RhinoRobot

User Manual

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Getting started

What is RhinoRobot?

RhinoRobot is a simple, easy to use and affordable robotic simulation and programming plugin for Rhinoceros 3D that enables you to:

-Import a custom environment and a custom tool collection
-Manipulate the robot by its TCP
-Record frame positions
-Define custom frames along a curve or points with orientation options
-Simulate the toolpath with different axes configurations
-Run the simulation of a third-party points file
-Perform singularity, accessibility and collision checks
-Fully customize the post-processor
-Export the simulated program to your robot
-Export the simulation as .gltf to be read on any device, including AR/VR

System requirements

Hardware:
-Intel Processor i5 or better
-8 GB or more RAM
-1 GB disk space, SSD disk drive is recommended
-OpenGL 4.1 capable video card recommended with 4GB RAM min
-Internet connection for installation and activation
-Multiple-button mouse with scroll wheel is recommended

Operating systems:
-Windows 10 64 bits and Windows 11, with licensed Rhinoceros 7

Installation

If you don't already have it, download the latest version of Rhinoceros 3D V7 (90-day trial period) here: https://www.rhino3d.com/download/rhino-for-windows/7/evaluation
RhinoRobot5 is not compatible with older versions than the V8.
You are ready to install the latest version of RhinoRobot. You will find it here :
https://www.kinematiq.net/fr/rhinorobot/telechargements/
After downloading the .rhi file just launch it with Rhinoceros 3D closed.

First start

At the end of the installation process, Rhinoceros 3D will start automatically.
If you cannot see the RhinoRobot panel, type "RhinoRobot" in the command line and the panel will appear.
Feel free to contact us if you have any trouble or question.
For a quick start up, go to page 9, "Starting a project".

License

RhinoRobot comes with a 30-day trial license, activation is automatic at first start, after accepting the EULA.
After that you can either geta standard license, that will be activated on a single machine, or a floating license that will activate and deactivate itself with the cloud server automatically.
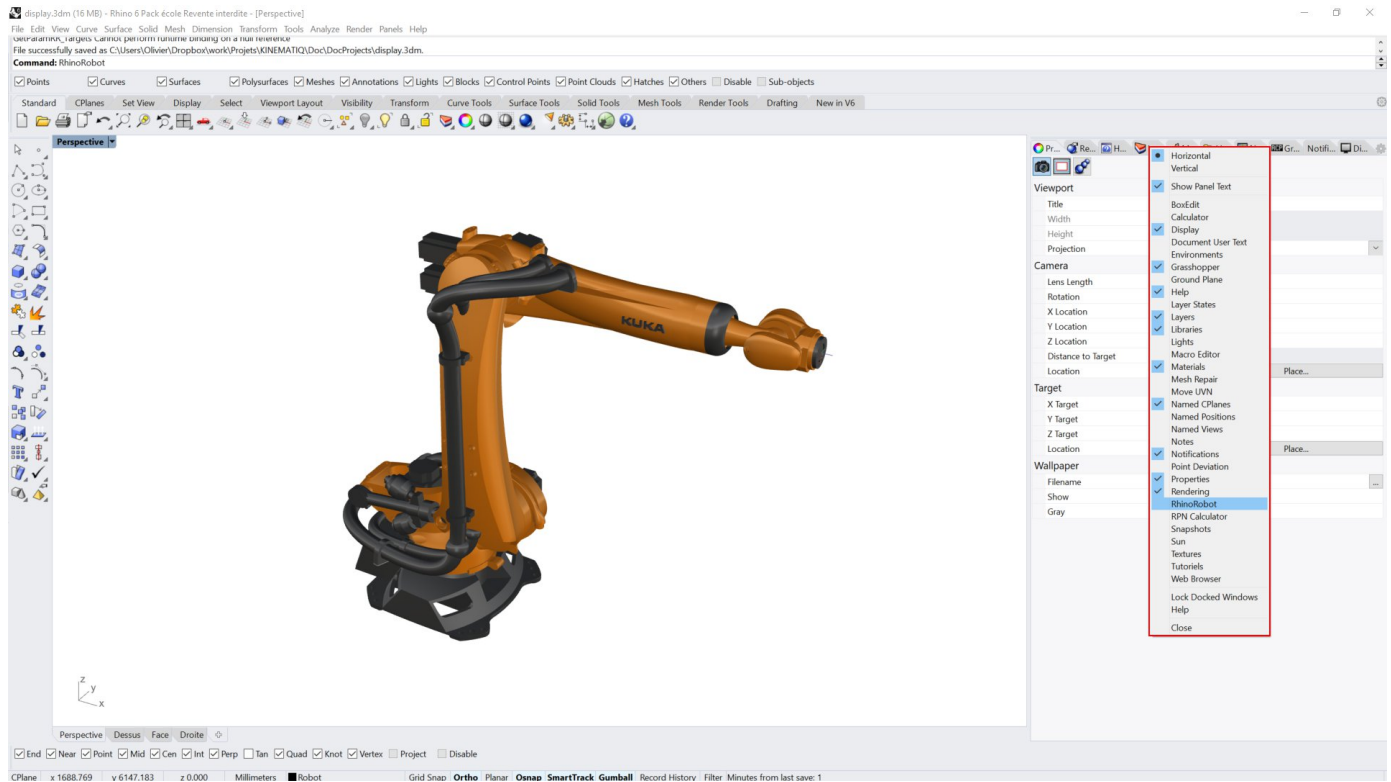
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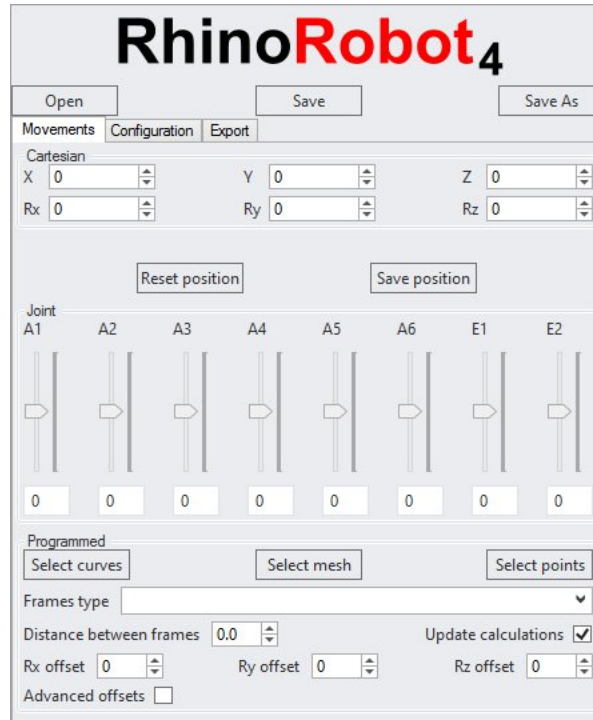## User Interface

Displaying RhinoRobot

The RhinoRobot installation file will install the user interface inside Rhinoceros 3D.
To display the user interface panel, you can either right click in the panels area and check RhinoRobot, or type "RhinoRobot" in the command line.

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The Movements tab

Open: Open a RhinoRobot project or model
Save: Save a RhinoRobot project
Save As: Save the RhinoRobot project with a different name

Cartesian
X, Y, Z: fields to enter Cartesian coordinates of the TCP
Rx, Ry, Rz: fields to enter Roll, Pitch and Yaw rotations of the TCP (in degrees)

Reset position: reset the position of the robot
Save position: save the position of the robot as a CPlane, it will be stored in the 'Named CPlanes' panel of Rhinoceros, (see the Record Position Section page 22) and can be used to build a toolpath.

Joint
A1, A2, A3, A4, A5, A6: fields to enter articular positions
E1, E2: fields to enter external axis positions

Programmed
Select curves: Select curves to program targets on
Select mesh: Select a mesh to program targets orientation on
Select points: Select points to program targets on
Frames type: Select the type of frame to use
Distance between frames: Distance between frames along the curve (in mm)
Update calculations: Disable the calculations of the inverse kinematics if needed
Rx, Ry, Rz offset: fields to enter a rotation offset of the frames (in degrees)
Advanced offsets: Apply offset to the toolpath with more option. See dedicated section

Play/Pause: Play and pause the simulation at the theoretical speed set in the Export tab

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The Configuration tab
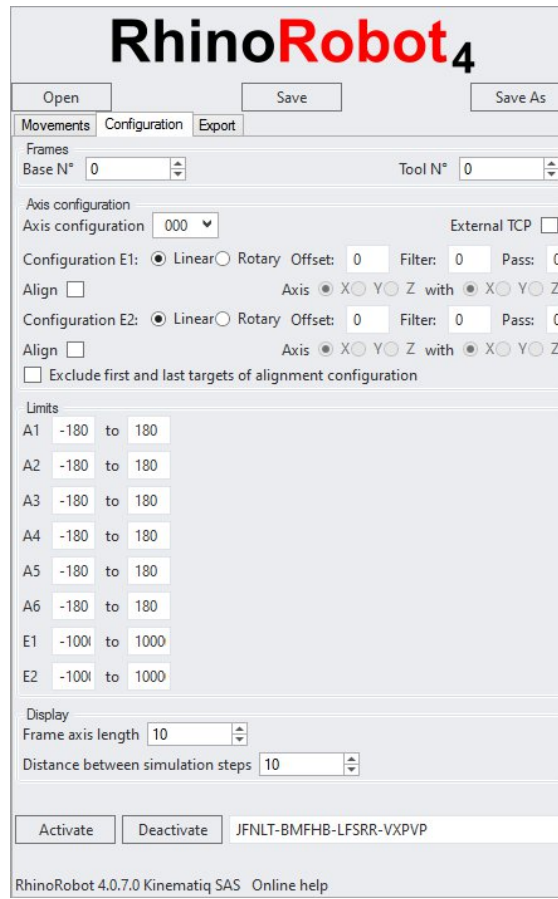
Frames
Base N°: field to select and display the preset base you want to work with
Tool N°: field to select and display the preset tool you want to work with

Axis configuration
Axis configuration: Choose the robot axis configuration
Configuration E1, E2: Select if you want a linear or rotary configuration and if the target axis must be continuously aligned with the CPlane axis.
Offsets and filters can be applied if the axis is configured aligned
First and last targets can be excluded from the aligned calculation
External TCP: Flag to set the robot motions as referring to an external TCP

Limits
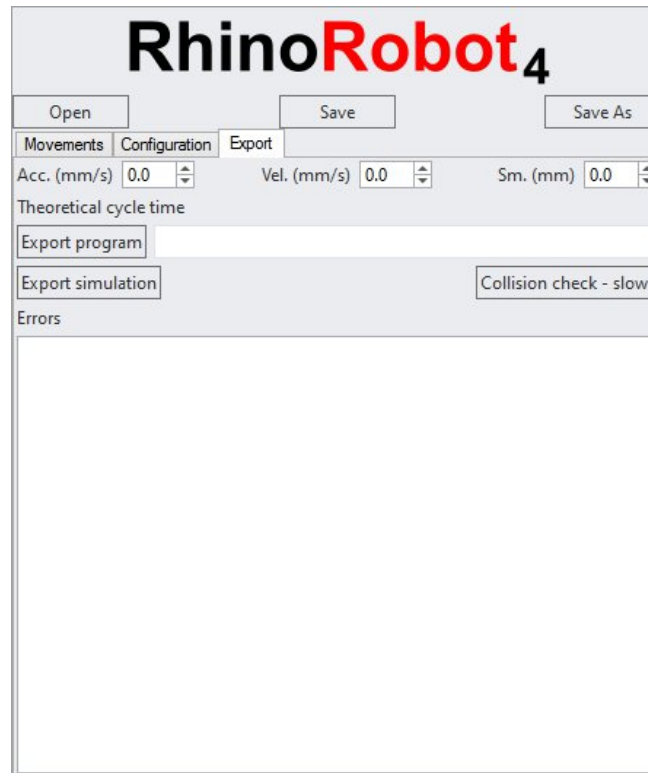A1, A2, A3, A4, A5, A6, E1, E2: fields to enter the limits of your axis (in degrees)

Display
Frame Axis length: field to enter the length of the displayed axis frames
Simulation steps: distance between each step displayed between two programmed targets

License can be activated and deactivated on demand by the user, to change computer for example

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Acc: field to set the acceleration (mm/s)
Vel: field to set the velocity (mm/s)
Sm: field to set the smoothing (mm/s)

Export program: Export the robot program
Export File path: Set the file path for the robot program export
Export simulation: Export the simulation as a .gltf file to be read on any device, including VR/AR
Collision check: performs a collision check between the tool and the robot, external axis, added objects
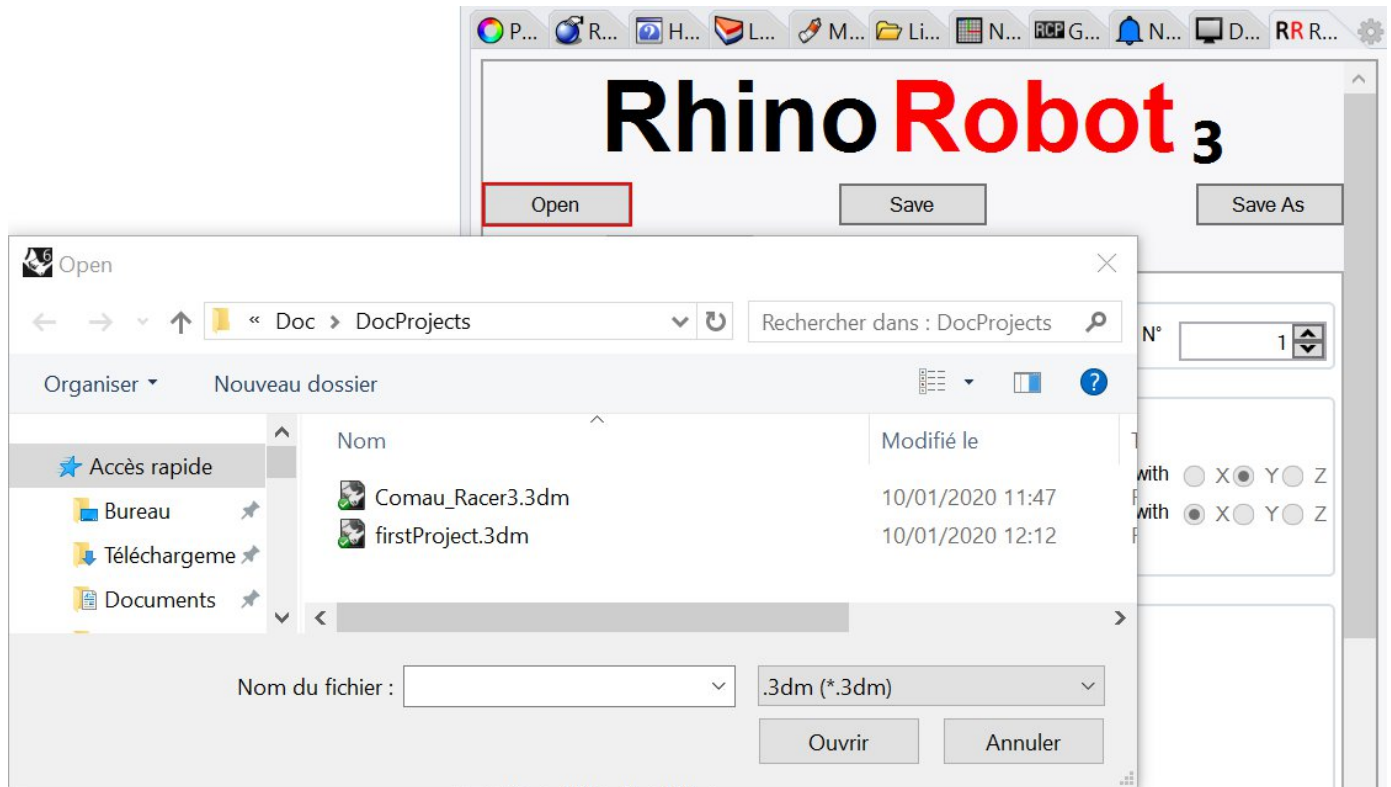Error field: Displays a list of errors (singularities, axis limits, distance limits, collisions…)
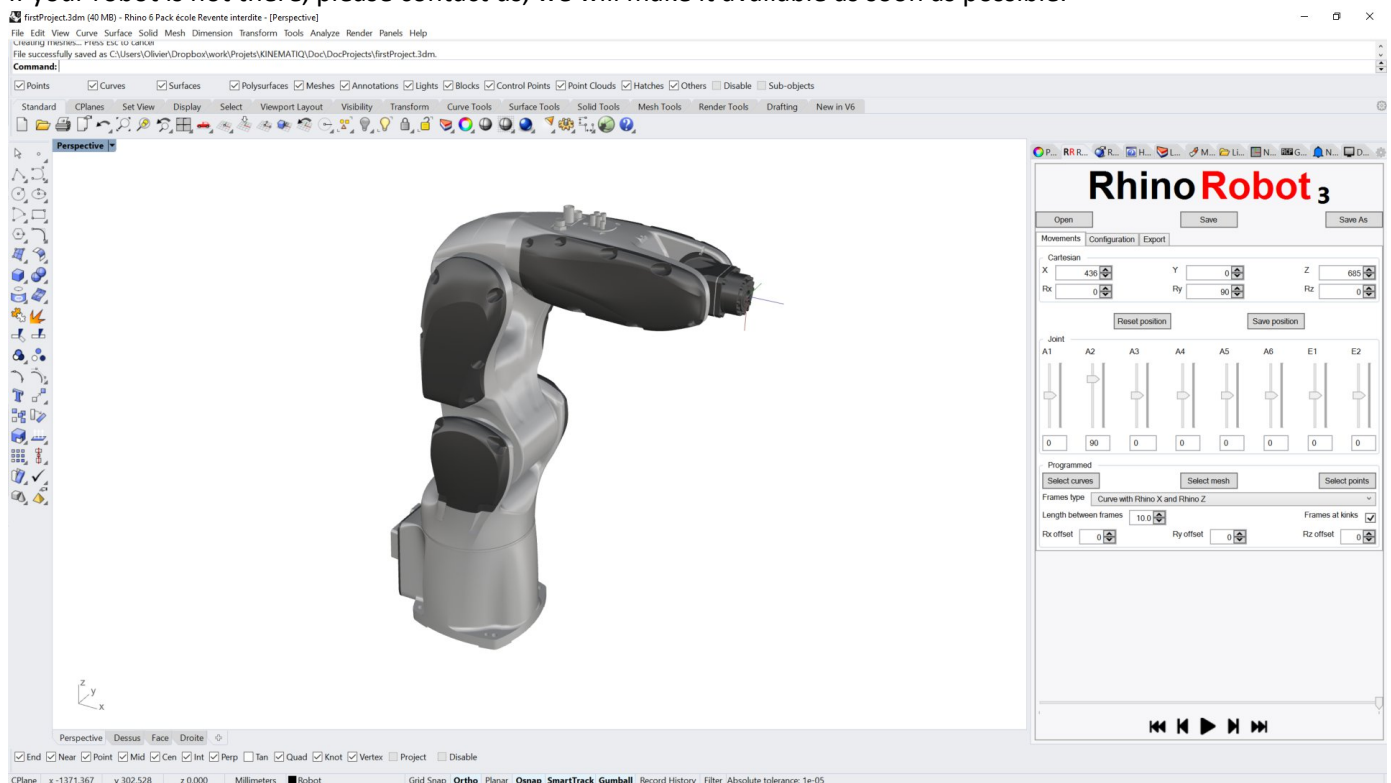
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Starting a project

Load a robot

When Rhinoceros is launched, first click on the RhinoRobot panel or type "RhinoRobot" in the command line. In RhinoRobot, Open the ready-made .3dm file of your robot.

