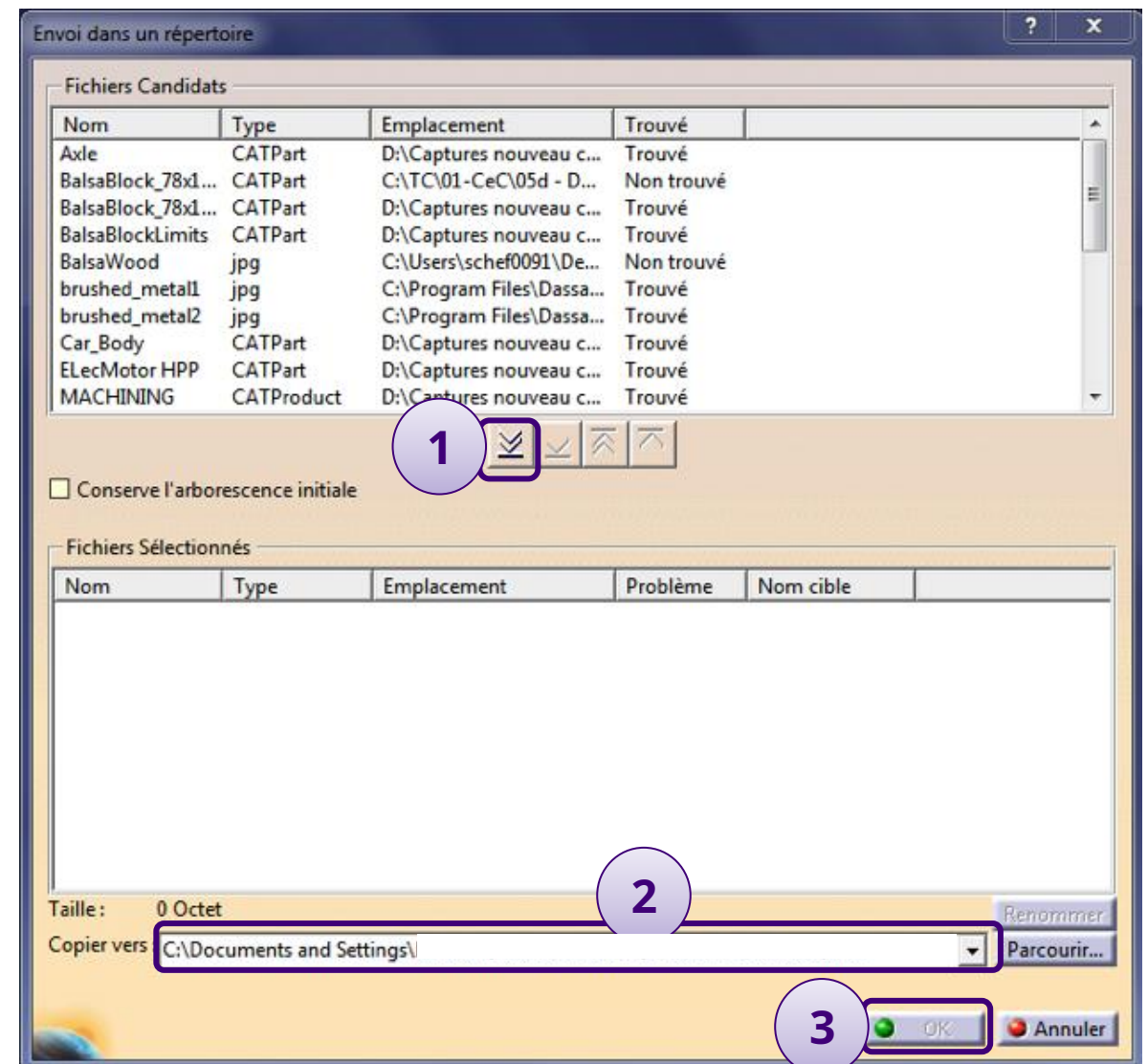


## Send to directory

You can also use the option **File > Send To > Directory** to send in a directory all the files that are used by your CATProduct. This includes basic files such as CATProducts and CATParts, **but also images serving as a painting, sticker, texture, etc.**

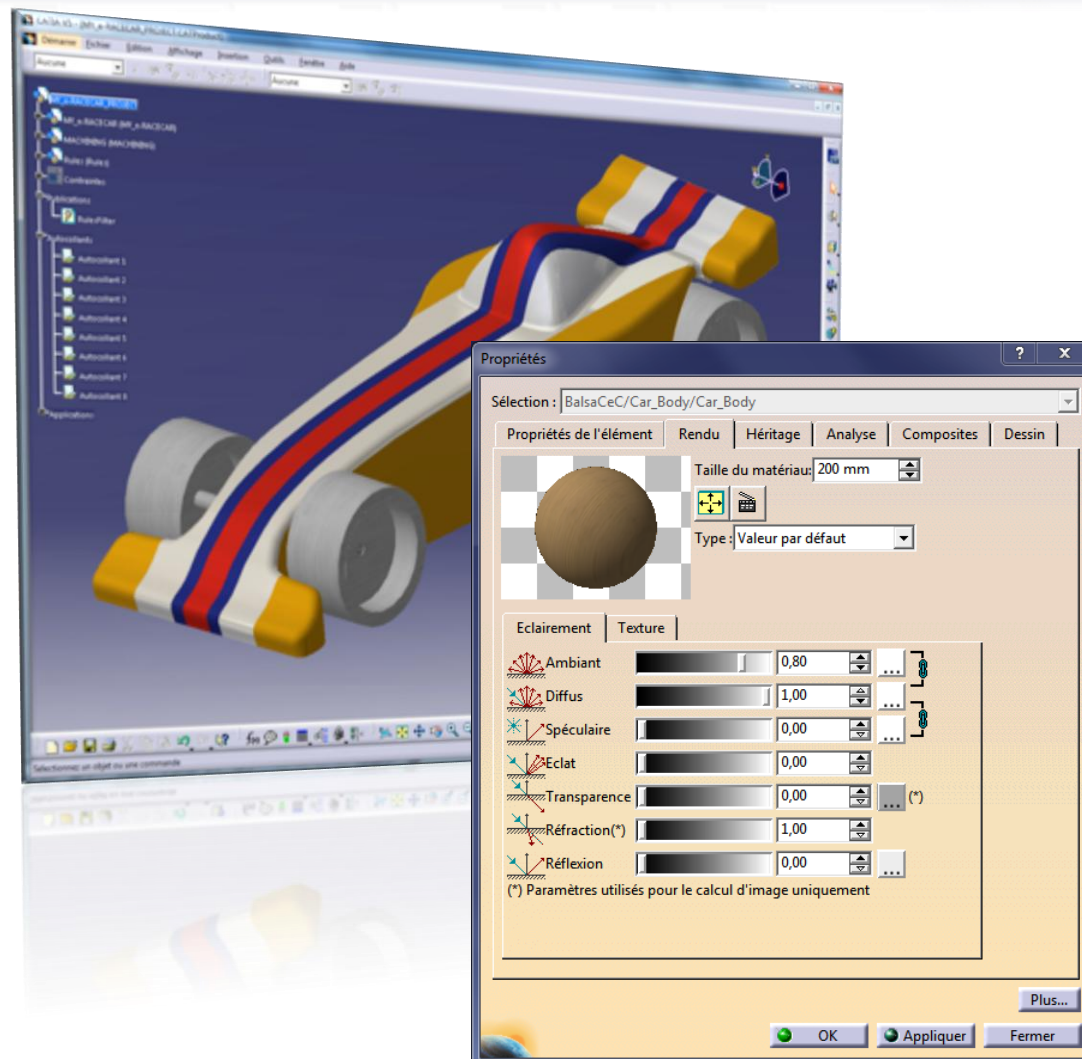
For that :

- ▶ Click the double arrow to send all files.
- ▶ Specify the destination folder for the files.
- ▶ Click on **okay**.



# Edit element properties

In this sheet we will see the means of accessing the properties of an element, and the different modifications that we can make to them.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. Your racing car.

#### **4. Method sheets.**

has. Configure CATIA.  
b. Manage your data.  
**vs. Modify element properties.**

- d. Specification tree.
- e. CATIA Sketcher.
- f. CATIA Imagine & Shape.
- g. CATIA Assembly Design.
- h. CATIA Surface Machining.

# Edit element properties

To access the properties of an element, you must identify it either in the construction tree or in 3D space.  
Below are some elements of different types:

► Products

► Rooms

► Materials

► Various operations

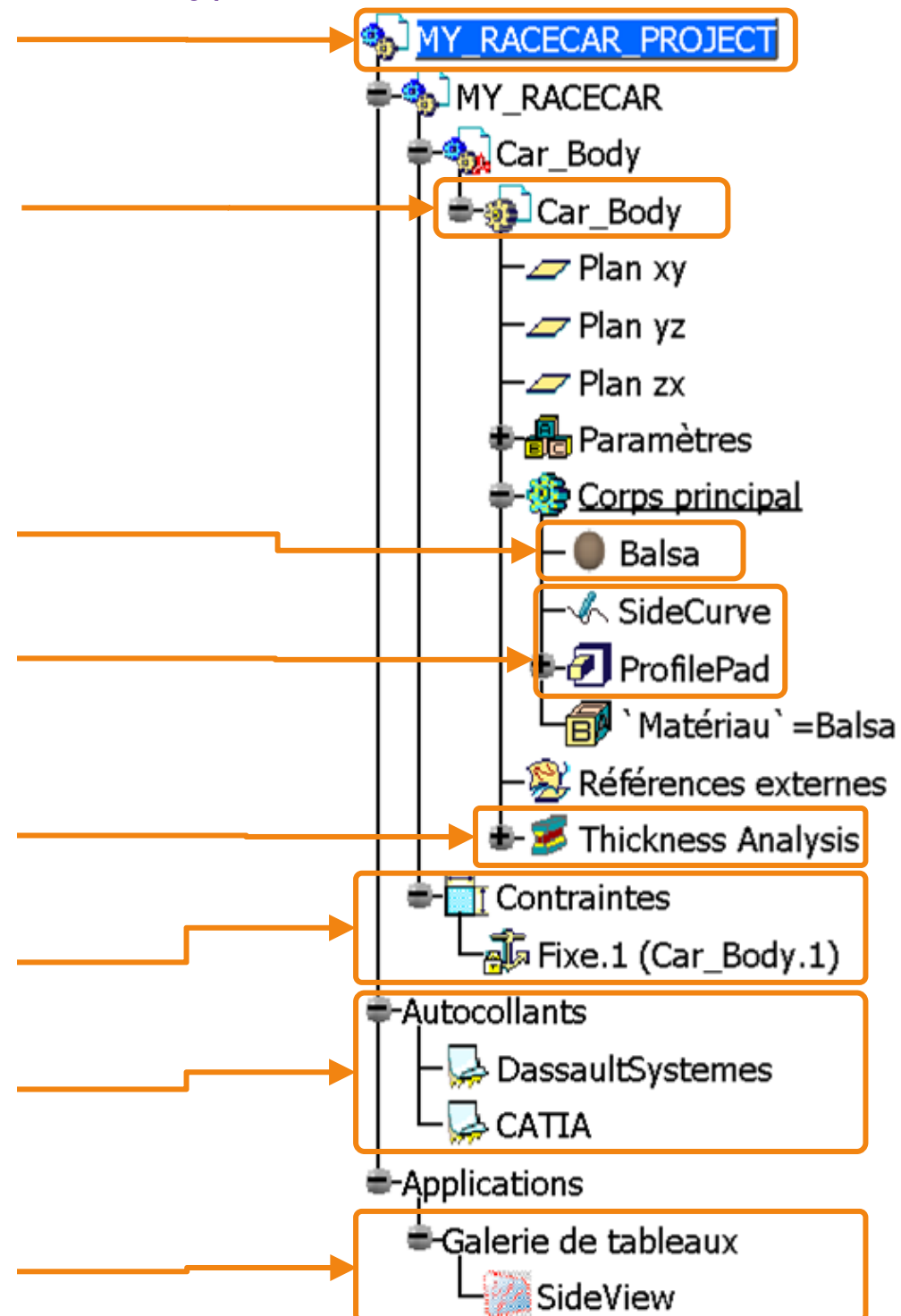
► Analyzes and results

► Constraints

► Stickers

► Paintings

► Etc...

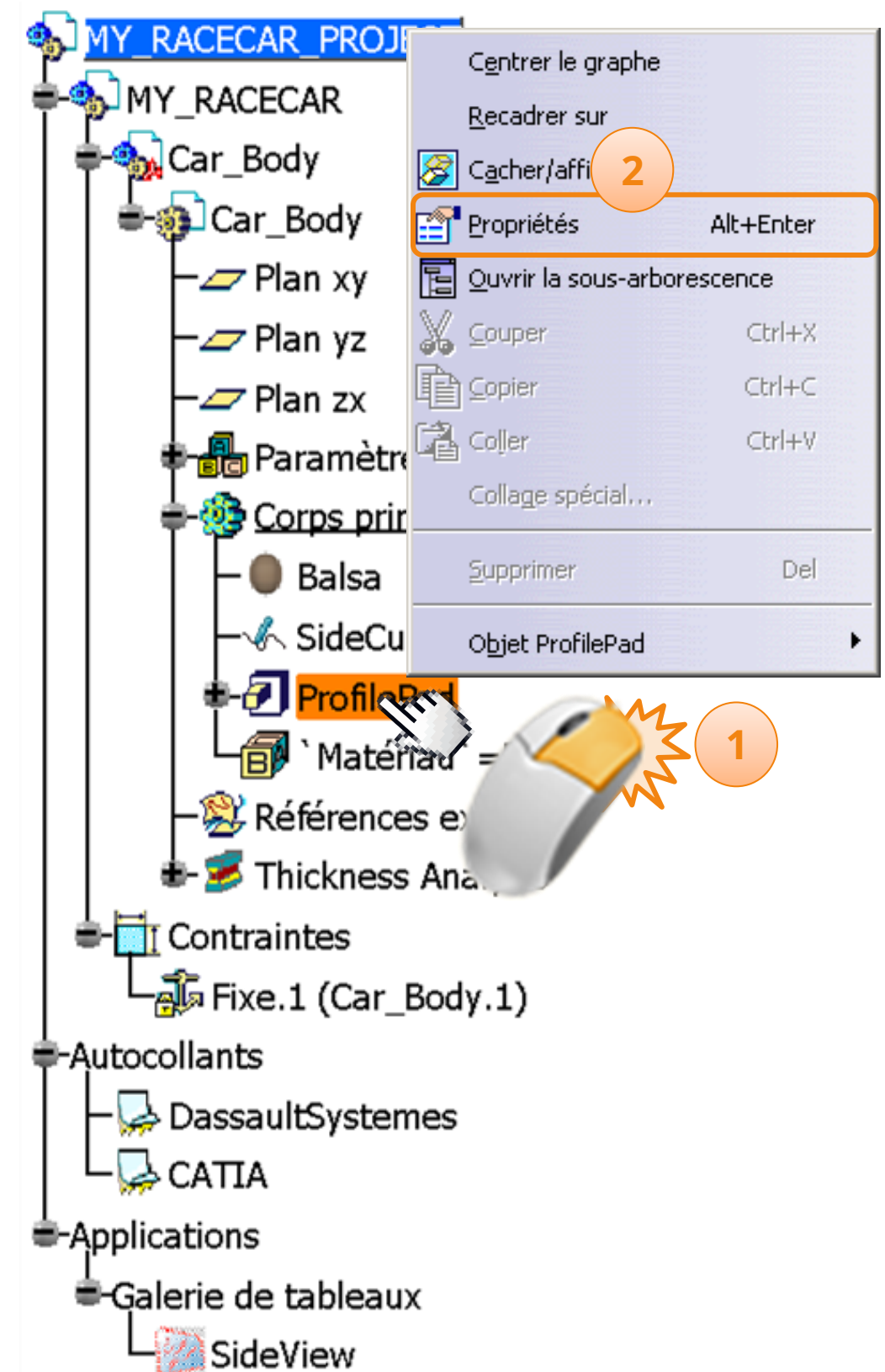
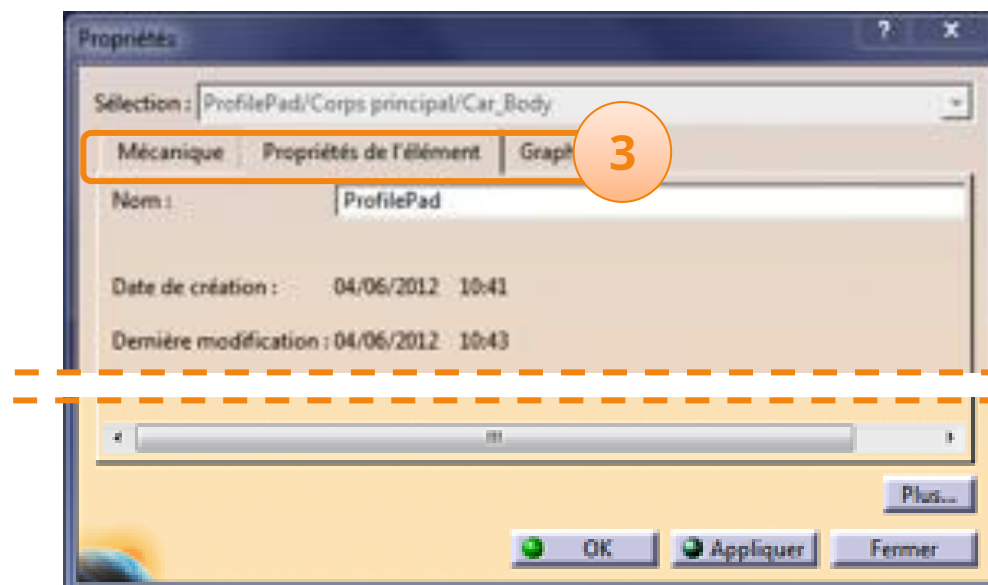




# Edit element properties

Once you have located the element, you can access its properties:

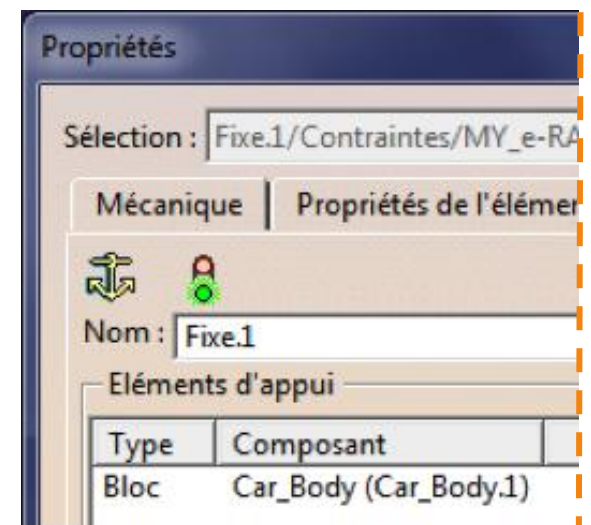
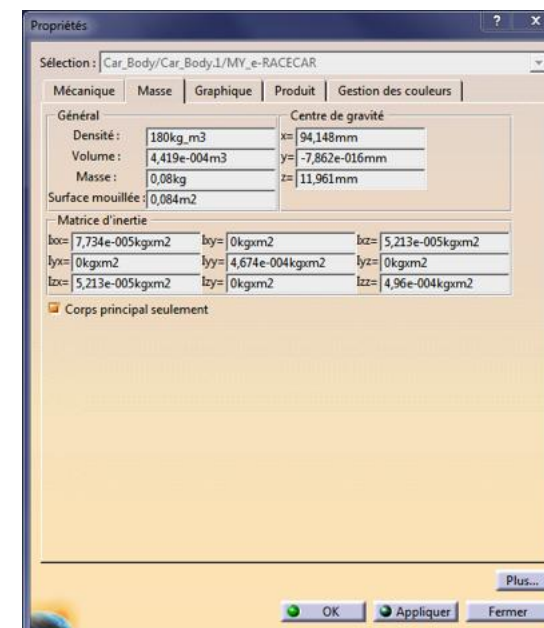
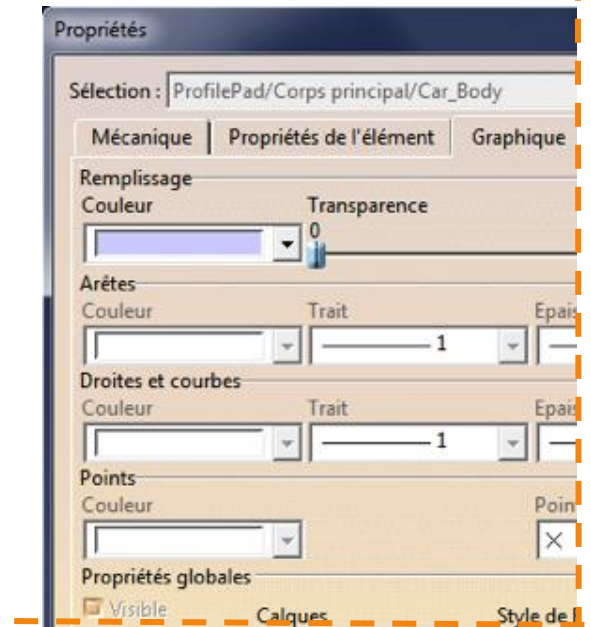
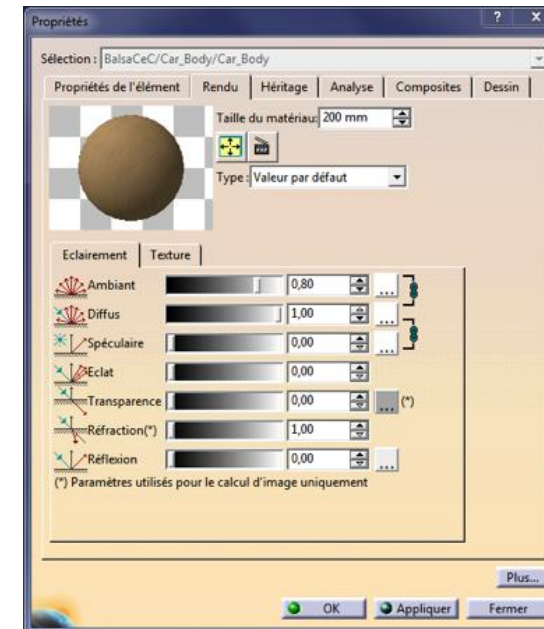
- 1.Right-click on it in the tree.
- 2.Select **Properties**.
- 3.This will open a window with tabs depending on the item you clicked on.



