PRI Notes by XVBJR

PRI version used: (updated)

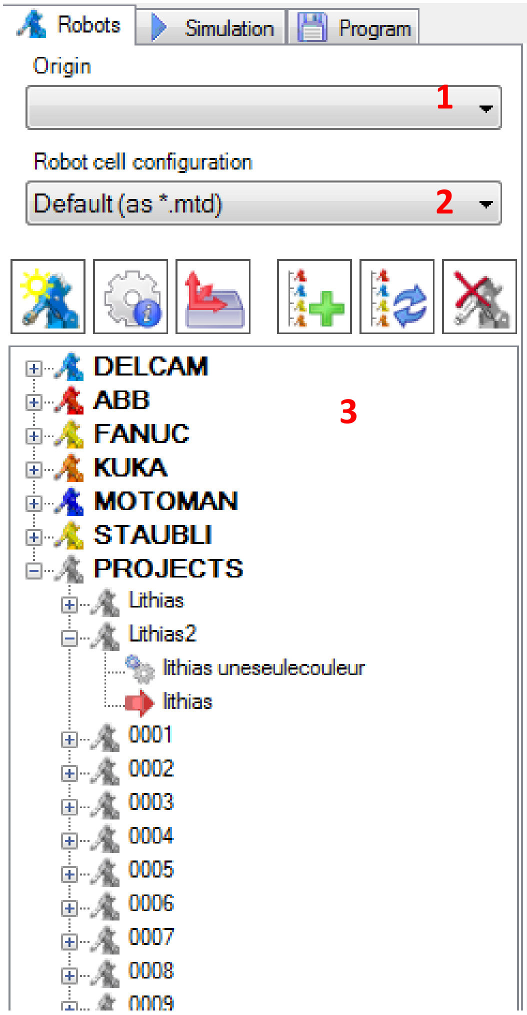
Contents:

Main interface

The PRI interface is divided into three tabs: Robots, Simulation and Program. To activate the interface, click on the  icon on the top right corner of the screen.

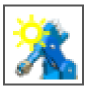
“Robots” tab

This tab is used to select and add a robot in the current project.

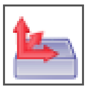


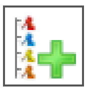
1: The origin selector.

2: Robot cell configuration. Allows the user to select a configuration of: home position / axis priorities / axis ranges. Theses configurations are saved in the ROBCONFIG file. (See “Robot positioning” section)

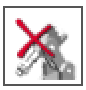
 : Display / Hide the currently loaded robot

 : Show robot information

 : Part positioning. Allow the user to create a new origin (new workplane)

 : Allows the user to add a robot in the library

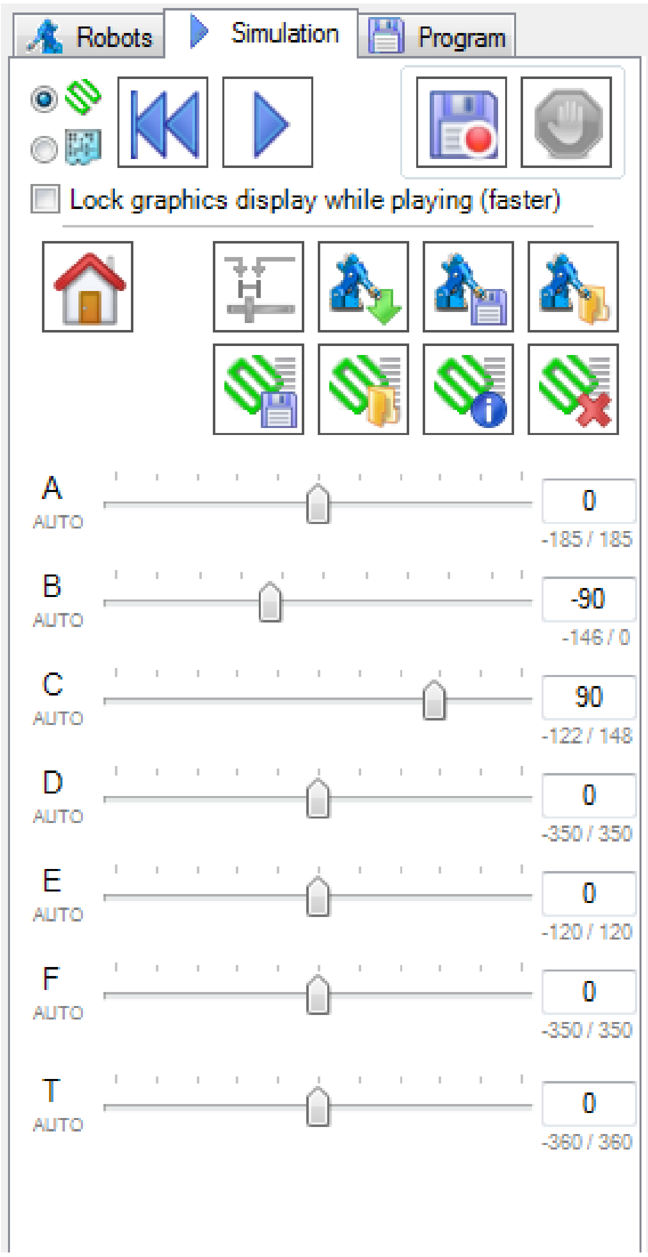
 : Reload the library (it will unload the current loaded robot).

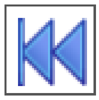
: Unload the currently selected robot.

3: Robot library. Double click to add the selected robot into the project.

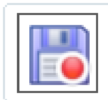
“Simulation” tab

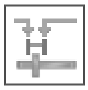
Use this tab to create simulations and perform analysis.



 Attach a tool to the robot from the selected toolpath.

Free robot head orientation, the robot will have the best posture.

 Use this button to record a toolpath into a RobFile. This file will contain a sequence of moves. After recording a file, the user can replay the sequence by using the timeline at the bottom of the screen.

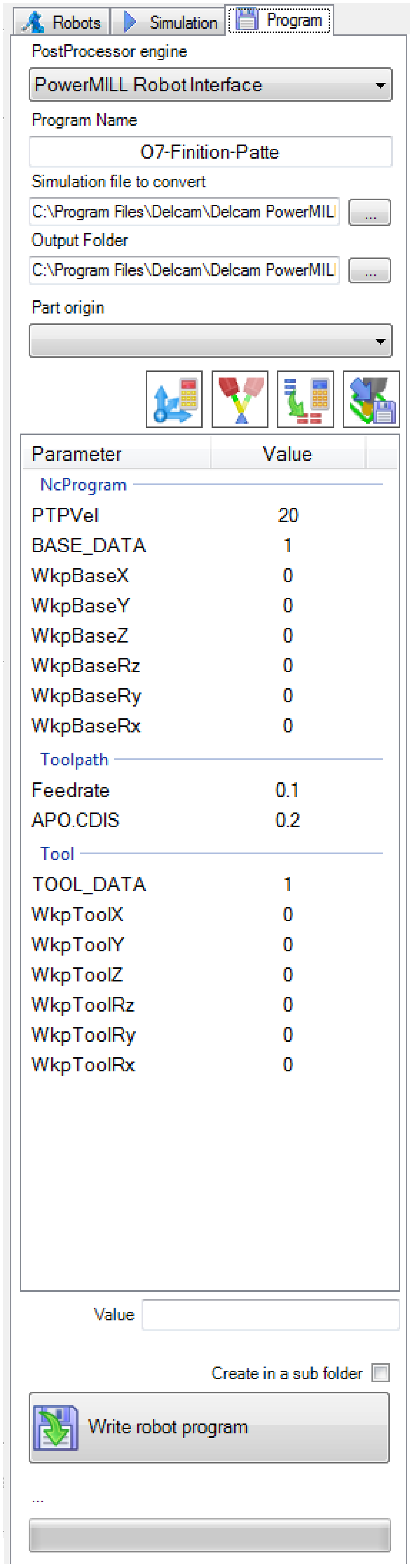
 Reset the axis limits in the “trace axis limits” mode.

Get the current robot position.

Save the current robot position into a file.

 Restore a previously saved robot position.

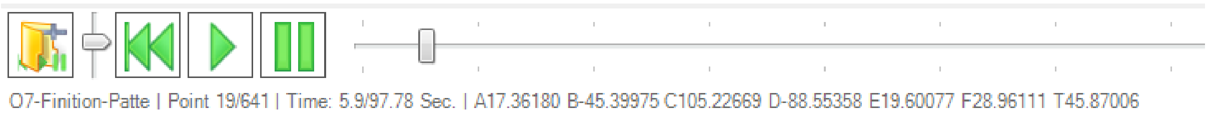
“Program” tab


Robots
Simulation
Program
PostProcessor engine
PowerMILL Robot Interface
Program Name
07-Finition-Patte
Simulation file to convert
C:\Program Files\Delcam\Delcam PowerMIL
Output Folder
C:\Program Files\Delcam\Delcam PowerMIL
Part origin
Parameter
Value
NcProgram
PTPVel 20
BASE_DATA 1
WkpBaseX 0
WkpBaseY 0
WkpBaseZ 0
WkpBaseRz 0
WkpBaseRy 0
WkpBaseRx 0
Toolpath
Feedrate 0.1
APO.CDIS 0.2
Tool
TOOL_DATA 1
WkpToolX 0
WkpToolY 0
WkpToolZ 0
WkpToolRz 0
WkpToolRy 0
WkpToolRx 0
Value
Create in a sub folder
Write robot program

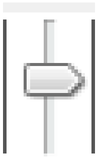
Timeline

(Is “Timeline” the correct word ??)

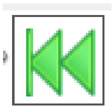
Use this tool to replay a recorded simulation.