Robot files are available here https://www.kinematiq.net/rhinorobot/telechargement/
If your robot is not there, please contact us, we will make it available as soon as possible.

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Click the Save As button in the RhinoRobot panel and choose a name for your project.

Be sure to **always** open and interact with a project through the "Open" and "Save" buttons of the RhinoRobot Panel.

Now if you enter Cartesian values or modify the Joint values in the Movement tab of RhinoRobot, the robot will move accordingly.
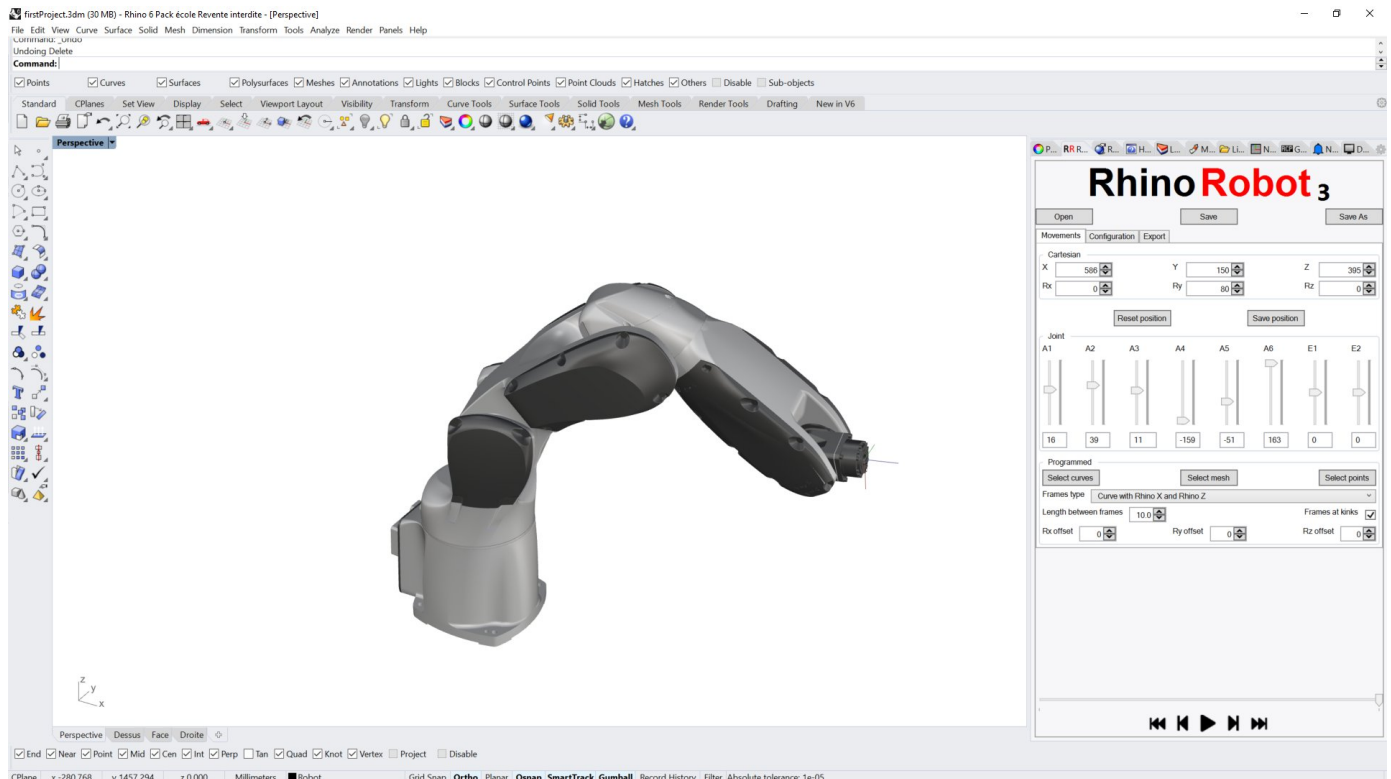
When you save your first project, two files are created, a Rhinoceros .3dm file that contains the geometries and a Grasshopper .gh file that contains the programming part. Those two files are connected through their name.

You should **always** open and save the Rhinoceros .3dm file via the RhinoRobot Panel so that its Grasshopper counterpart can be opened at the same time and be linked with it. If you click on the Grasshopper icon or type "grasshopper" in the command line, the already opened .gh file will be displayed.

Any robot environment can be added to build a full robot cell. Simply import any supported file format into Rhinoceros:

.3dm .3dmbak .rws .3mf .3ds .amf .sat .ai .dwg .dxf .dae .cd .x .e57 .dst, .exp .emf .eps .off .gf, .gft .pm .kmz .gts .igs, .i

ges .lwo .dgn .udo .fbx .

scn .obj .csv .x_t .pdf .ply .asc,

.csv, .xyz, .cgo_ascii,.cgo_asci, .pts .txt .raw .m .rib .svg .skp .slc .sldprt, .sldasm .stp, .step .stl .vda .wrl, .vrml, .vi .gdf .wmf .x3dv .xaml .xgl .zpr
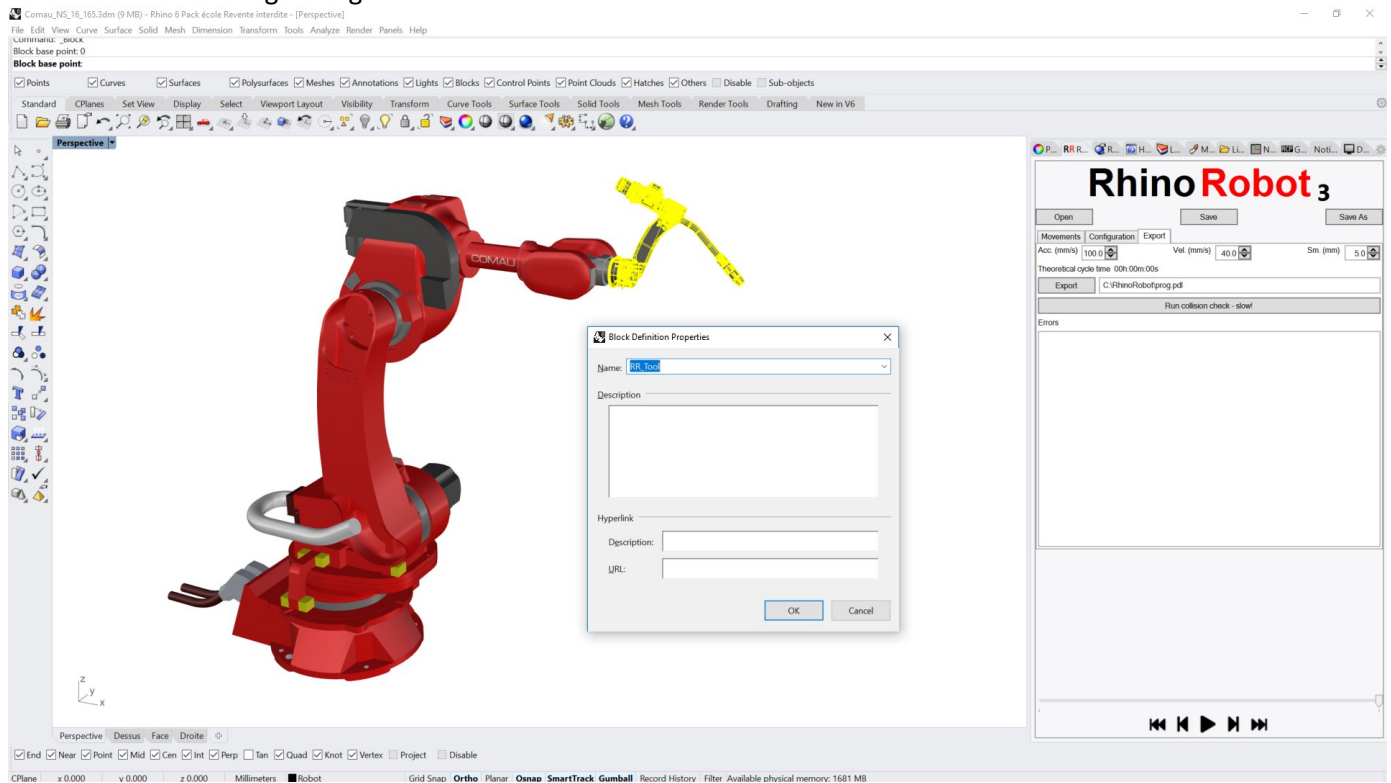
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Setting up a tool

First make sure that the current CPlane is the Rhino World CPlane

To set up a tool, first import in Rhino the geometry you want to use as a tool. You can either open two sessions of Rhino and copy and paste geometries or import as a block and then explode it.
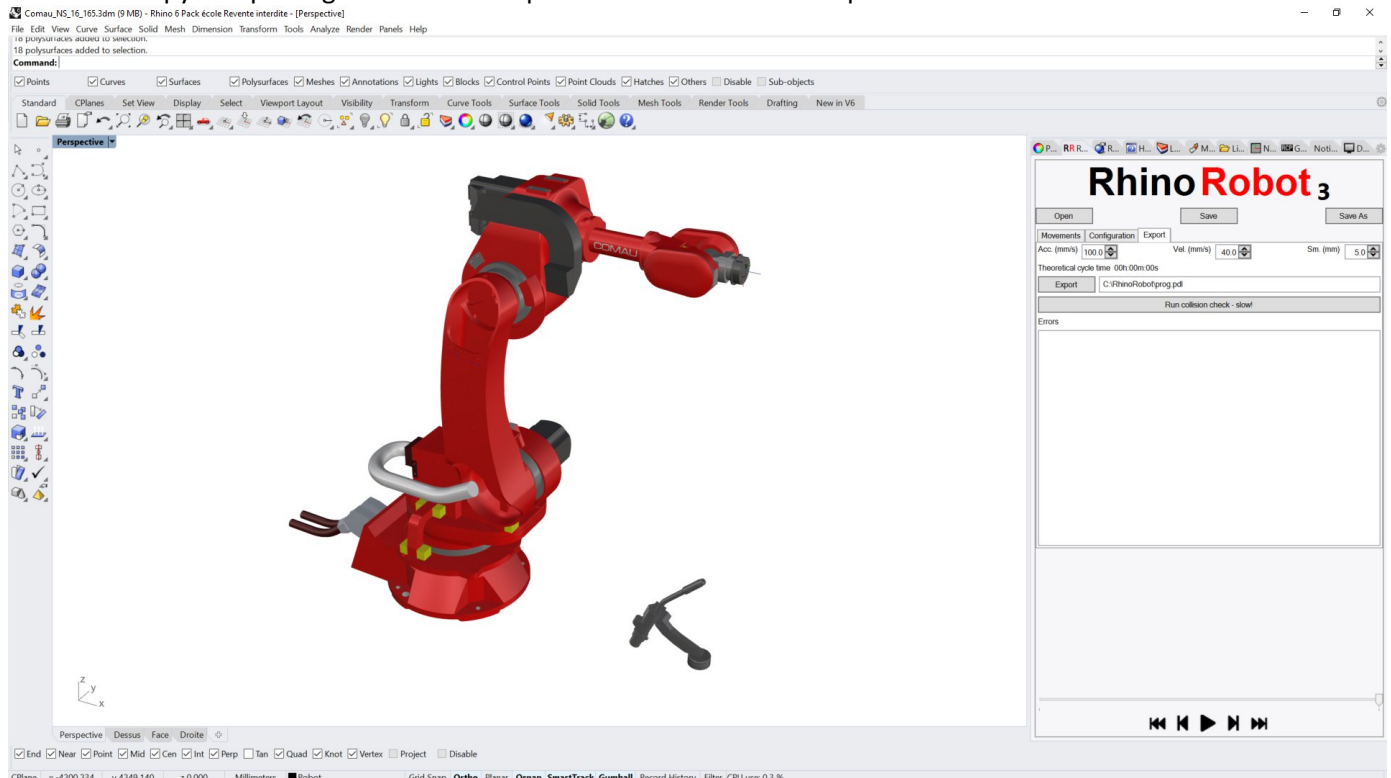
Make sure the robot is at his default angle by clicking on the reset position in the RhinoRobot panel, then place the tool on the robot flange using the Rhino "orient" command.


Once it is in place, select the tool geometries and type "block" in the command line, type 0 and press Enter to set (Rhino World 0) as the base point and name your block "RR_Tool", replacing the default existing block.
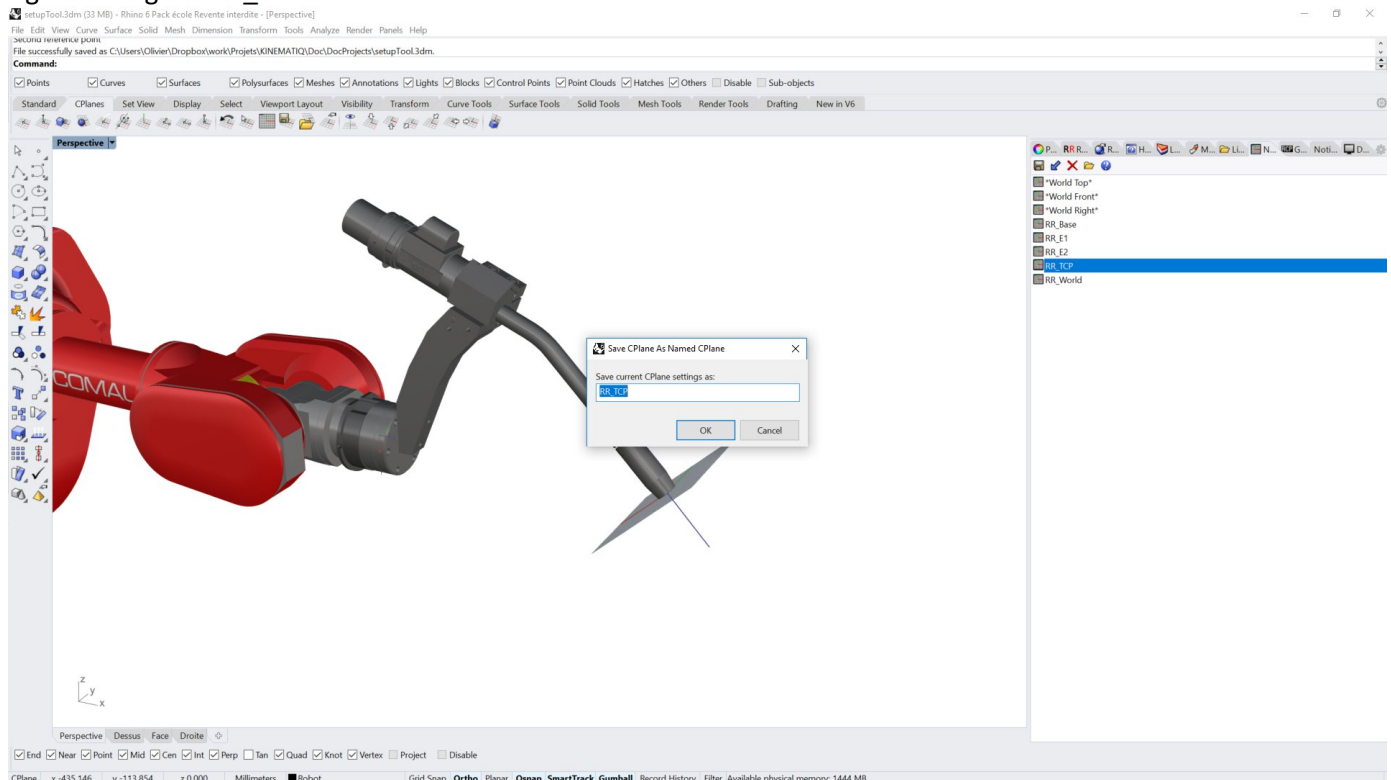
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Now you have to set up the TCP plane, right click on the RhinoRobot panel tab and make sure "Named CPlanes" is checked, then click on that tab and select the already existing RR_TCP plane.