# Edit element properties

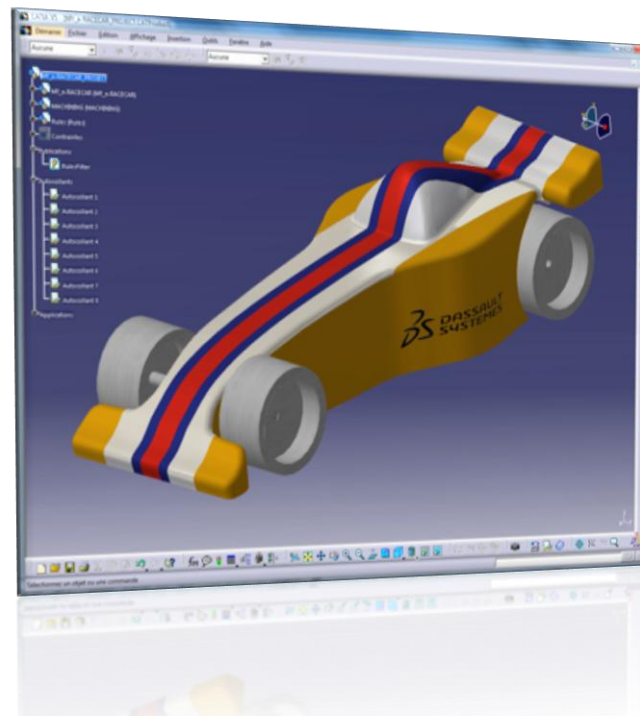
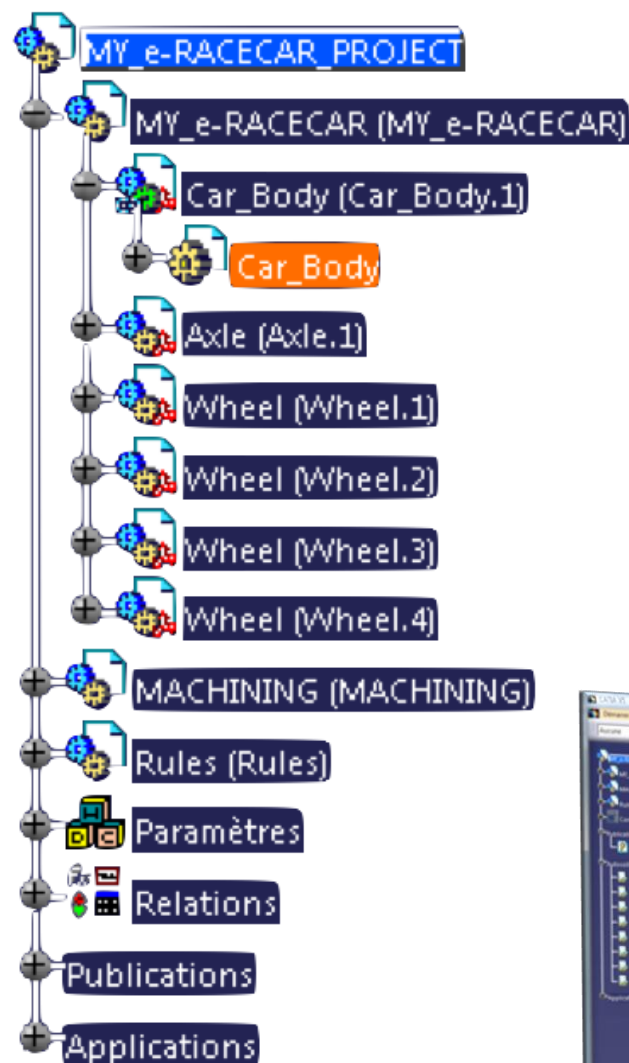
In this way you can access, depending on the element, the following properties:

- ▶ The name.
- ▶ Appearance (colors, transparency, etc...).
- ▶ Attributes (enabled/disabled, updated, etc.).
- ▶ Physical properties (Weight, Volume, etc.).



# Specification tree

The specification tree helps to organize its design. In this part you will learn how to manipulate the different states of the specification tree. This is useful for a good understanding of the design.



Here are the steps to follow:

1. Introduction.
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3. Your racing car.
- 4. Method sheets.**
  - has. Configure CATIA.
  - b. Manage your data.
  - vs. Manage item properties.
  - d. Specification tree.**
  - e. CATIA Sketcher.
  - f. CATIA Imagine & Shape.
  - g. CATIA Assembly Design.
  - h. CATIA Surface Machining.

# Specification tree

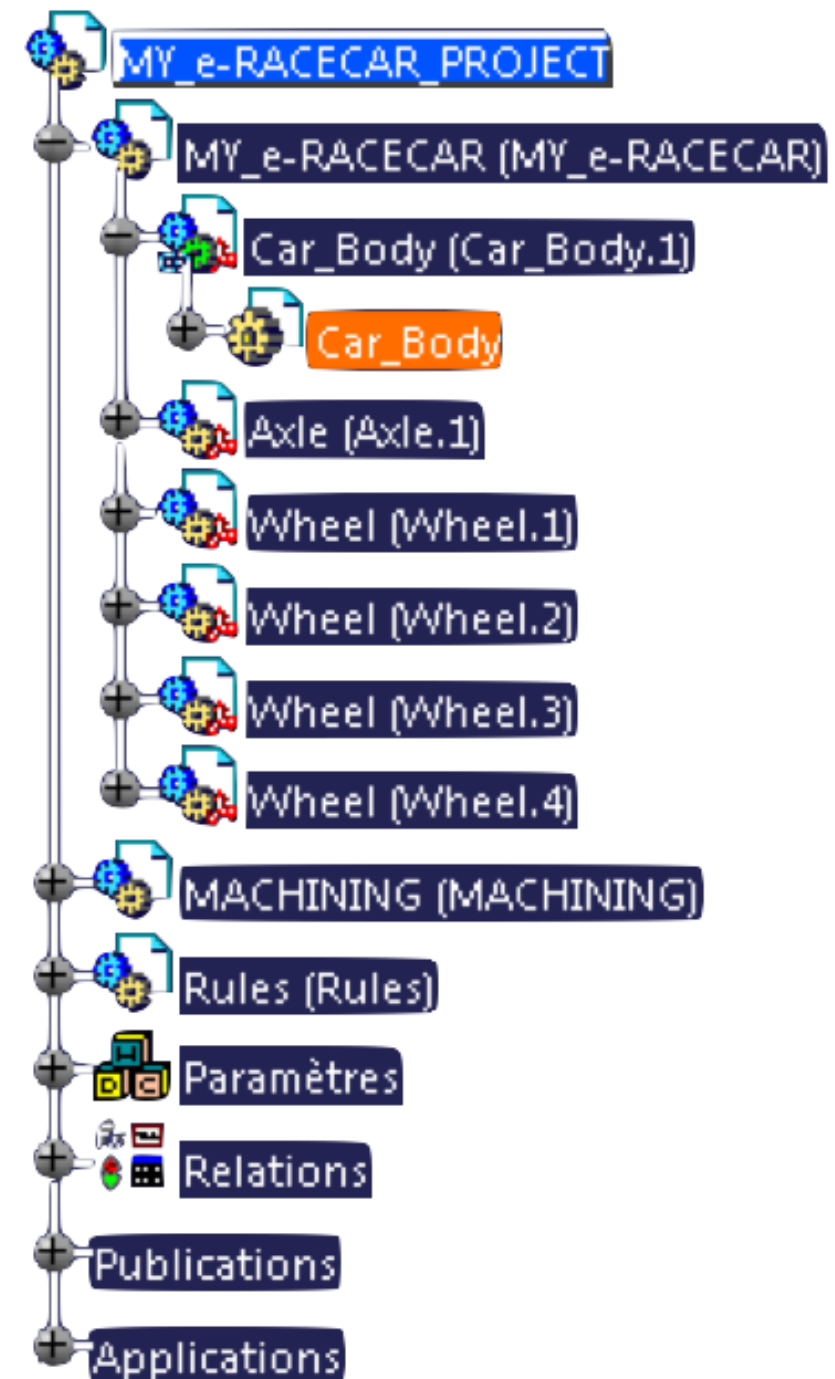
- In the specification tree you can find the following important information.

- **Active Component**

- A component is active when highlighted in blue.
- To activate a component, double-click it in the specification tree.

- **Selected Component**

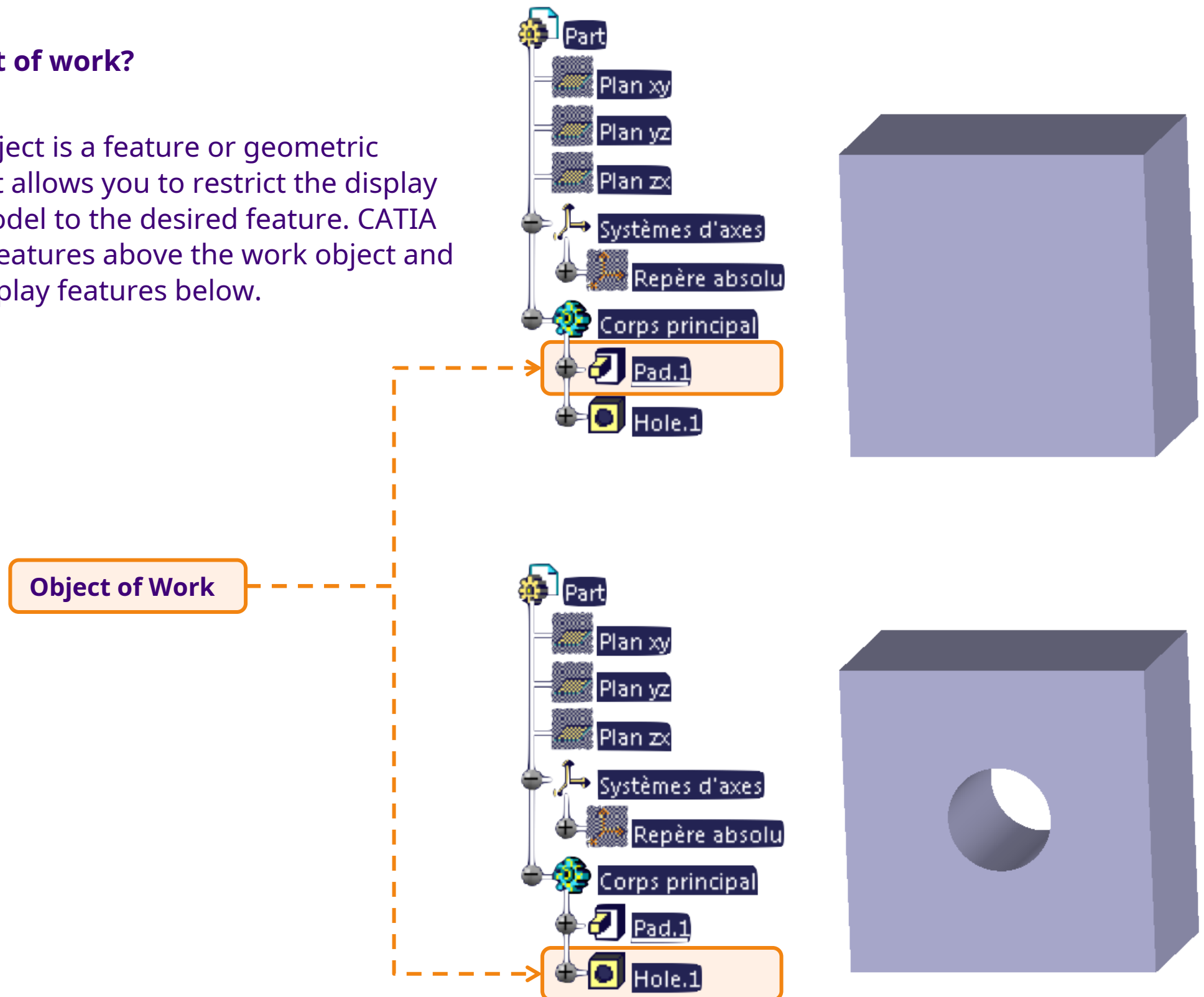
- A component is selected when highlighted in orange.
- To select a component, click on it in the specification tree.



# Specification tree

## ► What is the object of work?

- The work object is a feature or geometric element that allows you to restrict the display of the 3D model to the desired feature. CATIA displays all features above the work object and does not display features below.





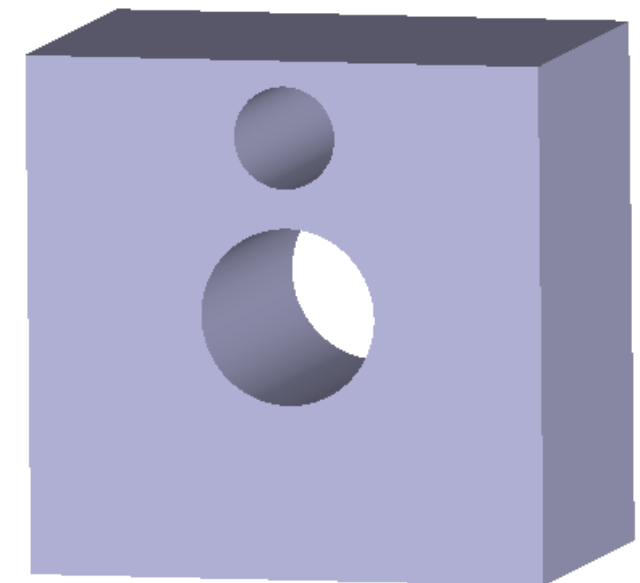
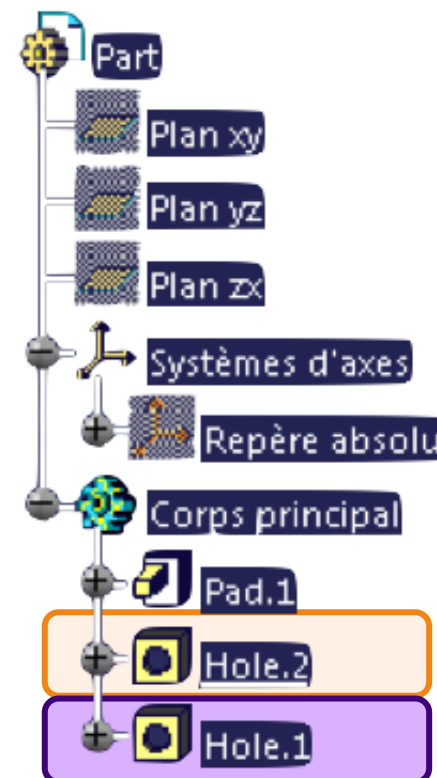
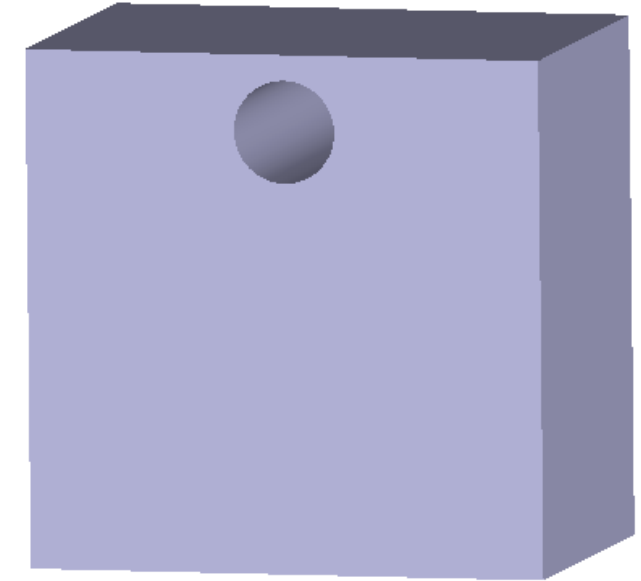
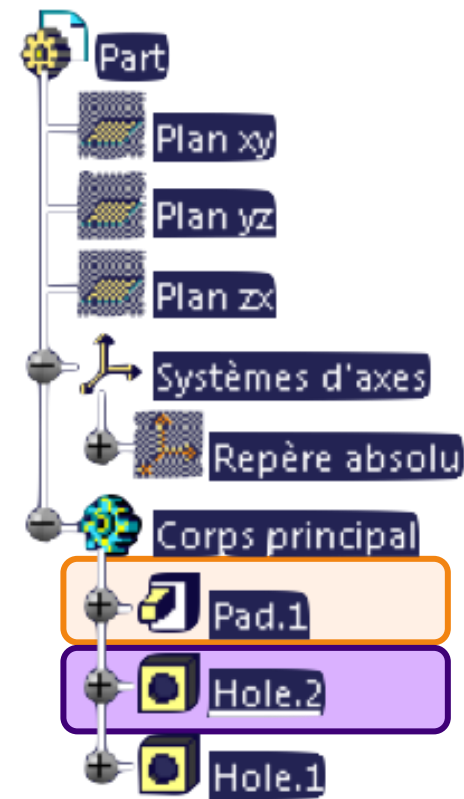
# Specification tree

## ► What is the object of work?

- The work object tells you where in the specification tree the next feature will be created.
- New functions are added just below the work object.

Previous Work Object

New Function



# Specification tree

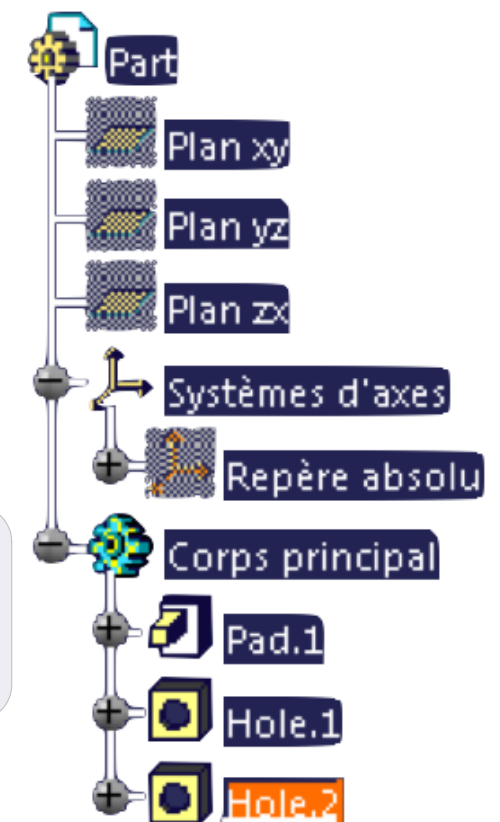
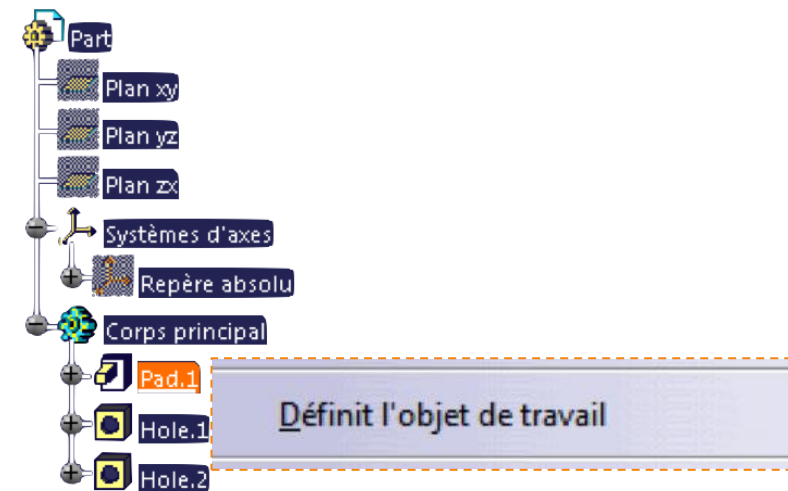
## ► How to define the object of work?

- **Automatically:** when you create a new function, this is automatically the work object.
- Using **Defines the work object**. For various reasons, you may need to define the work object. You can do this using the context menu **Defines the work object**, as explained below.

- In the specification tree, right-click on the feature you want to define as a work object,
- Select **Defines the work object**.
- The function is now defined as a work object.