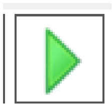
Open a recorded simulation



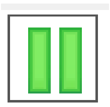
Select the simulation replay speed



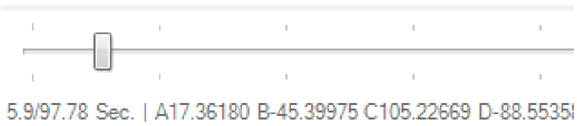
Move to the beginning of the simulation



Plays the simulation



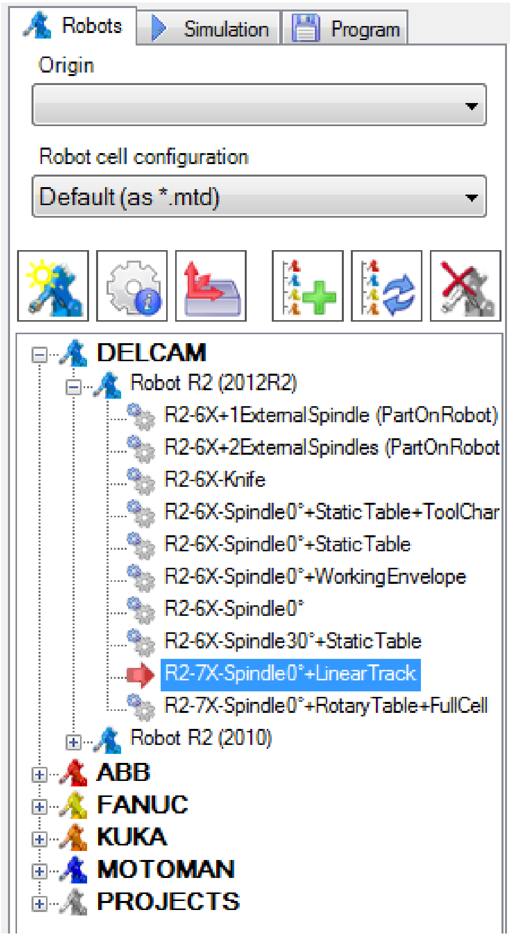
Pauses the simulation



Drag this cursor to play from a specific point of the current simulation

Adding and moving a robot

In this section we will see how to add a robot the current project and how to move manually the head. In the PRI interface, open the **Robots** tab and double click on the **R2-7X-Spindle0+LinearTrack** robot in the DELCAM folder. A red arrow appears behind the name and the robot is loaded



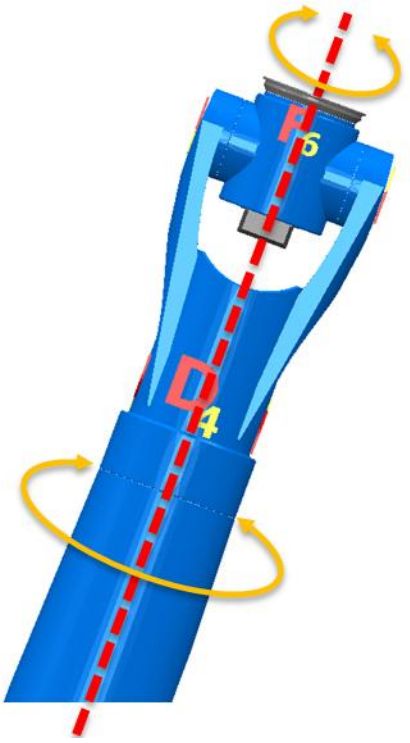
Open the **Simulation** tab. The robot axes are displayed. The A, B, C, D, E and F are the robot axes; the Y axis is the rail. Actually, the robot is in his home position, it's a secure (a better word ?) position. The currents axis positions are displayed on the right. The small values under the axes positions are the minimum and maximum values (the range).

Click and drag the cursor of the **A** axis. The robot is moving around his first axis. Try to move the other axes. Moving the Y axis will move the robot along the rail. To restore the robot in the first position, click on the **Home** button. Click on the **Move** tab at the bottom of the Simulation tab. Use the different arrows to move the robot head. Now the robot head is moving along the workplane attached to the robot head. You can see that a simple movement in the X axis for example causes modifications on almost all the robots axes (A to F). An internal solver is used to generate the motion of all axes.

Open now the **Rotate** tab and try the Roll (turn around the X axis), Pitch (around the Y axis) and Yaw (around the Z axis) buttons. Robots have the great ability to reach a random orientation.

But sometimes, the internal solver can have difficulties to move the robot in a particular position. These issues are called **singularities**. They append when the solver have no or more than one solution to move to a particular point. We can have three kinds of singularities[1]:

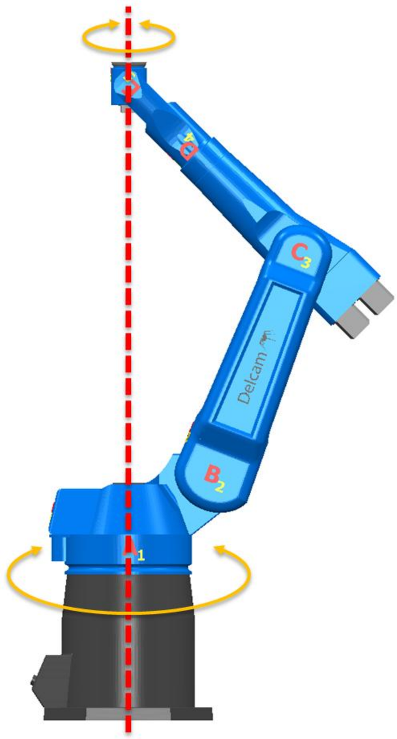
- The Wrist Singularity: Alignment of axis 4 and 6. (the most common one)



- The Elbow Singularity: When the arm is fully extended. Alignment of axis 2,3 and 5

[1] Source : \\ns2\Marketing\AMS\Products\PowerMILL Robot Interface\COMET\ PRI - Introduction to Robotics & PowerMILL Robot Interface - 30-08-2010

- The alignment Singularity: Alignment of axis 1 and 6



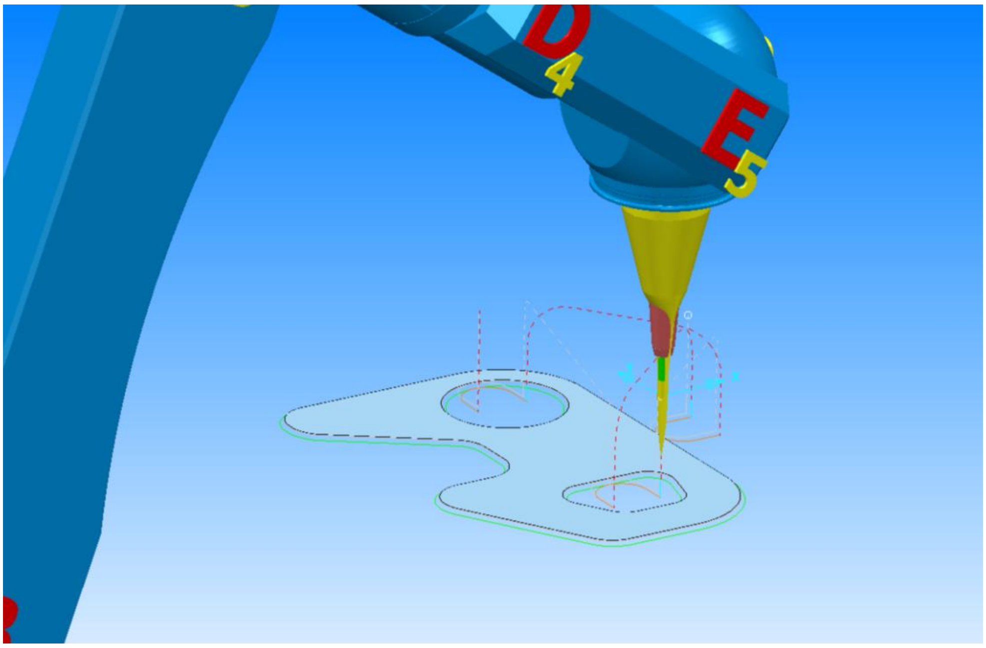
Additional constraints or modifications in the toolpath are sometimes essential to prevent the robot to have a singularity.

How to make a simple simulation

In this section, we will learn how to create a robot simulation from an existing toolpath. The simulation will be recorded and replay in a RobFile, and we will create the final robot program.

Open the **Trimming Blade** example:

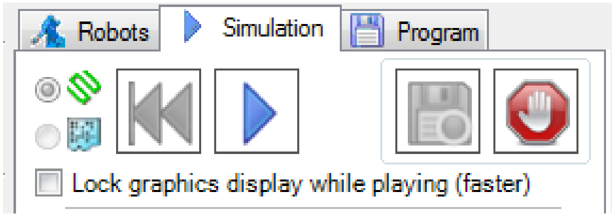
Select and activate the Milling toolpath. Then, select the **Attach a tool to start** button in the **Simulation** tab. The robot will move on the starting point on the selected toolpath:



Make sure that the **Lock graphics display** is unticked and click on the **play** button. The robot will move along the toolpath.

Click on the **Attach a tool to start** to move the robot in the beginning position, then select **Start recording the simulation....**

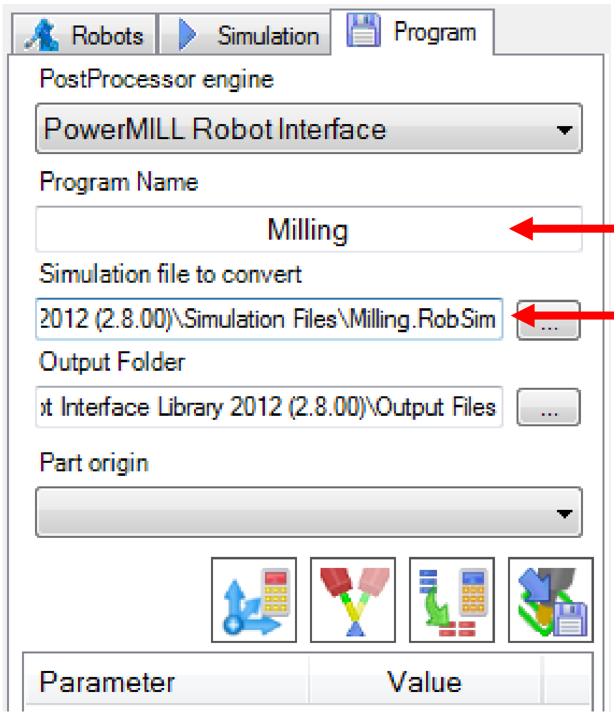
Select a name for the RobFile. This file will contain all the move sequences. The icon is grey, and the Stop recording button is enabled:



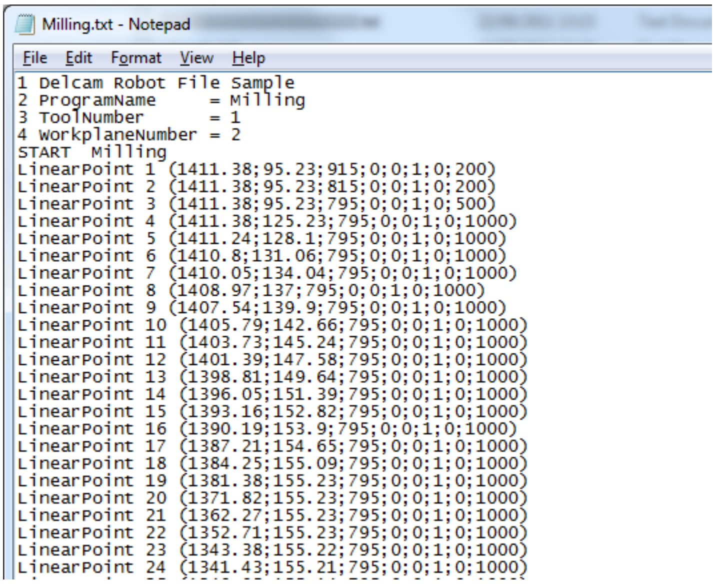
Tick the **Lock graphics display** option and click on **Play.** The simulation will be recorded into the RobFile. The **Lock graphics display** option allows saving time (The calculation will be 100 times faster). After the simulation, click the **Stop recording** button.