Type "CPlane" in the command line and place the plane where you want it to be on your TCP.
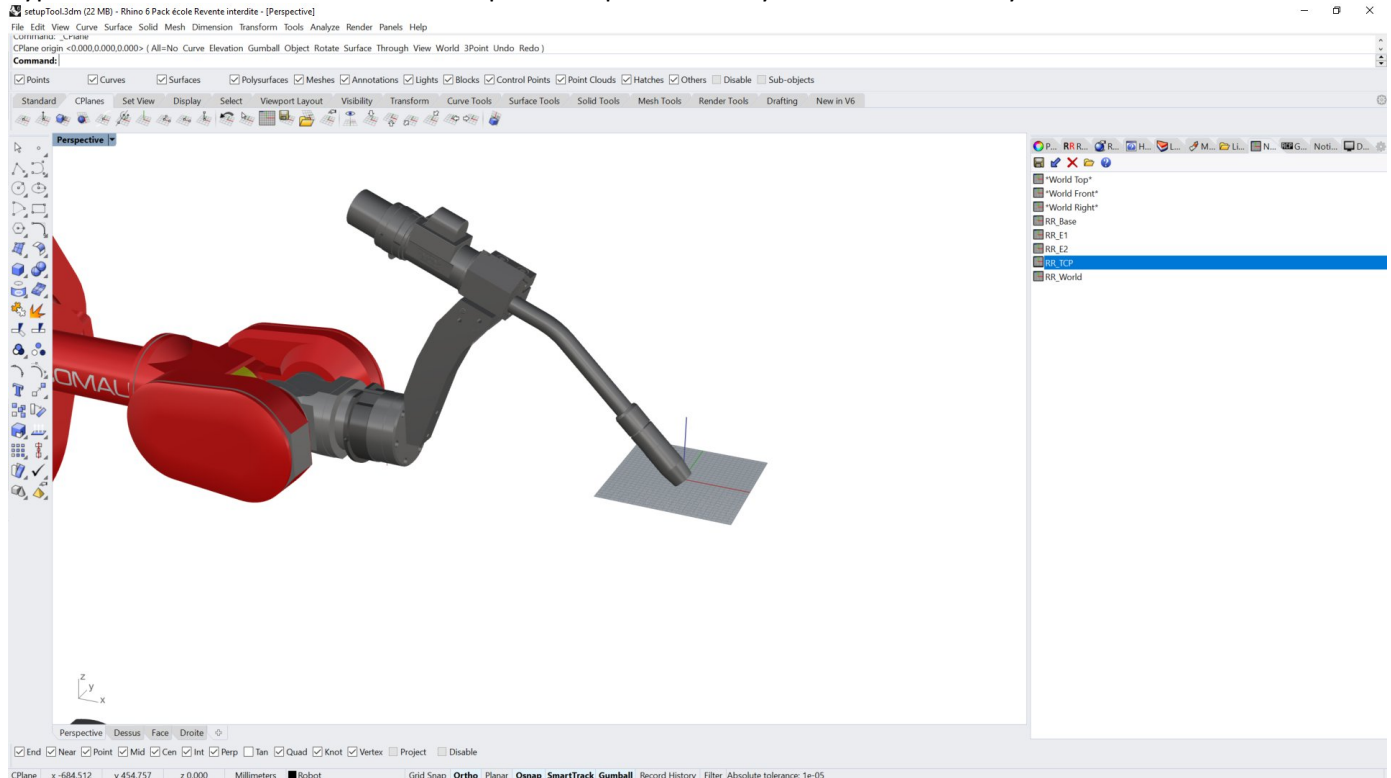
Orient it the way you want with the rotate CPlane command in the CPlane tab. Lastly save the new TCP CPlane by right clicking the RR_TCP in the list of the Named CPlanes and select "Save current CPlane as named CPlane".


Save the project, the tool is now attached and moves as you move the robot.

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Setting up multiple tools and bases

You may repeat the above process (setting up a tool) to add multiple tools, just increment the name of the new tool: RR_Tool_1, RR_Tool_2, … And then add the corresponding named TCP CPlanes: RR_TCP_1, RR_TCP_2, …

A base is already set up in the robot file you downloaded from us, to change it or add multiple bases, follow the same process as for the tools, just name your blocks RR_Base, and then RR_Base_1, RR_Base_2,… and define their corresponding named CPlanes RR_Base, RR_Base_1, RR_Base_2, …
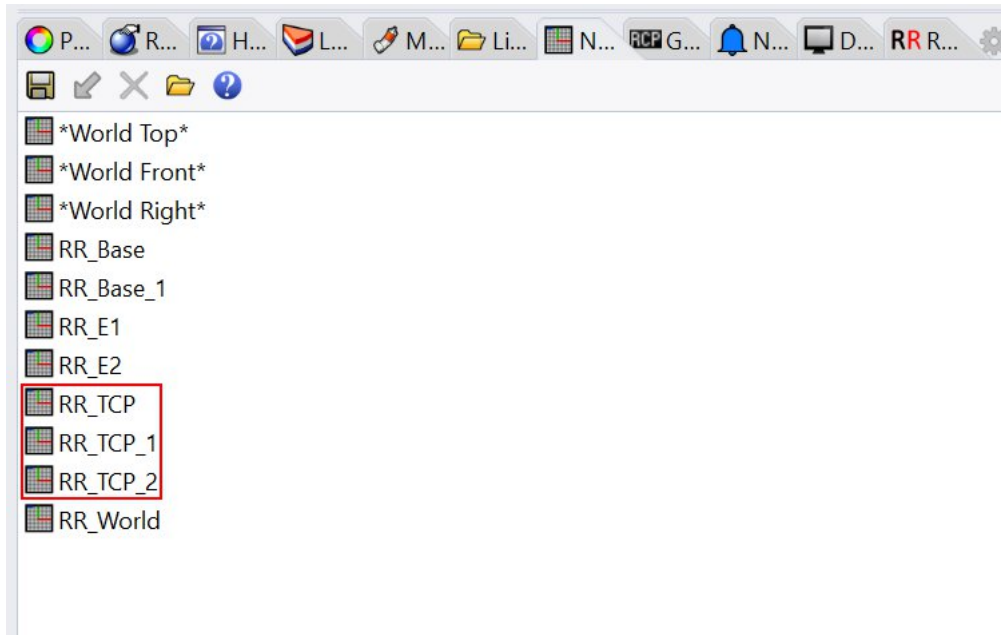
You will be able to specify the tool and the base you want to work with inside the "Frames" part of the Configuration tab.

In Grasshopper, you can get the tool and base number by using two Integer variables named "RR_ToolNbr" and "RR_BaseNbr". They will be automatically affected when you change those numbers from the RhinoRobot interface.

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Setting up external axes
First make sure that the current CPlane is the Rhino World CPlane

Please repeat the general process of the previous section (setting up a tool): import your geometries, set them in place and then create a block with (Rhino World 0) as the base point. Name it appropriately (RR_E1 or RR_E2 more coming on this subject), set up a dedicated named CPlane RR_E1 or RR_E2.

You can configure one or two external axes, which can be linear or rotary.
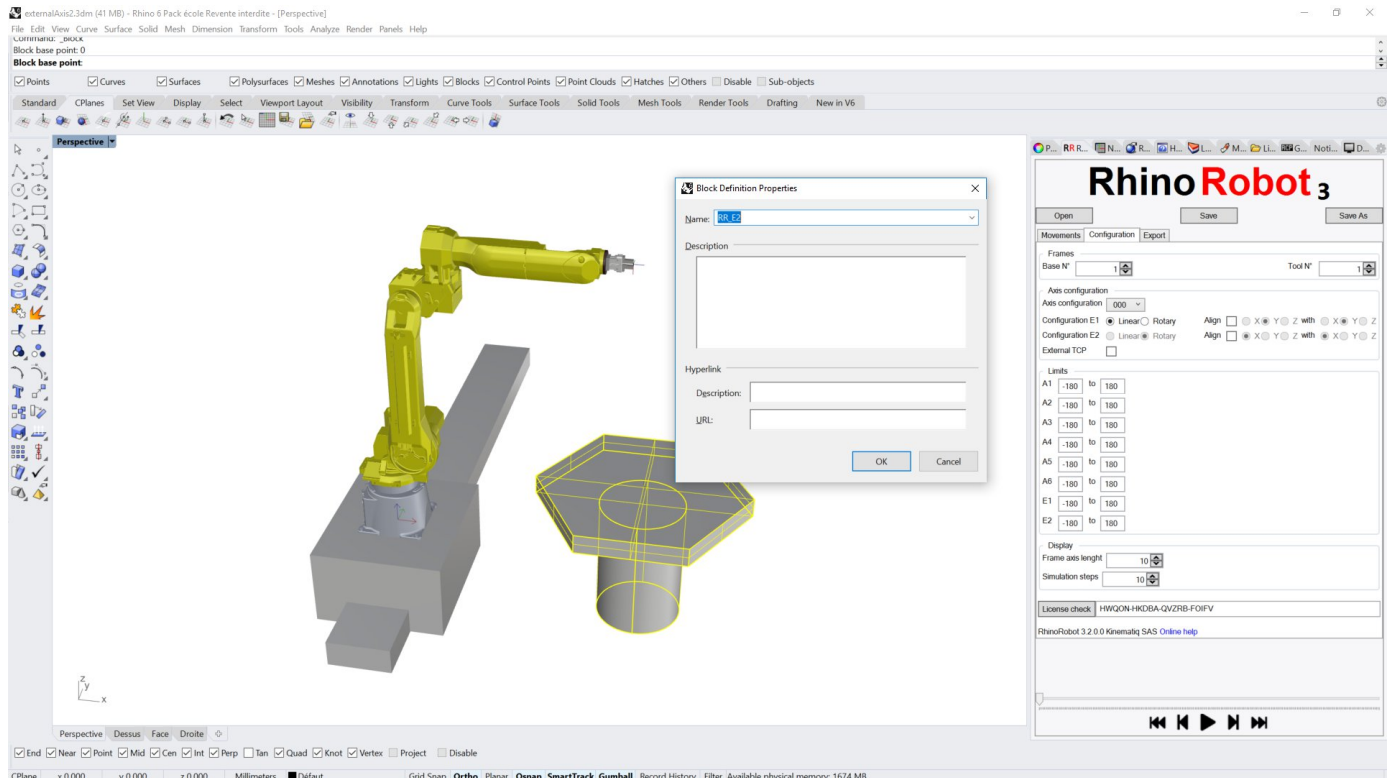
Linear axis
The linear axis will move along the y axis (green) direction of the named CPlane RR_E1 or RR_E2.
Those CPlane must be placed at Rhino World 0, and only the direction of the Y axis will matter.
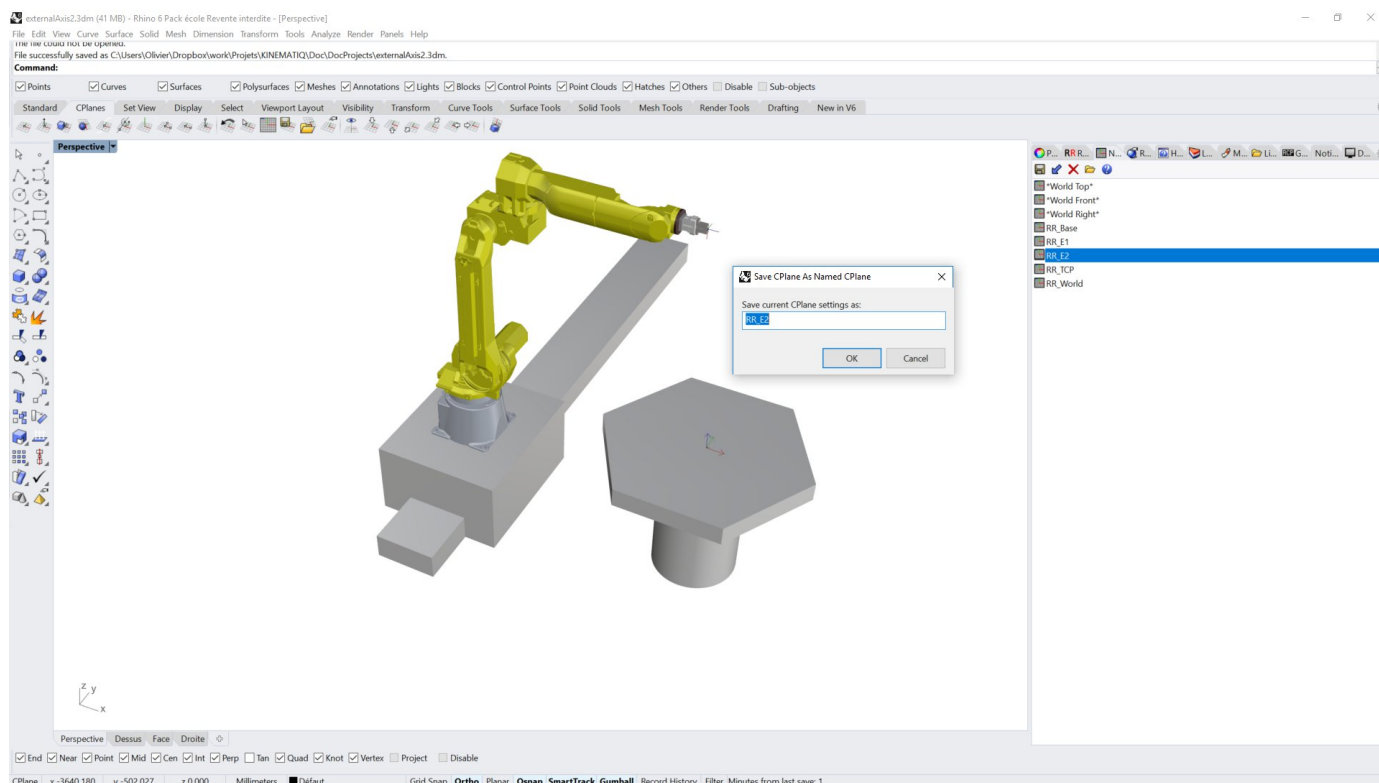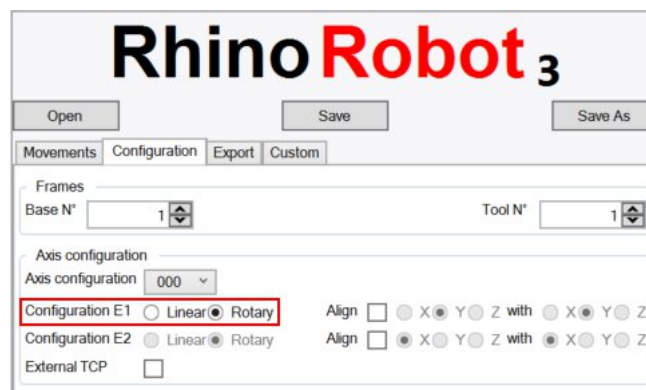
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Rotary axis

The rotary axis must rotate around the Z axis (blue) of the named CPlane.

The rotary axis can be named E1 or E2, if you name it E1 you have to specify that E1 is rotary in the configuration tab of the RhinoRobot panel.

External axis configuration

External axis can be setup as indexed, in that case the linear or angle value will be setup from RhinoRobot first panel, or they can be setup as align/synchronize with the toolpath.

If an external axis is setup as align, the linear or angle value will be automatically calculated to follow the TCP along the toolpath, with the chosen offset.

You can also add a filter value, that will filter the movement of the external axis to snap the value at certain positions.
If a filter is applied, you can also add smoothing pass to soften or harden the snapping to certain values.
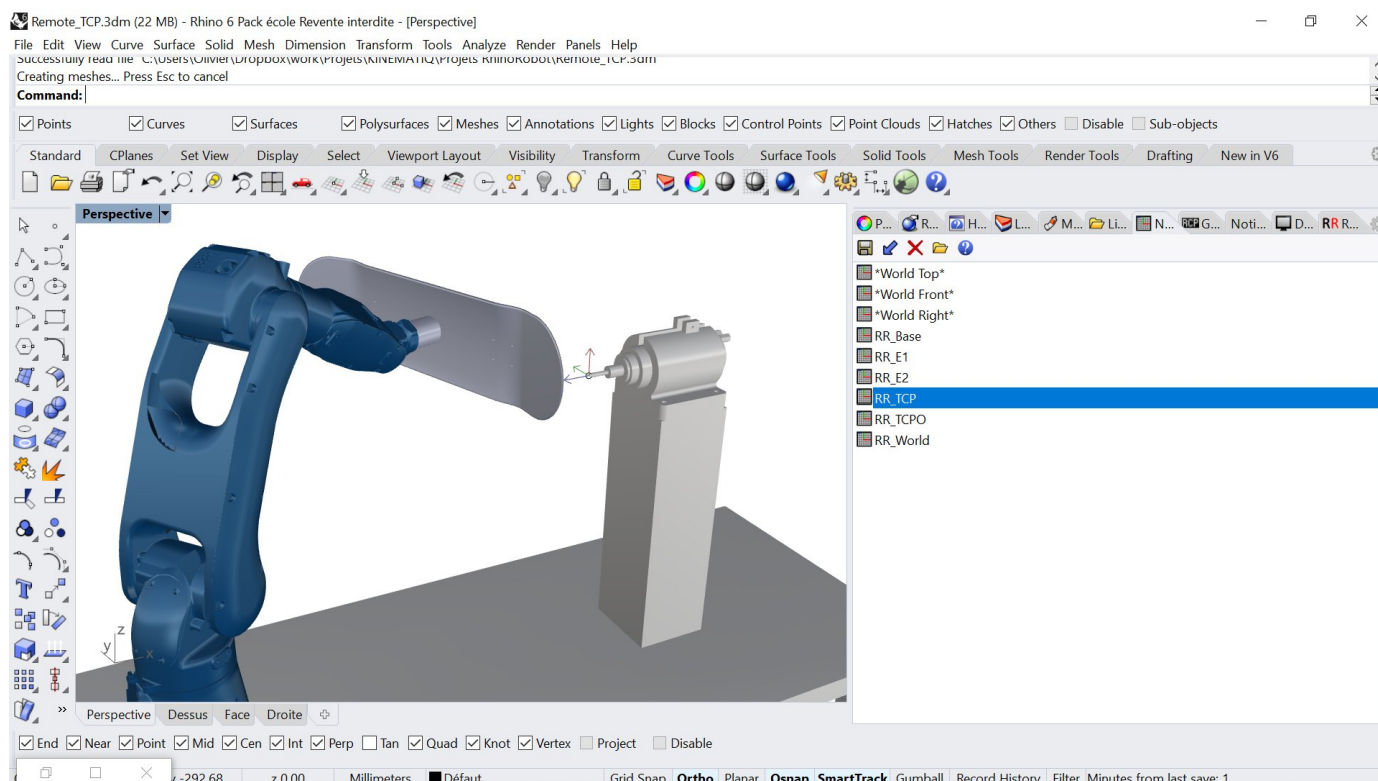
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Setting up a remote TCP

First check the External TCP checkbox in the configuration tab of the RhinoRobot panel.