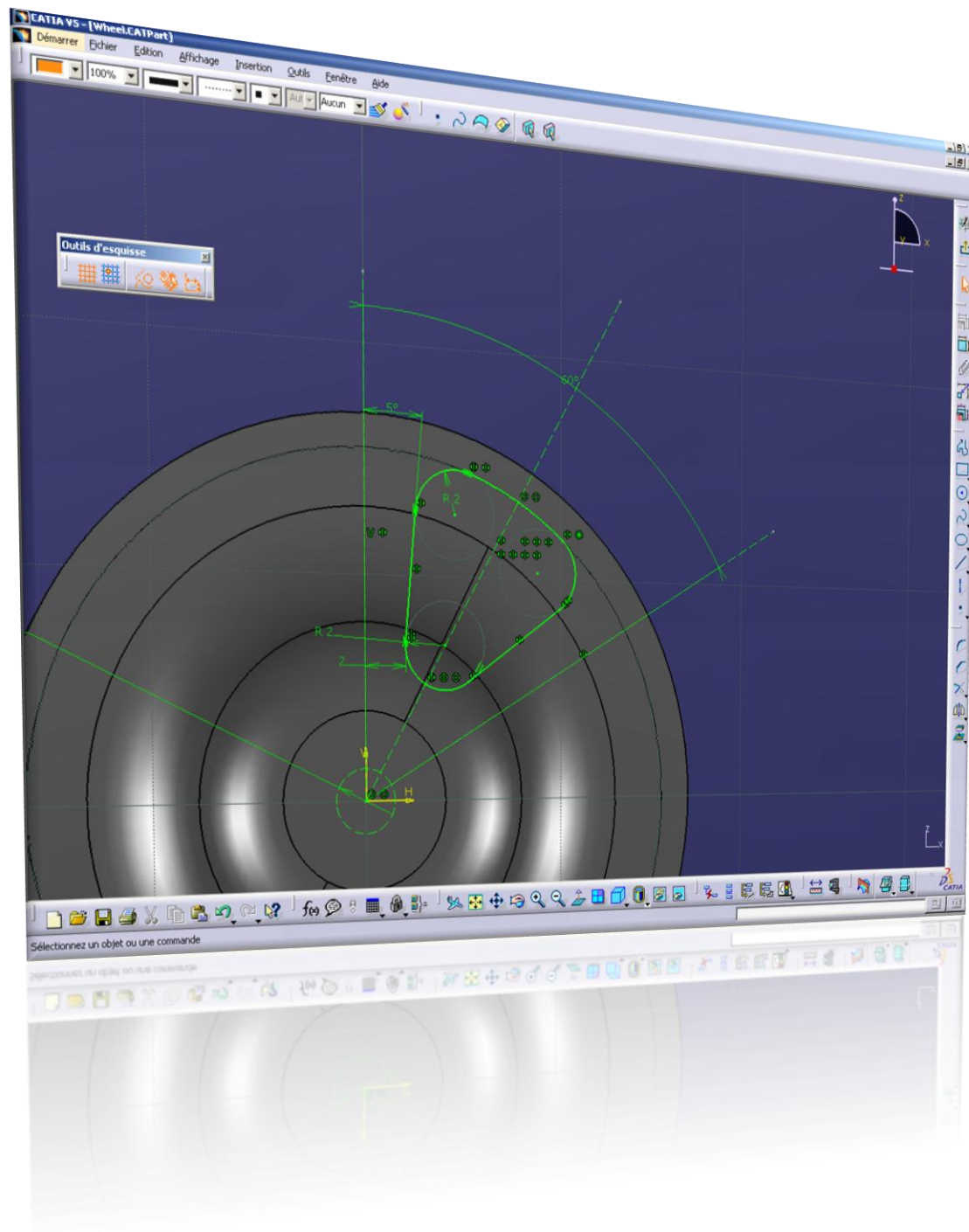


To quickly identify the work object is the element that is underlined.



# CATIA Sketcher

In this step you will see how to constrain a sketch.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. Your racing car.
4. *Method sheets.*
  - has. Configure CATIA.
  - b. Manage your data.
  - vs. Manage item properties.
  - d. Specification tree.
  - e. *CATIA Sketcher.*
    - i. Constraints dimensional.
    - ii. Geometric constraints.
  - f. CATIA Imagine & Shape.
  - g. CATIA Assembly Design.
  - h. CATIA Surface Machining.

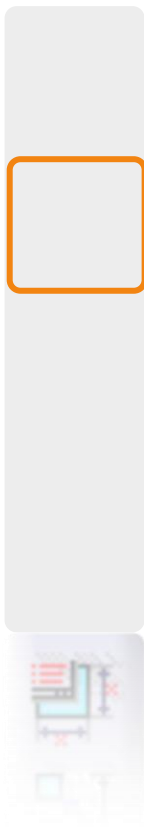
# Constraint

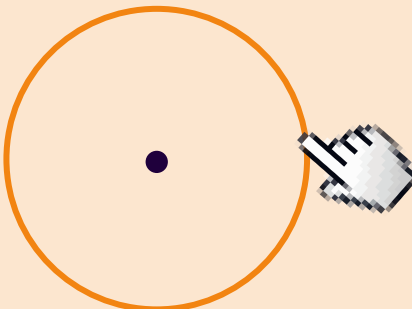
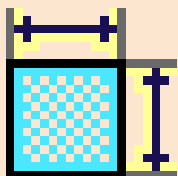
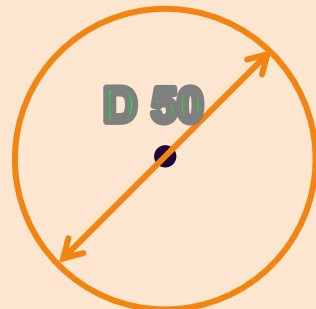
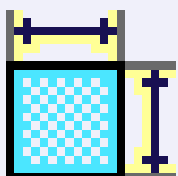
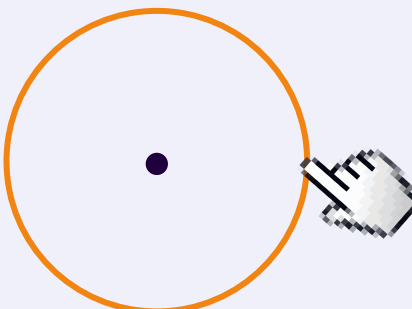
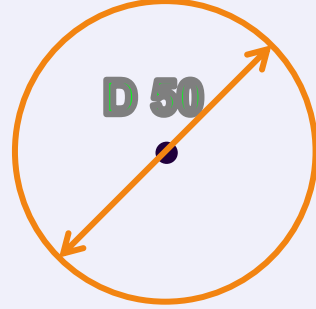
To dimensionally constrain a geometric element, use the icon **Constraint**.

You have two possibilities, select the element before or after clicking on the icon **Constraint**.



In order to constrain several elements relative to each other, you can select them by holding hold down the <CTRL> key on the keyboard.

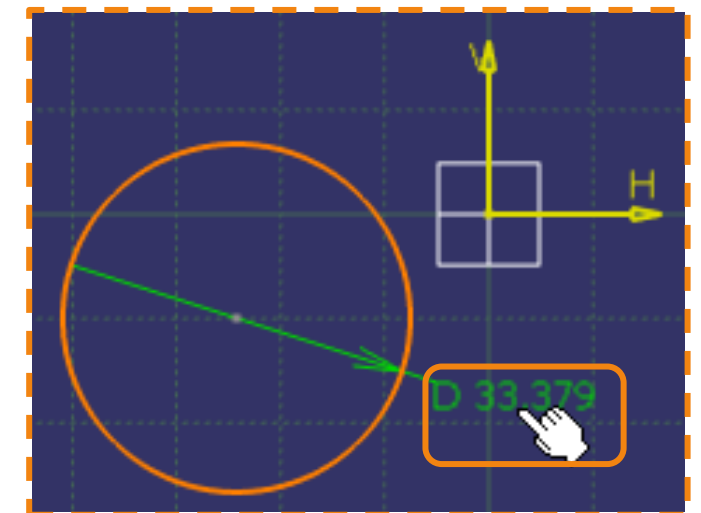


	<u>Step 1</u>	<u>2nd step</u>	<u>Result</u>
<u>First Method</u>	Select 	Constraint Icon 	
<u>Second Method</u>	Constraint Icon 	Select 	

# Dimensional constraints

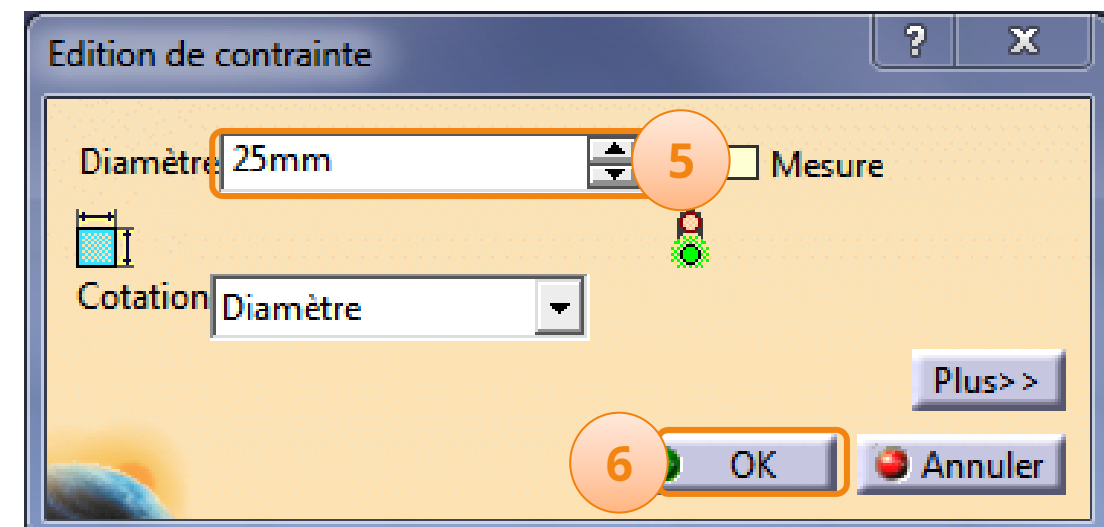
The purpose of this exercise is to constrain a circle to the coordinate system. For that :

1. Draw a circle. After drawing it, the circle remains selected. If not, select it by clicking on it.
2. Click the icon **Constraint** to display a diameter dimension. By moving the mouse you can choose where to display it.
3. When the location suits you, click, being careful not to be on another element.



Now we are going to modify the value of this diameter.  
For that :

4. Double-click **the value of this rating** to bring up the edit window.
5. Enter a diameter of [25mm].
6. Click on **okay** to validate.

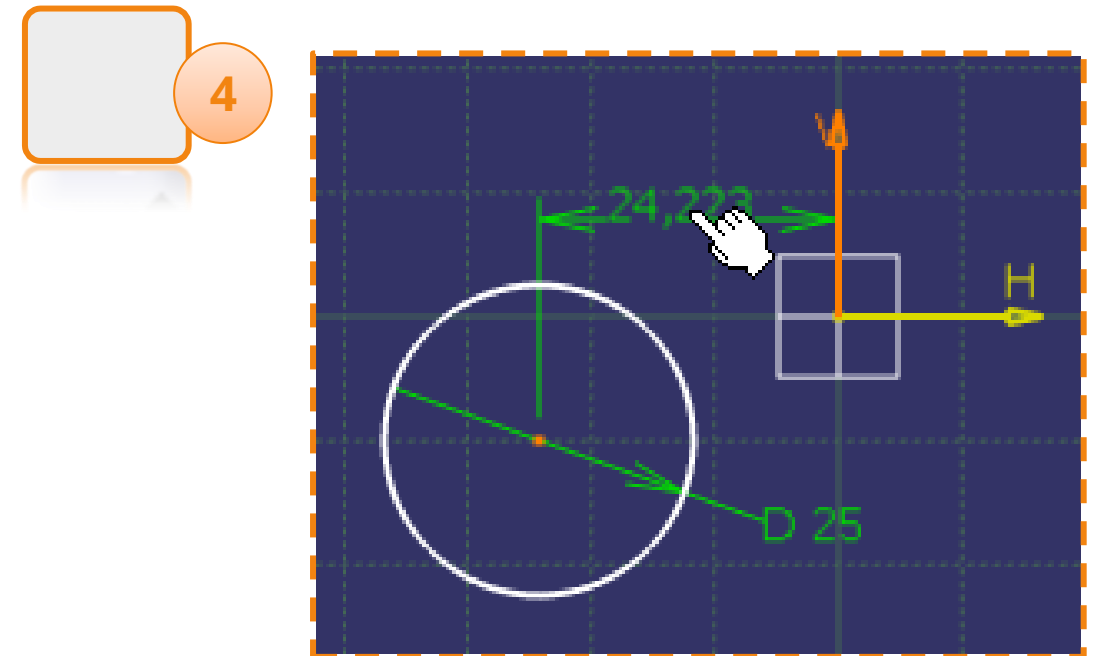




# Dimensional constraints

Now that you have a circle with the correct diameter, you still have to constrain it with respect to the mark. For that :

1. Hold down **<CTRL>** depressed.
2. Select **the center of the circle**.
3. Select **the vertical axis of the marker**.
4. Click the icon **Constraint**. This causes a dimension to appear between the center of the circle and the mark. By moving the mouse you can choose where to display it.
5. When the location suits you, click, being careful not to be on another element.
6. You can edit the dimension in the same way as for the previous one. By indicating a value of [25 mm].

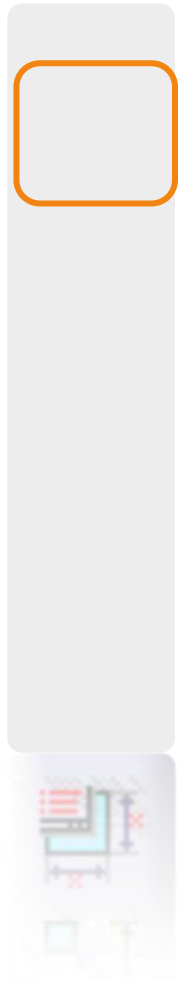
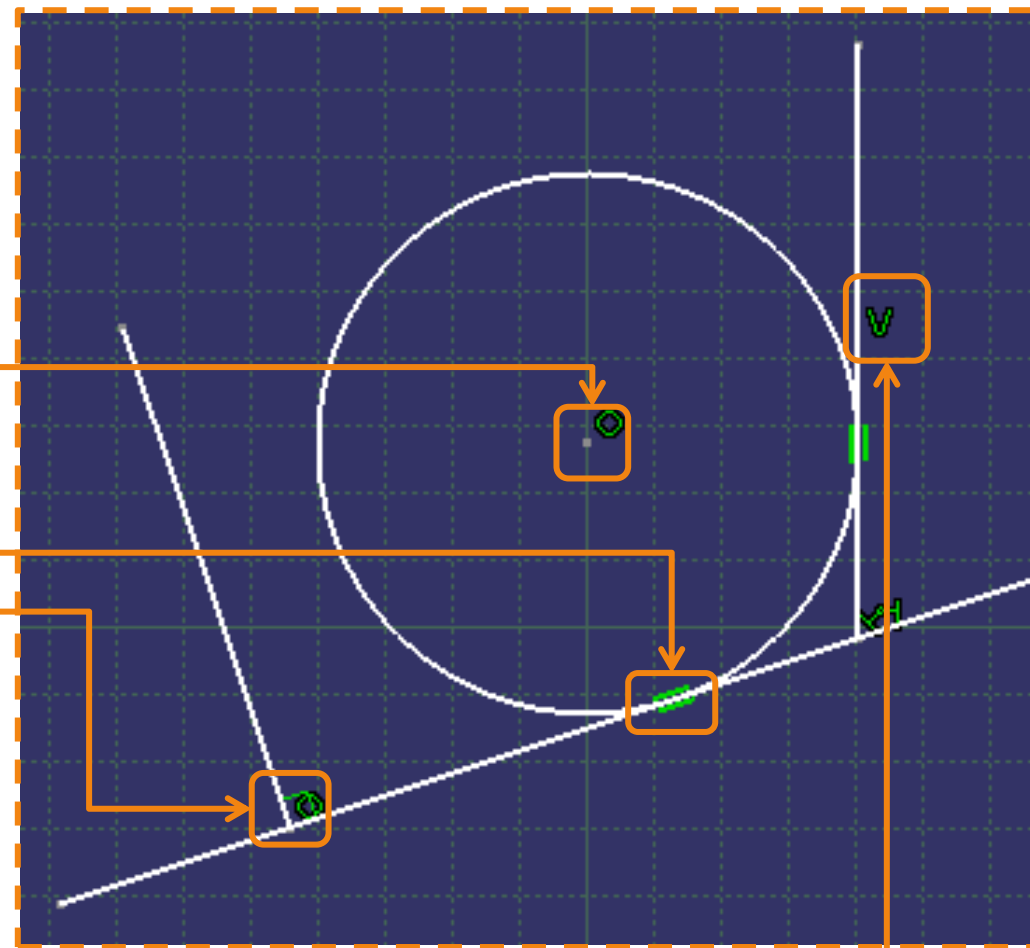
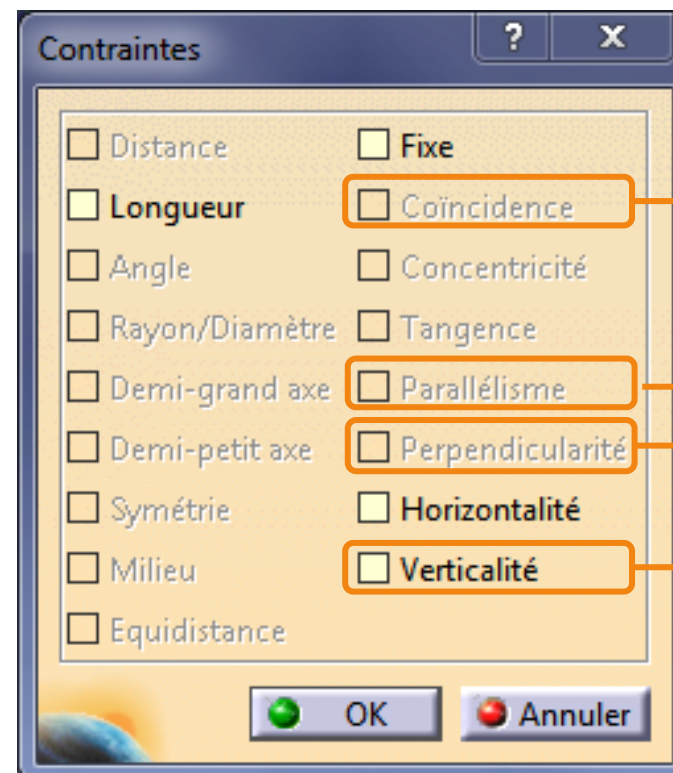


The center of the circle will always be 25mm from the vertical axis.

## Constraints chosen in a dialog box

To geometrically constrain elements you can use the icon **Constraints chosen in a dialog box**.

The icon is accessible when you have selected enough elements to constrain by holding the key <CTRL>.