To replay the simulation, use the toolbar in the bottom of the screen. The **Start**, **Pause** and **Go to the beginning** options are used to replay the simulation. The **Loop** option on the right will play the simulation in a loop.

To create the final robot program, open the Program tab. The program name and Simulation file to convert are already filled with the good parameters (the current simulation):



Use the ... buttons to change the simulation file or the output folder. When everything is ok, simply click the **Write robot program** button to create the final file. The file is written on the disc and the output folder is opened.



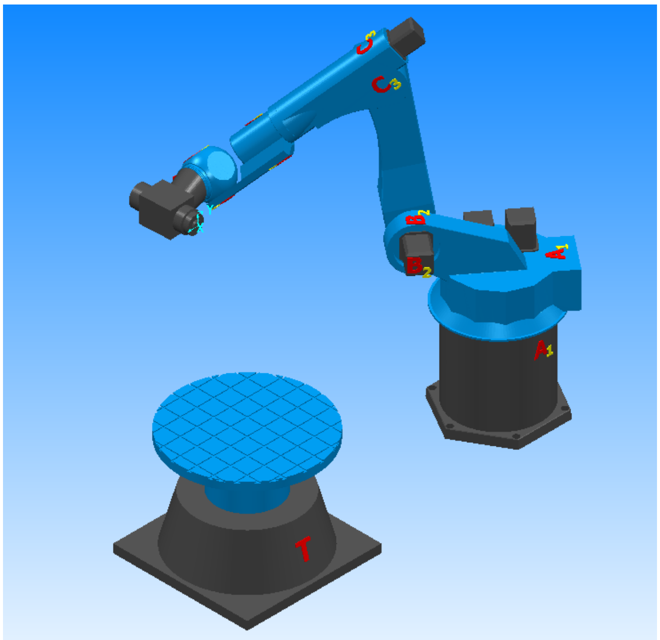
```
Milling.txt - Notepad
File  Edit  Format  View  Help
1 Delcam Robot File Sample
2 ProgramName     = Milling
3 ToolNumber      = 1
4 WorkplaneNumber = 2
START  Milling
LinearPoint 1  (1411.38;95.23;915;0;0;1;0;200)
LinearPoint 2  (1411.38;95.23;815;0;0;1;0;200)
LinearPoint 3  (1411.38;95.23;795;0;0;1;0;500)
LinearPoint 4  (1411.38;125.23;795;0;0;1;0;1000)
LinearPoint 5  (1411.24;128.1;795;0;0;1;0;1000)
LinearPoint 6  (1410.8;131.06;795;0;0;1;0;1000)
LinearPoint 7  (1410.05;134.04;795;0;0;1;0;1000)
LinearPoint 8  (1408.97;137;795;0;0;1;0;1000)
LinearPoint 9  (1407.54;139.9;795;0;0;1;0;1000)
LinearPoint 10 (1405.79;142.66;795;0;0;1;0;1000)
LinearPoint 11 (1403.73;145.24;795;0;0;1;0;1000)
LinearPoint 12 (1401.39;147.58;795;0;0;1;0;1000)
LinearPoint 13 (1398.81;149.64;795;0;0;1;0;1000)
LinearPoint 14 (1396.05;151.39;795;0;0;1;0;1000)
LinearPoint 15 (1393.16;152.82;795;0;0;1;0;1000)
LinearPoint 16 (1390.19;153.9;795;0;0;1;0;1000)
LinearPoint 17 (1387.21;154.65;795;0;0;1;0;1000)
LinearPoint 18 (1384.25;155.09;795;0;0;1;0;1000)
LinearPoint 19 (1381.38;155.23;795;0;0;1;0;1000)
LinearPoint 20 (1371.82;155.23;795;0;0;1;0;1000)
LinearPoint 21 (1362.27;155.23;795;0;0;1;0;1000)
LinearPoint 22 (1352.71;155.23;795;0;0;1;0;1000)
LinearPoint 23 (1343.38;155.22;795;0;0;1;0;1000)
LinearPoint 24 (1341.43;155.21;795;0;0;1;0;1000)
```

The program language varies for each robot manufacturer.

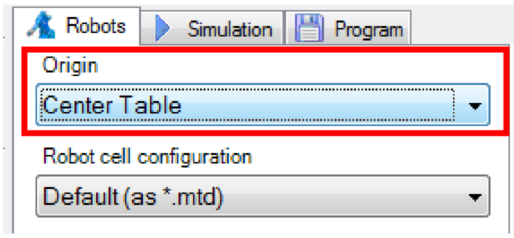
Advanced simulation analysis

This tool allows the user to graph the evolution of the different axis against the time or the point numbers and to collect information about the singularities.

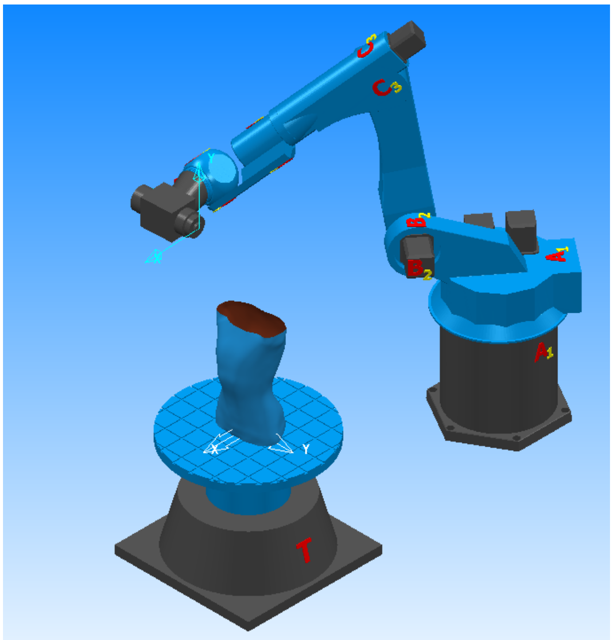
Open the **Rotary table – Body Cast** project.

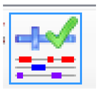


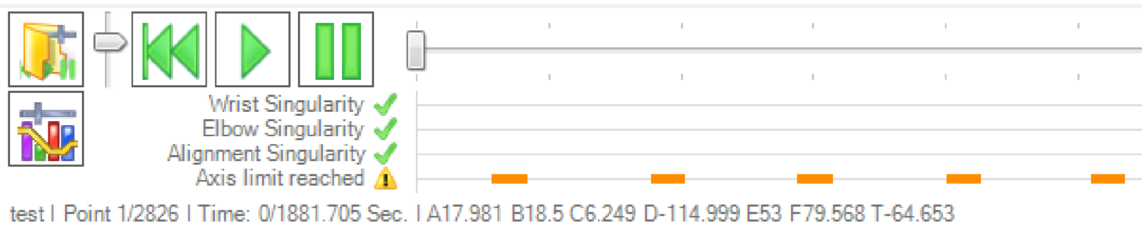
The default origin is wrong (the part is inside the robot base). To put the part on the rotary table, select the **Center Table** origin in the Robots tab:

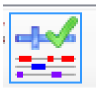


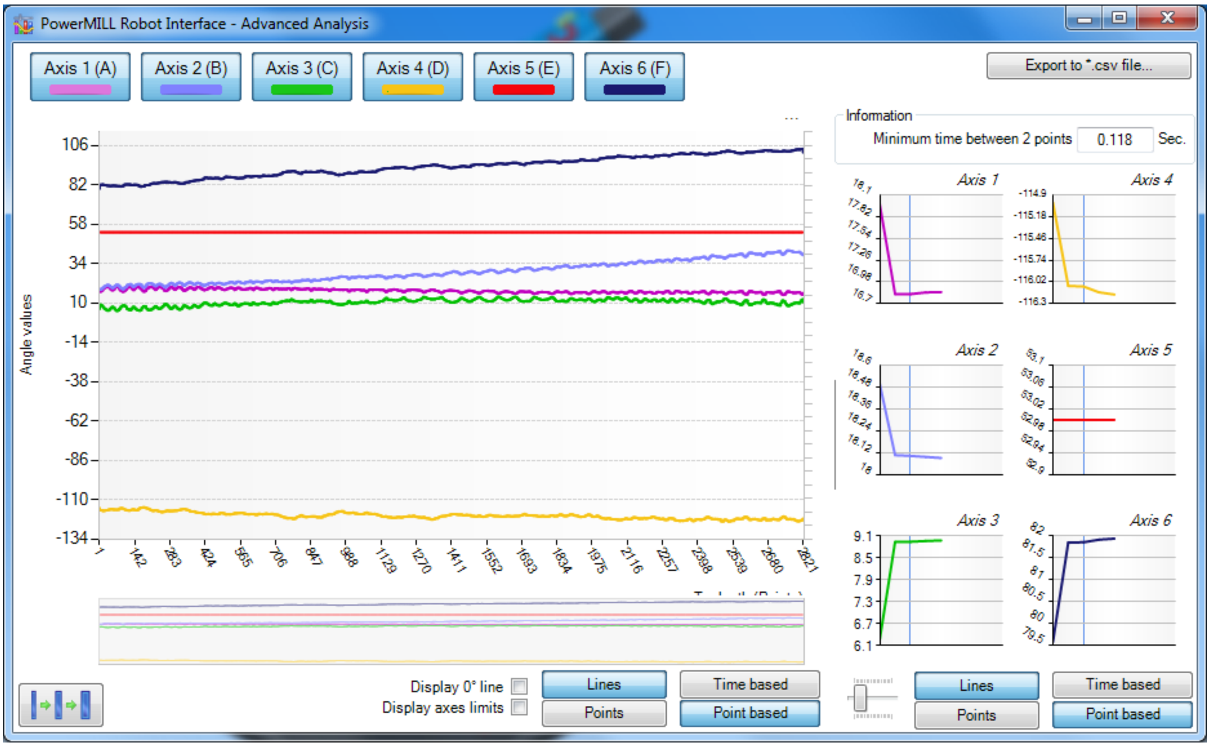
The part is now correctly positioned in the centre of the rotary table:



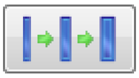
Create and record a simulation of the **Full Bidirectional 45°** toolpath, click on the **Advanced simulation analysis**  in the bottom of the **Simulation** tab. Additional information is displayed under the timeline:



This tool shows information about the singularities and the axis limits. In this example, no singularities are detected and some axis limits are reached. To add more information, open the **Advanced analysis** form by clicking on the  button. This form will appear:

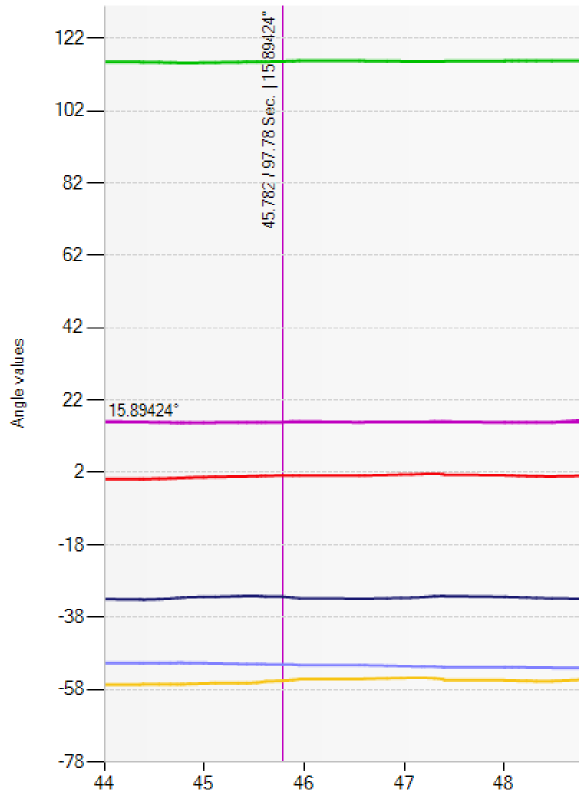


It shows the evolution of axis values against the point's number. To change this, the **Time based** / **Point based** buttons can be used to draw the information against the time or against the points numbers.