Then follow the general process of the 'setting up a tool' section: import your geometries, set them in place and create a block with (Rhino World 0) as the base point. Name it accordingly (RR_Tool).
What changes is that the CPlane RR_Tool you will set is not on the RR_Tool Rhino Block but at the location of your choosing (your remote tool).
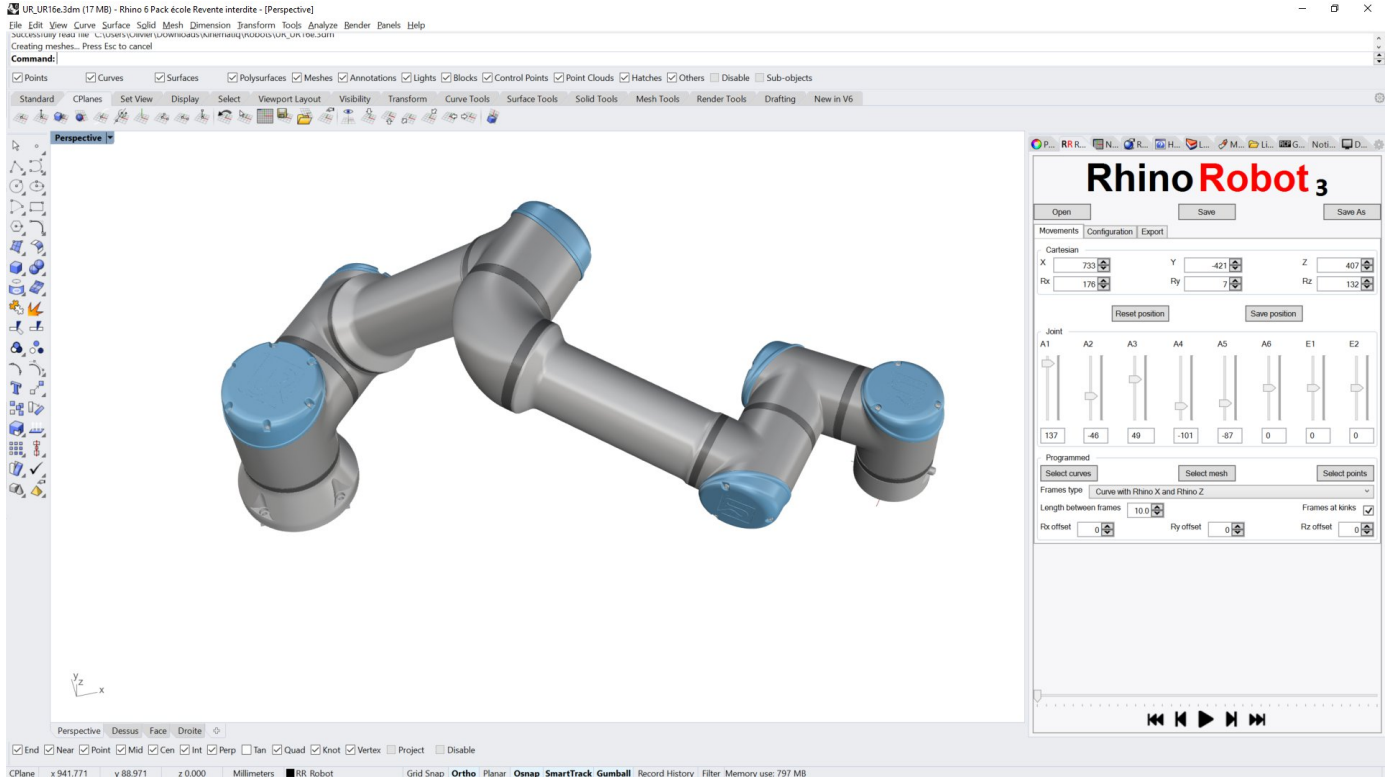

All the frames of the kinematic are calculated by reference to the robot flange, so you have to set up the curves, mesh or points in place on the tool with the robot in his default position.

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Moving the Robot

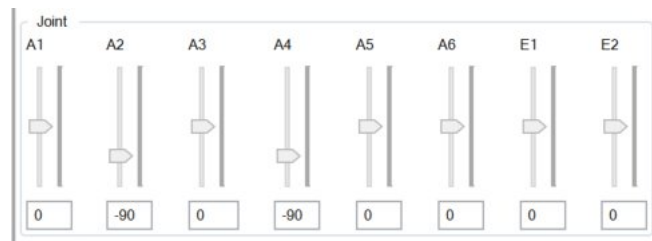There are three ways to move the robot.

Cartesian movement
You can enter Cartesian coordinates as well as Roll, Pitch and Yaw.
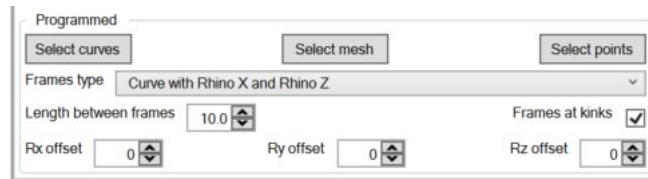

Articular movement
You can interact with the Joint handles or enter articular values.
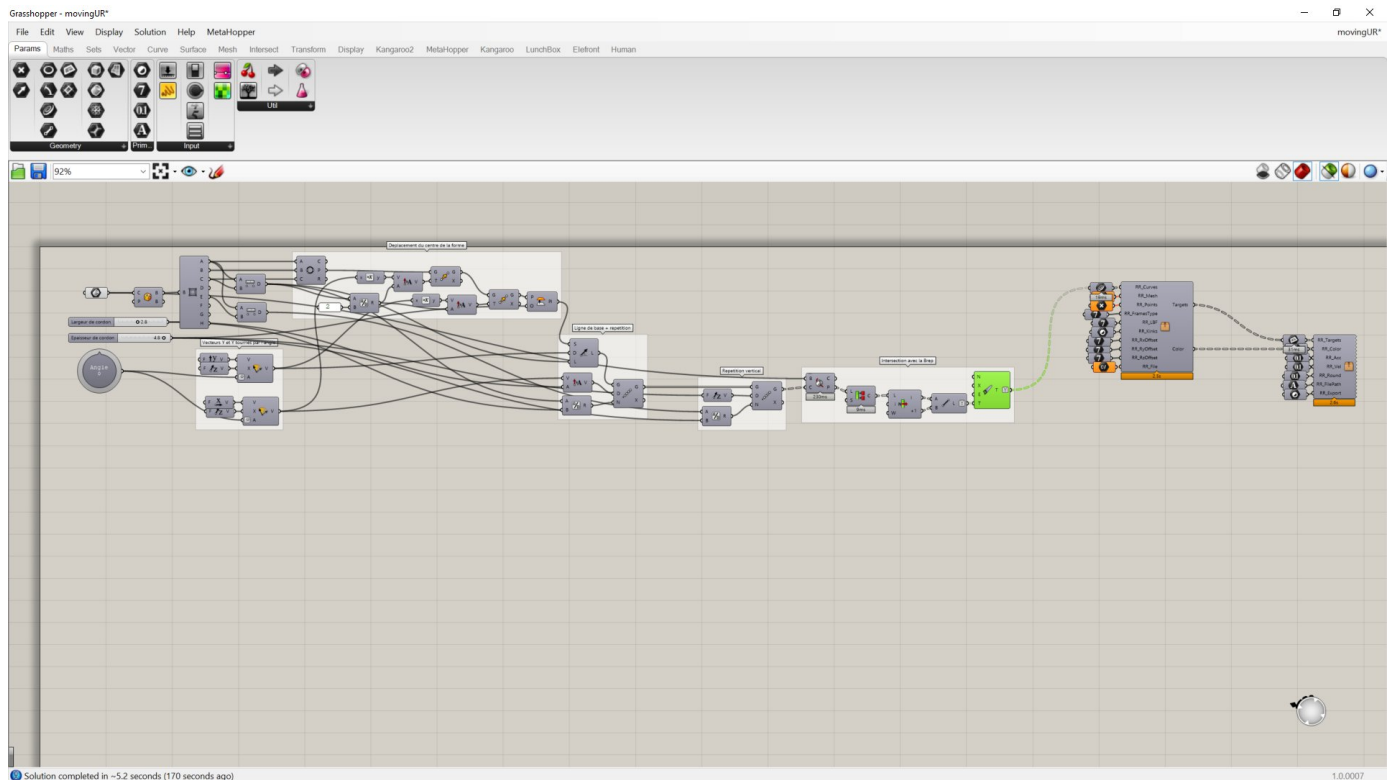
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Programmed movement

You can program a geometry-based toolpath in the program section. This is covered in detail in the following 'Programming a toolpath' section.

Programming a toolpath

RhinoRobot consists of a Grasshopper program interacting with a Rhinoceros 3D interface. You can therefore very easily integrate any Grasshopper definition to it and generate your custom toolpath. See 'Anatomy of your grasshopper project file' for more information.
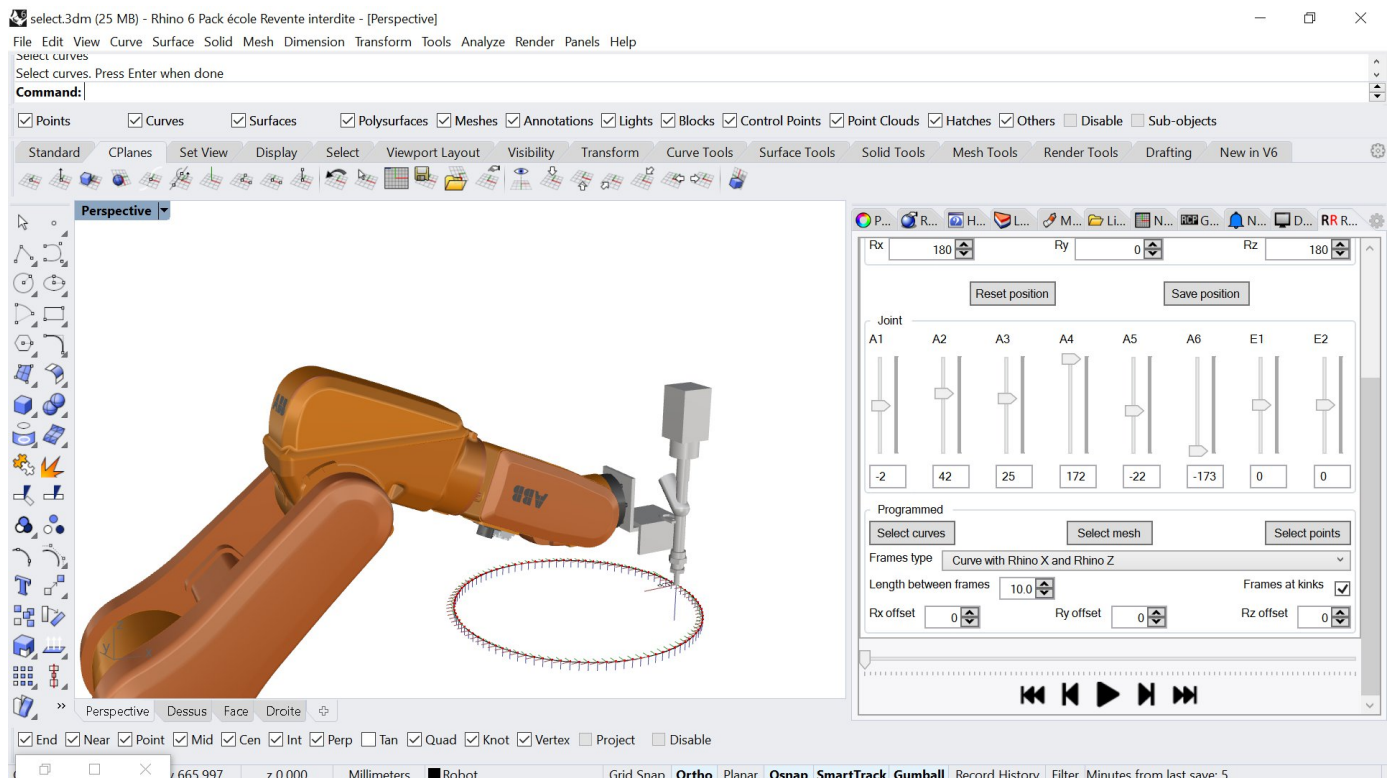
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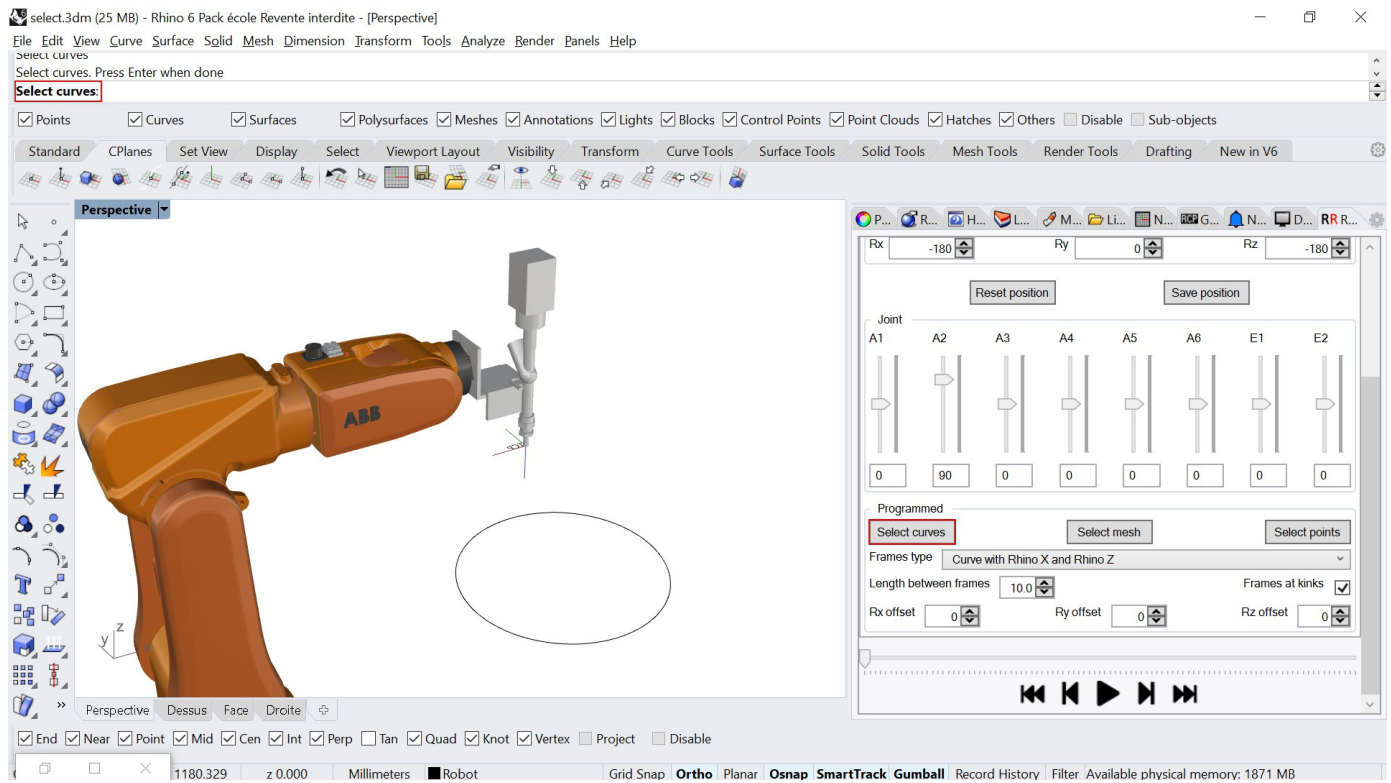
Selections

Selecting curves

RhinoRobot will generate a toolpath based on the curve(s) you select, and optionally a supporting mesh.
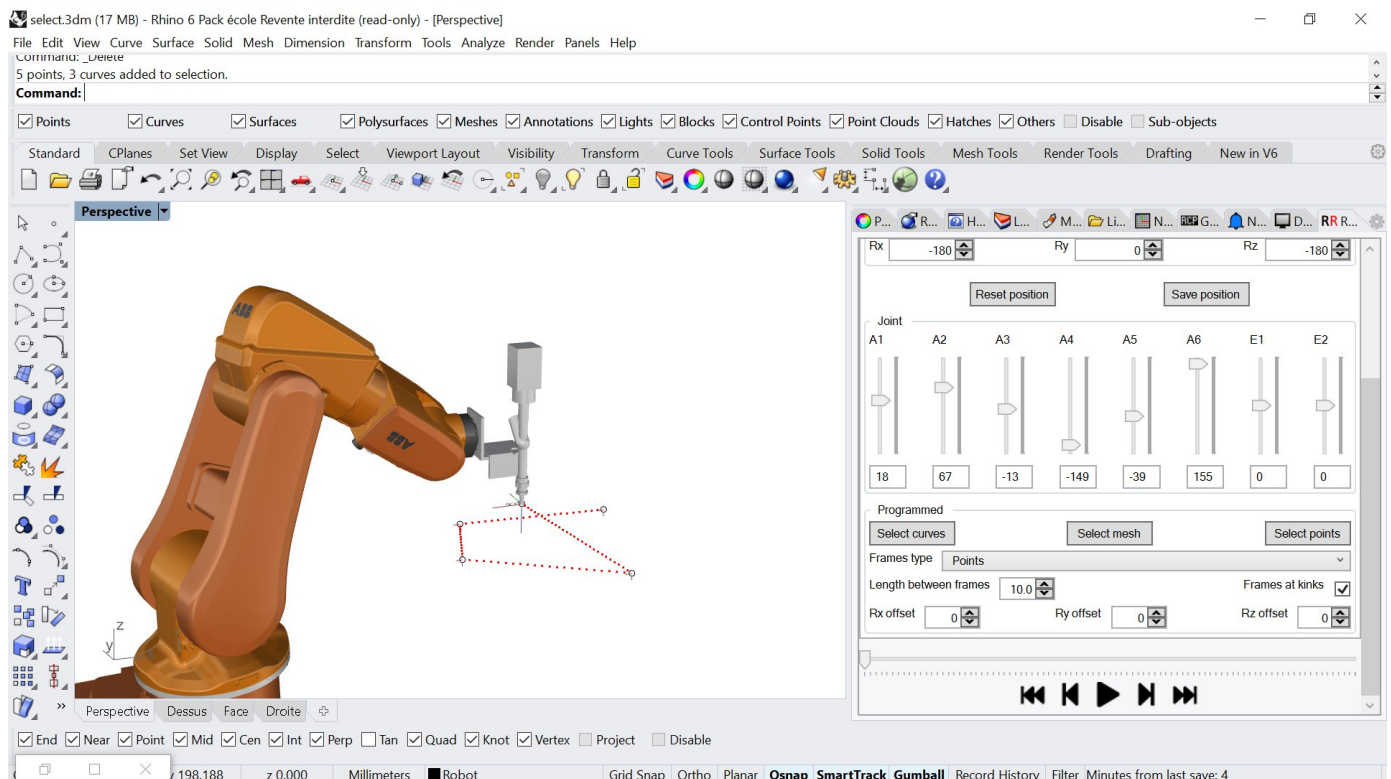

Selecting a mesh

Depending on the Frames type you choose, you may need to select a mesh. The selected mesh must be a single joint mesh, and for good results it's better to have a finely defined mesh with all its normals directions well aligned.

