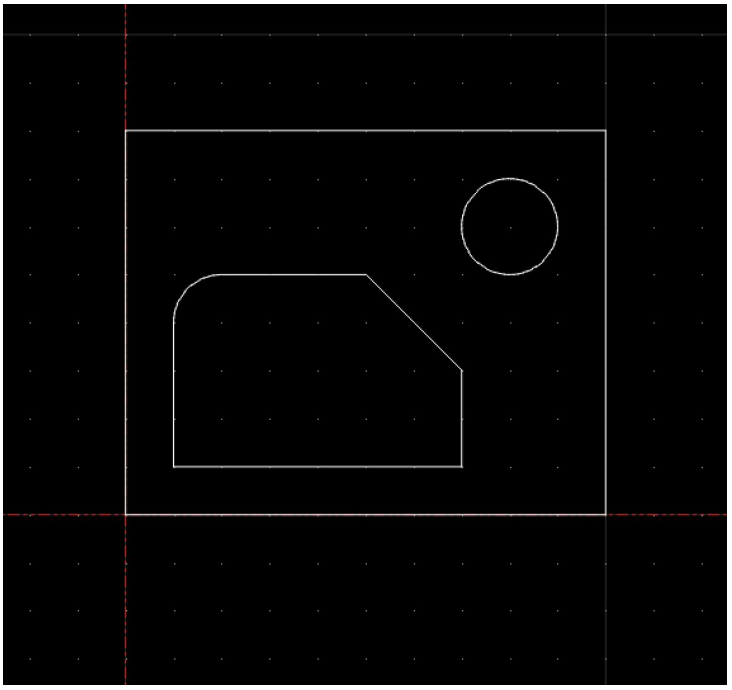
QCAD/CAM Tutorial

Caution should be exercised when working with hazardous machinery. Simulation is no substitute for the careful verification of the accuracy and safety of your CNC programs. QCAD/CAM or any other software can make mistakes. Programs that appear correctly in the simulations can be very dangerous for operation on real machines. The safety and correctness of any CNC program when working on a real machine is the sole responsibility of the CNC machinist.
QCAD/CAM is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.

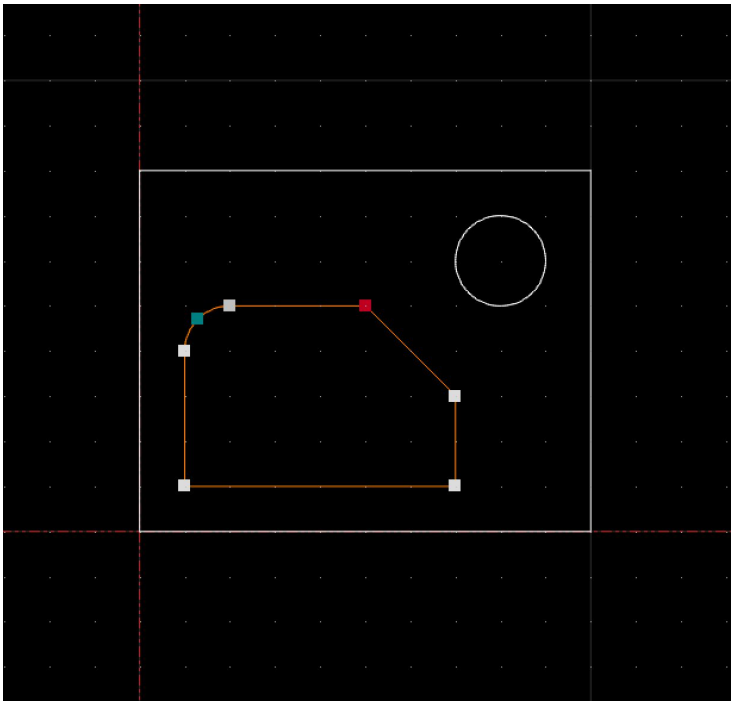
Content

Creation, Preparation and Selection of the Geometry

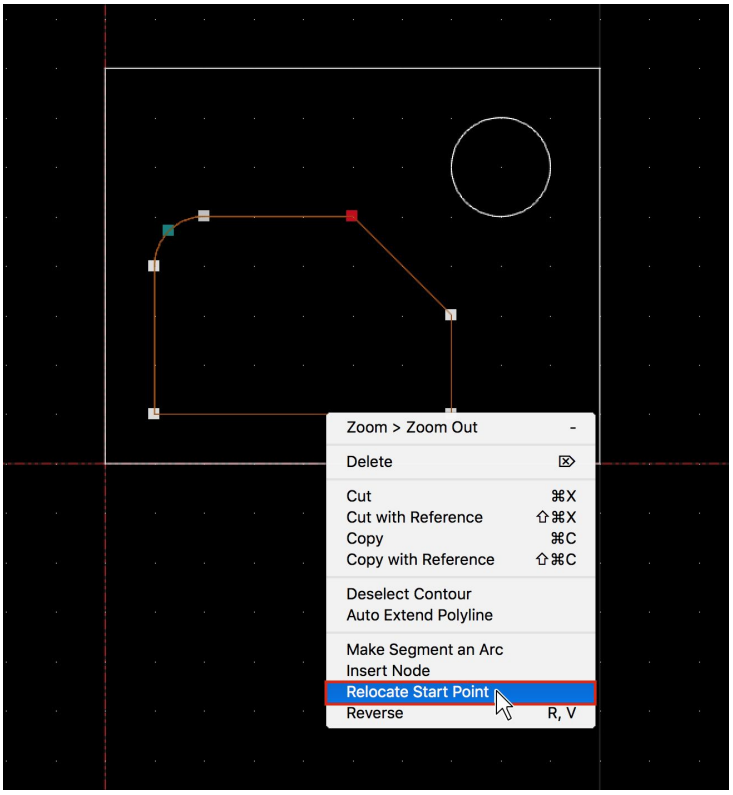
1. In the first step, the polylines are created which represent the geometry that has to be cut:



2. All contours must have a suitable start point. This must be a start point at which the contour can be approached using a lead in. For our example, this is not the case for the inner contour (the red reference point is the start point of the selected polyline):

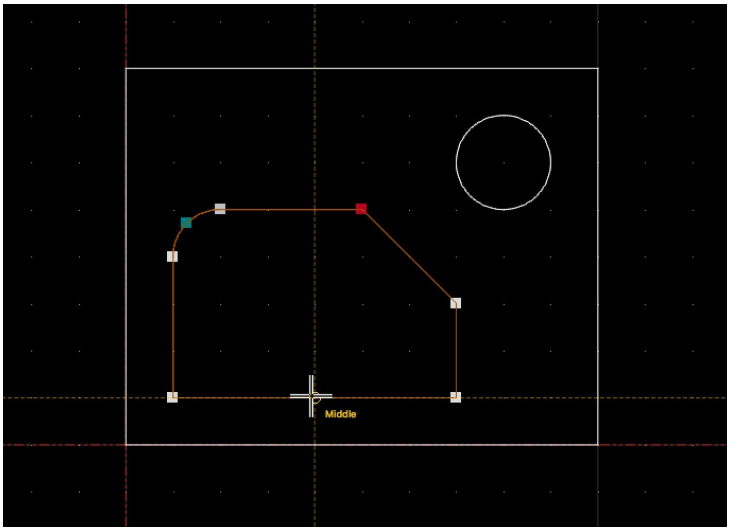


By right-clicking on the contour, the start point can be set to another position:



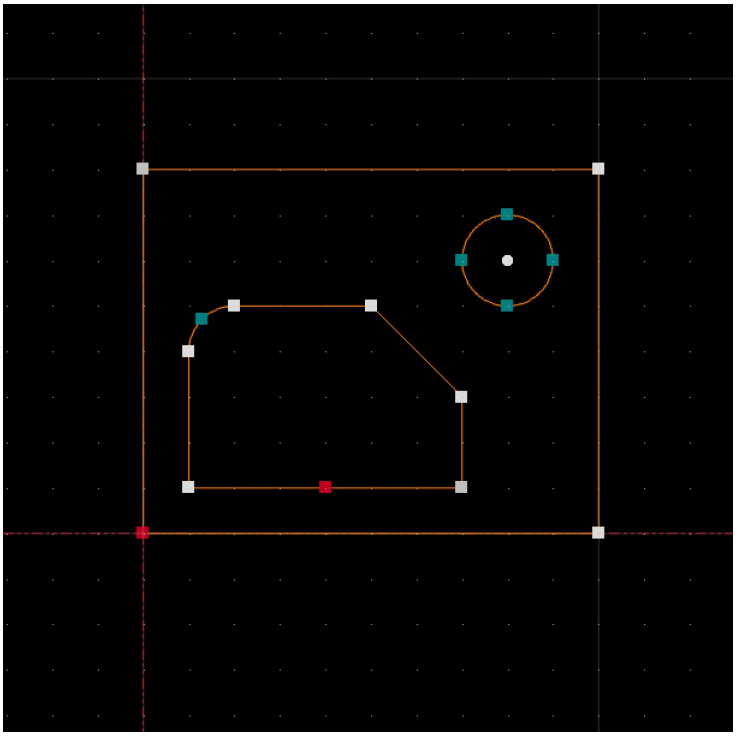
Click on the new position of the start point. This can be an existing node, or any other point on the polyline. Here the center of the lower segment is selected:

All contours are now ready for processing. With the function *Select* > *Select All* all start points of all contours can be highlighted (red):



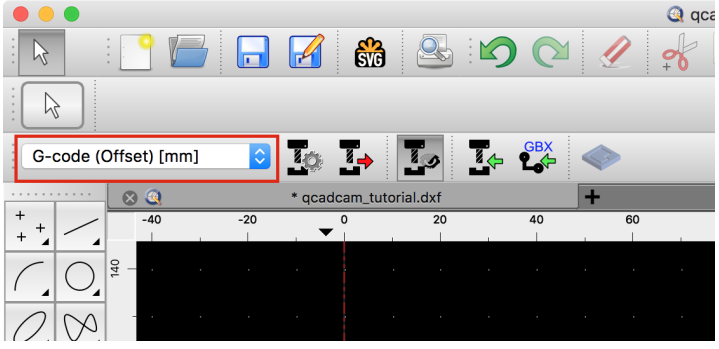
The circle is not a polyline, and may have its start point at any point.