Use the coloured buttons in the top of the form to draw/undrawn the curves. The  button can be used to change the drawing thickness. The **display 0 deg line** and **Display axes limits** options can be used too. To zoom on a specific area, use the white rectangle under the main graph:

Drag and drop the rectangle horizontally, and resize it by click and drag the edges to zoom on a specific part of the simulation.

Move your cursor on the curves to display the current axes values:



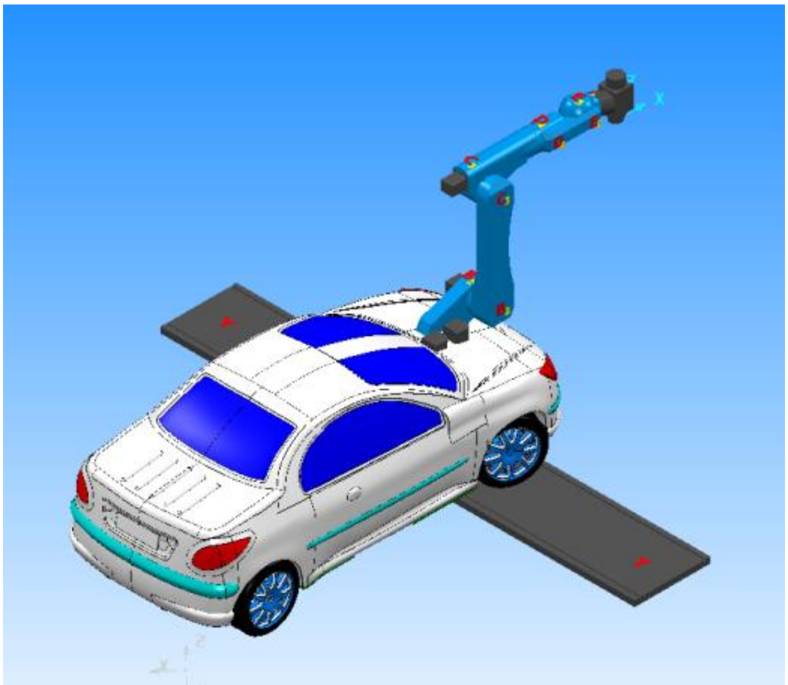
To draw only the points (and not the lines) use the **Lines/Points** buttons to toggle the view.

If you want to export the curves in order to use them in additional software (like Excel), use the **Export to *.csv file...** button.

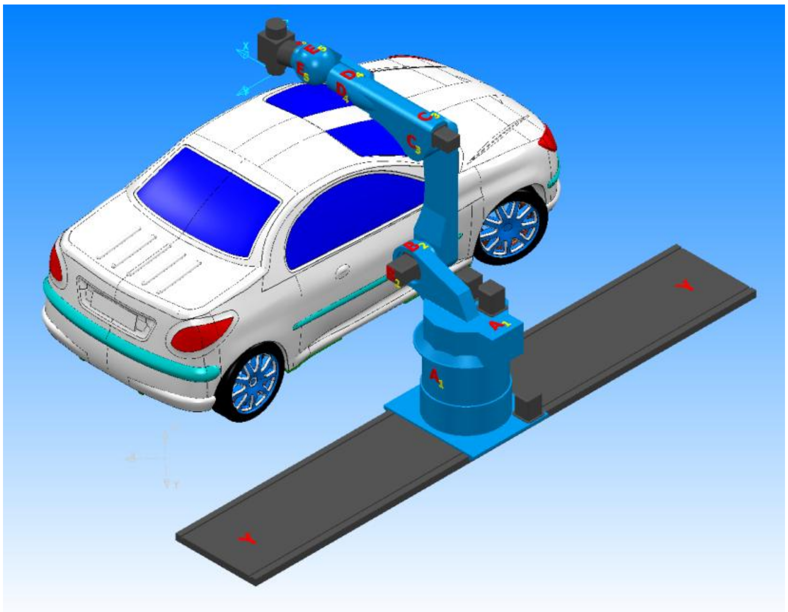
Robot positioning / Axes priorities

In this section, we'll use a robot on a rail to understand how to control the motion of the robot.

Open the Surface **Milling with Rail – Car** project:



As we can see, the robot is not positioned correctly. In the **Robots** tab, select the **Center Right (Rail)** origin. Select an ISO 4 view (Ctrl + 7).

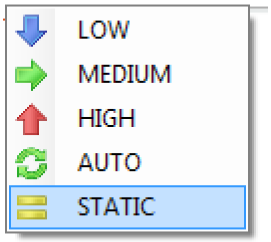


Select and activate the **Back window 3+2** toolpath (in the **Milling** folder), then create and record a simulation. As you can see, the motion range of the Y axis rail is wide. We'll see how to fix that.

It the simulation tab, the axis values are displayed with a priority parameter. By default, for this robot, all axes are in an automatic priority. It means that the solver will use all of them with the same priority.

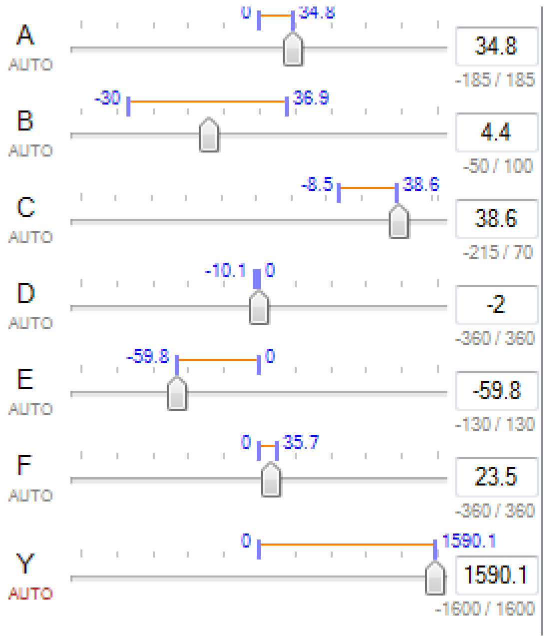
Select and drag the cursor of the Y axis and move it at about 1000 mm.

Right click on the Y letter, and in the menu select **STATIC**.

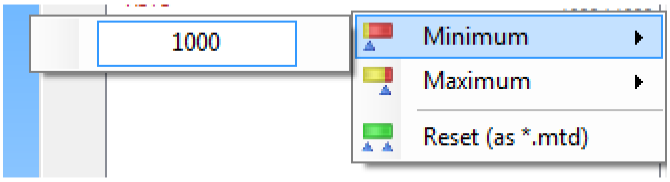


The Y axis is now disabled. Click Attach tool to start and replay the simulation. Now, the A axis is used to move the robot in the correct position. The same result can be obtained with the **LOW** priority settings. A good combination of these priority parameters can be used to get a better robot motion.

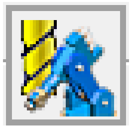
Right click on the Y axis and select the **AUTO** parameter. In the bottom right of the screen, select the **Trace Axis limits** option. Play a simulation of the **Back window 3+2** toolpath. The motion ranges of all the axes are displayed in the **Simulation** tab:

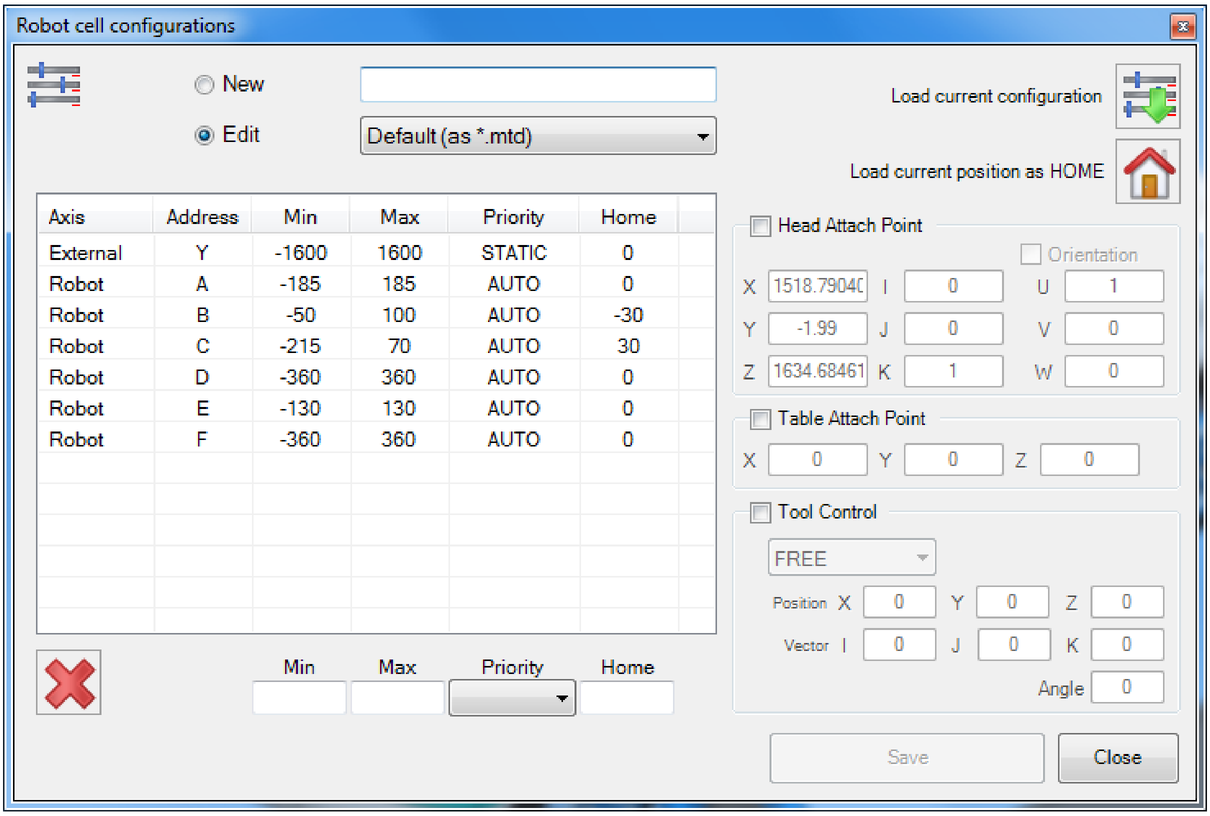


It is possible to apply additional constraints on an axis. Right click on the **-1600/1600** on the Y axis. In the menu, select and apply the value of 1200 for the maximum value and 1000 for the minimum value. Use the Enter key after entering the values.