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Selecting points

RhinoRobot will generate a toolpath based on the selected points. The order will be the selection order, or the historical one if you select a group of points.
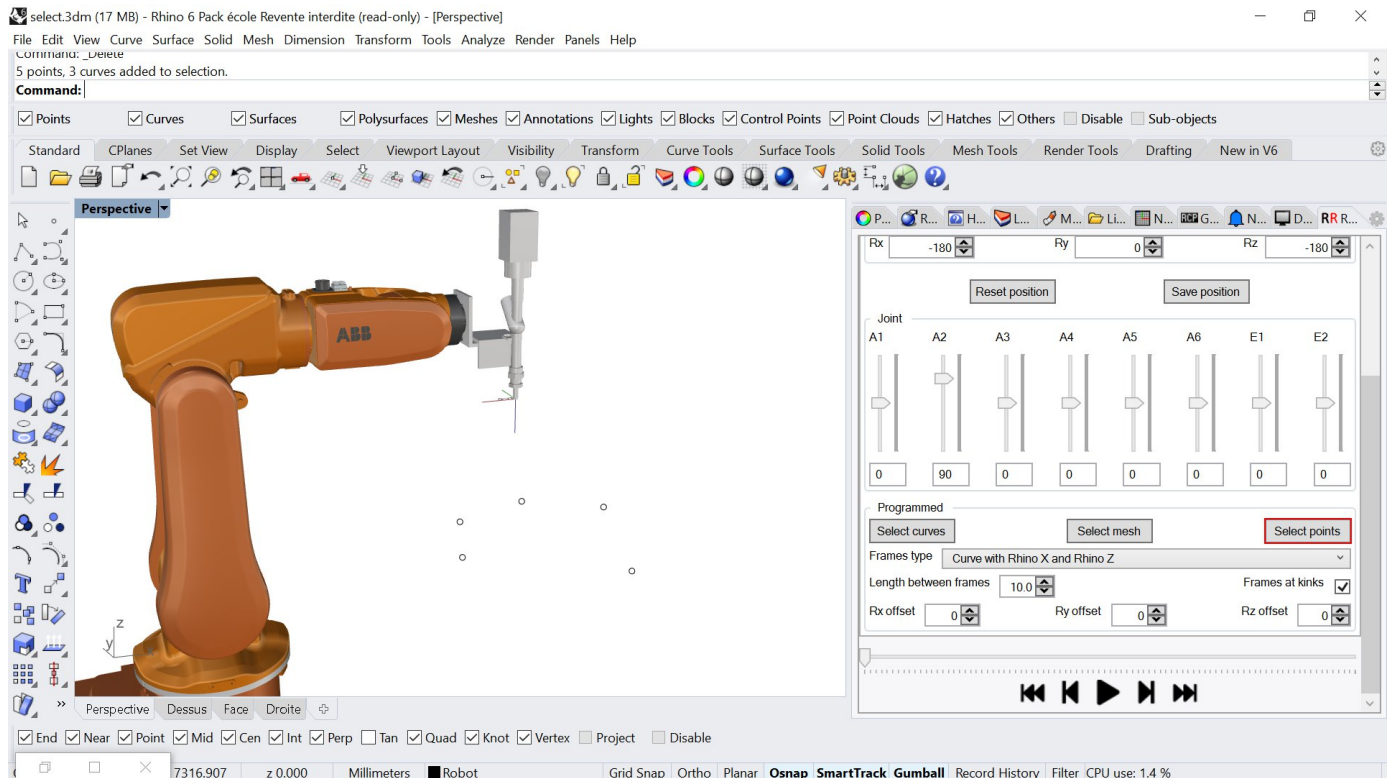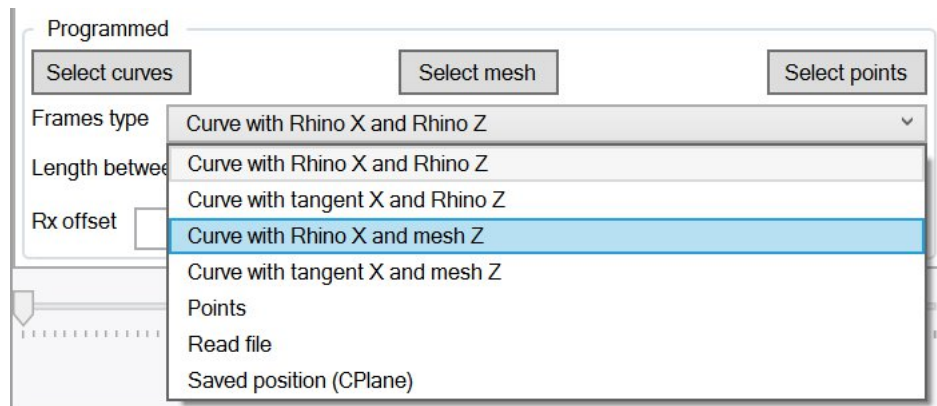

Unselecting

You can unselect any curves, mesh or points by right clicking the according button

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Frame options

Frames type

If you have selected curve(s) the Frames type can be:

'Curve with Rhino X and Rhino Z'
The frames' X axis will be aligned with the Rhino X axis and the frames' Z axes will be aligned with the Rhino Z axis.
'Curve with tangent X and Rhino Z'
The frames' X axis will be aligned with the curve tangents and the frames' Z axes will be aligned with the Rhino Z axis.
'Curve with Rhino X and mesh Z'
The frames' X axis will be aligned with the Rhino X axis and the frames' Z axes will be aligned with the mesh normals.
'Curve with tangent X and mesh Z'
The frames' X axis will be aligned with the curve tangents and the frames' Z axes will be aligned with the mesh normals.
If you have selected 'points' you have to select points as a Frames type.
If you choose 'Read file' as a Frames type, you will be prompted for a file containing points. The syntax of that file can be standard 3d printing Gcode, or milling x,y,z,I,j,k for example, or even a custom reader from Grasshopper. You can select multiple files
If you choose 'Saved position (CPlane)' the toolpath will be an interpolation of the saved positions in their names' order.

Distance between frames
You can set the distance between the frames along the curve in mm.
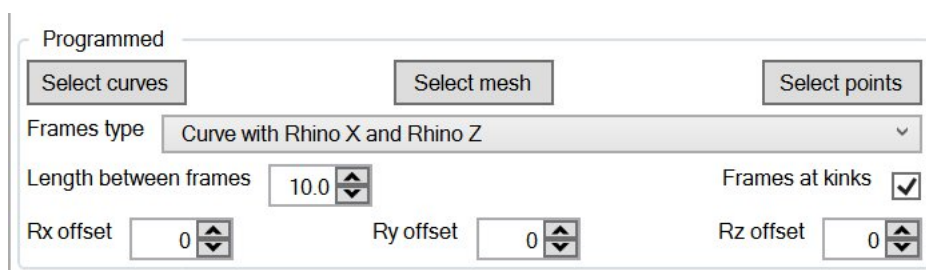

Rotation offset
If you need to adjust the orientation of your curve frames, you can use the rotations' offsets of -180/+180 around the X, Y and Z axes.

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Recording positions

You have the ability to save a position of the TCP by clicking the Save position button. You can record as many positions as you want. Each time you save a position it is saved as a named CPlane as 'RR_Pos_xxx' and it is linked through a line to the previous position. If you select 'Saved position (CPlane)' as a "Frames type" in the programmed movement part the TCP will follow a toolpath consisting in a polyline between all your saved positions.
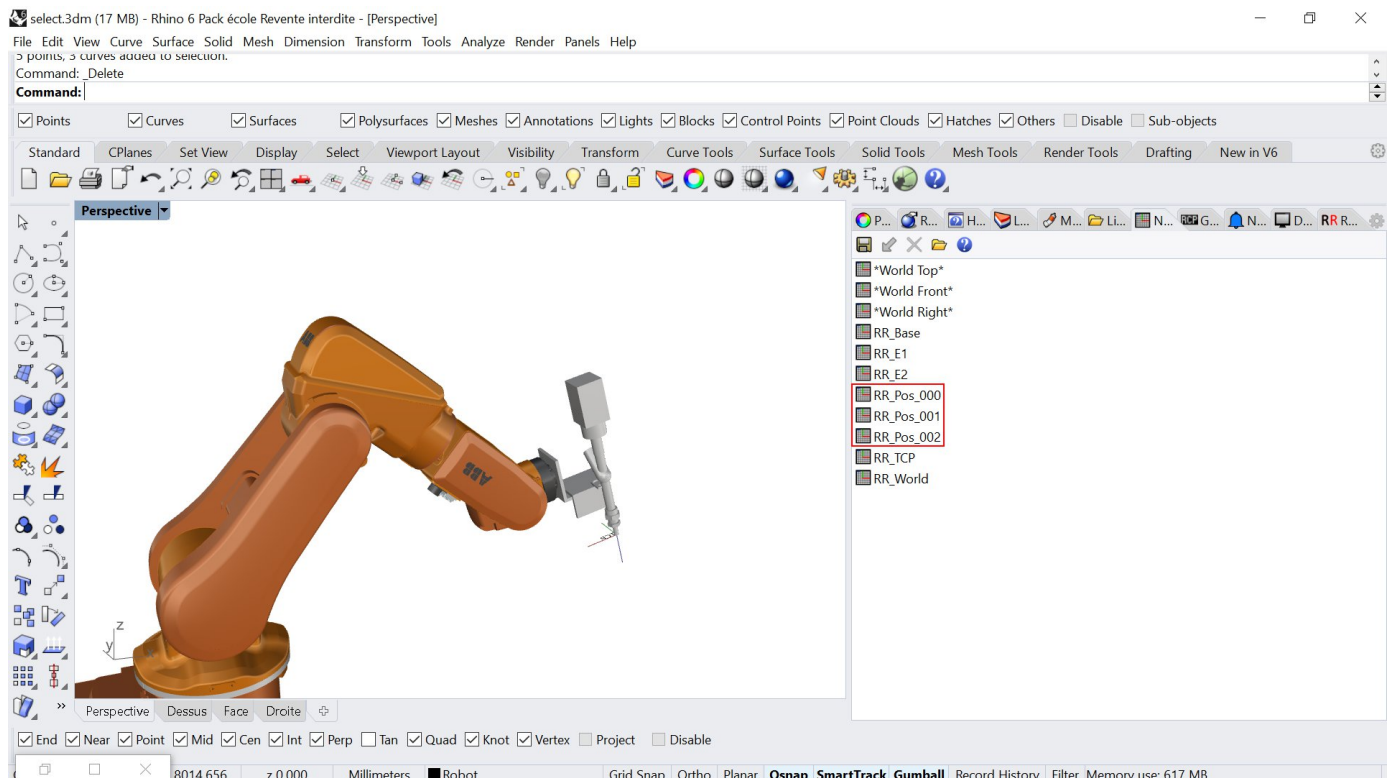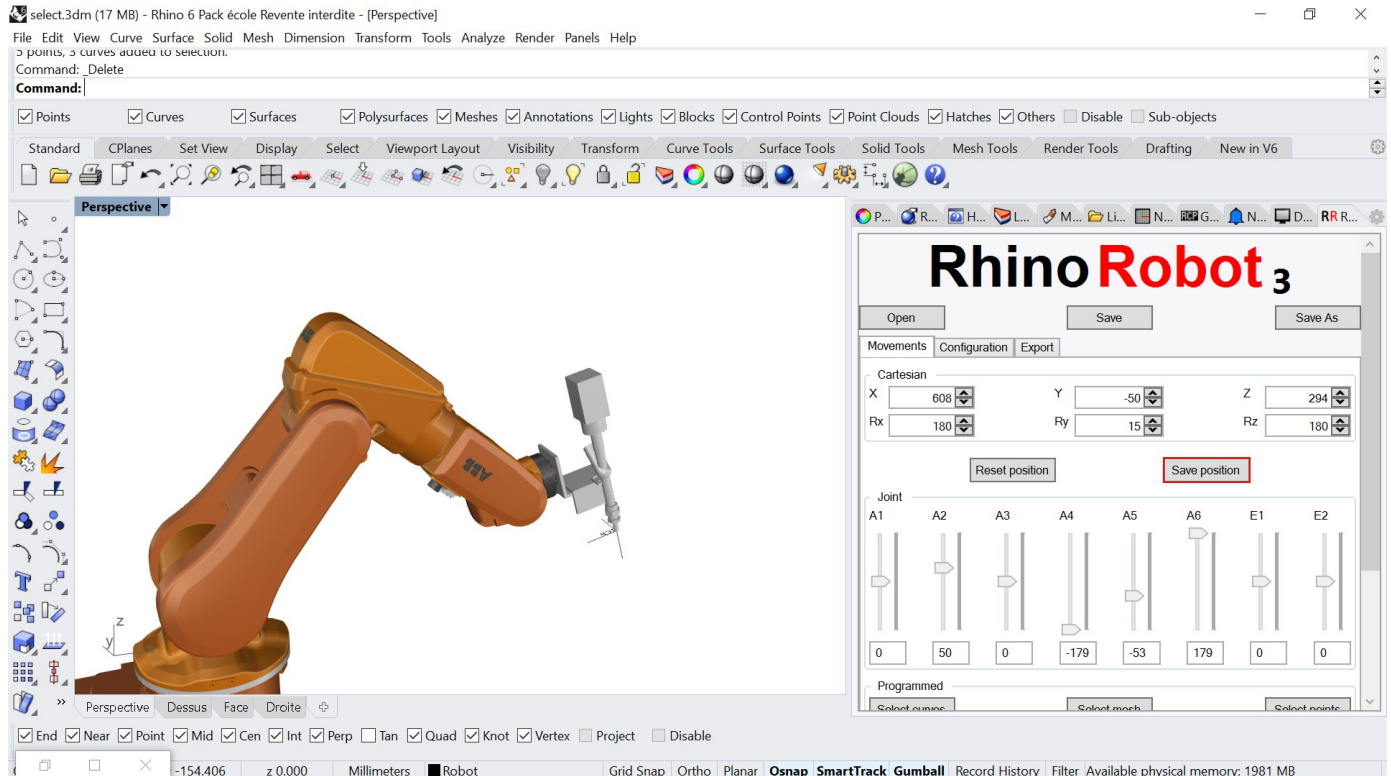
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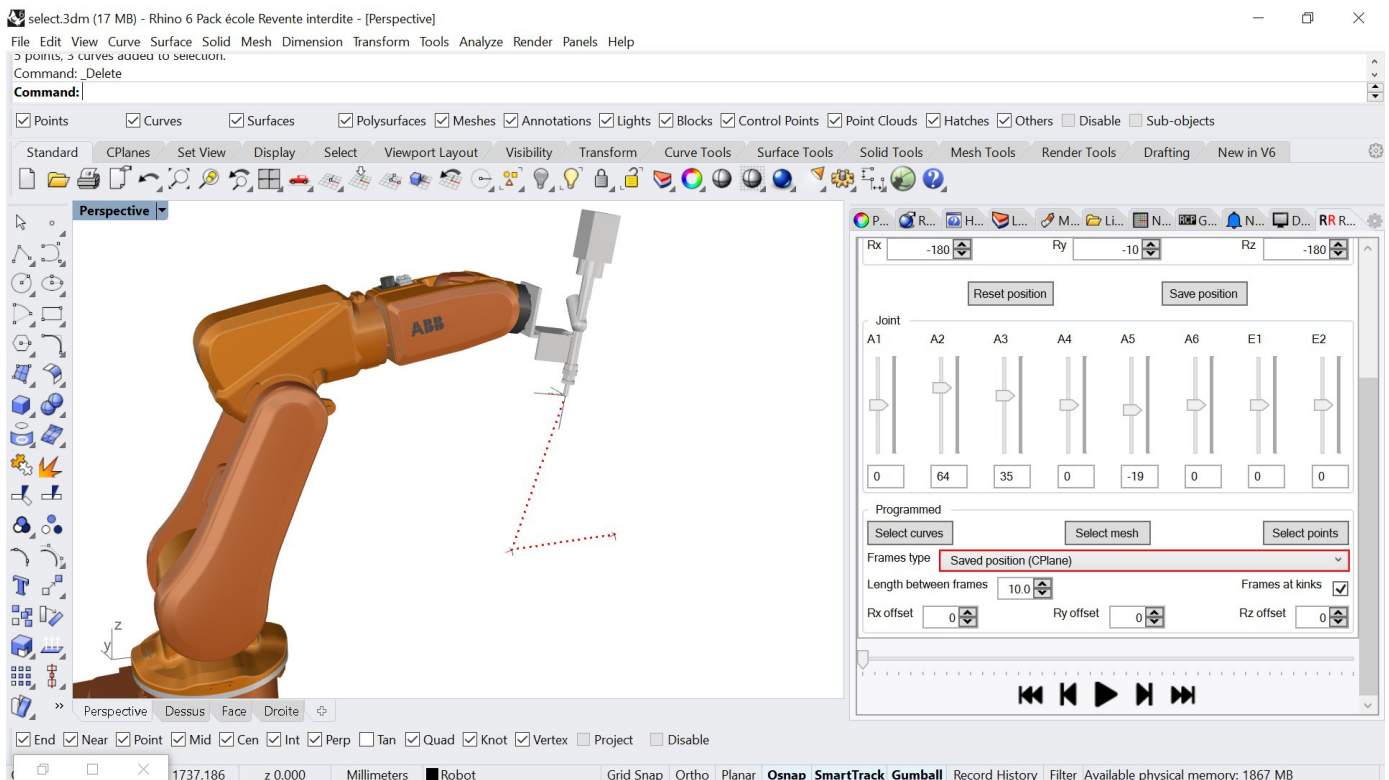
Editing recorded positions