3. Select all the contours that are to be processed with the same machining parameters and the same tool. In this example, these are all the contours in the drawing:



Choosing a Configuration

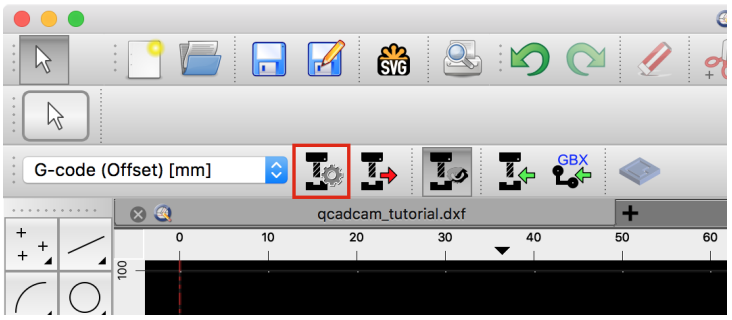
1. Choose the desired configuration at the top left:



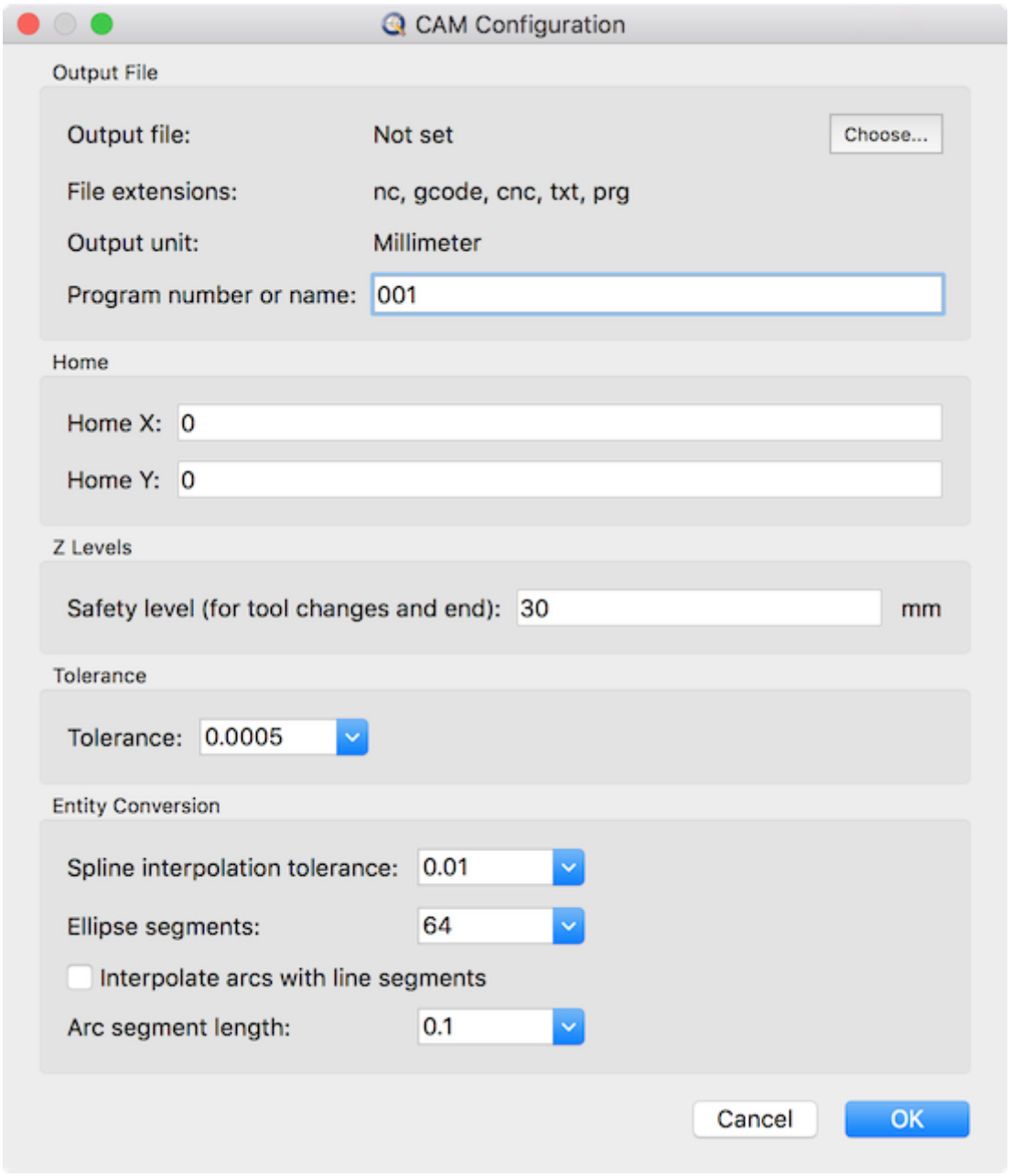
   - *G-code (G41/G41) [mm]*
     This configuration can be used when the controller supports the commands G41 and G42 for radius compensation.

- *G-code (Offset) [mm]*
  This configuration can be used if no G41/G42 commands are available or G41/G42 should not be used. QCAD/CAM computes the tool path of the tool center based on the tool radius automatically in this case and outputs the appropriate offset coordinates.