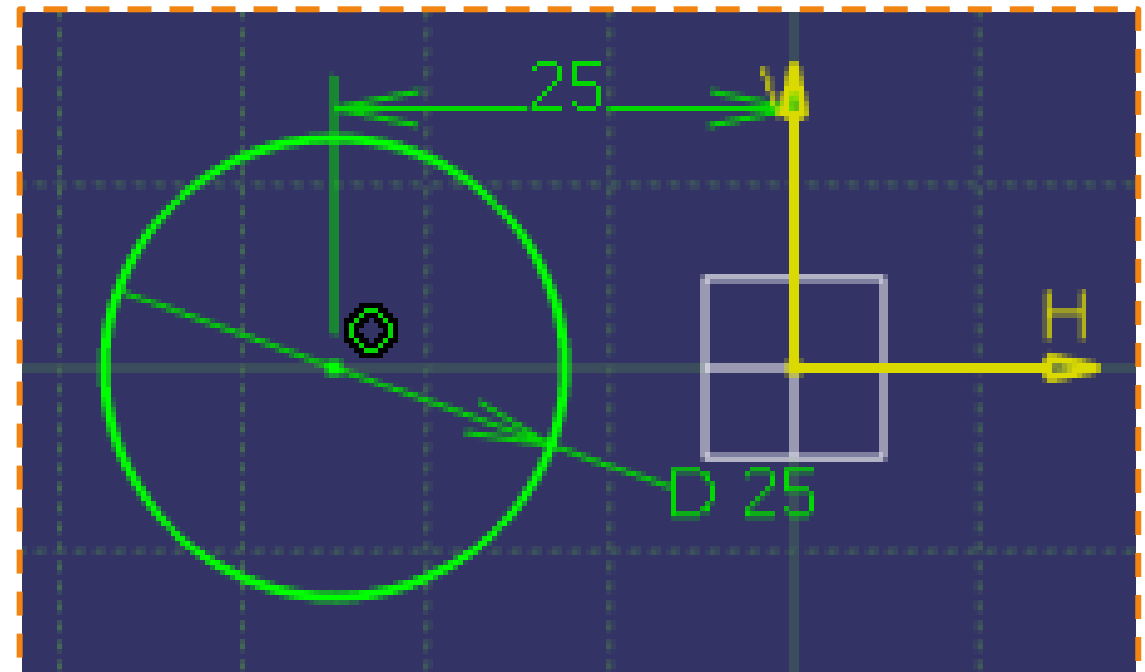
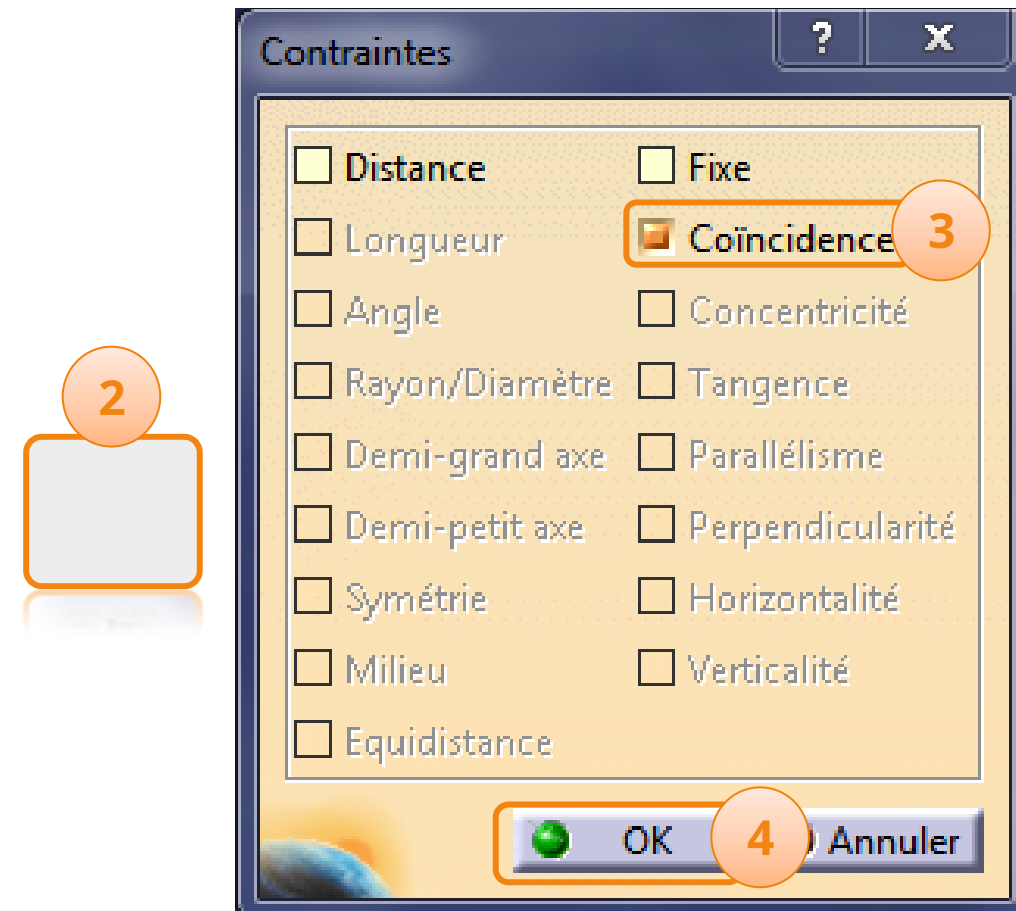


# Geometric constraints

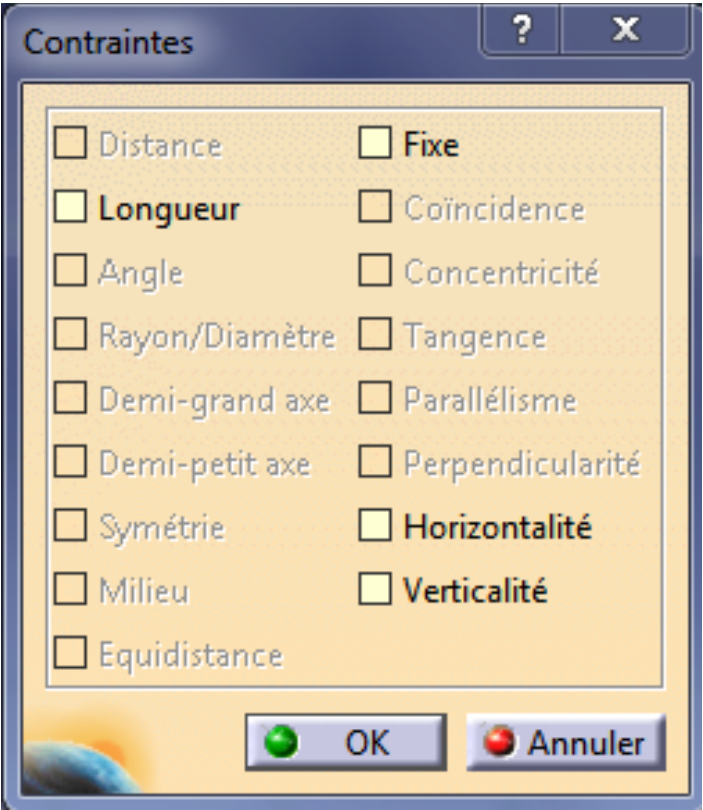
We will now apply a coincidence constraint. This constrains the center of the circle to the horizontal axis H. To do this:

1. Select by holding the key<CTRL>the center of the circle and the horizontal axis.
2. Click the icon **Constraints** chosen in a dialog box.
3. Select **Coincidence**.
4. Click on **okay**.

You will notice that the center of the circle aligns with the horizontal H axis, and the presence of a small green circle next to it, indicating a coincidence constraint.



# Geometric constraints

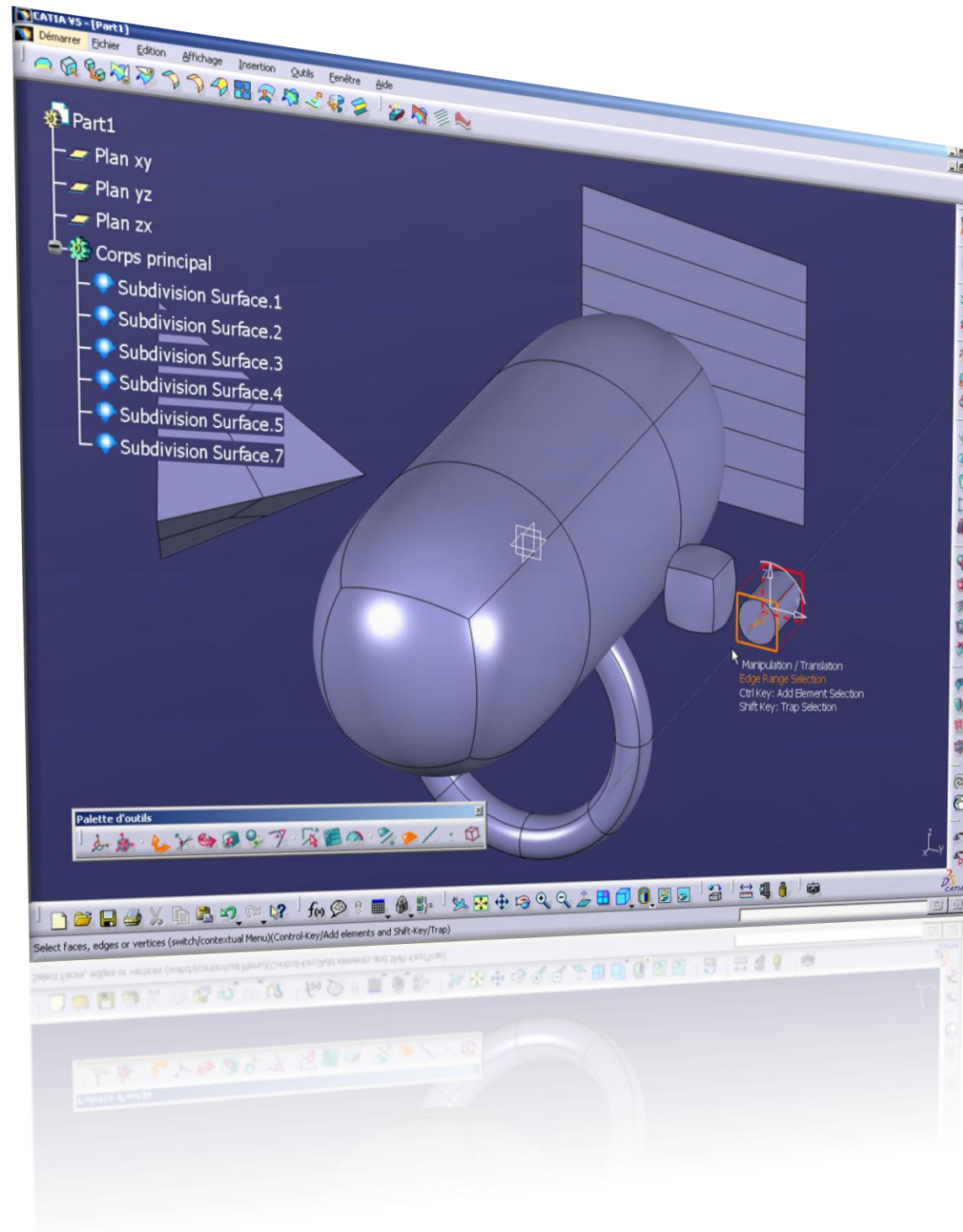


Symbols	Meaning
R 25 / D 50	Radius / Diameter
	Symmetry
	Environment

Symbols	Meaning
	Fixity
	Coincidence
	Concentricity
	Tangency
	Parallelism
	Perpendicularity
H	Horizontality
V	Verticality

# CATIA Imagine & Shape

In this step you will discover the Imagine & Shape workbench of CATIA V5.



Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. Your racing car.

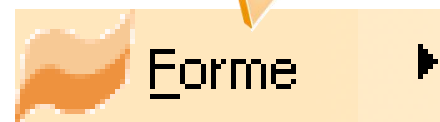
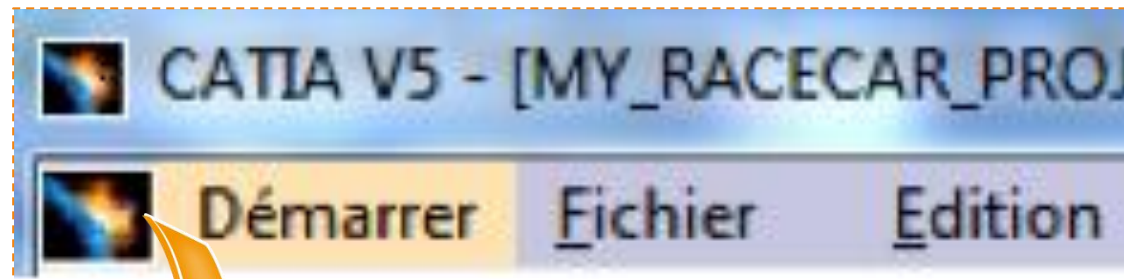
#### **4. *Method sheets.***

- has. Configure CATIA.
- b. Manage your data.
- vs. Manage item properties.
- d. Specification tree.
- e. CATIA Sketcher.
- f. *CATIA Imagine & Shape.***
  - i. Introduction.
  - ii. Basic shapes.
  - iii. Modification and manipulation tools.
- g. CATIA Assembly Design.
- h. CATIA Surface Machining.

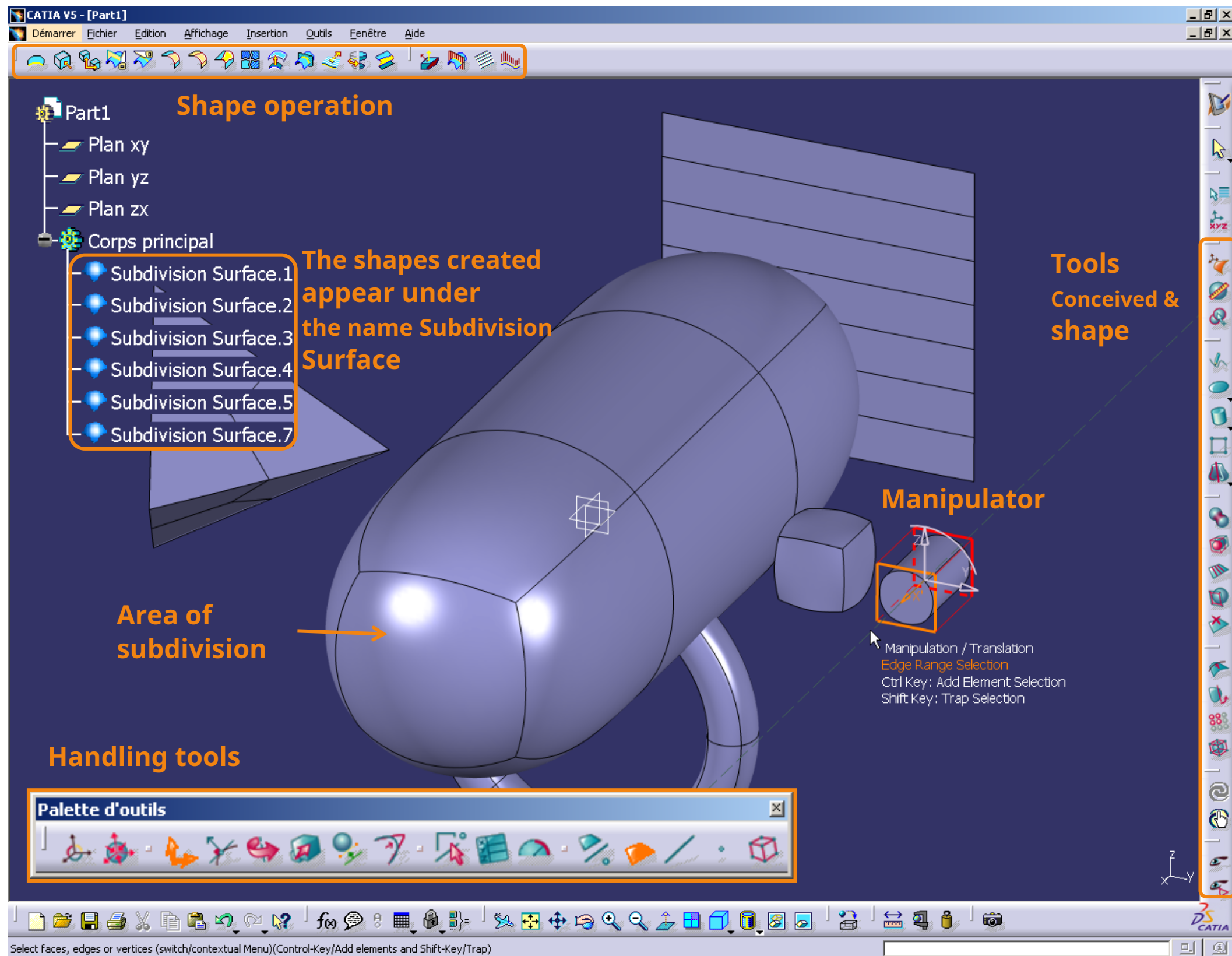


## Imagine & Shape

This workshop is dedicated to the creation of shapes for manufacturers and designers. Its surface approach is particular, which makes it easy to use even for a new user. It allows you to quickly transcribe your ideas in 3D. To access it click on **Start > Shape > Imagine & Shape**.

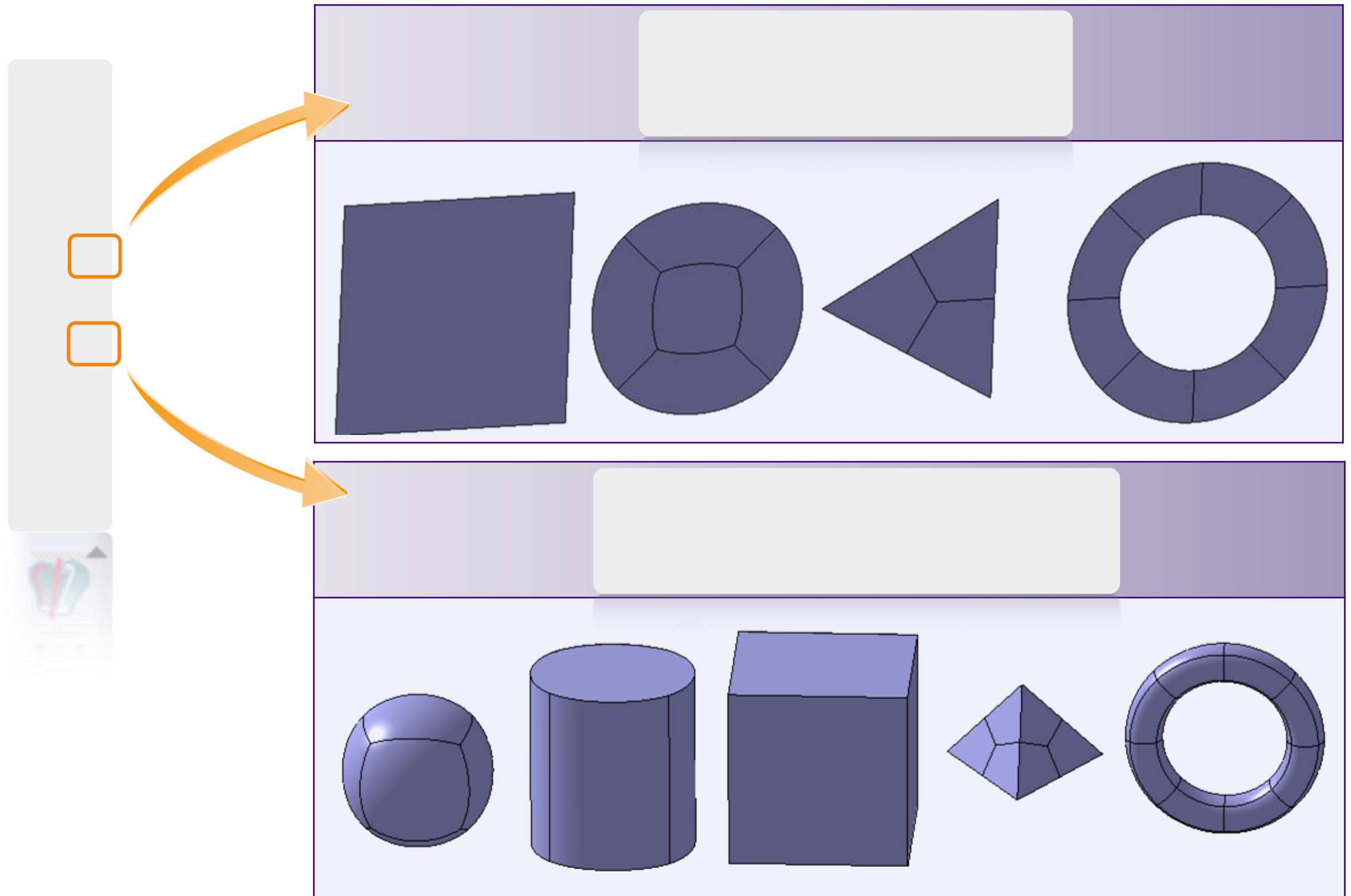


# Introduction



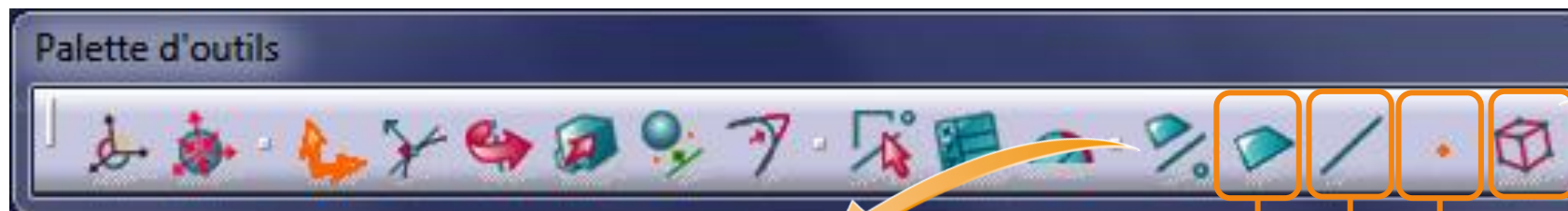
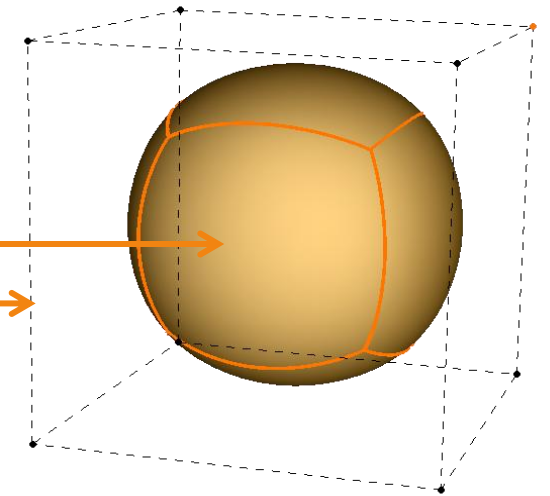
## Basic forms

- Imagine & Shape allows you to create different types of basic subdivision surfaces.



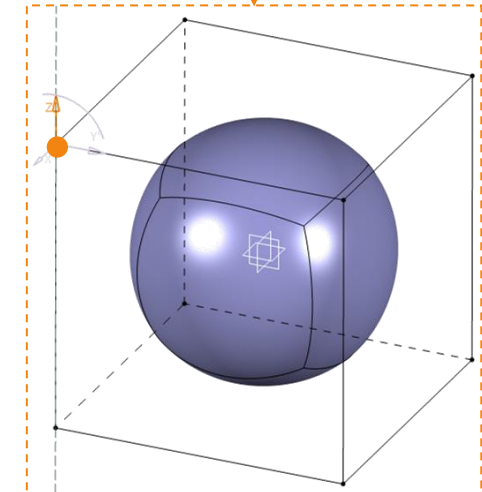
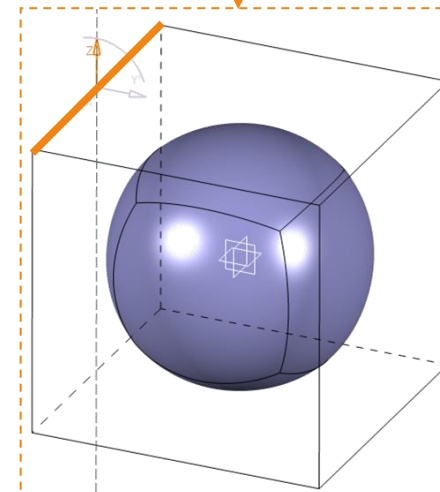
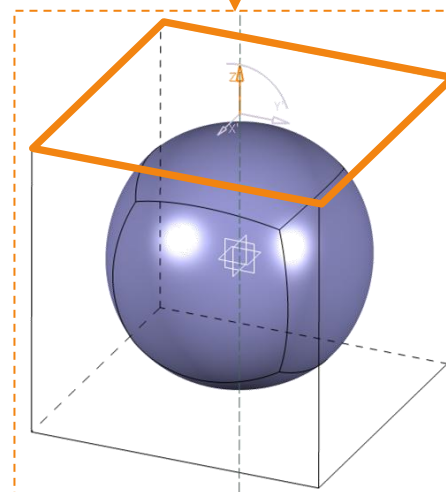
# Manipulation and modification tools

- ▶ The shapes you create are subdivision surfaces that you model using the control mesh.
- ▶ Modifying the control mesh is equivalent to modifying the surface. One applies for that a transformation on sub-elements of the mesh: faces, edges, points or the whole of the elements of the mesh.