Named CPlanes are listed and editable in the 'Named Views' panel of Rhinoceros.
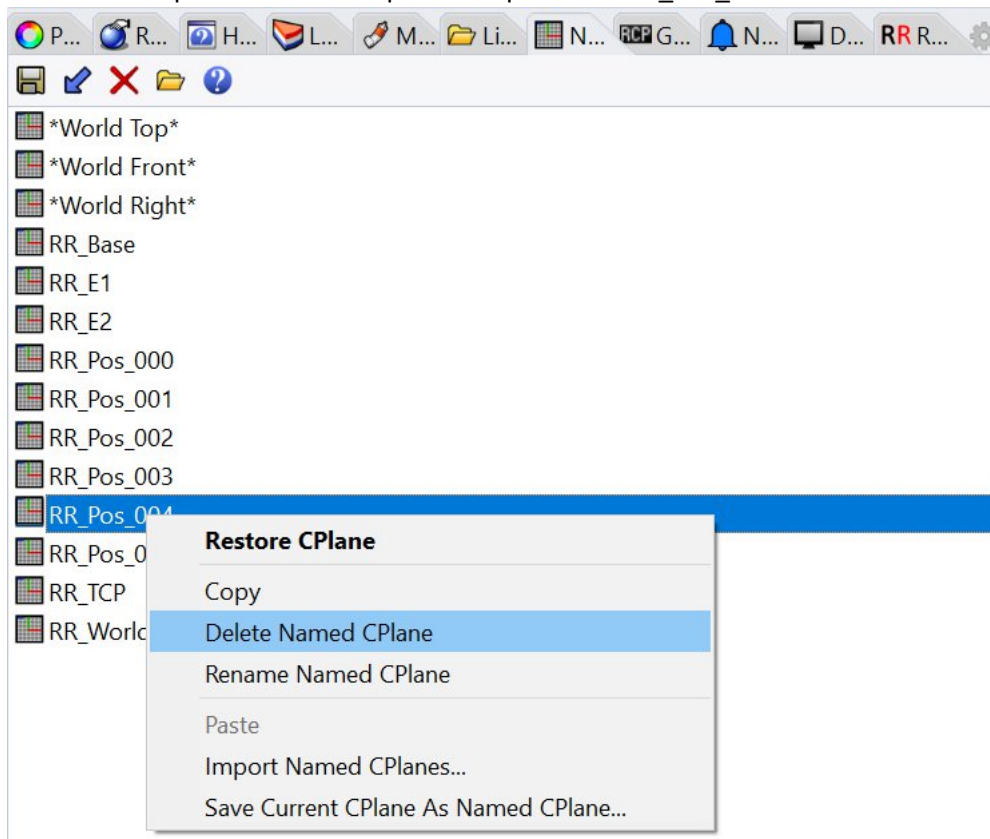You can also copy and paste CPlanes and edit their positions. You will have to accurately rename the copied CPlane for it to be treated as a Saved position for a toolpath. The pattern is RR_Pos_XXX.

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Advanced offsets

Advanced offsets, if activated, is a way to add local offsets, as well as speeds and commands to the different selected toolpath.
When activated, you will see the Grasshopper branch and points number next to the targets.

The offsets let you move and rotate the targets locally with a diffuse behavior, regarding the other targets.
If you want to apply a very local offset, set the offsets to 0 on the target just before and after the one you edited.

The advanced offset section also let you input extra points from the recorded position to another toolpath, like one from a curve or a milling file.
To do so, just input the name of the recorded position CPlane to the Command section:

This can be useful for simply add entry and extraction points to a milling path, or simply add an extra position to a trajectory.
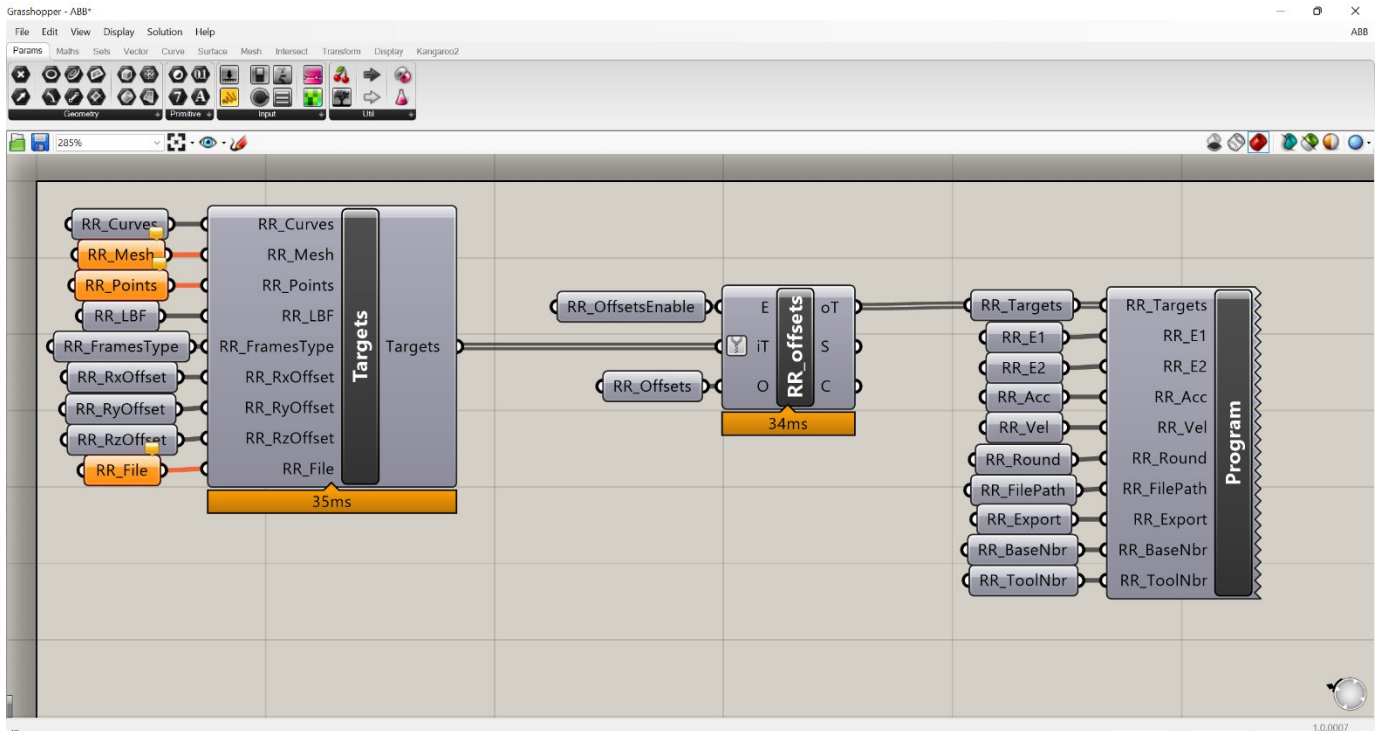
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Customizing your RhinoRobot

Anatomy of your Grasshopper project file

Display the grasshopper definition of your project by clicking on the Grasshopper icon or typing "grasshopper" in the command line.

A RhinoRobot definition consists of two main Grasshopper clusters (a cluster is a program within a Grasshopper definition).

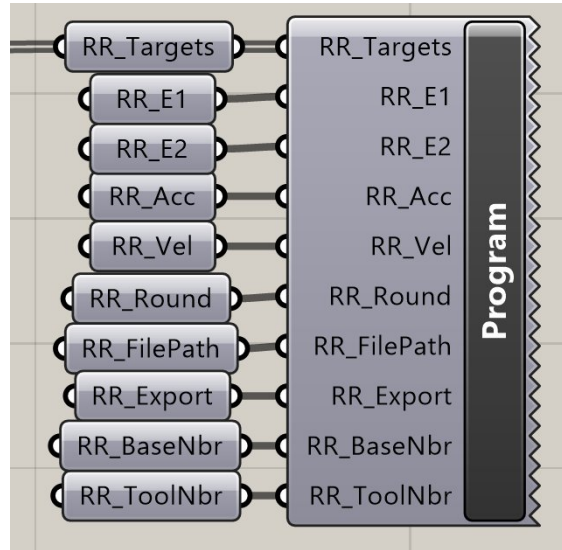The first one, called 'Targets, is where and how the targets are constructed.

Its INPUT parameters are primarily a list of geometries and other values passed on by the interface (the RhinoRobot Rhino panel).

Its OUTPUT parameter is a list of frames.
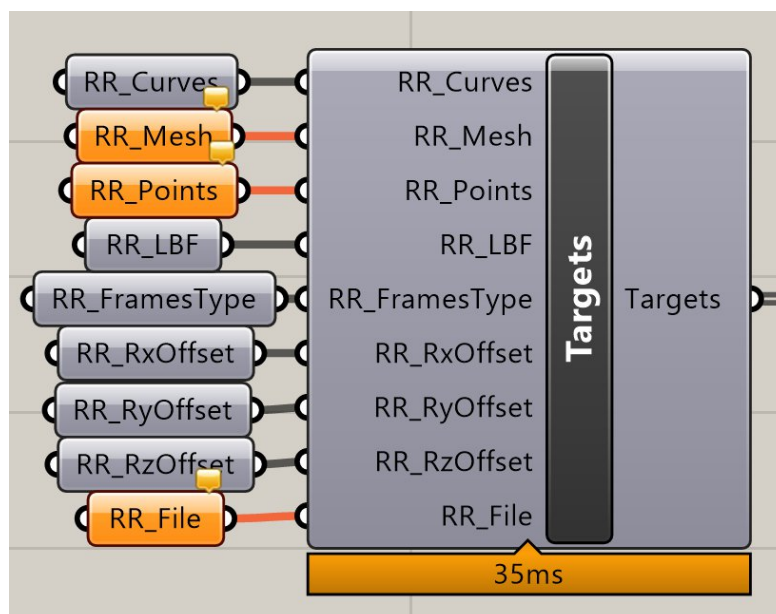
The second one, 'Program' is where and how the program can be exported.

Its INPUT parameters are primarily the list of frames given by the 'Targets cluster called 'RR_Targets' and a path for the file export. It contains the post processor specific to your robot or robot brand.
To enter a cluster, just double click on its name.

The name of the parameters is what allows the passing of the values between the interface and the Grasshopper definition, for example the 'Targets' cluster is listening for some curves that the interface might pass on via a RR_Curves parameter.

Between those two main clusters, you will find the Advance offsets component, who is responsible for applying the advanced offset to you list of frames.
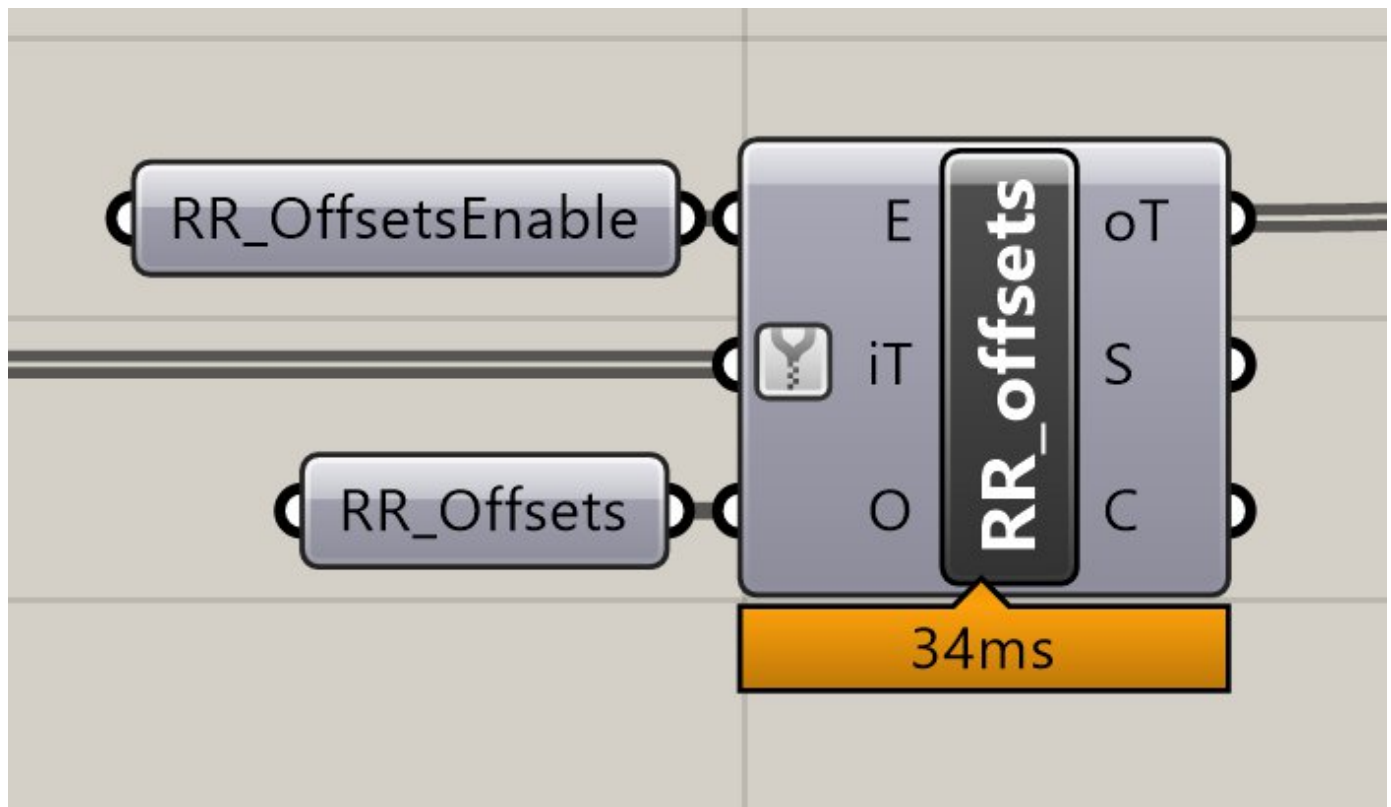
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Directly feeding geometry to the Grasshopper part of RhinoRobot

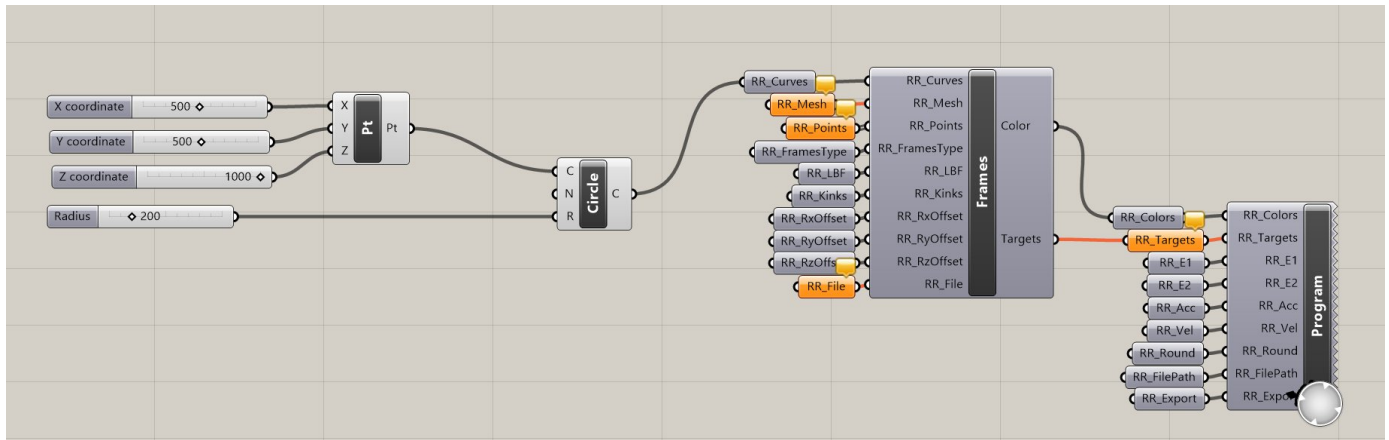You can bypass the selection of curves via the interface and feed directly the Grasshopper part of RhinoRobot. For example, you can connect your own Grasshopper definition to the RR_Curves container linked to the 'Frames' cluster and then bypass the curve selection system of the interface (the RhinoRobot panel).