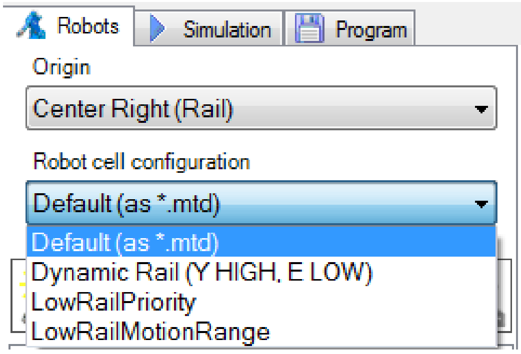
The motion range of the Y axis is now limited from 1 000 to 1 200 mm. To reset the values, select the **Reset (as *.mtd)** option.

In some cases, it can be interesting to keep these values for a further usage. It is possible in PRI to create and store different cells configurations. Right click on the PRI icon  and select Tools/Robot cell configuration editor:



Select the **New** option and type **LowRailPriority** as a name. Select the Y line in the table and change the priority to **Low**, then click the **Save** button. Rename the **LowRailPriority** into **LowRailMotionRange** and modify the **Min** and **Max** values to 1000 and 1200. Restore the axis priority to **AUTO.** Click **Save** and **Close** the dialog.

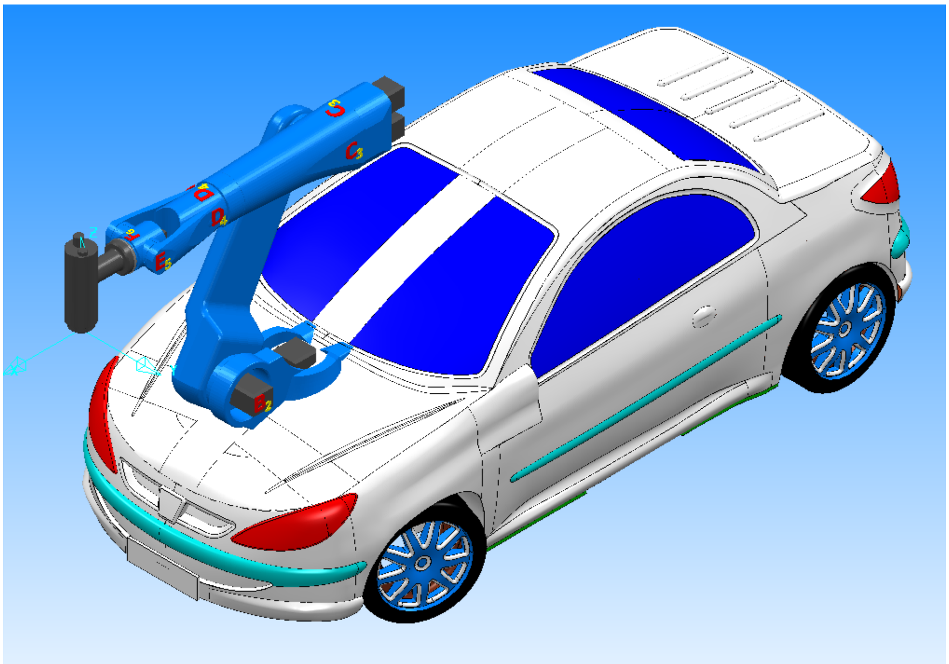
Open the Robots tab and click in the list box Robot cell configuration:



All the cells configurations are listed (and one more previously created). By selecting them, the associated values are loaded.

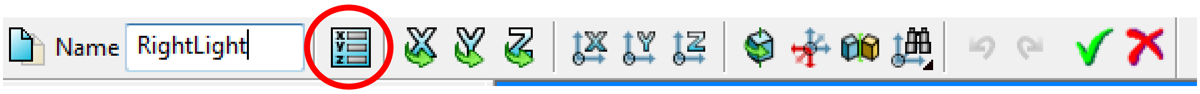
It is possible to save axis priorities/ranges, robots workplace, home position... into a cell configuration. They will be saved in the ROBCONFIG file.

We will now see how to put the robot in a specific place by creating a new workplane. In the **Robots** tabs, click on the **Unload current robot**. In the library, select the "+" before DELCAM and double click on "R2-6X-Spindle0" to add a new robot.



Make sure that the workplane **4** is active. To locate the robot in a correct position, we'll create a new workplane. In the **Explorer**, right click on the **Workplanes** line and select **Create workplane...**

In the workplace toolbar, input the name RightLight then select Position to open the position form.



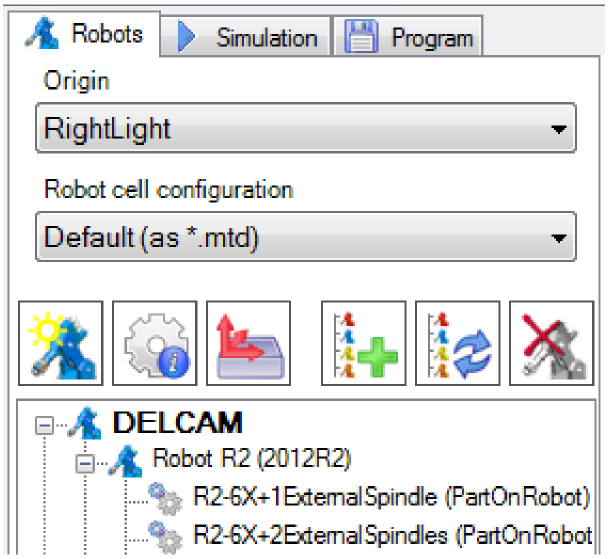
In the form, enter the following coordinates:

X: 500 mm

Y: -1 500 mm

Z: 0 mm

Apply the form and tick the green button in the workplace toolbar. In the Robots tabs, select the **RightLight** origin:



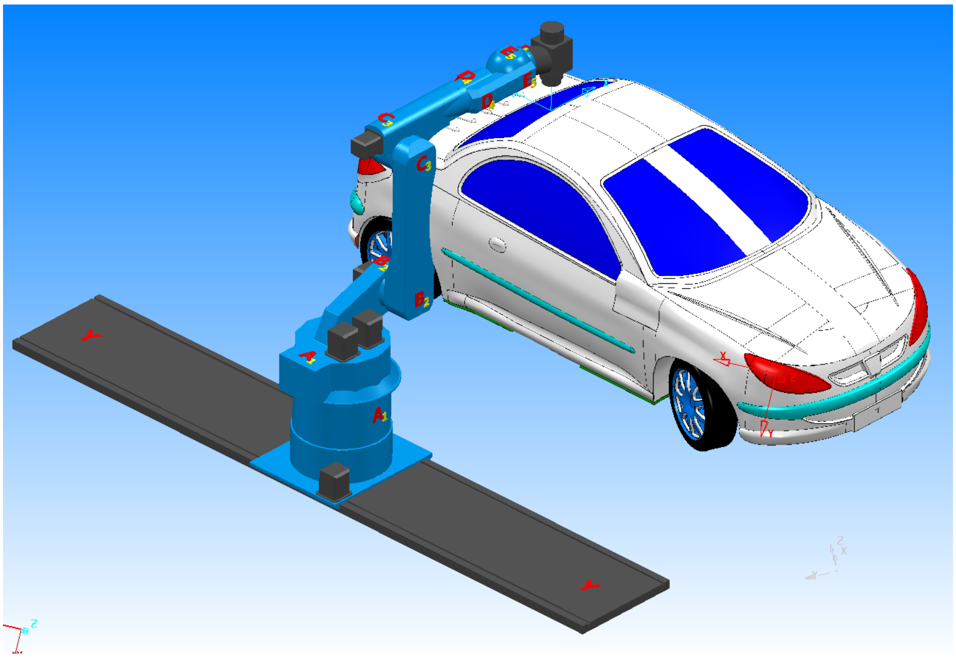
The robot is now positioned to mill the right light of the car.

Exercise: create a workplane to mill the front logo toolpath. Tip: use the **Twist around Z** option in the workplane toolbar to reverse the side of the robot!

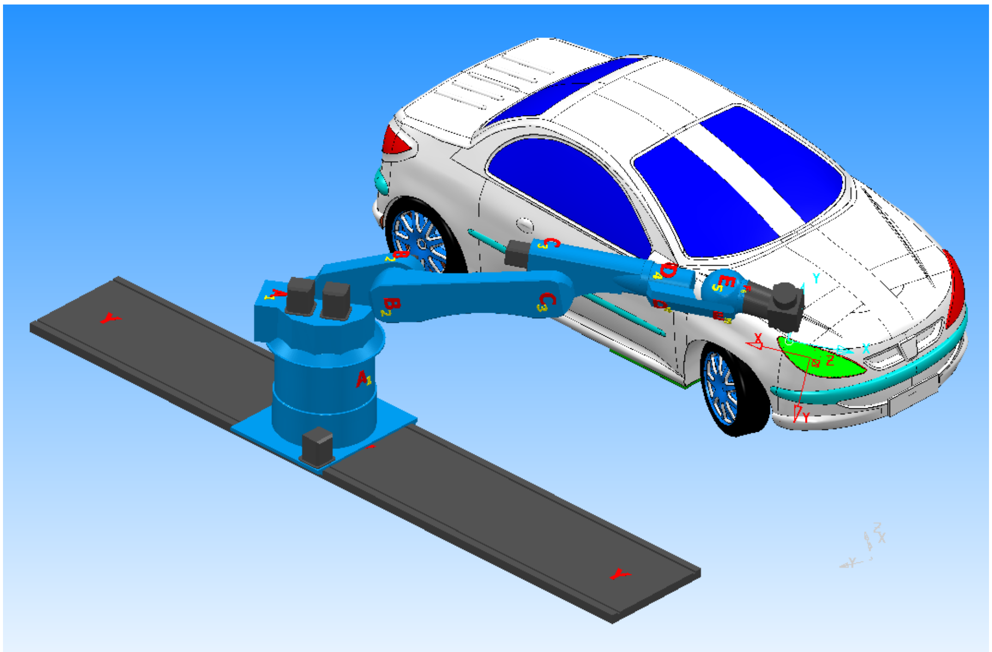
Robot Prepositioning

When you attach a robot to an existing toolpath, sometimes the default robot position is bad. A better position can be created and saved in the considered toolpath.