Selection Any Type

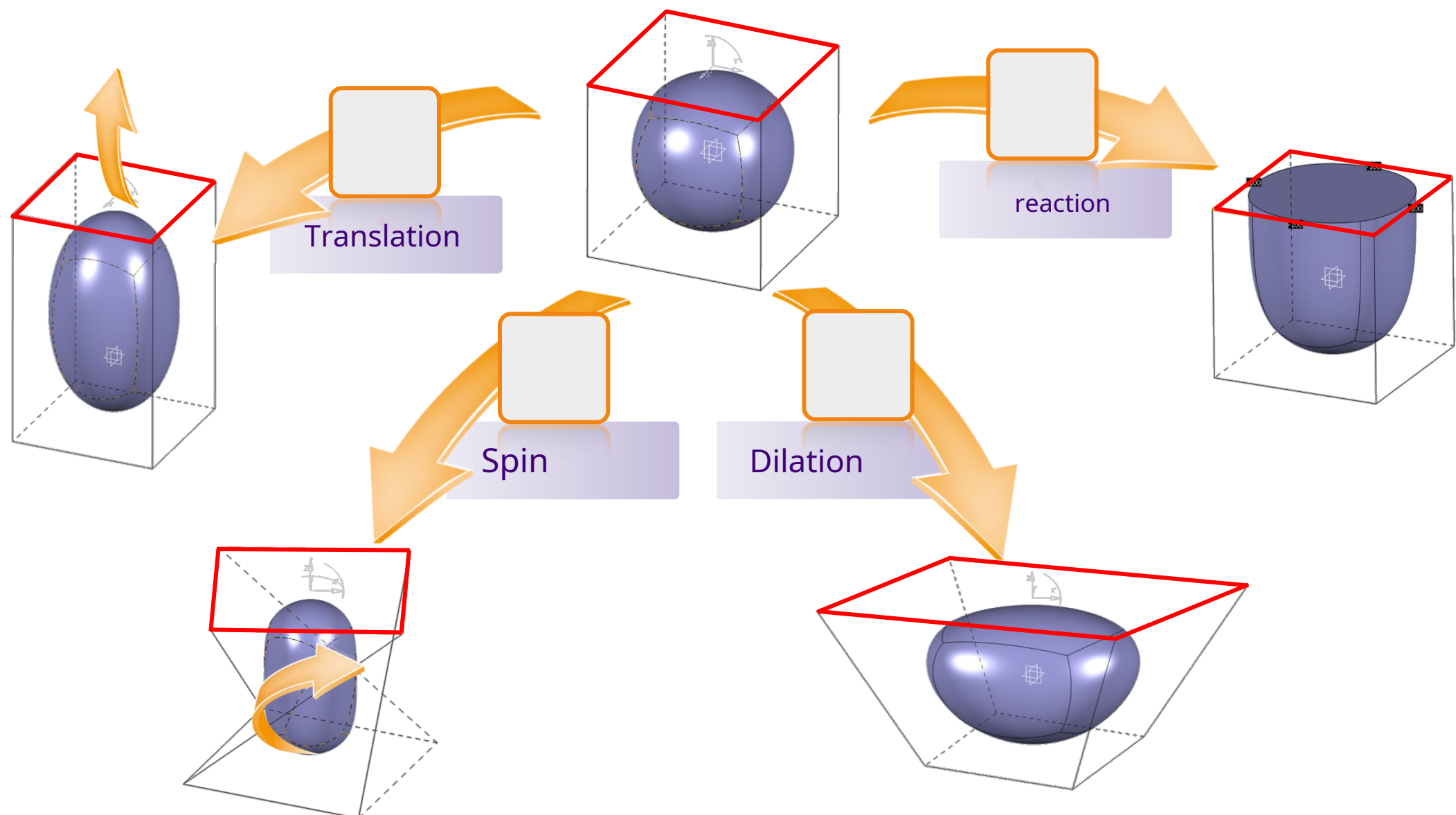
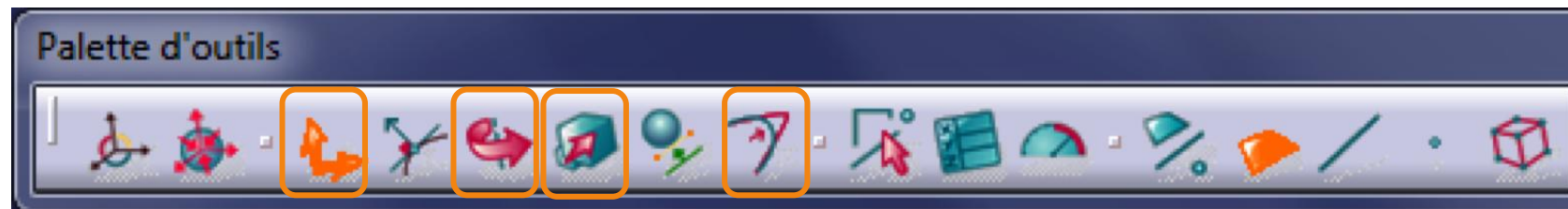
Item Selection Complete





# Manipulation and modification tools

- You can manipulate the different thanks to different tations.



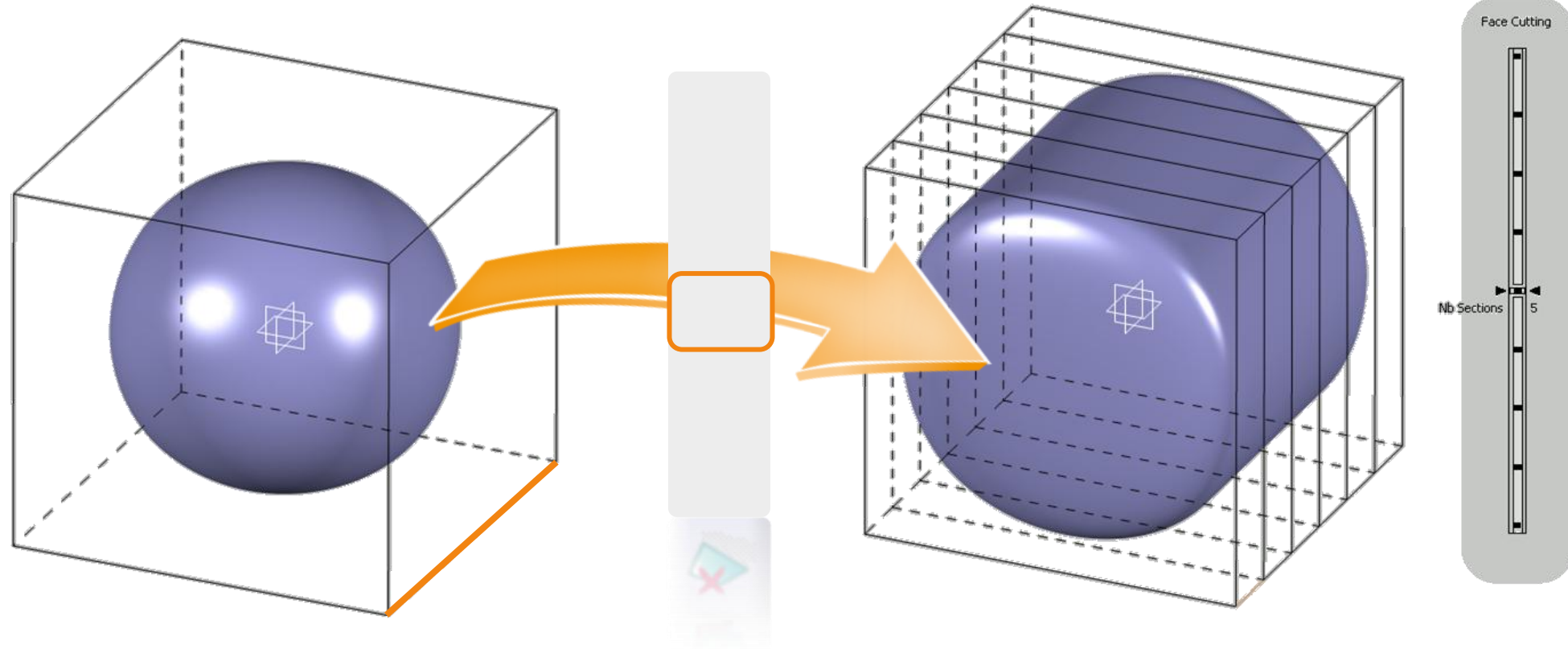


# Manipulation tools e

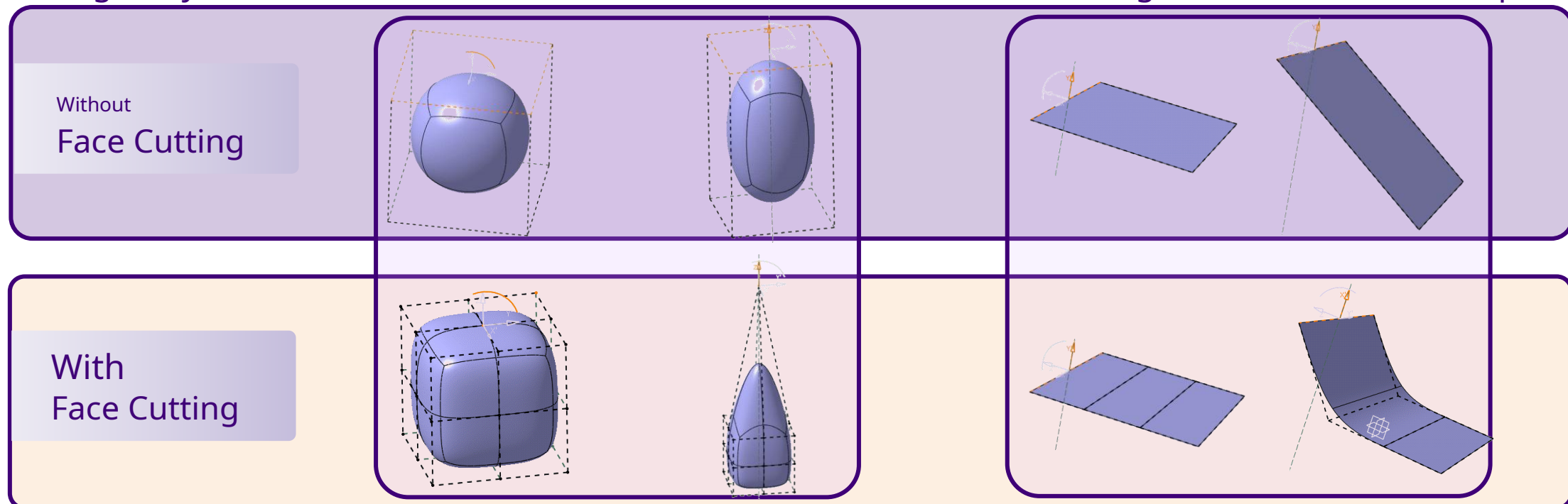
## modifications

- You can also change the

control, using the icon **Face Cutting**.



- This greatly influences the behavior of a s e of subdivision during a translation for example.

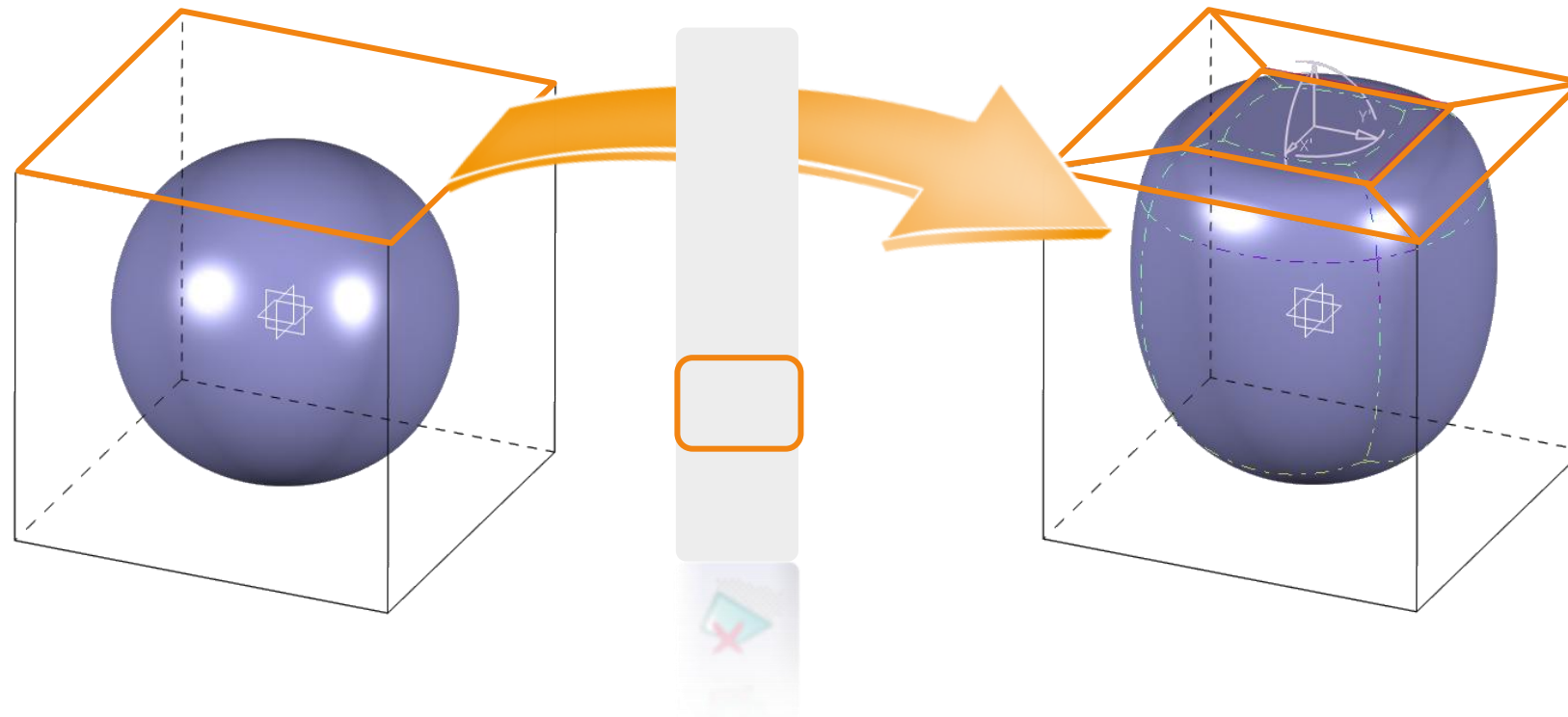


# Manipulation tools e

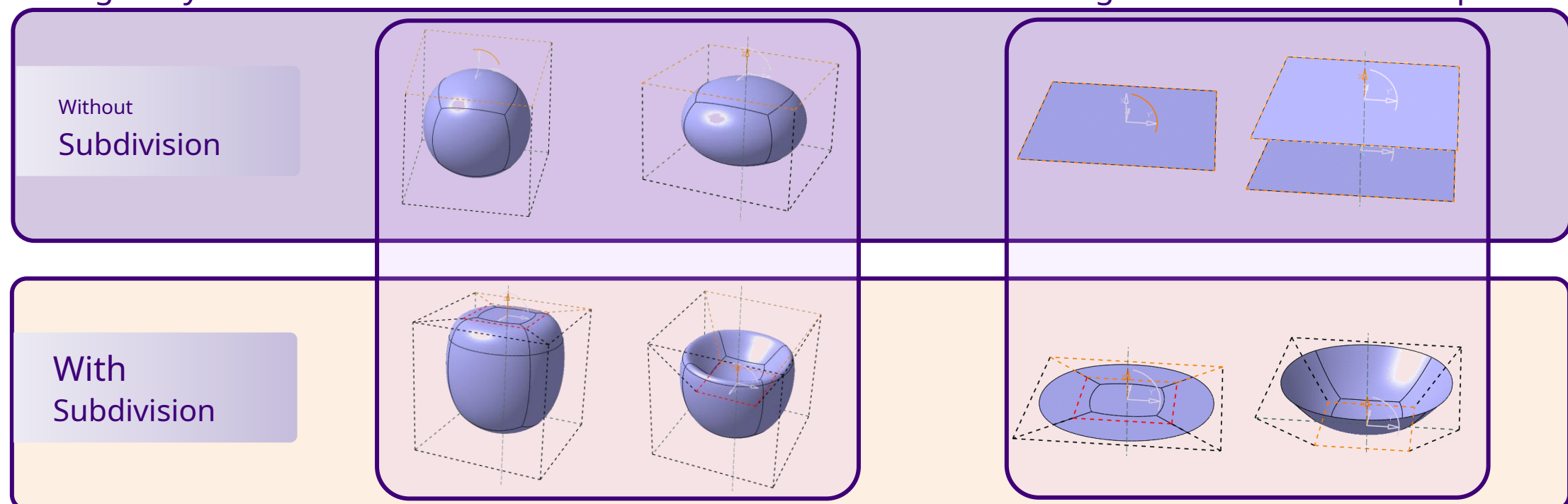
## modifications

- ▶ Another way to modify the m

control is **subdivide faces**.



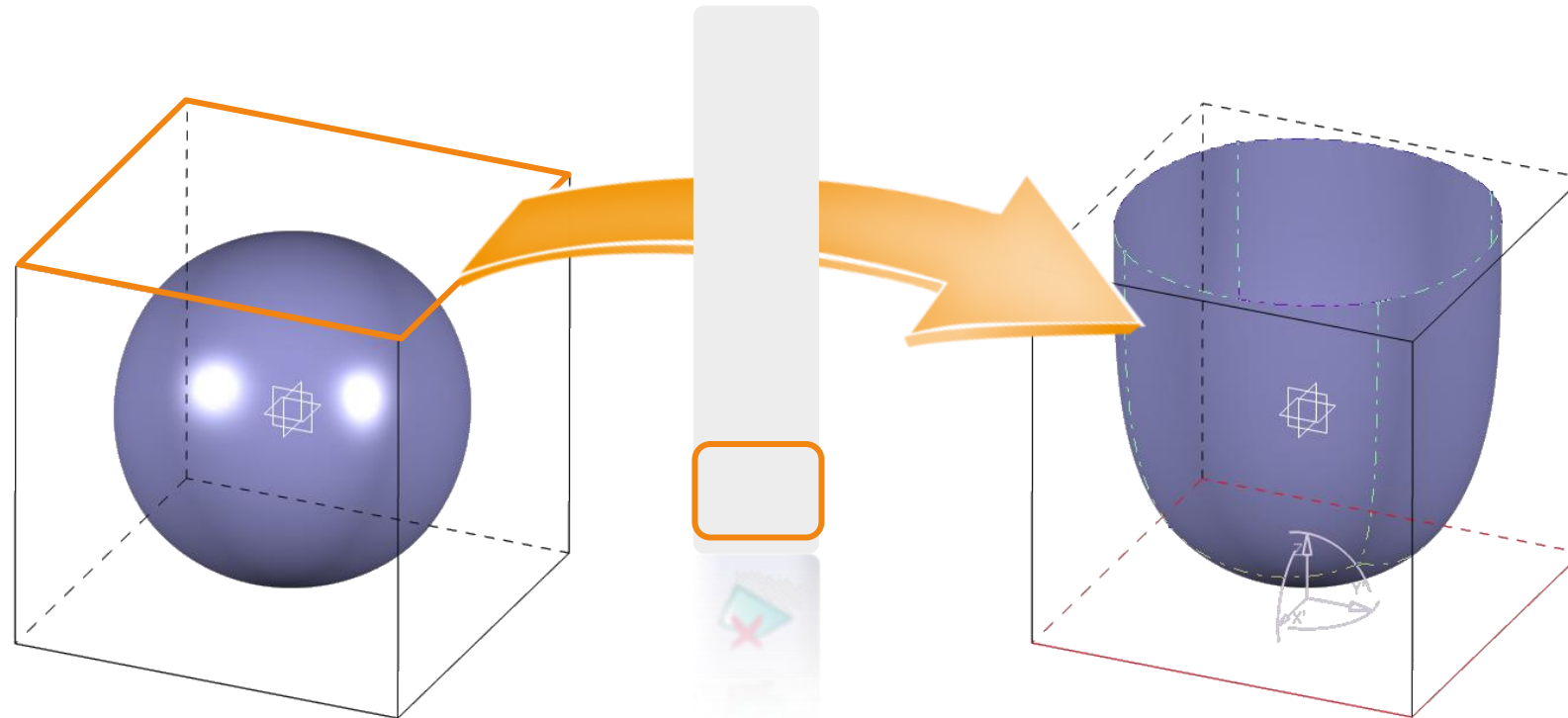
- ▶ This greatly influences the behavior of a subdivision face during a translation for example.