2. Adjust the parameters of the selected configuration *(CAM > CAM Configuration...)*:



3. Enter the desired parameters:

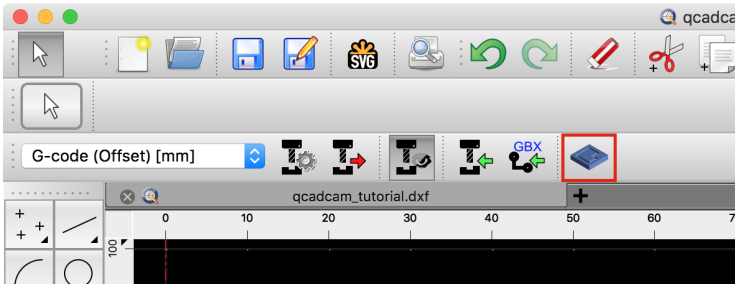


The output file can be set here or later when the output is generated for the first time.
The safety Z level is used at the beginning and end of the program as well as before tool changes.
Splines are interpolated as tangentially connected arcs within the given tolerance.
Ellipses are interpolated as arcs with the given number of segments (per full ellipse).

Arcs can optionally be interpolated as lines with the given segment length if the targeted controller does not support circular movements.

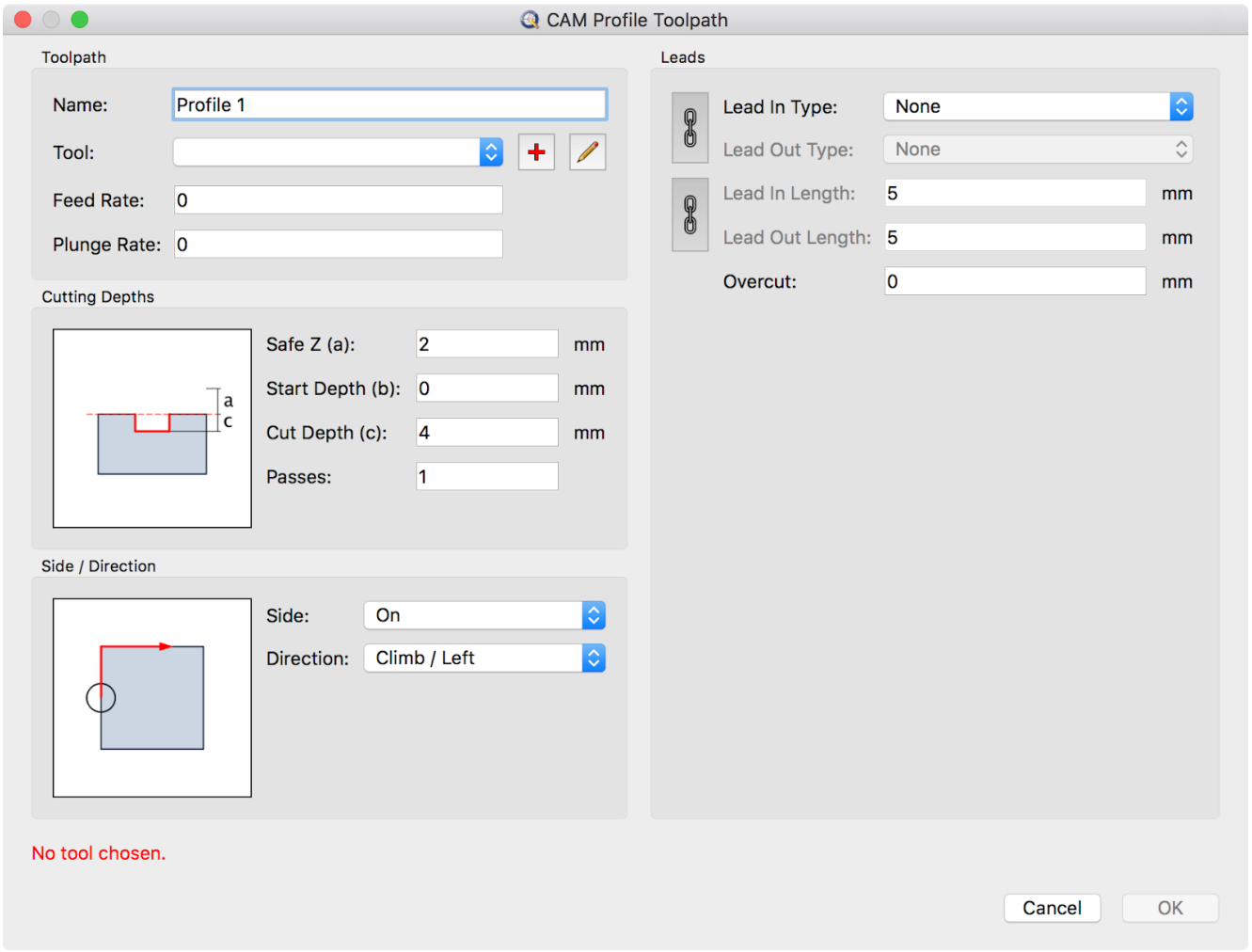
Create a Profile Toolpath

1. Start the tool to create a profile toolpath *(CAM > Add Profile Toolpath)*:

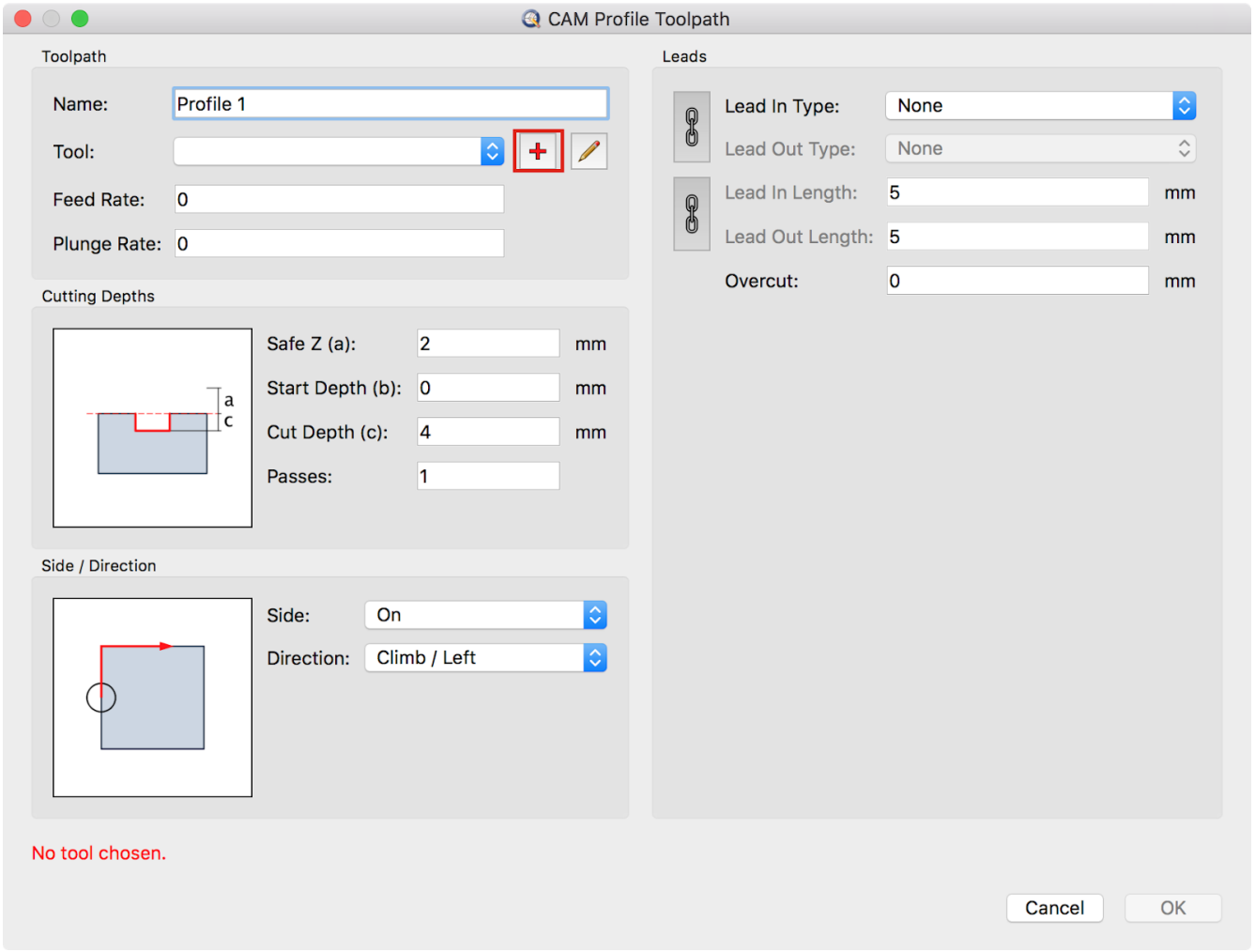


In future releases, other types of toolpaths will be available, for example for pockets.

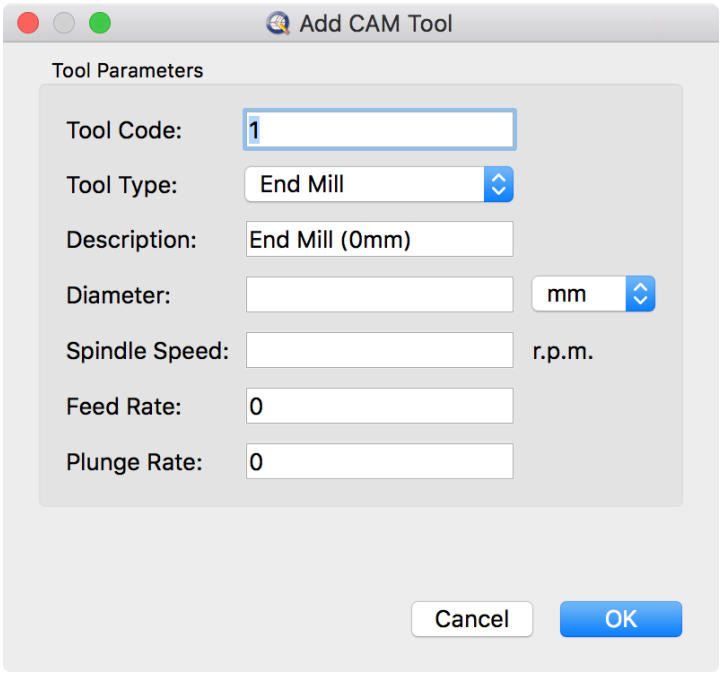
2. The dialog with profile toolpath parameters appears:



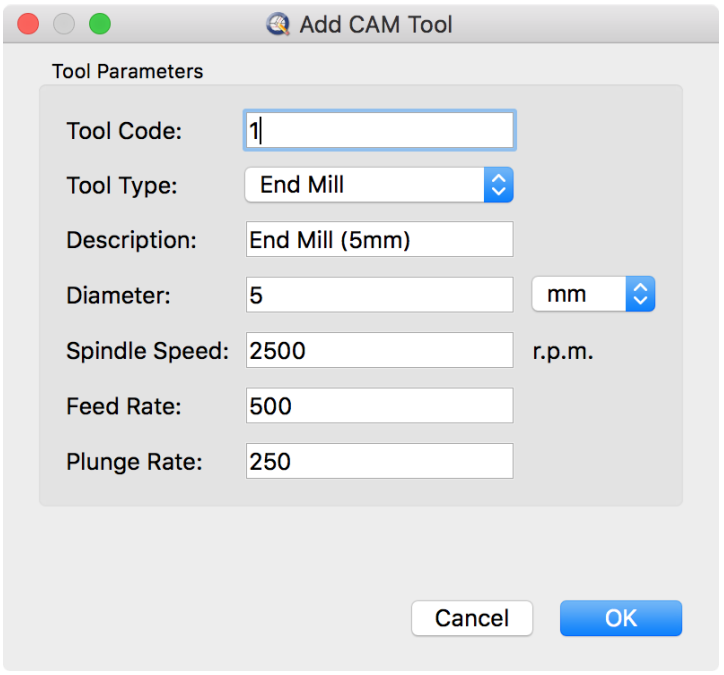
3. At the moment there are no tools defined yet, so at tool must be added first by clicking the button to add tools:



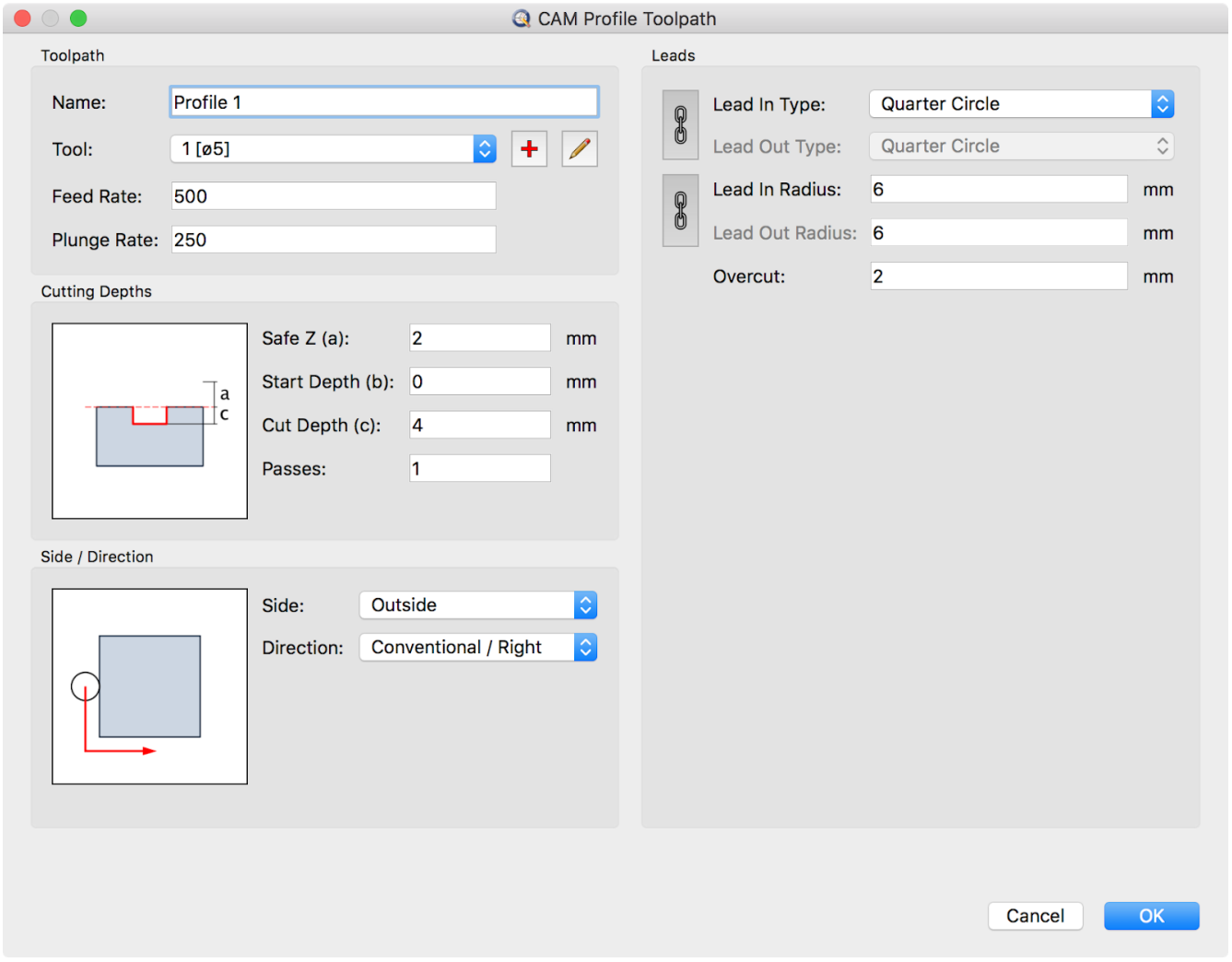
4. The dialog to add the new tool is displayed:



5. Enter the desired tool parameters and click *OK*:



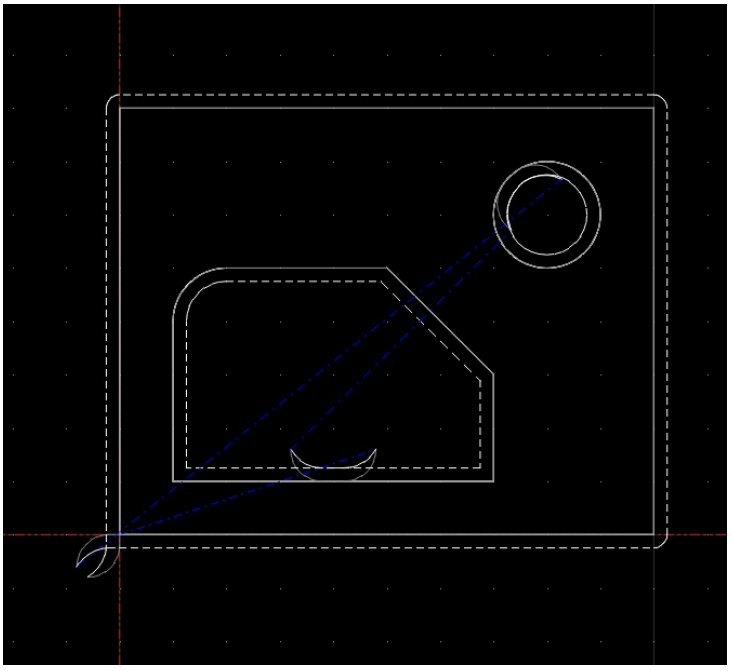
6. Adjust the parameters for the profile toolpath. For contours with radius compensation (side: *Outside* or *Inside)* a lead in path and a lead out path must be used. In this example, these lead paths are quarter circles with a radius which is slightly larger than the tool diameter:



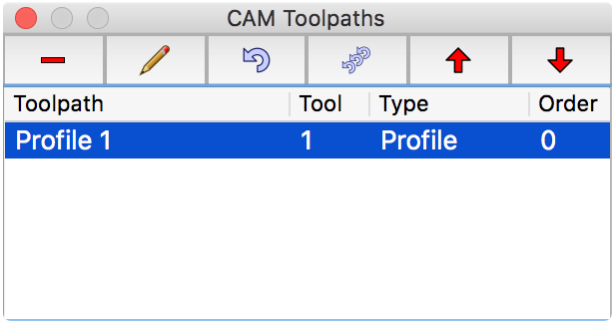
The *overcut* indicates by how much overlap is used at the start and end when processing the contour.

7. Click *OK* to generate the toolpath for this profile.

8. The toolpath is generated and displayed accordingly:



9. All generated toolpaths are also listed in the widget CAM Toolpaths (*CAM* > *Show CAM Toolpath List*):



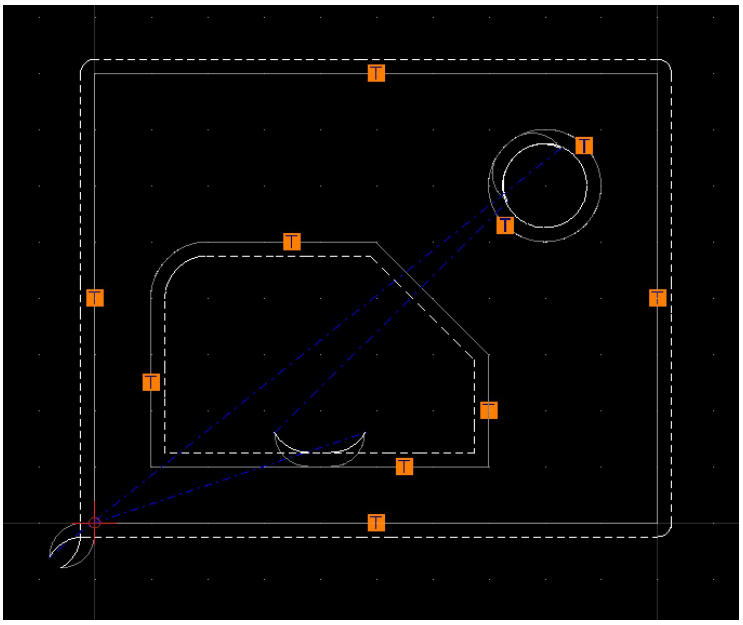
10. When G-code is generated, these toolpaths are processed in the order shown.