Open the Surface **Milling with Rail – Car** project and select the **CenterRight (singularity) 2010** origin. WRONG DEFAULT ROBOT ???

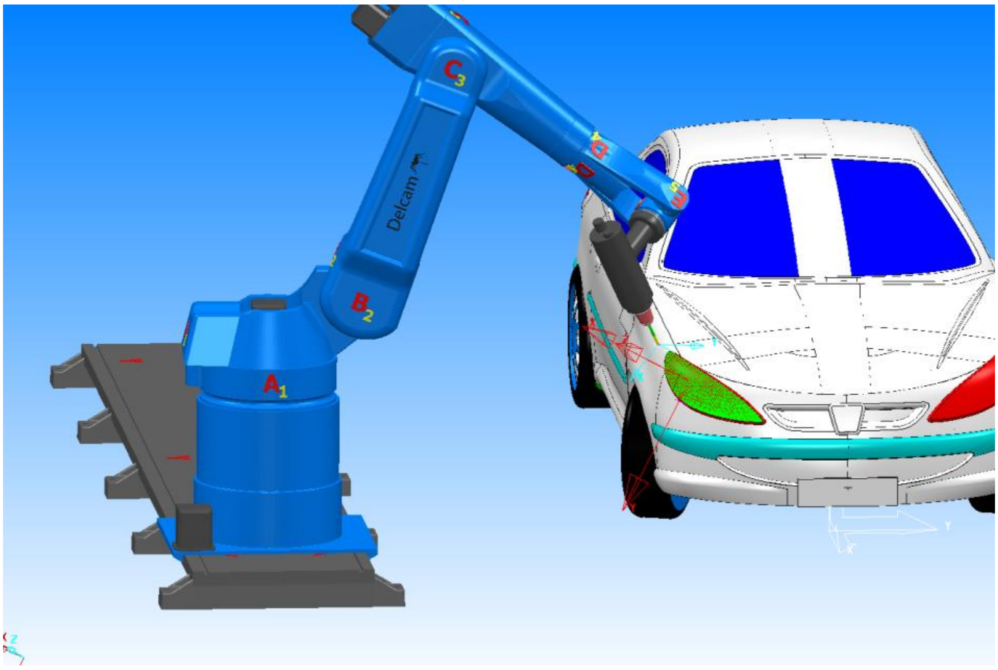


Select and activate the **Right light** toolpath and attach the robot to the tool.



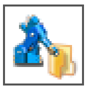
The suggested position is not very good. Hit the Play button. The toolpath cannot be executed, since a singularity appears. It should be better to move the robot on the Y axis and orient the head.

Open the **Simulation** tab and hit the **Home** button, then move the Y axis to a value of about -1400. Attach the robot to the toll and then, in the same tab, **Rotate** the head around the Z axis (Yaw) to put the robot approximately in this position:

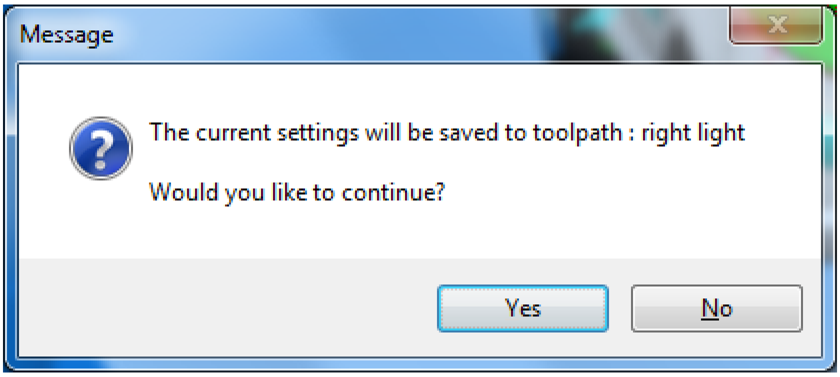


Click again on the **Attach tool to start** button. The robot is moved to a beginning position near as possible from the previous one. Play the simulation. The robot is now able to perform all the milling toolpath without singularities.

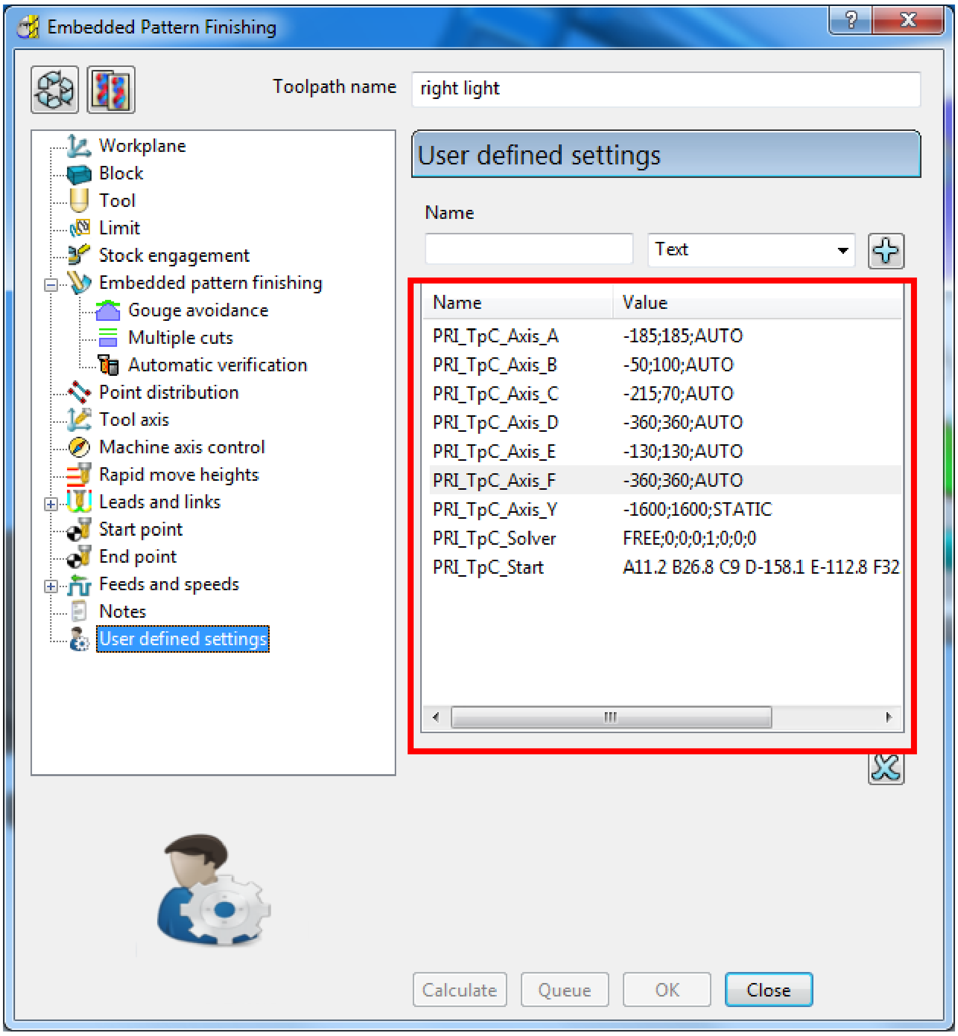
It can be interesting at this point to save the chosen robot position in order to use it later in the project. We will see how to store the position into a file and into a toolpath.

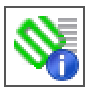
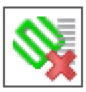
Make sure that the robot is in the good starting position. To save the position in a file, click on the  button is the **Simulation** tab. You'll be asked for a file name. Choose **RightLight** for the RobPos file name. Hit now the **Home** button (or manually move the robot in another position). To restore the previous position, click on the  button to restore the contents of the previous file. The robot is moved into the previous position.

To save the chosen position into a toolpath, hit the  button. A message box will ask for confirmation:



Say **Yes**. The robot position is saved as user parameters in the current toolpath. In the explorer, right click on the **Right Light** toolpath and select **Settings**... and **User defined settings**. The values are stored here:



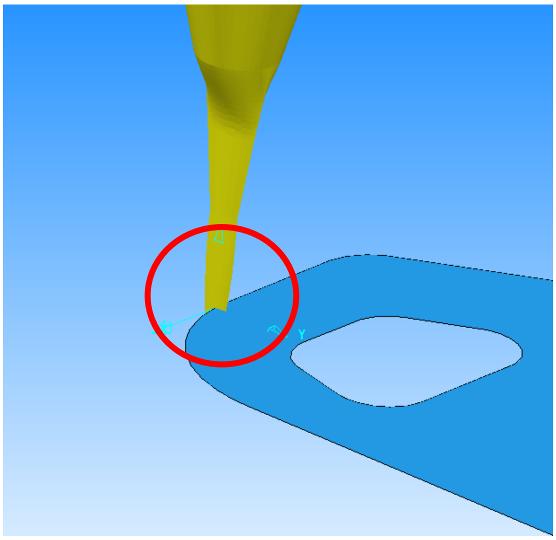
Select **Close** to hide the form. This information can be also checked by using the  button. Use  to restore the stored information. To clear them, use the  button.

Tool orientation control

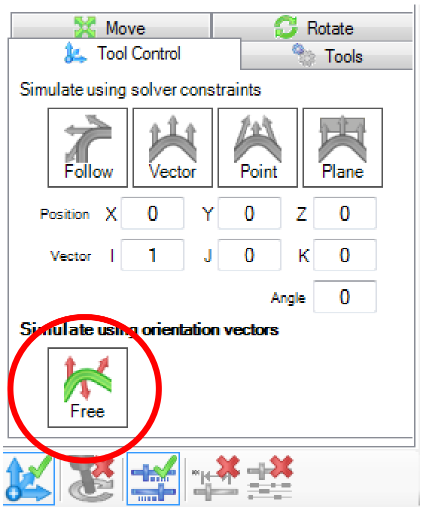
In this section, we will see how to control and change the orientation of the robot head. By default, the "free" orientation mode is selected  . It means that the robot will try to reach the best posture. But sometimes it is necessary to control the head orientation with more accuracy.