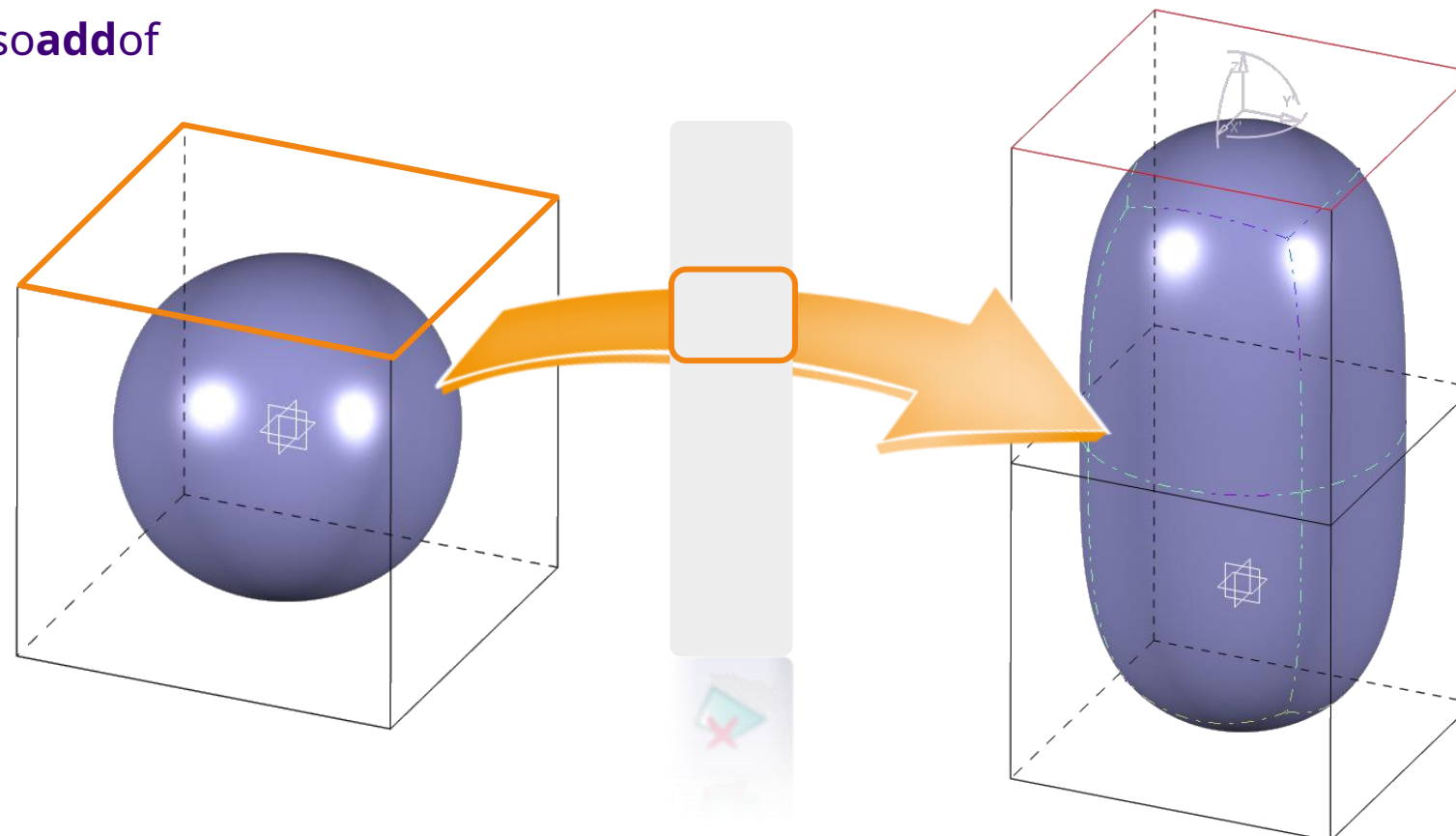
# Manipulation tools e

## modifications

- You can **remove** faces



- You can also **add** of



# CATIA Assembly Design

In this method sheet we will explore several assembly options. In particular, we will see two component insertion methods. Finally, we will see how to delete a component and its associated assembly constraints.

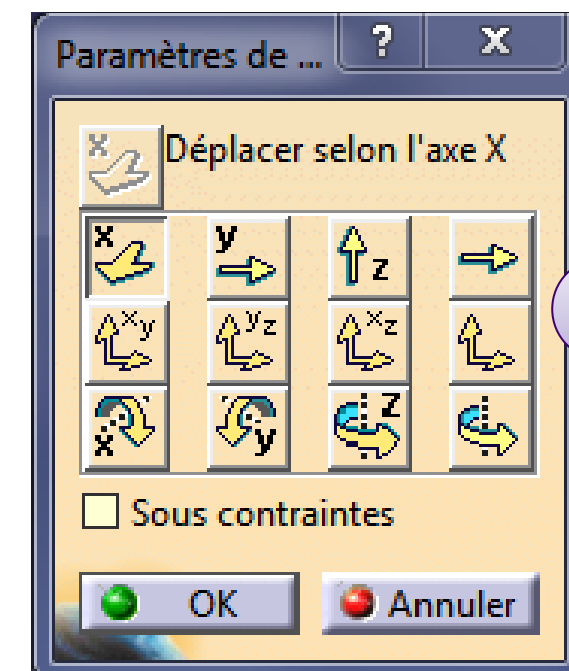
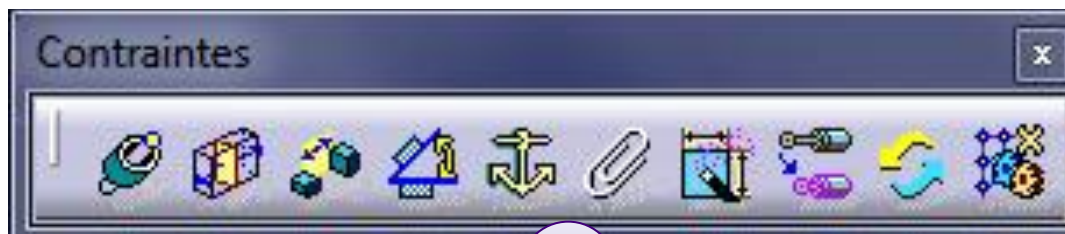


Here are the steps to follow:

1. Introduction.
2. CATIA V5 QuickStart.
3. Your racing car.
- 4. Method sheets.**
  - has. Configure CATIA.
  - b. Manage your data.
  - vs. Manage item properties.
  - d. Specification tree.
  - e. CATIA Sketcher.
  - f. CATIA Imagine & Shape.
  - g. CATIA Assembly Design.**
    - i. Existing Component
    - ii. Existing component with positioning.
    - iii. Delete a component.
  - h. CATIA Surface Machining.

## Existing component

- 1 The corresponding icon is accessible when you are in the Assembly Design workbench. By default, it is on the right of your screen. This tool allows you to insert a part into your assembly.
- 2 The tool **Handling** will allow you to move this piece along the axes you want.
- 3 The various constraint icons will allow you to apply assembly constraints to your parts. This bar is vertical on the right of your screen by default.





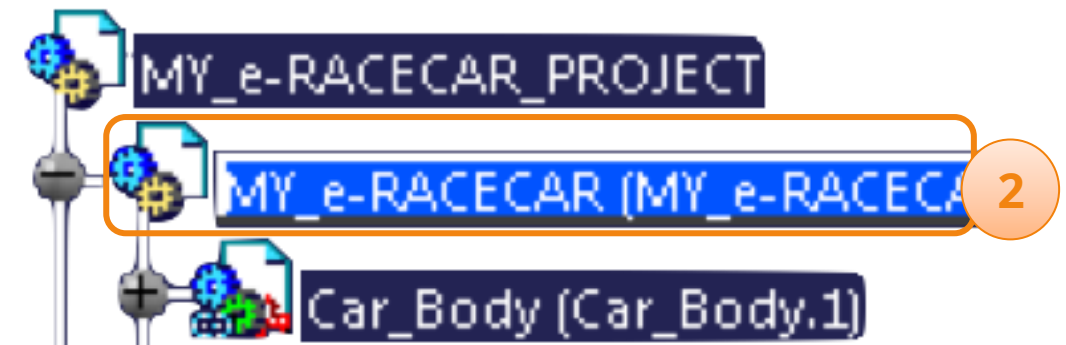
# Insert parts with "Existing Component"

We will start by adding the part to the assembly.  
For that :

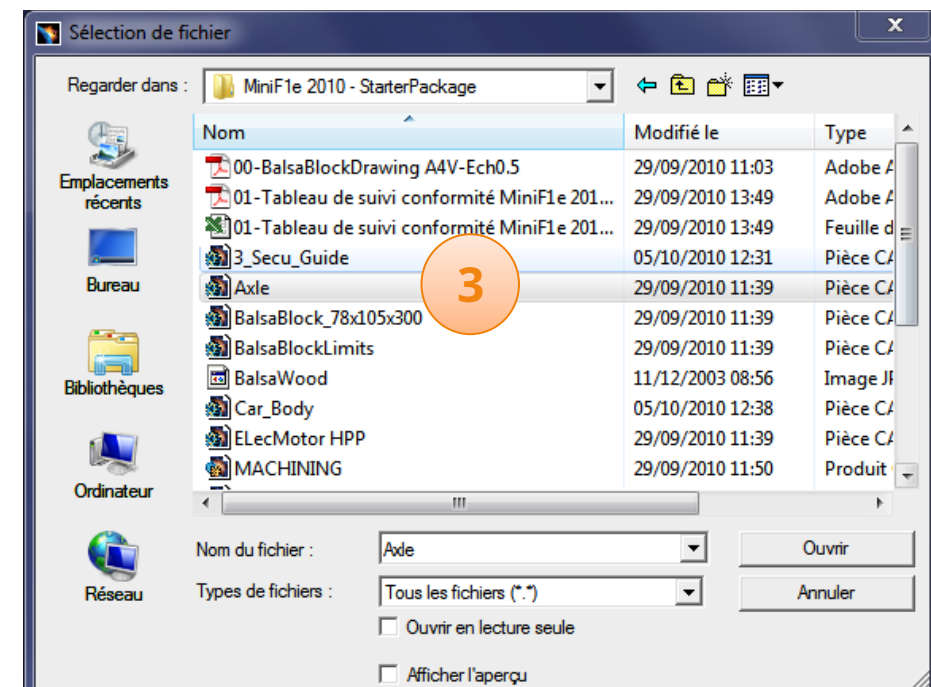
1. Click on **Existing component**.



2. Click on the product in which you want to add your part.  
Here we will click on **MY\_e-RACECAR**.



3. A file selection window opens. Select the file you want to insert. Here we will select **Axel**.



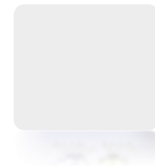
# Insert parts with "Existing Component"

1. You will notice that the part is inserted at the origin of the assembly mark.

We will pre-position the part and then constrain it. As the coordinate system used is that of the assembly, we will first perform a rotation.

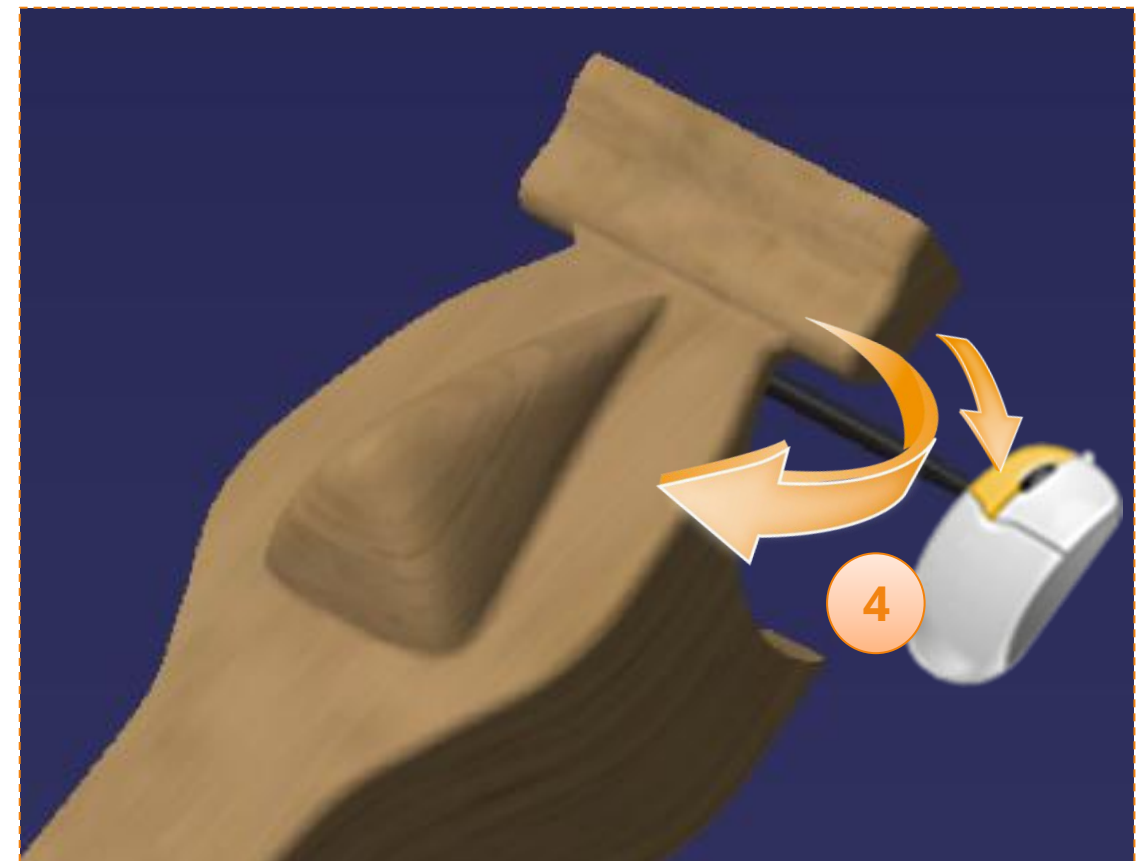
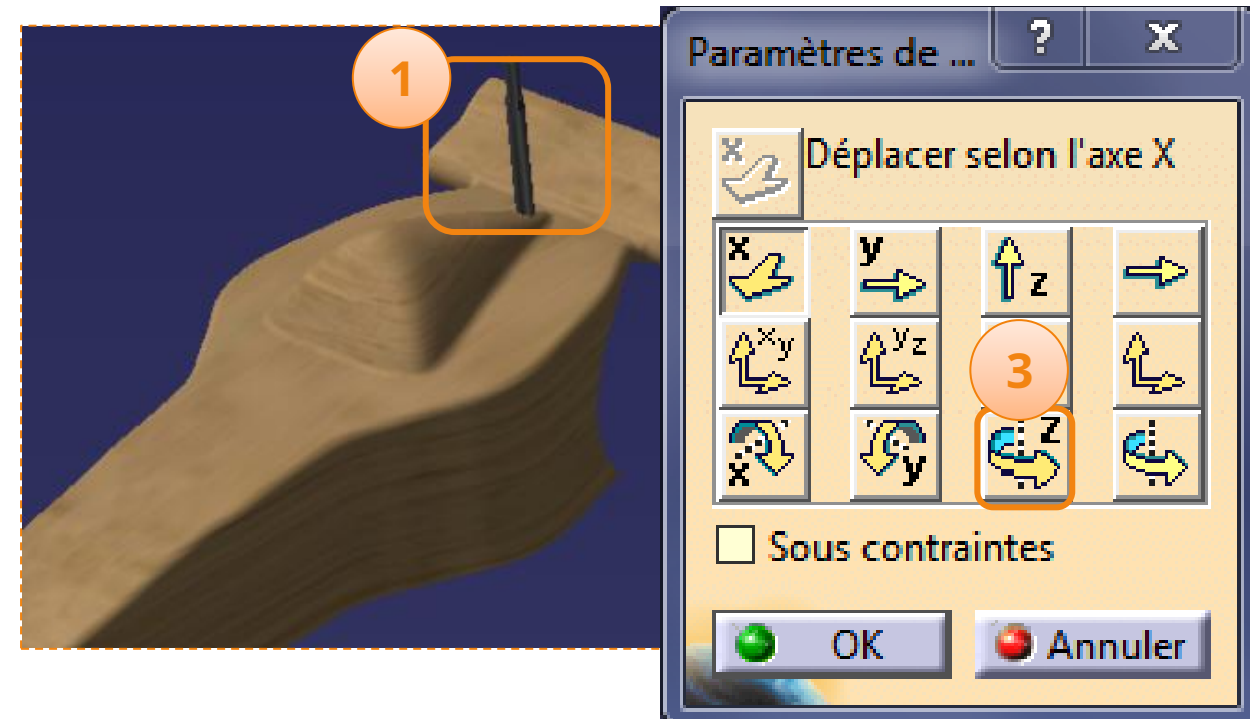
For that :

2. Click the icon **Handling**.



3. Select the **rotation along the Z axis**.

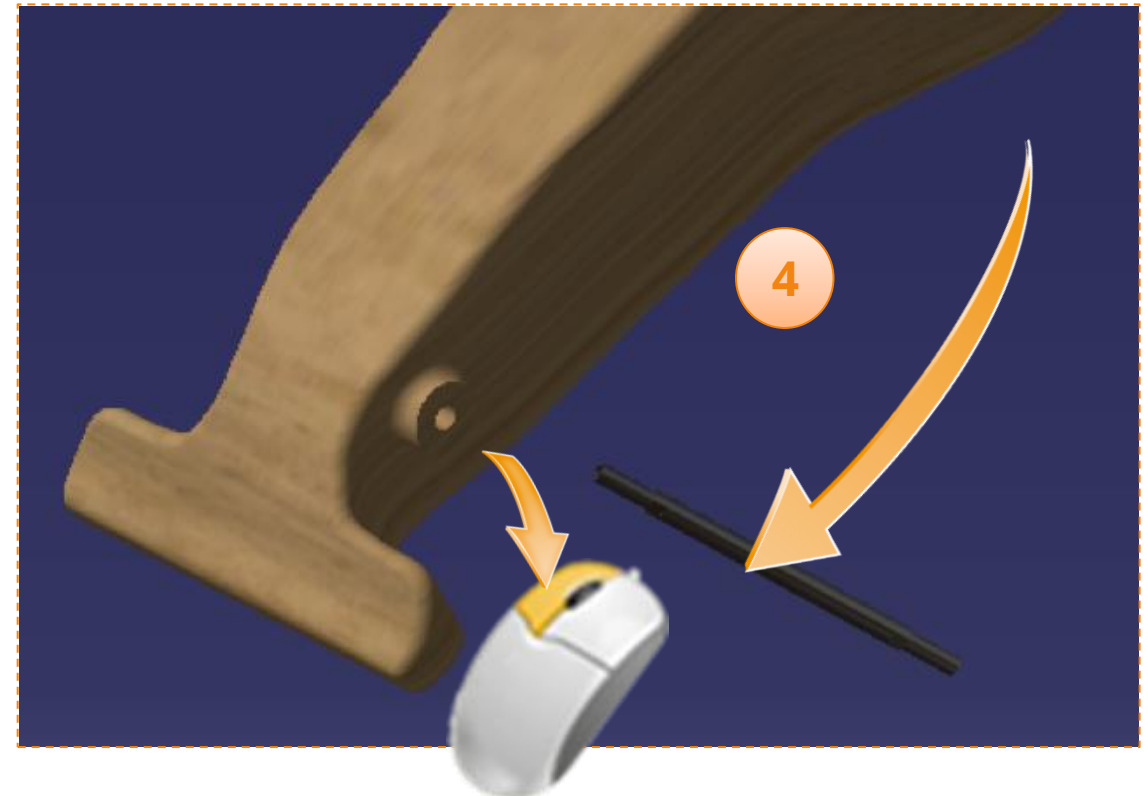
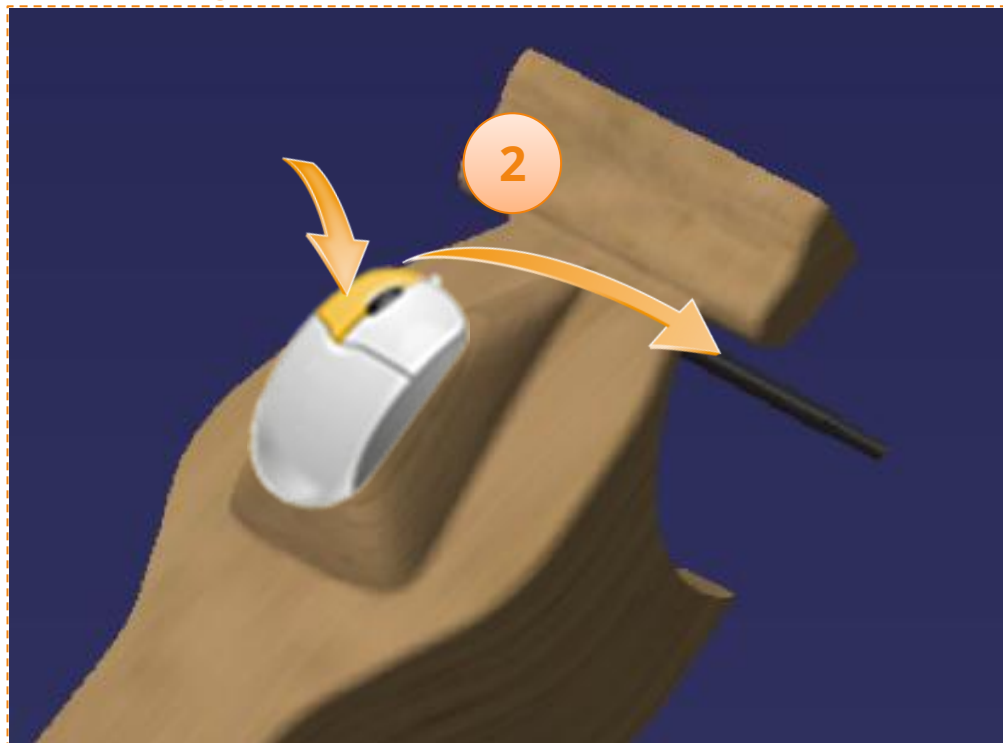
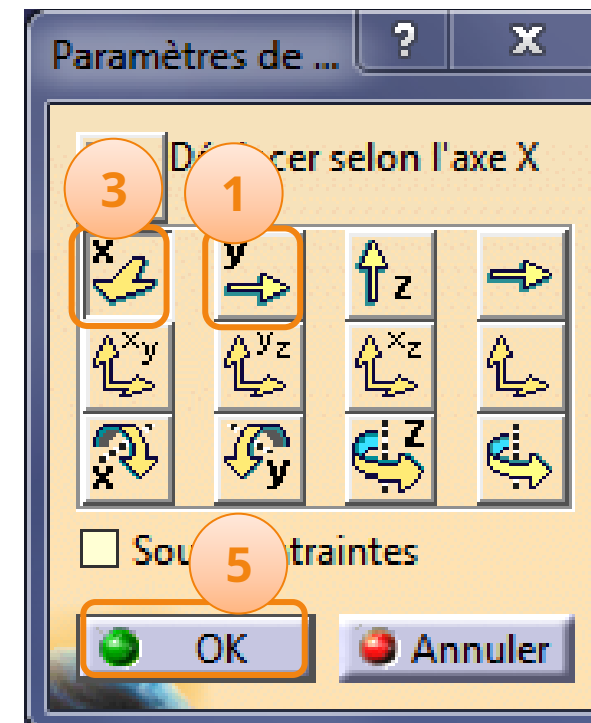
4. Click and hold the click on the piece to rotate it by moving the mouse. Orient the piece as shown.