Customizing your toolpath

You can also very easily customize or replace the default toolpath/set of frames created by the "Frames" cluster. For example, you can add approach and retraction points to your toolpath as follows:
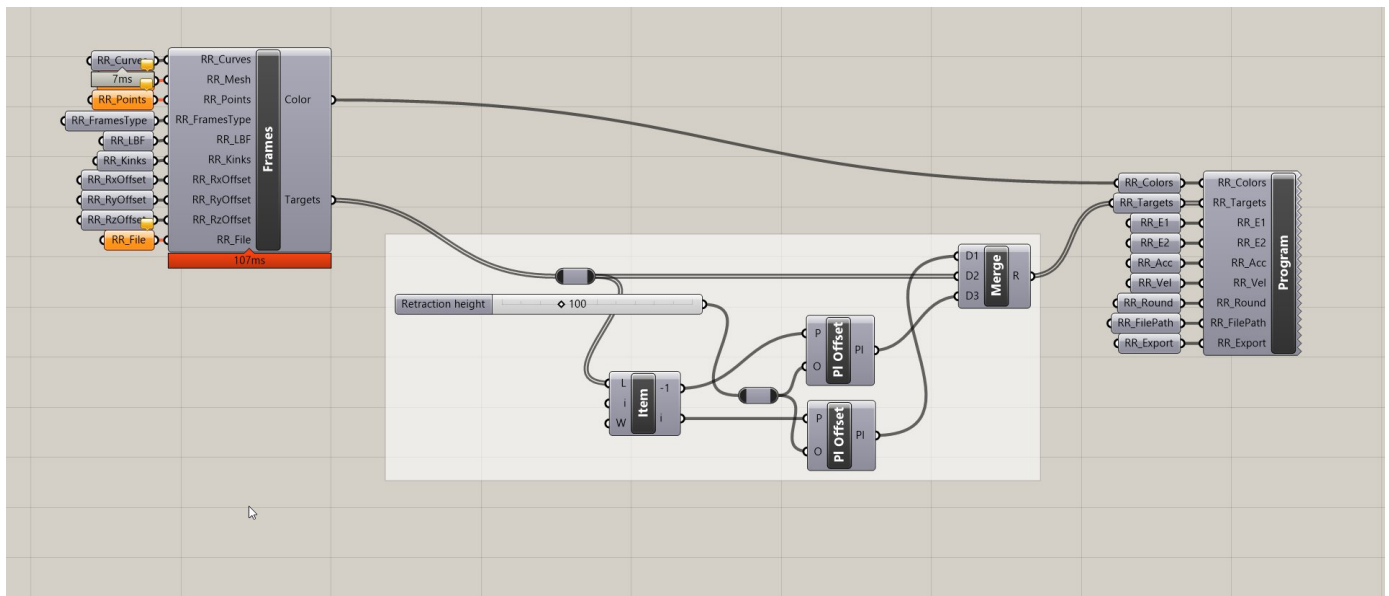
The only thing that you must continue to use if you customize you Grasshopper definition is the "RR_Targets" component.
This component is the one that RhinoRobot will "listen to" for changes and where it will load the list of targets from, in order to simulate them via the interface.
This link between the "RR_Targets" component and the RhinoRobot interface is established when you open a project via the "Open" button.

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Customizing the RhinoRobot interface

Inside your Grasshopper definition if you right click an input component like a slider, button, toggle or panel, you can select 'Publish to remote panel'. A new tab appears in your RhinoRobot panel, called 'custom', where you can find your custom published component, and you can add as many publishable components as you like.
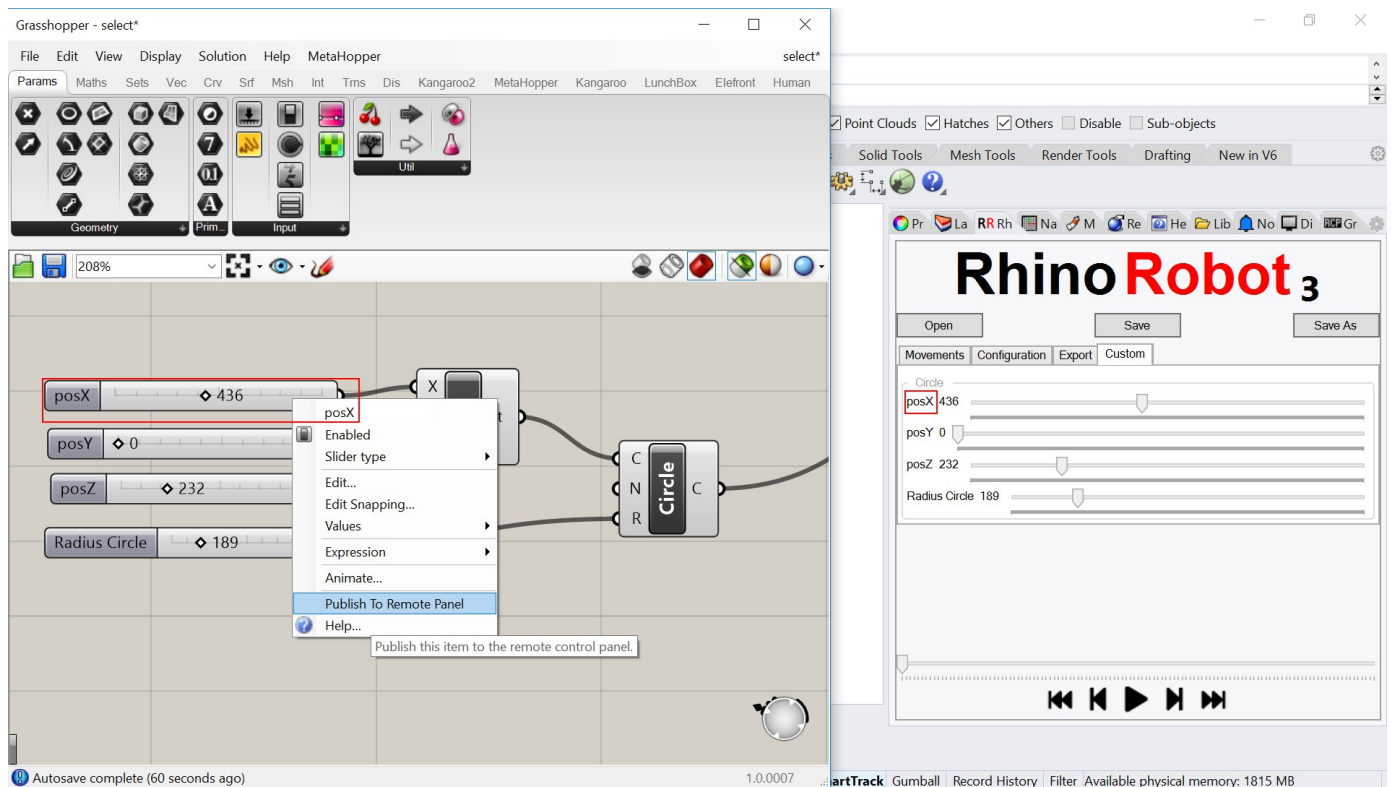
To edit your custom tab, click on the Grasshopper panel of Rhinoceros and click on the 'edit' button in the top right corner.
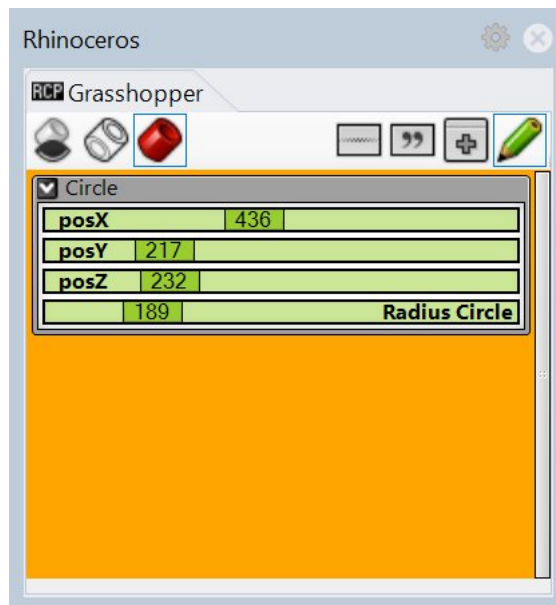
It is also possible to use a completely different interface for your project, for example if you are reselling RhinoRobot to your client with a custom machine, or if you need a lot of new controls and don't need the default ones.

This is made possible using the RhinoRobot_Core API that you (or we!) can include in your custom Rhinoceros plugin.

Contact us for more information on the subject.

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Customizing the Post-processor

As the post-processor is pure Grasshopper programming (as well as in the "Frames" cluster), it's fully open and you are completely free to customize it for your needs, by adding custom commands, sub-programs, etc.
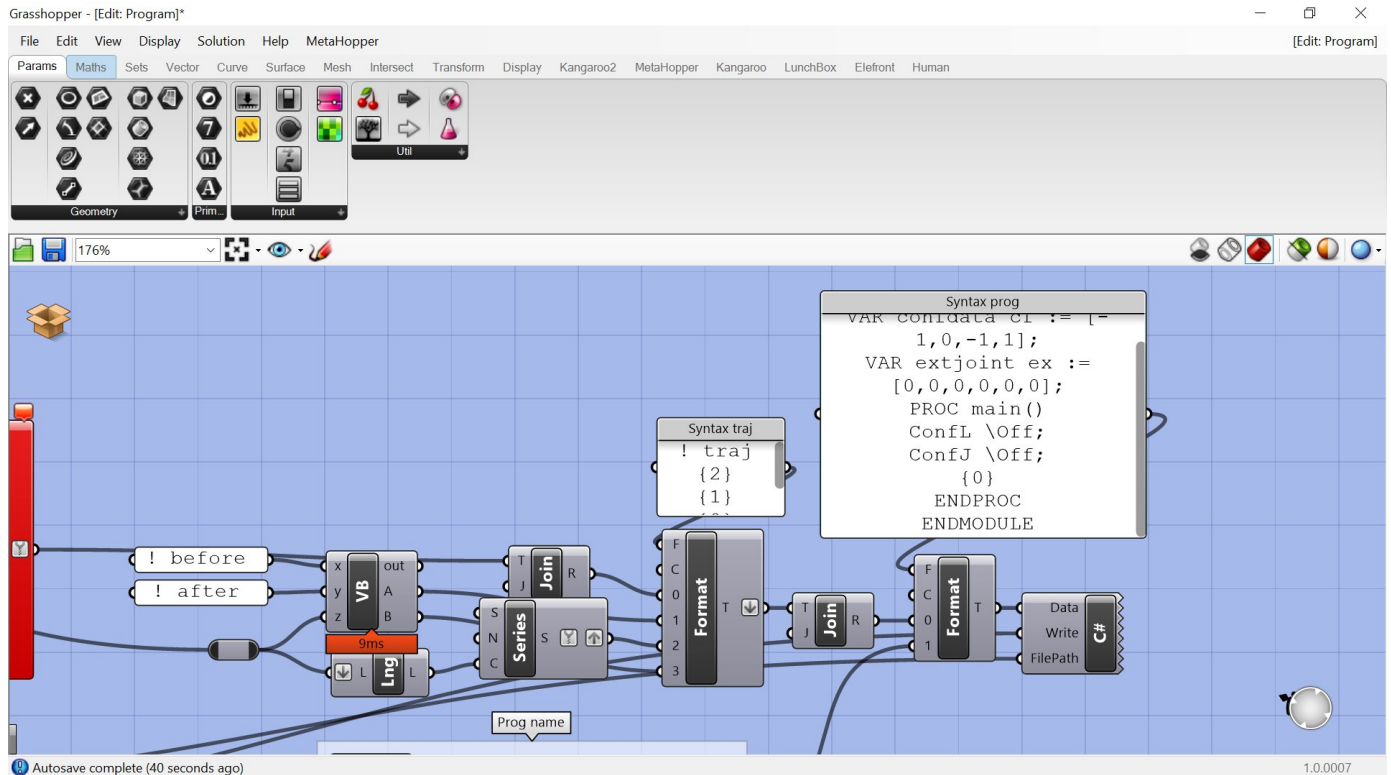
Project naming and Post-Processor matching

When you open a .3dm file via the RhinoRobot panel
-it first looks for its Grasshopper counterpart (a file with the same name and a .gh extension)
-if it cannot find it, it looks into the filename for the brand name of a robot, before a '_' character
-if it cannot find a match, a generic post-processor file named Other.gh which contains a standard points and vectors post processor is opened.

Extra variables

There are extra variables that are passed to the Grasshopper definition that you can use in a custom project. Those are not present by default by you can simply add them and name them correctly to see them updated as the simulation is played:
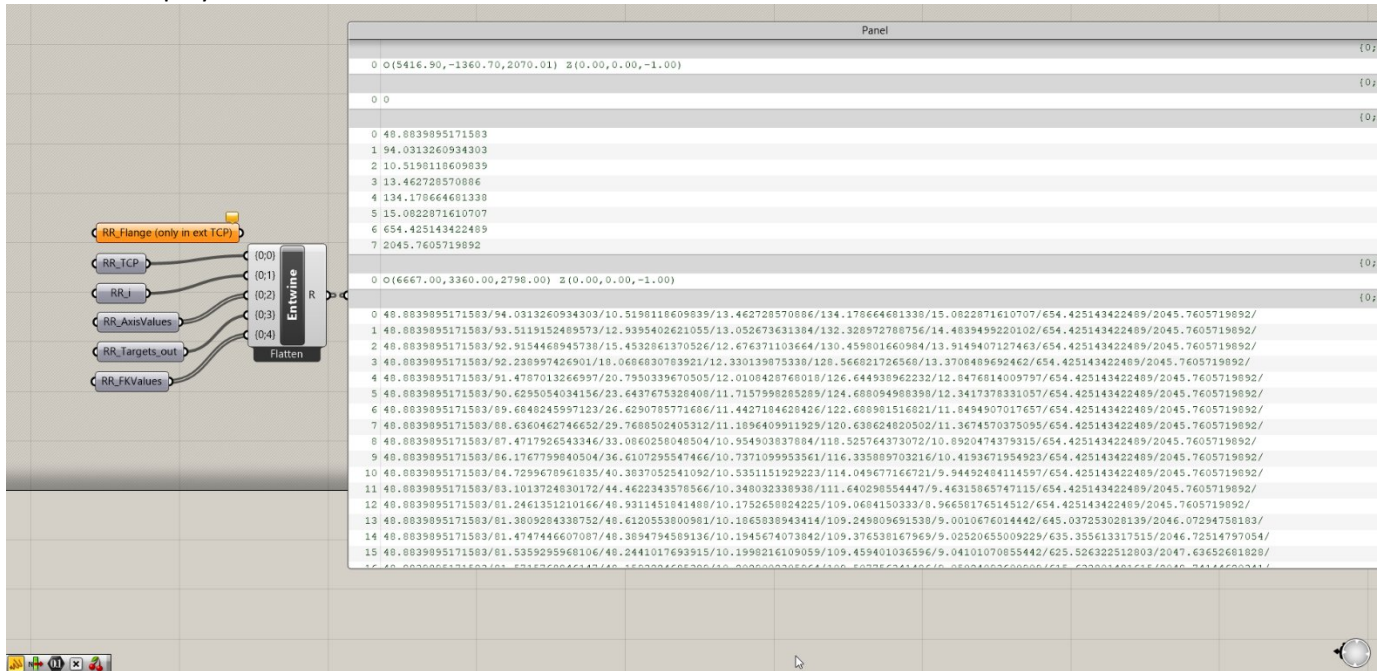
RR_Flange: Current flange plane position calculated by the FK engine, available only if external TCP mode

RR_TCP: Current TCP plane position calculated by the FK engine

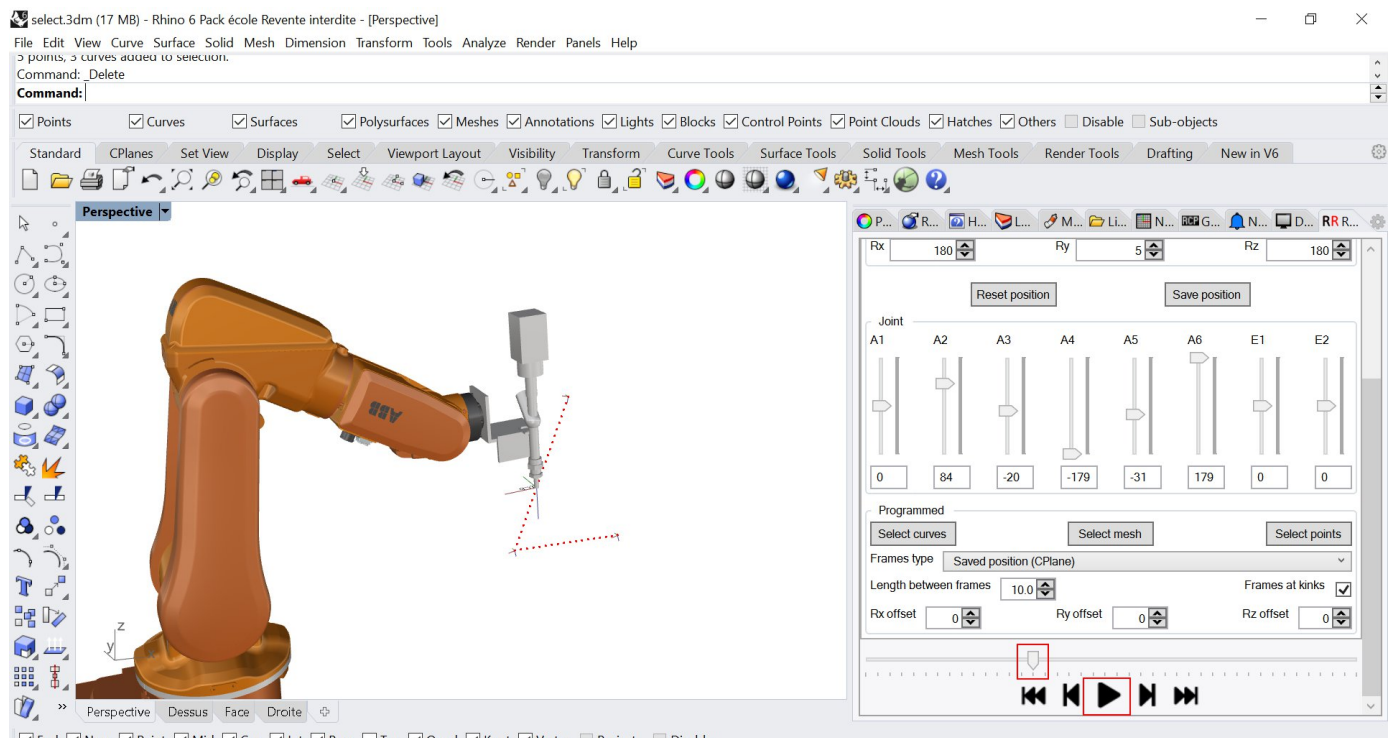RR_i: Index of the current simulated position

RR_AxisValues: Axis values of the current simulated position

RR_Targets_out: List of saved position CPlane

RR_FKValues: All the axis values calculated by the IK engine for the current list of Targets

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Simulating the toolpath

If you press the next button the robot will go to next the simulation step. Those simulation steps are different from the actual programmed frames.