Open the **Trimming Blade** example:

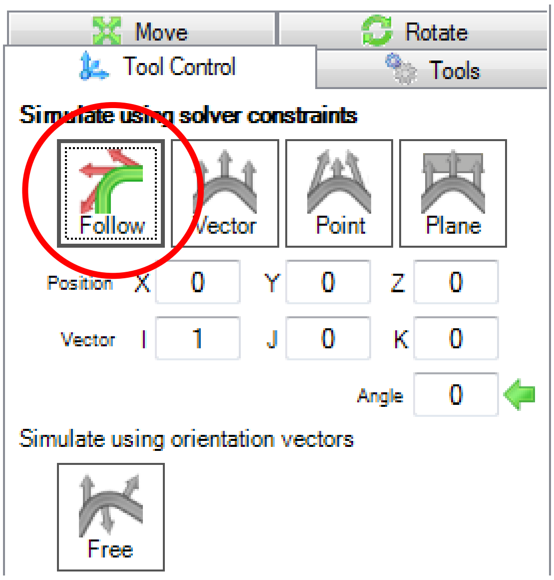
Select and activate the **Knife** toolpath and perform a simulation. The result is bad as the robot head (the F axis) doesn't follow the edge of the part:



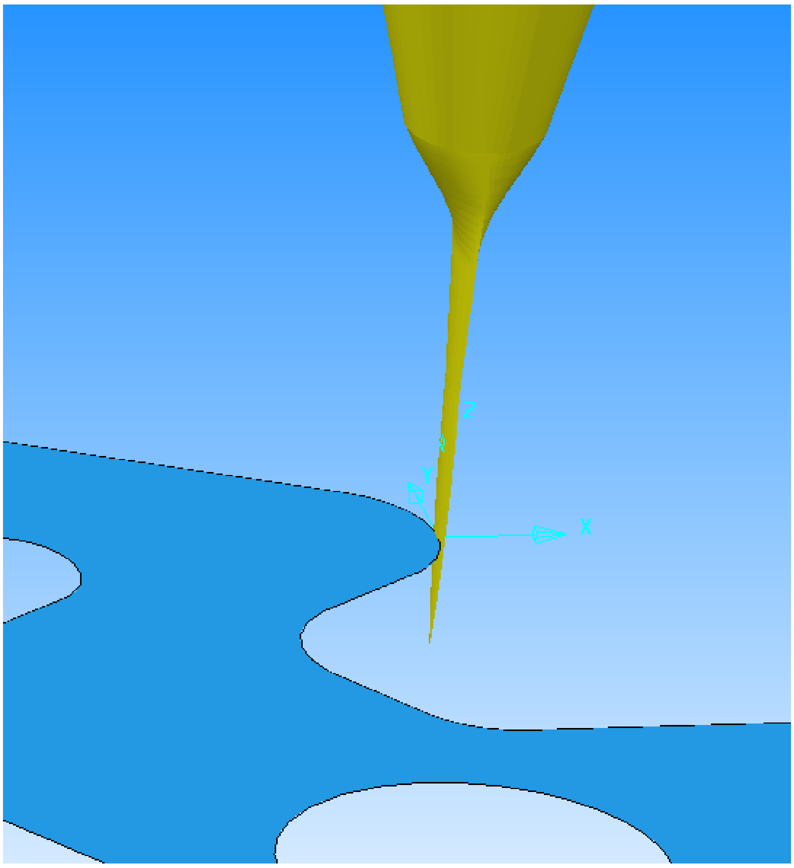
Open the **Tool control** tab in the **Simulation** tab. The robot is actually in the **Free orientation** mode:



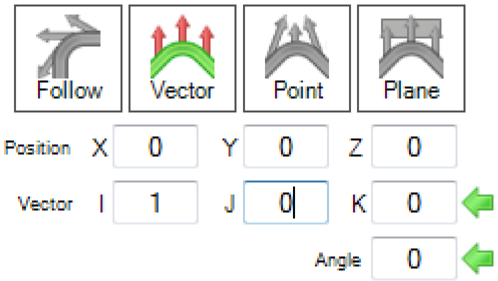
We will add orientation constraints in order to force the knife to follow the tool path. Select the Follow option and perform a simulation:



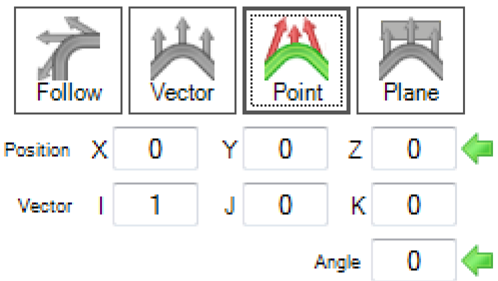
Now, the robot head will follow the toolpath, but with the wrong orientation. To orientate the tool in the correct position, we will have to add an additional angle parameter. The green arrow in the previous figure shows where it is possible to add information. By adding the 90 (**followed by enter)** value in the angle textbox, the tool knife will follow perfectly the toolpath:



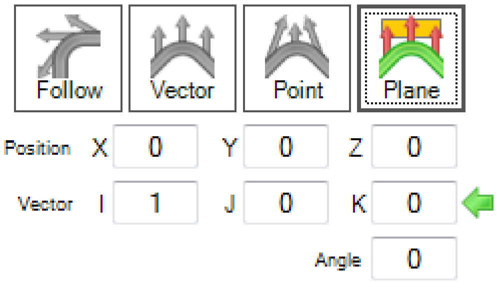
The other options available to add constraints on the solver are :



Vector: Keep a constant orientation of the head.

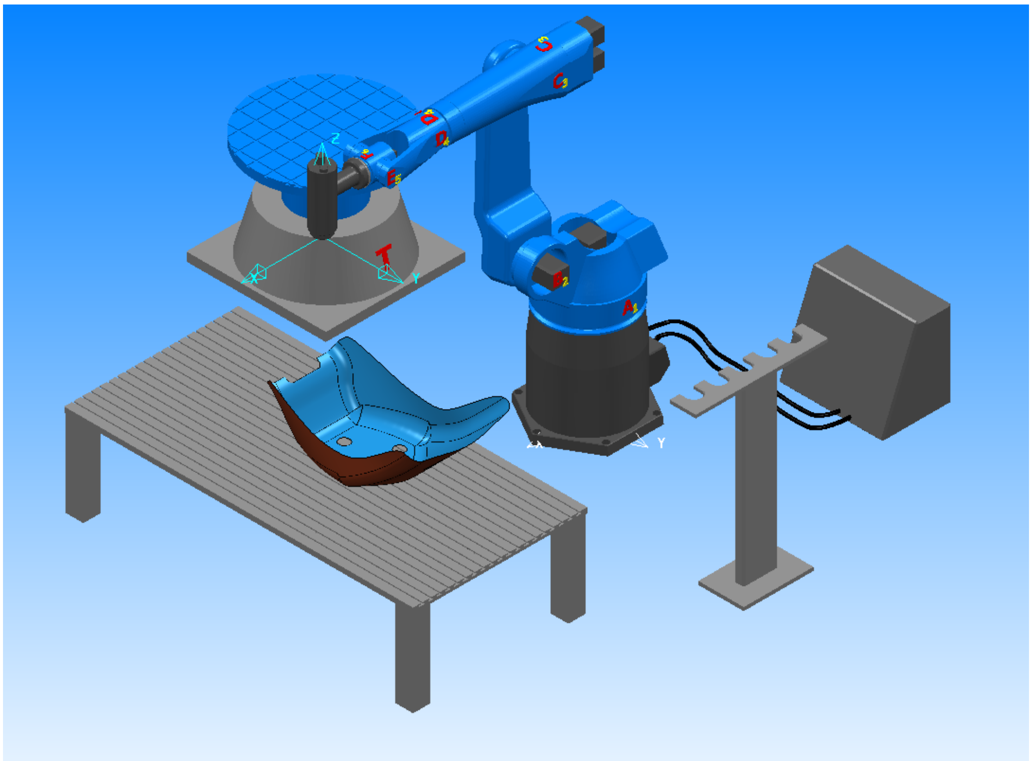


Point: The head will point on a specified point.



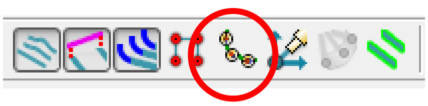
Plane: Keep the head perpendicular to a plane (What is the difference between the “vector” orientation and this one?)

Another tool is available if you want to change manually the orientation of the head of a specific area on an existing toolpath. Delete all and open **the Surface Trimming – Bucket Seat** project:

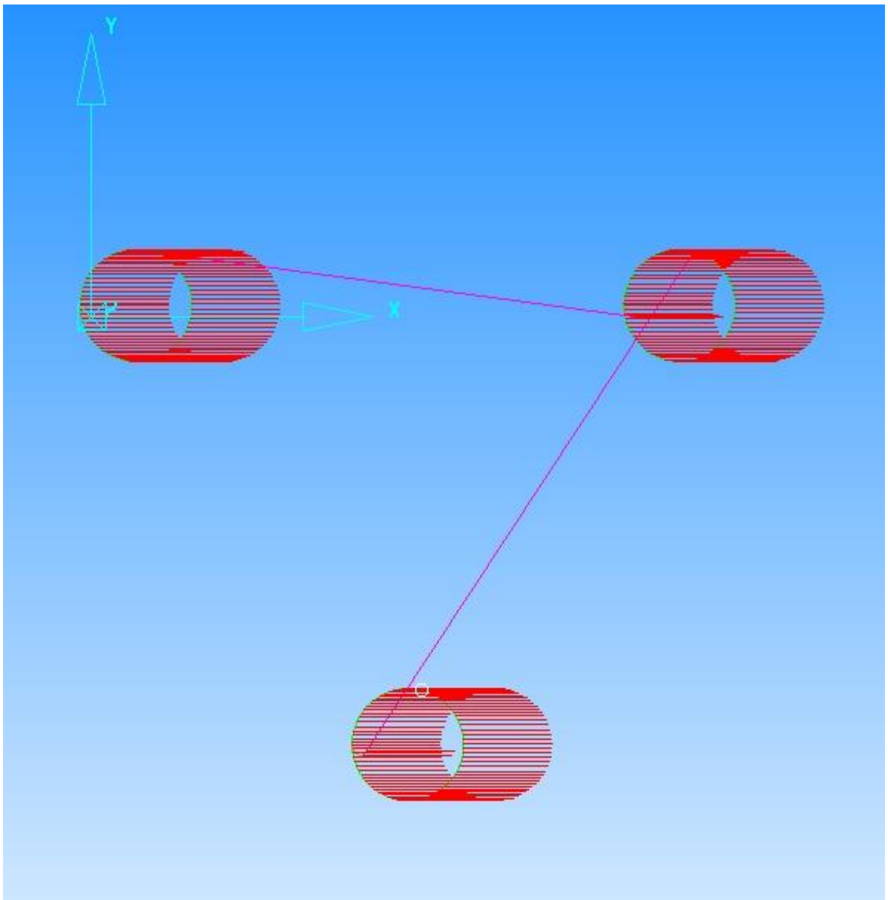


Create and perform a simulation on the **Holes (original)** toolpath. The simulation works perfectly, but we will modify the head orientation for each hole, in order to put the robot wrist in the middle of the part.