# Insert parts with "Existing Component"

We are going to place the piece on the front right of the car.  
For that :

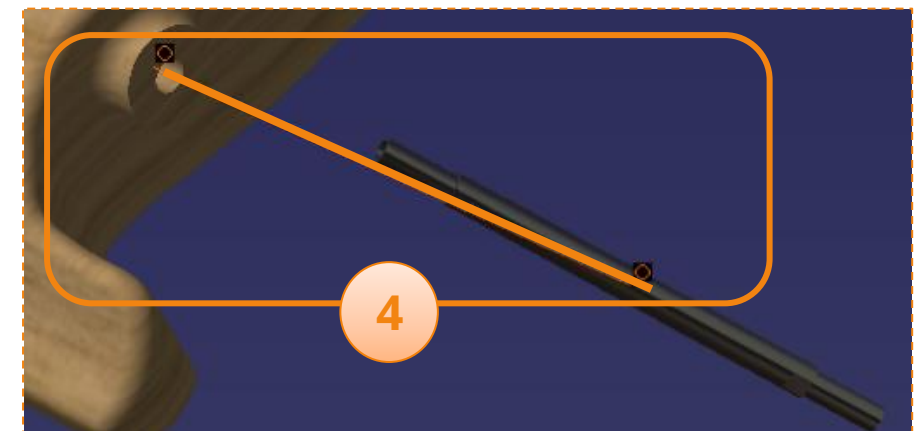
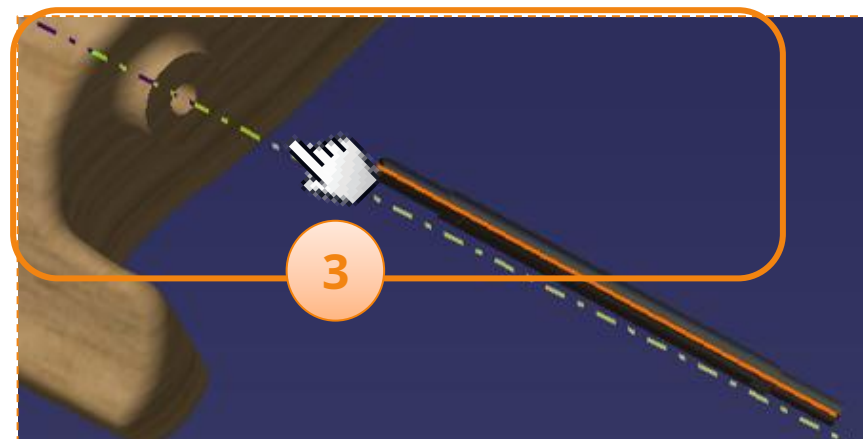
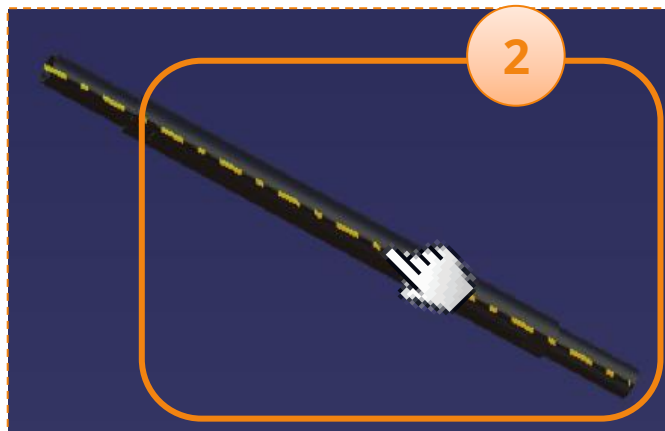
1. Select the **translation along the Y axis**.
2. Click and hold the part to move it to the side of the car.
3. Select the **translation along the X axis**.
4. Click and hold the part to move it to the front of the car.
5. Click on **okay**.



# Insert parts with "Existing Component"

We are now going to constrain the part by making the axis of the part coincide with that of the car. For that :

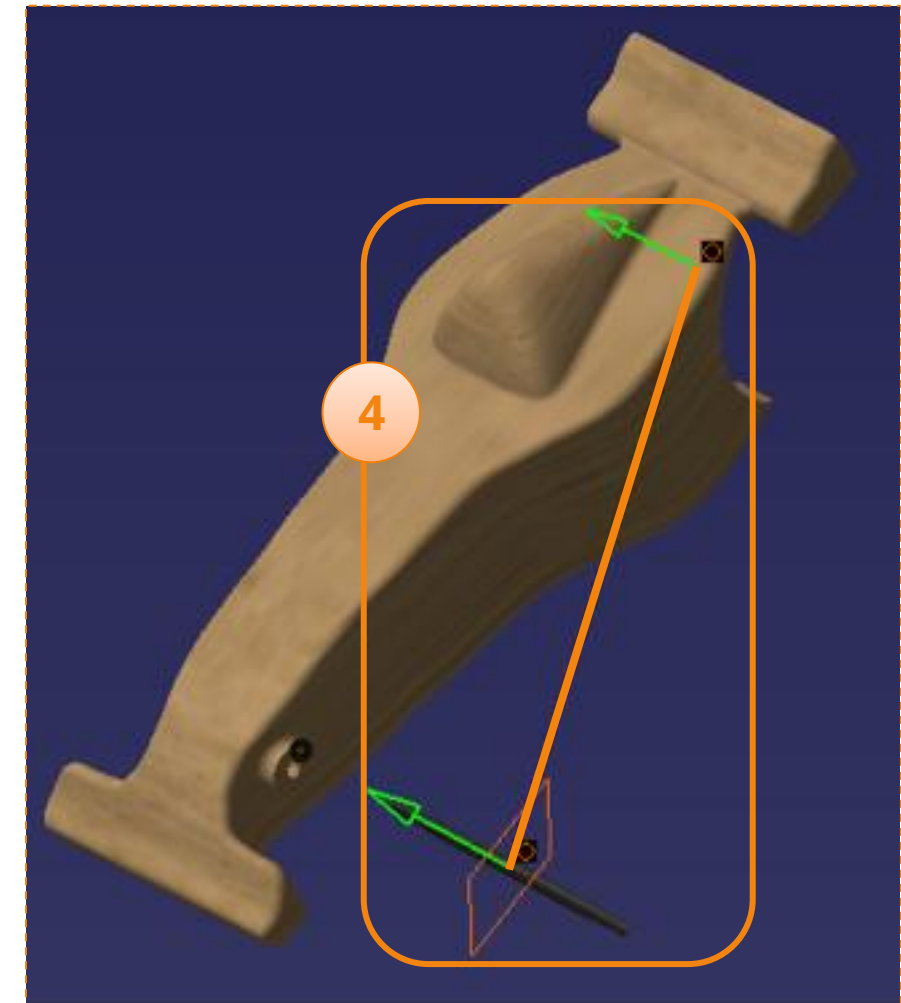
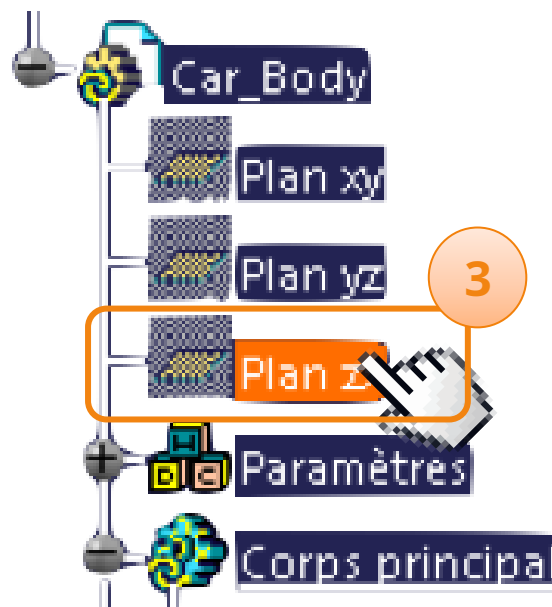
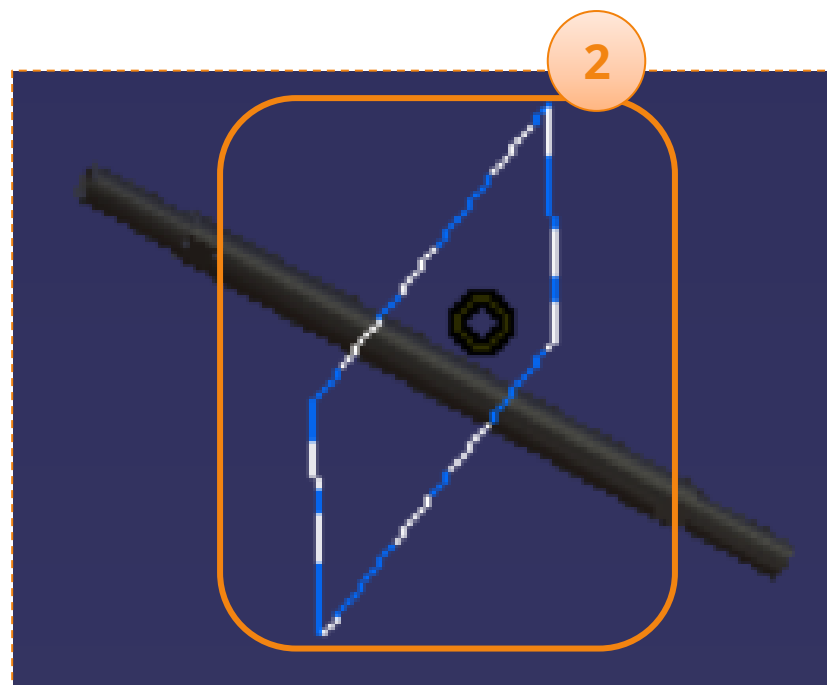
1. Click on the icon of **Coincidence constraint**.
2. Click on **the axis of the part**.
3. Click on **the axis of the car**.
4. You will notice the link between the two parts indicating a constraint.



# Insert parts with "Existing Component"

Now we are going to press the part against the car. For that :

1. Click on the icon of **Contact stress**.
2. Click on the **Map.1** of Axle which represents the plane of symmetry of the axis.
3. Click on the **ZX plane**, plane of symmetry of the car.
4. You will notice the link between the two parts indicating a constraint.





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You now notice the p symbols representing the gray moment cont, because the constraints To apply them:

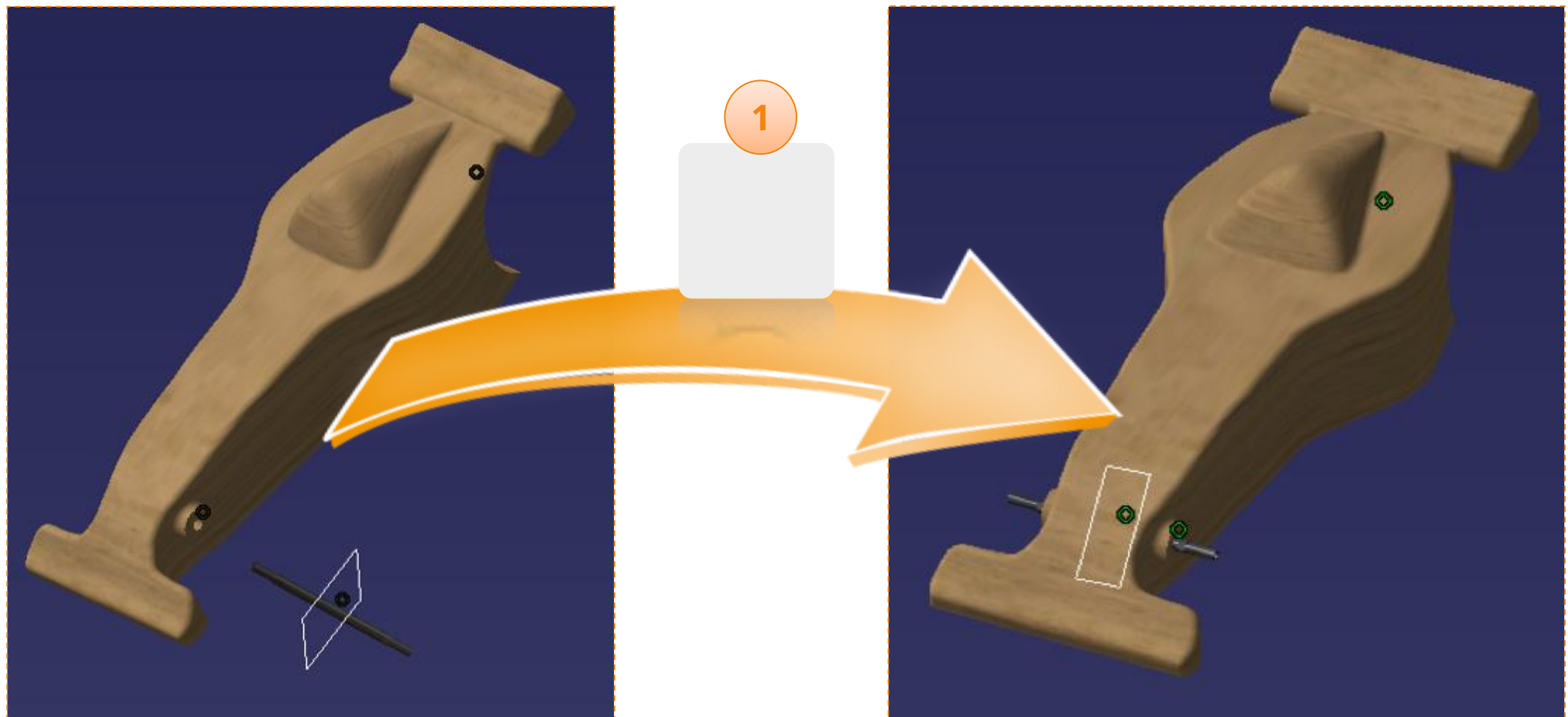
1.Click the icon **Update**

2.You can now co are green, so up to date, and

## Health Existing »

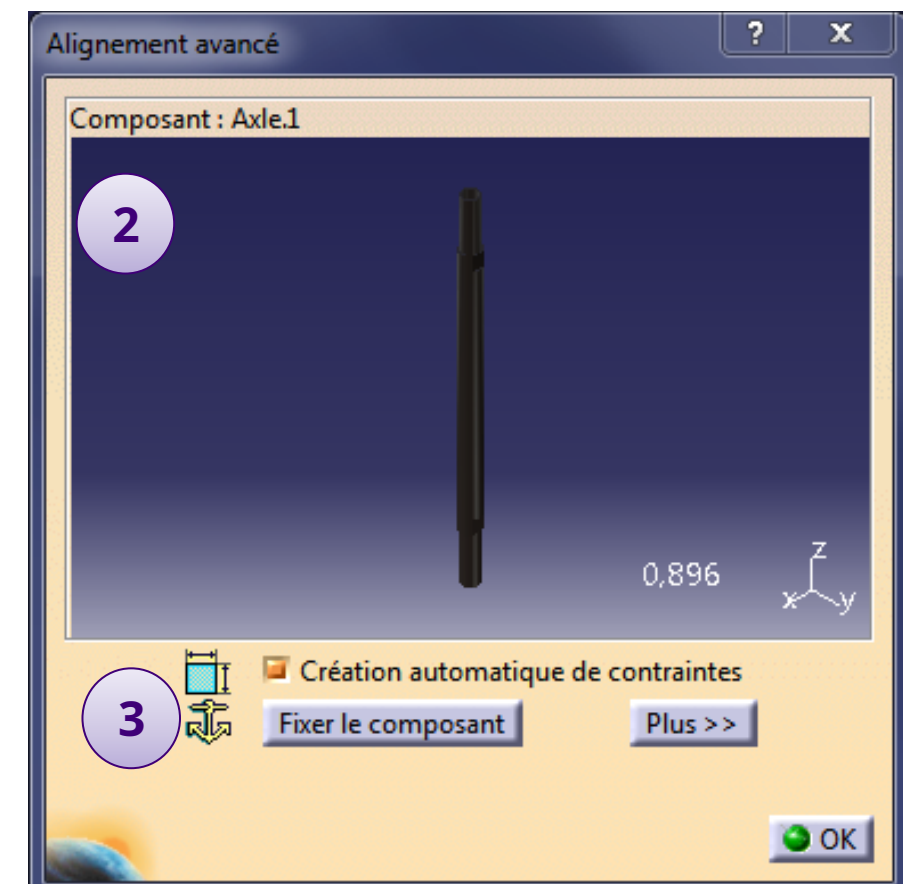
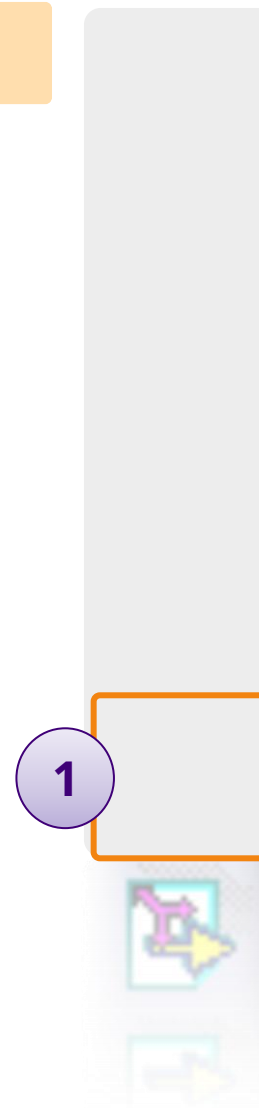
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## Existing component with positioning

- 1 The corresponding icon is accessible when you are in the Assembly Design workbench. By default, it is on the right of your screen.
- 2 This tool allows you to have a view of the part that we are going to insert. This view is handled in the same way as the CATIA environment. Thanks to this view we have easy access to the surfaces of the part without being disturbed by the rest of the assembly.
- 3 The interest of this tool lies in the option of automatic creation of constraints. You will be able to create the constraints with this tool, and it makes the assembly more ergonomic.