The simulation steps distance can be set down in the "Configuration" tab. You will find a 'Display' group of features. Frame axis length is self-explanatory and allows you to hide or enhance the frames display.
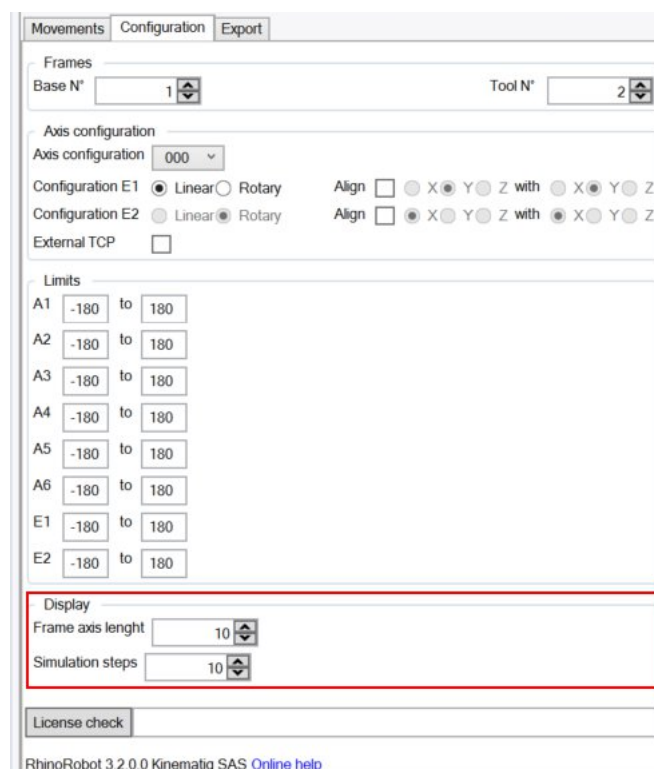
You can also simply scroll the simulation track bar or press play for an automatic simulation at the theoretical speed set in the Export tab. Note that if your computer is not capable of rendering the robot positions at this set speed, the displayed simulation will be slower than reality.

To get a more accurate simulation speed you can render your simulation in "Wireframe" view.

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Testing different axes configurations

You can test up to 8 different axes configurations to find the best one for your program (note that depending on your robot type and the chosen movement types, this axes configuration may not be chosen by the real robot).
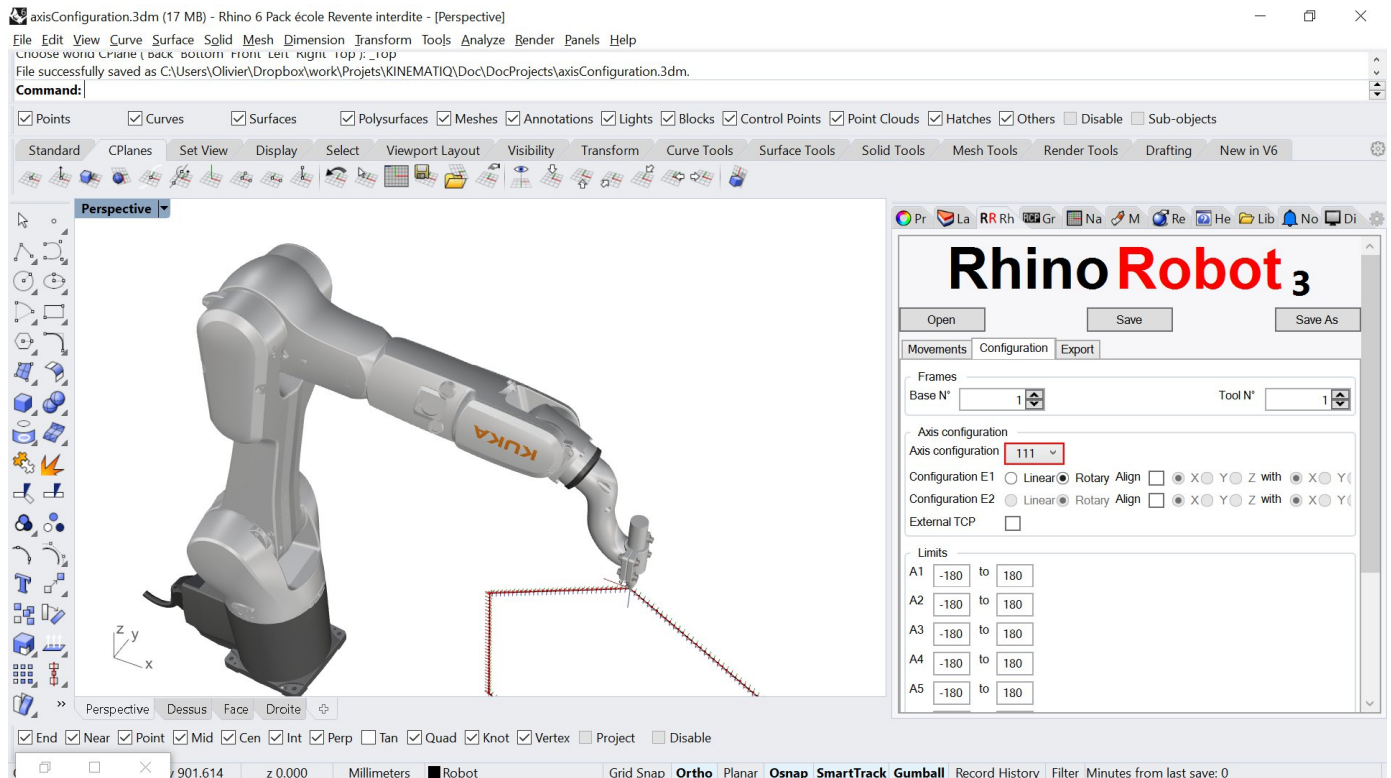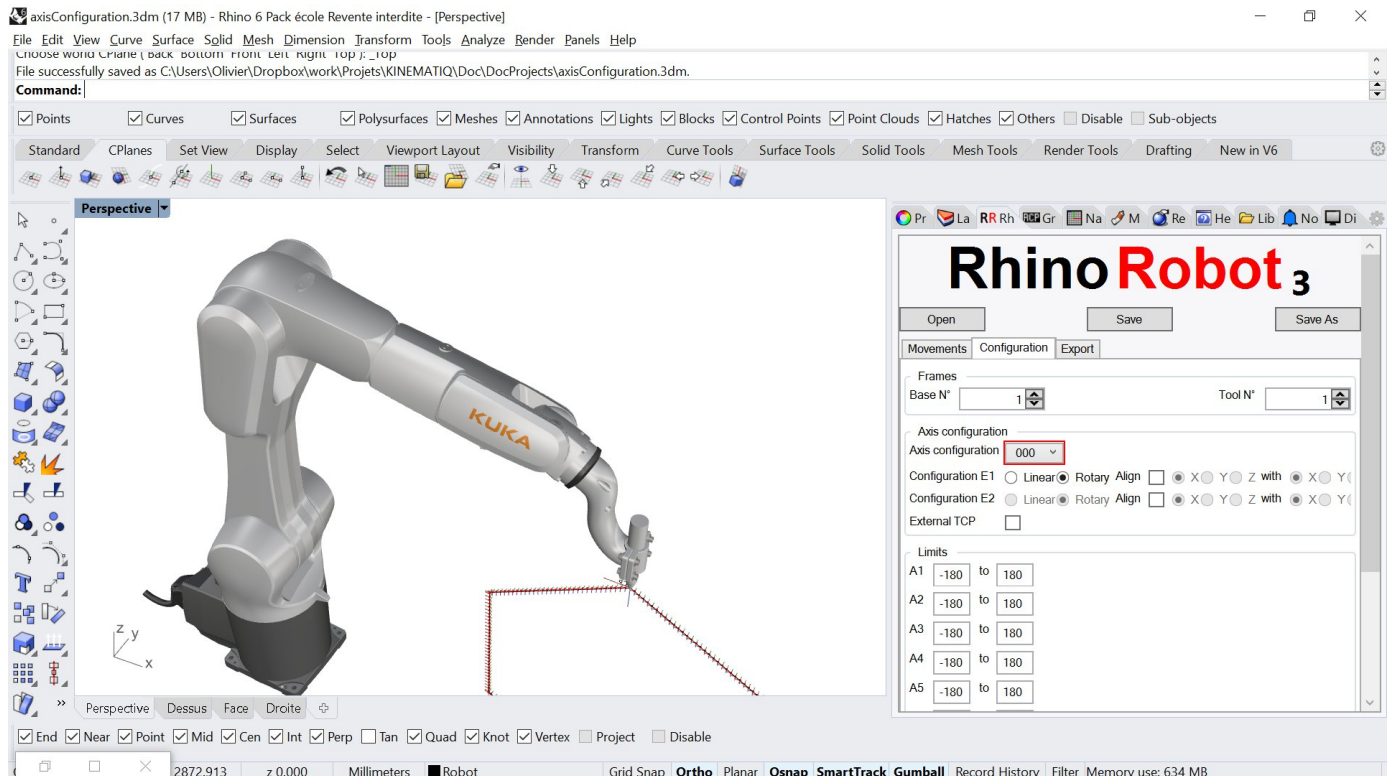
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Running a collision check

In the export tab you can click to run a collision check.
A list of potential problems will be displayed underneath at the end of the check. In this collision check, you have the possibility to add any chosen geometry that will be tested for collision with the tool.

Note that the robot itself, the tool and the external axis geometry are already included in this collision check.

As it tests every single simulation step for collision, this check can take very long, depending on the length of your toolpath and computer power.
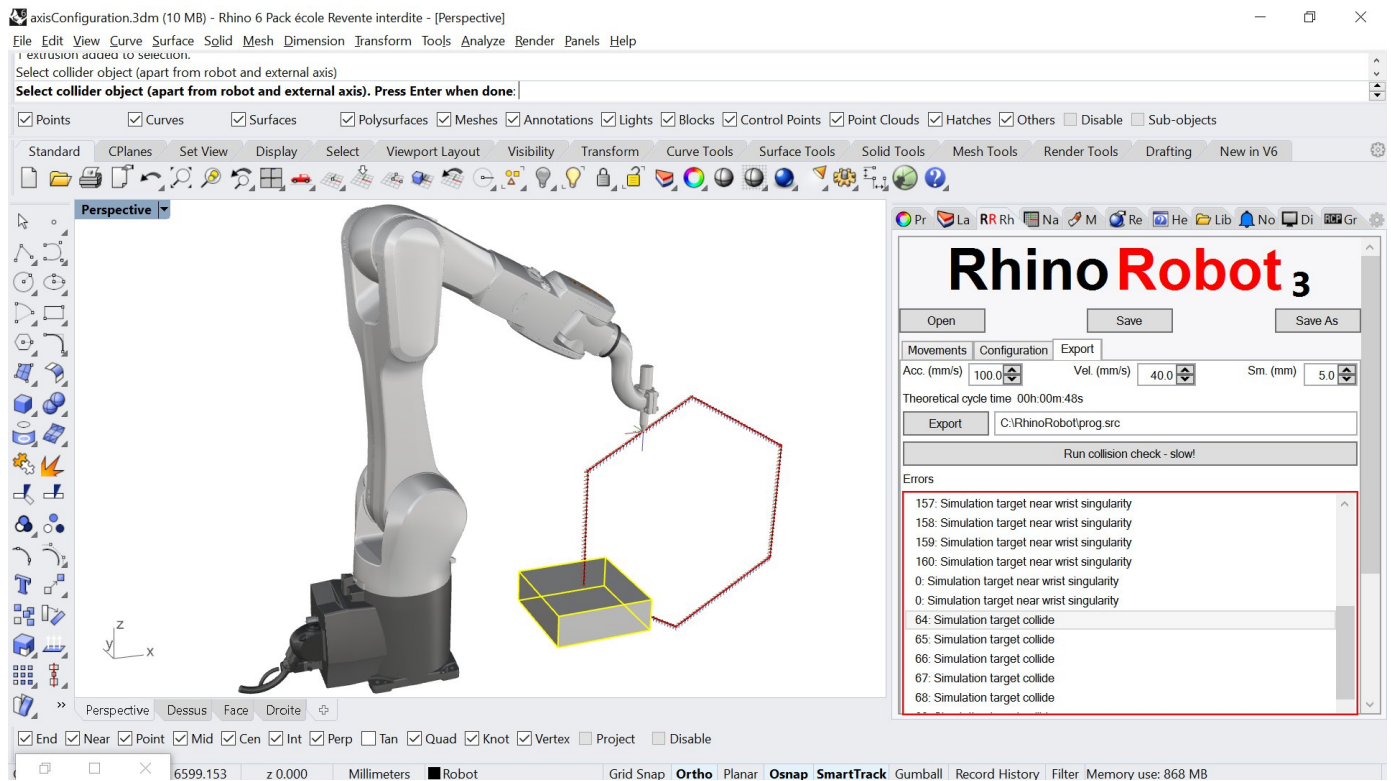
Exporting the simulation

You can export your simulation, including the environment and the Grasshopper customs display.

The export format is .gltf/.glb, and the file is named and exported as the program file path.

GlTF is an open file format to contain and display 3D geometry and their animations, its creators have described it as the "JPEG of 3D." more information here: https://en.wikipedia.org/wiki/GlTF

The exported .gltf/.glb simulations can be read on any PC/Mac natively, as well as in web-browser on online viewer like https://gltf-viewer.donmccurdy.com/, shared and embed on website like https://sketchfab.com/ and view on almost any VR and AR goggles.
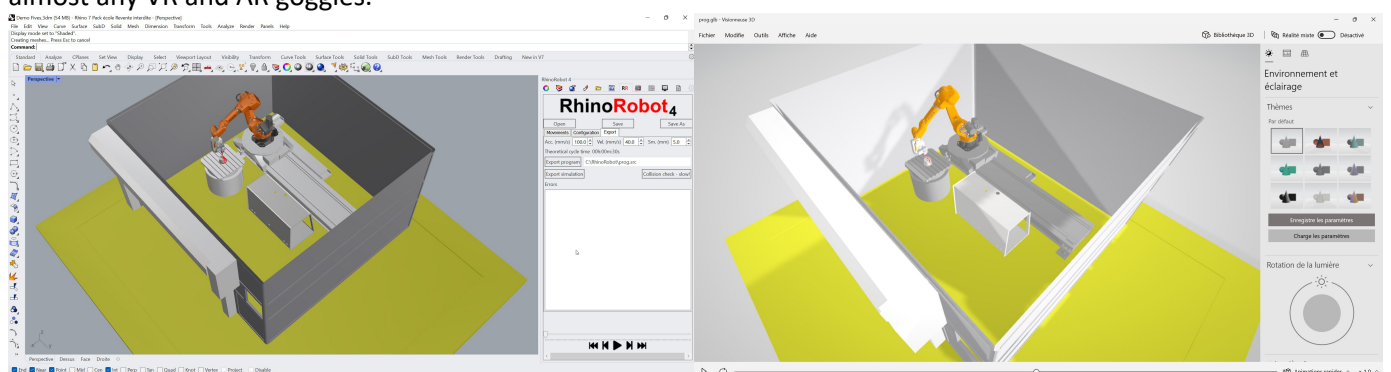
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Exporting the program

When you are done with your frames construction, and satisfied with the simulation, you can set the acceleration, the velocity and the smoothing for the toolpath, a theoretical cycle time is also displayed.

You can export your program as a robot program file by clicking the Export button.


```
DEF prog(prog)

;Programme avec RhinoRobot
;www.kinematiq.net

;FOLD DECL
;ENDFOLD

;FOLD INI
  ;FOLD BASISTECH INI
    GLOBAL INTERRUPT DECL 3 WHEN $STOPMESS==TRUE DO IR_STOPM ( )
    INTERRUPT ON 3
    BAS (#INITMOV,0 )
  ;ENDFOLD (BASISTECH INI)
;ENDFOLD (INI)

$BASE=BASE_DATA[1]
$TOOL=TOOL_DATA[1]

BAS(#VEL_PTP,50)

$VEL.CP=0.4 ;m.s
$ACC.CP=0.1 ;m.s.s
$APO.CDIS=5 ;mm

PTP XHOME
;Fold traj0
;before
LIN {X 125.000,Y 402.230,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 134.364,Y 403.152,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 143.369,Y 405.884,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 151.667,Y 410.320,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 158.941,Y 416.289,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 164.911,Y 423.563,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 169.346,Y 431.861,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 172.078,Y 440.866,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 173.000,Y 450.230,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 172.078,Y 459.594,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 169.346,Y 468.599,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 164.911,Y 476.897,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 158.941,Y 484.171,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 151.667,Y 490.141,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 143.369,Y 494.576,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 134.364,Y 497.308,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 125.000,Y 498.230,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 115.636,Y 497.308,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 106.631,Y 494.576,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 98.333,Y 490.141,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 91.059,Y 484.171,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 85.089,Y 476.897,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
LIN {X 80.654,Y 468.599,Z 24.205,A 180.000,B 0.000,C 180.000} C_DIS
```