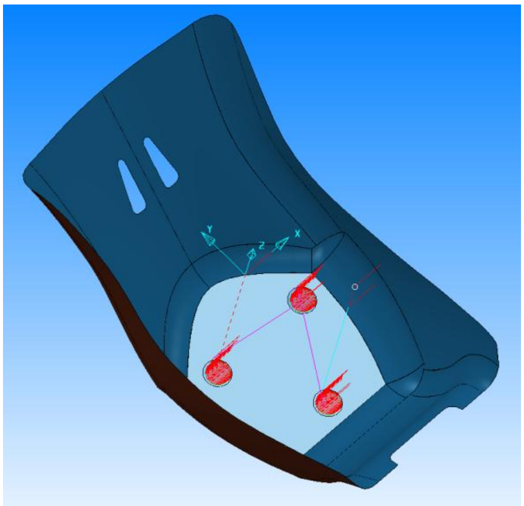
Select a **top view** (Ctrl+5) and hide the part, the robot and the **EndMill10** tool. Select and activate the **Holes (original)** toolpath.



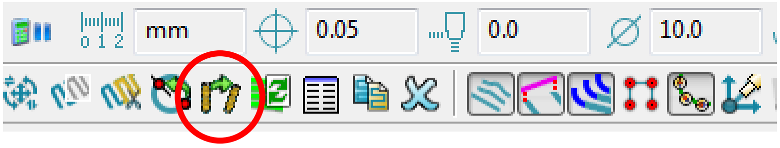
Use this button shown in red (toolbar in the bottom of the screen) to draw the tool orientation. The red line shows the direction of the x axis:



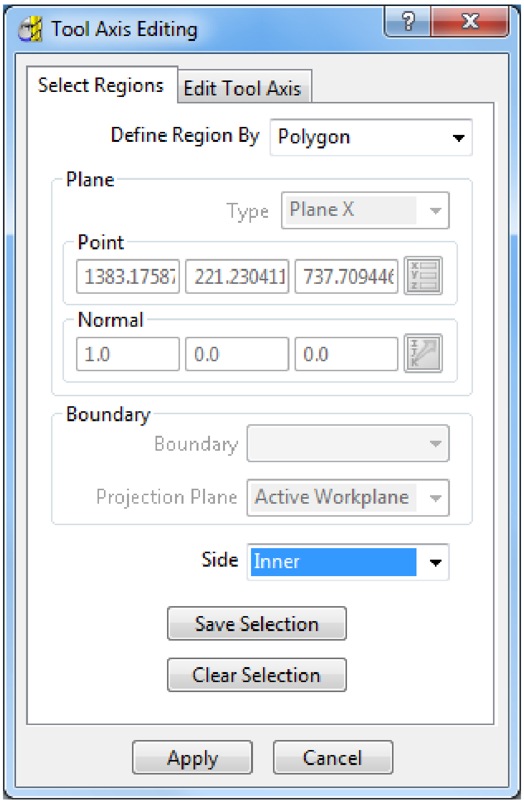
But for this part, the better should be to orient the robot head in the middle of the part.



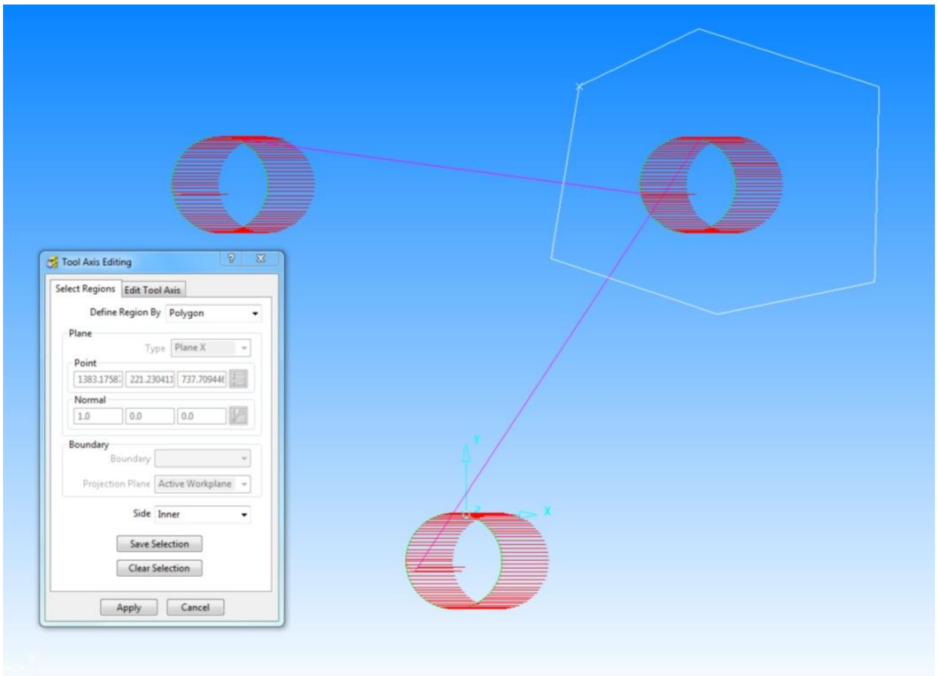
We will use the **Tool axis editor** feature. This tool is activated by clicking on this icon in the bottom of the screen:



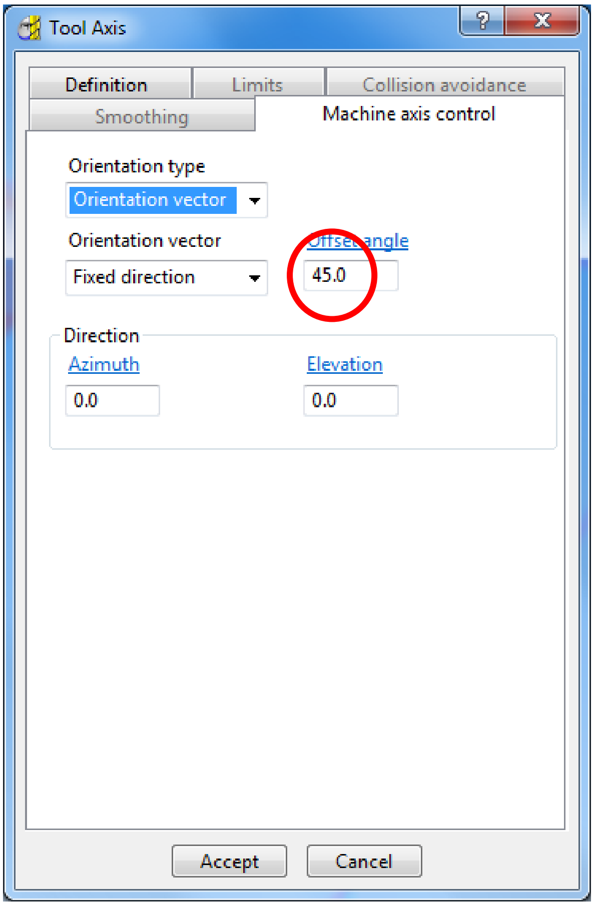
This form will appear:



The first step consists in selecting the area to modify. Select Define region by **Polygon** and click on the main view to create a polygon around the top right hole:

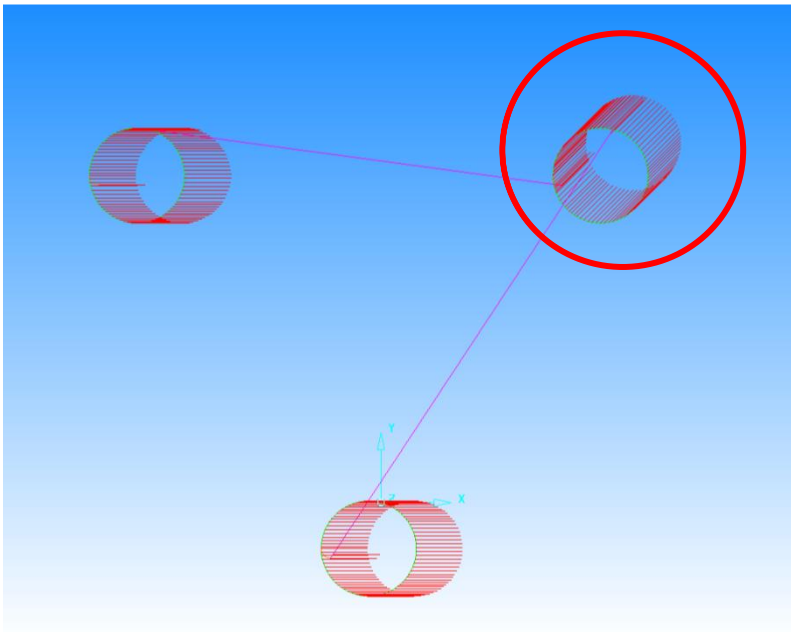


Select the **Edit Tool Axis** tab and select the **Tool Axis button**,  then select the **Machine axis control** tab:

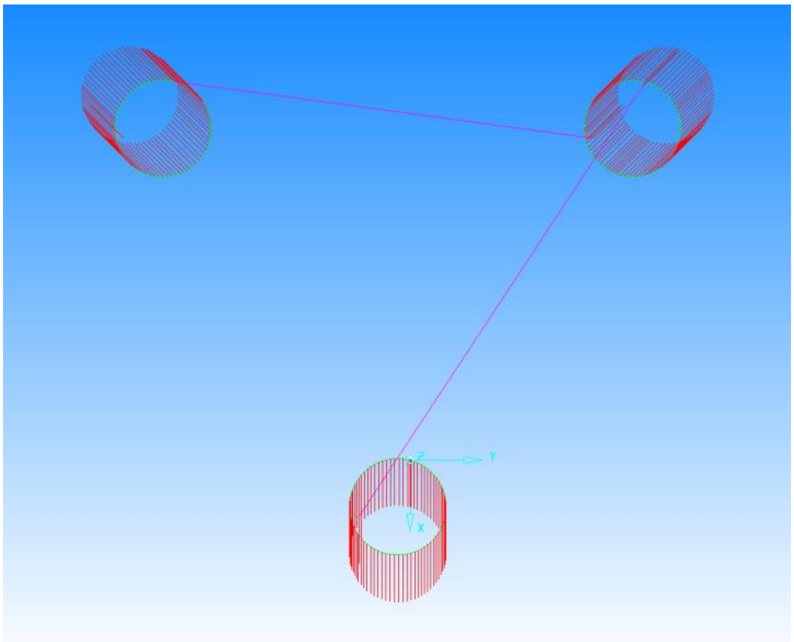


Select an **Orientation vector** orientation type and a **Fixed direction**. Add an **offset angle** of 45 deg in order to change the head orientation.

Click Accept and Apply. The tool orientation is now in the correct direction:



Use the same method to modify the two other holes. Use an offset angle of **-90** for the bottom hole and **135** for the top left hole. The final result should look like this:



Perform a simulation to see the modifications in the robot head orientation.

Tool calibration

TODO: How to add the user menu