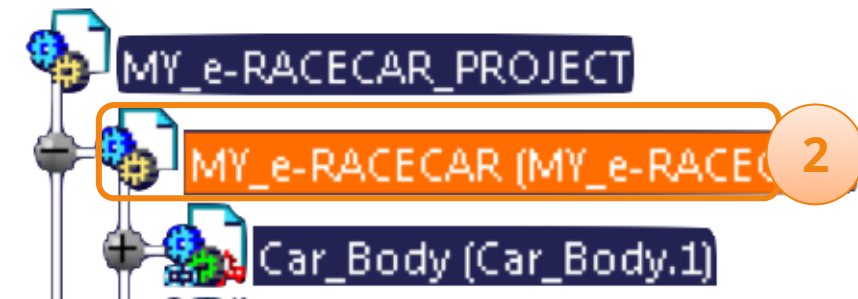
# Insert parts with positioning

We will start by adding the part to the assembly.  
For that :

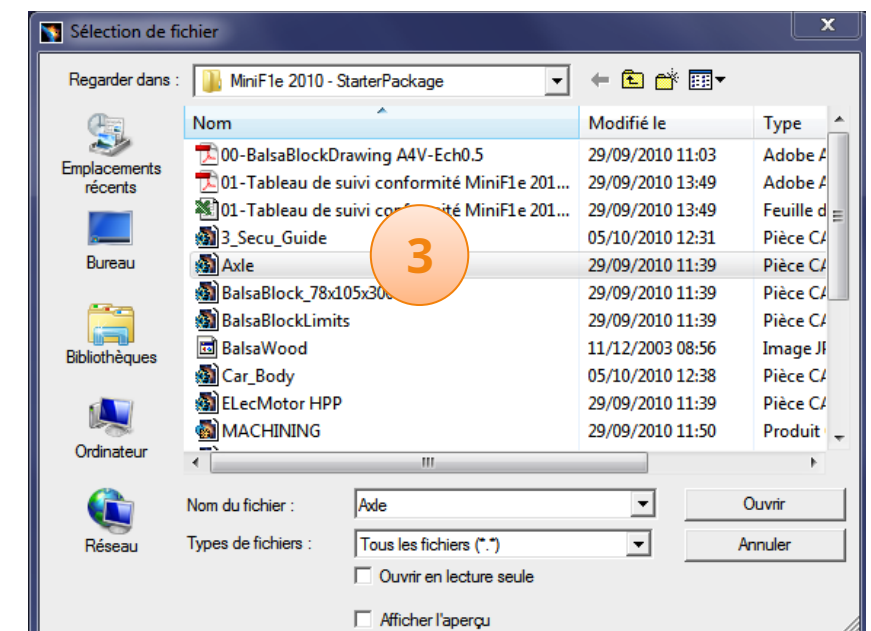
1. Click on **Existing component with positioning**.



2. Click on the product in which you want to add your part.  
Here we will click on **MY\_e-RACECAR**.



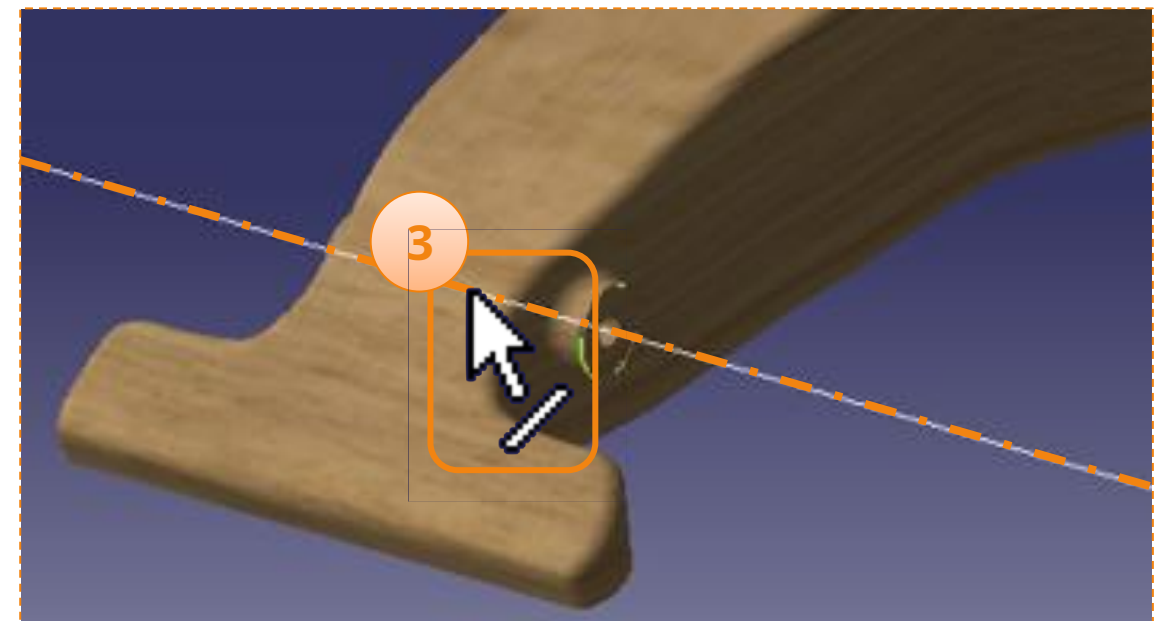
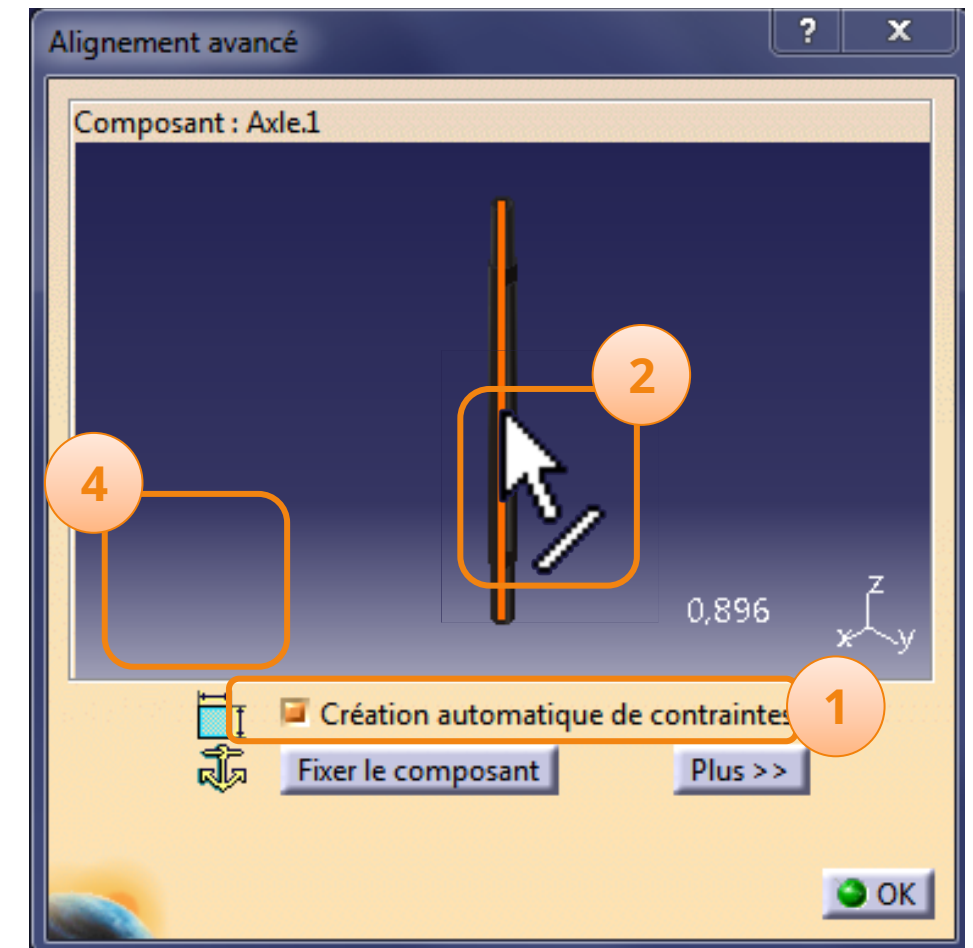
3. A file selection window opens. Select the file you want to insert. Here we will select **Axel**.



# Insert parts with positioning

Now that the inserted part visualization window has appeared, we will start by constraining the bearing axially. For that :

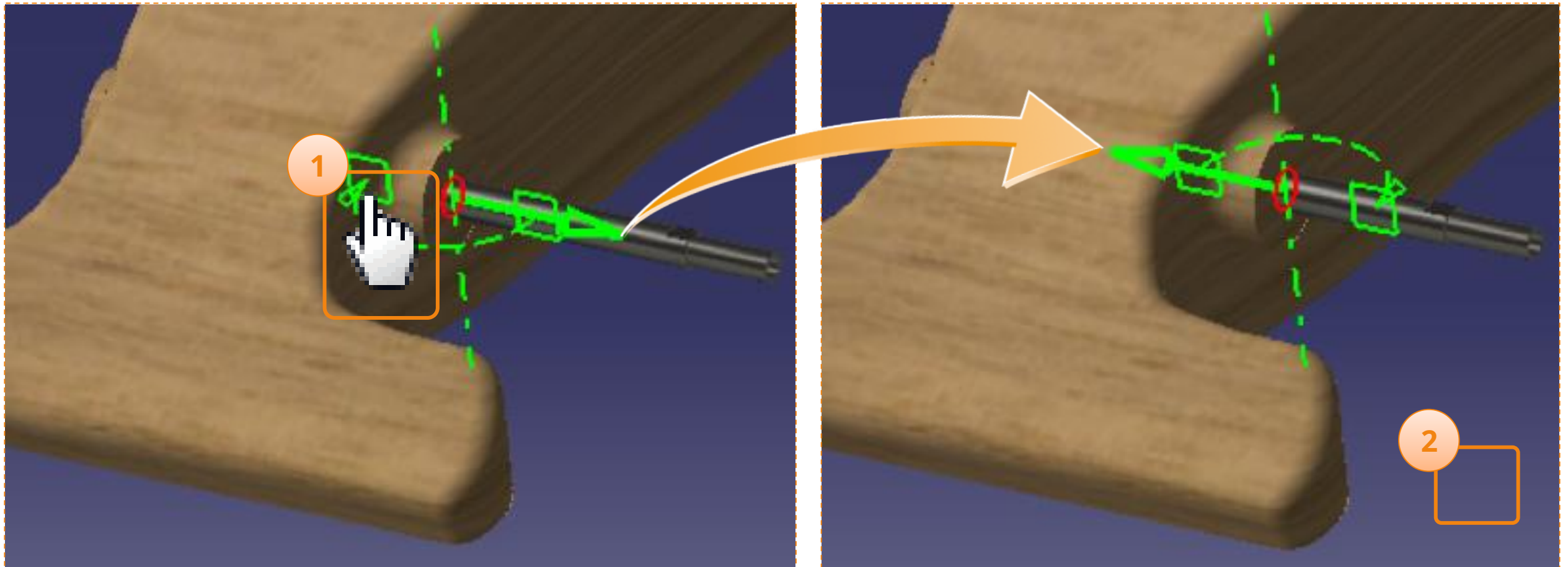
1. Check the automatic creation of constraints option.
2. Click on the axis axis.
3. Click the centerline of the front axle hole on the other side of the car.
4. Then click in a blank area to create the constraint.



## Insert parts with positioning

In some cases, depending on the origin mark of the imported part, its orientation may be reversed. If that is the case :

1. Click on the plane as below to reverse the direction of the room.
2. You can now validate the constraint by clicking in a blank part of the screen.



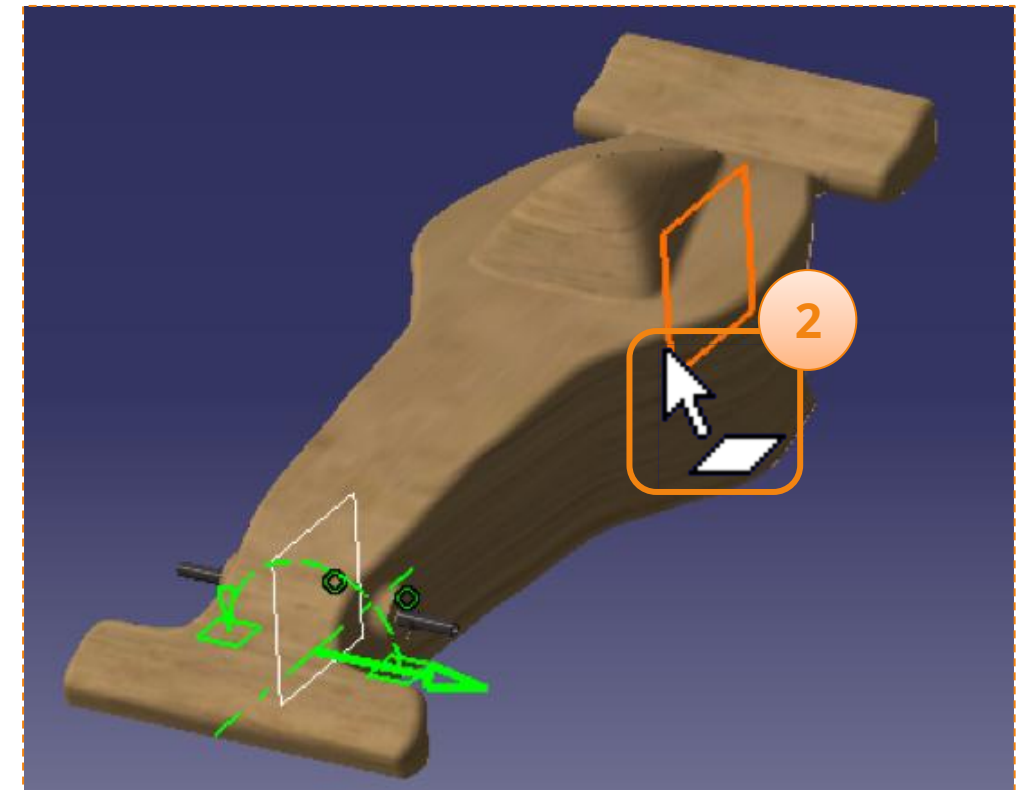
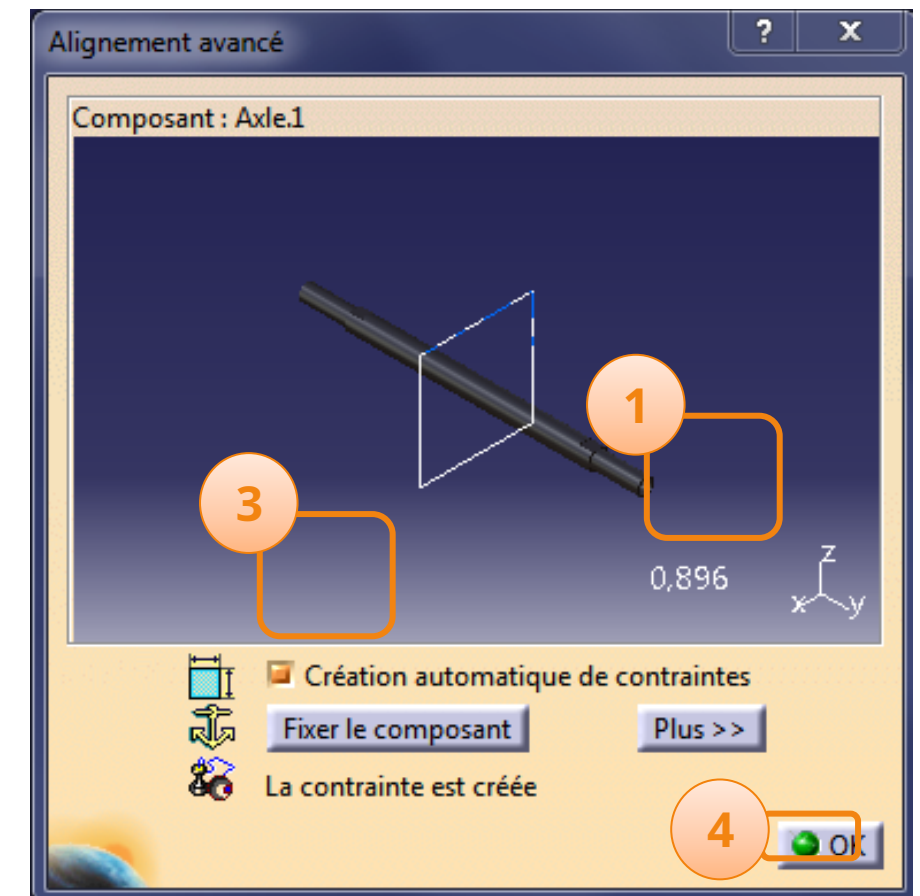


# Insert parts with positioning

Let's continue by centering the axis on the car. For that :

1. Click on the **Map.1** in Axle's tree.
2. Click on the **ZX plane** of the car, common plane of symmetry.
3. Click in the blank part of the viewing window.
4. Click on **okay**.

Congratulation ! You have inserted and constrained the part!



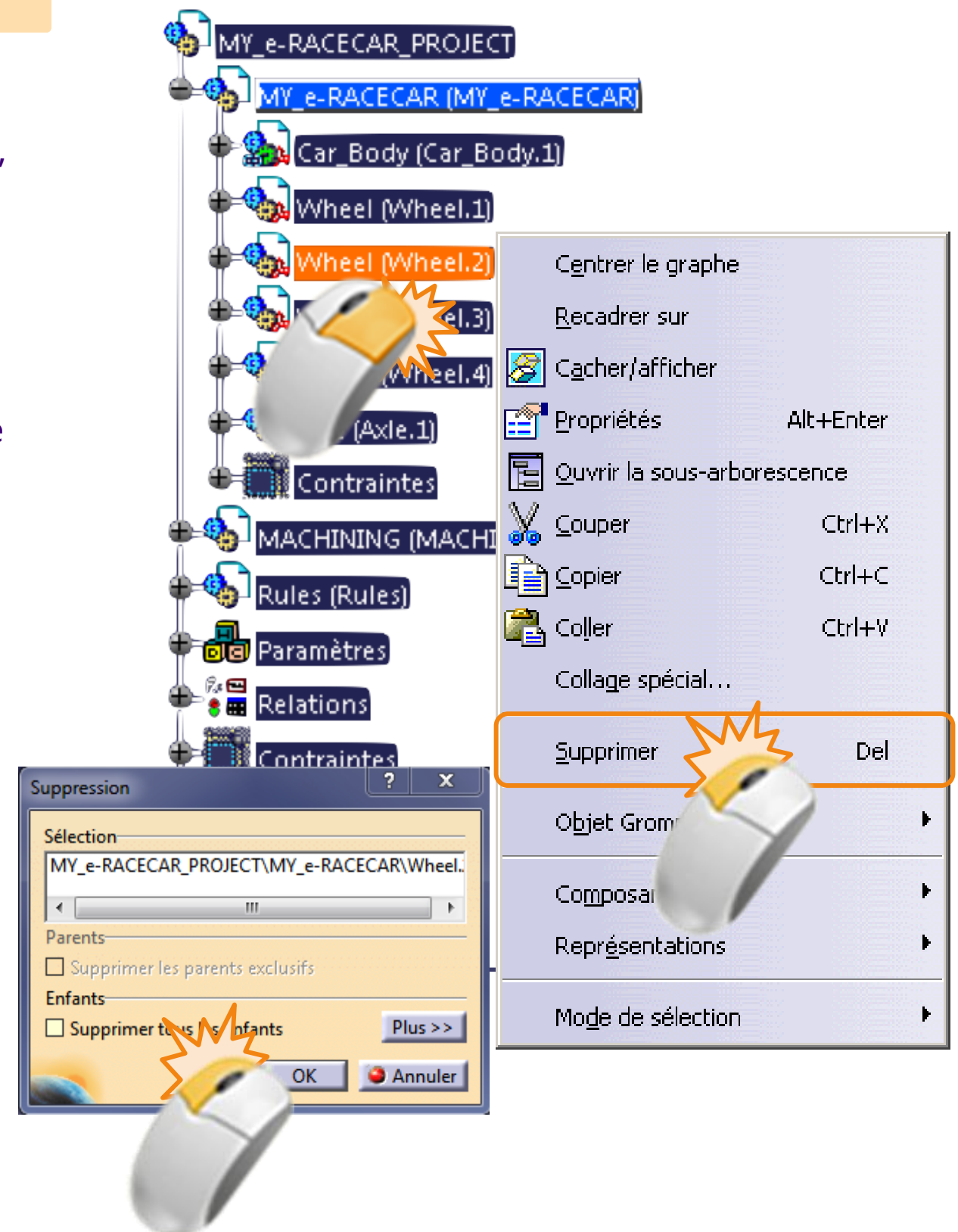
## Delete an item

To remove an element from an assembly, the product in which it is located must be active. If not, double-click it to make it active.

Here the Wheel part we want to remove is included in the product **MY\_RACECAR**. This is the active product because highlighted in blue. We are ready to delete the item.

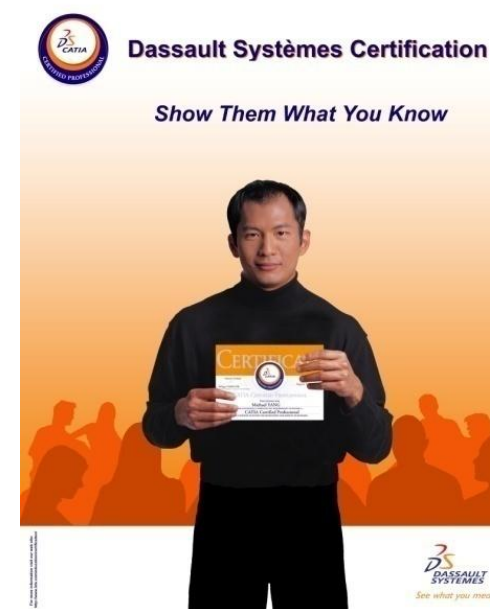
Right-click on the element to delete in the tree then choose **Remove** in the context menu.

In the window that opens, click **okay**.



# Academic Certification Program

- ▶ Dassault Systèmes has set up an academic certification partnership program, which allows students to take the standard CATIA certification exams at a reduced price.
  - ▶ This is a unique opportunity for them to enrich their CV, and thus promote their technical recognition on the industrial market.
- 
- For students, the title of CATIA Certified Professional is an effective way to enhance their skills on DS products and obtain professional recognition, which will allow them to access better job opportunities in a highly competitive industrial world. .
  - Dassault Systèmes Academic Certification partners can also become assessment centers, facilitating registration and organization of exams. It is also a good way to extend your influence and strengthen the recognition of your teaching abilities in academic and industrial circles.



<http://www.3ds.com/en/education/certification/academic-certification/>