Working with Tabs

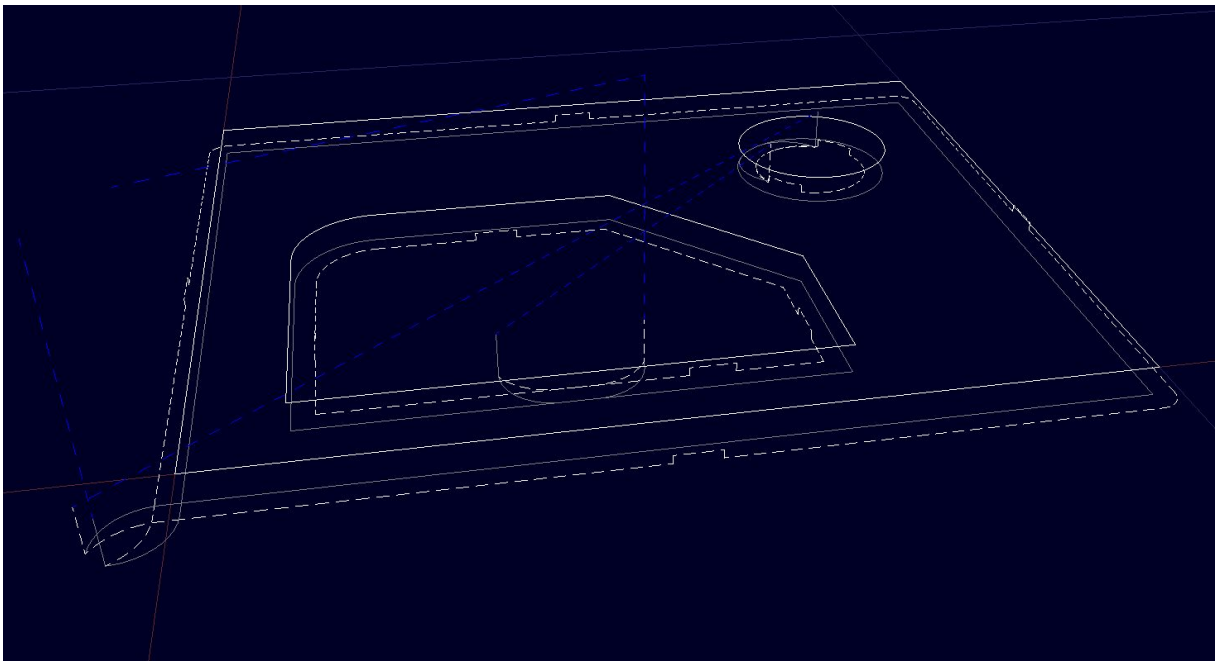
When cutting holes into sheet material, the material inside the hole becomes loose when the cut is almost complete. Such a loose piece of material can move into the cutter and get damaged in the process. Tabs are small connections between a cutout shape and its surrounding material which hold a piece in place while cutting it. Tabs are typically very small, so the piece can be easily broken or cut out of the sheet material once the toolpath is complete. To securely hold a piece in place, three or more tabs are typically used.

1. Create a separate layer for the tabs you want to insert (e.g. layer "tab" with a distinguished color).
2. Click the menu *CAM* > *Add Tab* to insert tabs.
3. Enter the tab length and thickness in the options toolbar. The tab length is the total length of the bridge that is left in place. The thickness is the tab height, measured from the maximum cutting depth of the toolpath.

4. Position the tab on the contour. Note that the tab is always placed on the originally drawn contour, even if cutter compensation is used:

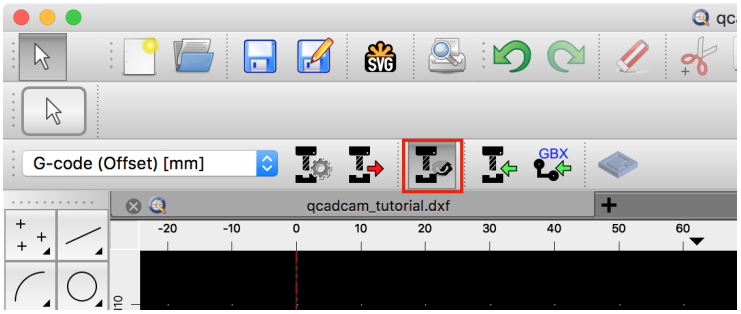


5. Tabs are shown as small squares with a *T* inside.
6. The effect that tabs have on the toolpath can best be observed in the 3D simulation view (see also *Simulation* below):



Showing / Hiding CAM Output

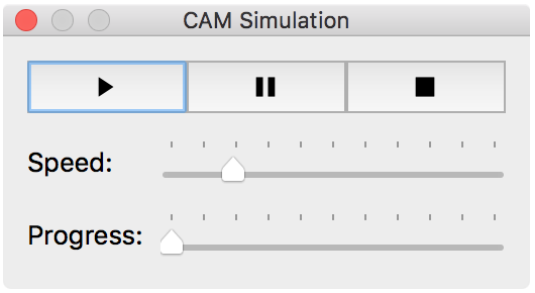
1. The tool paths can be shown or hidden with the button or menu *CAM* > *Show Toolpaths*:



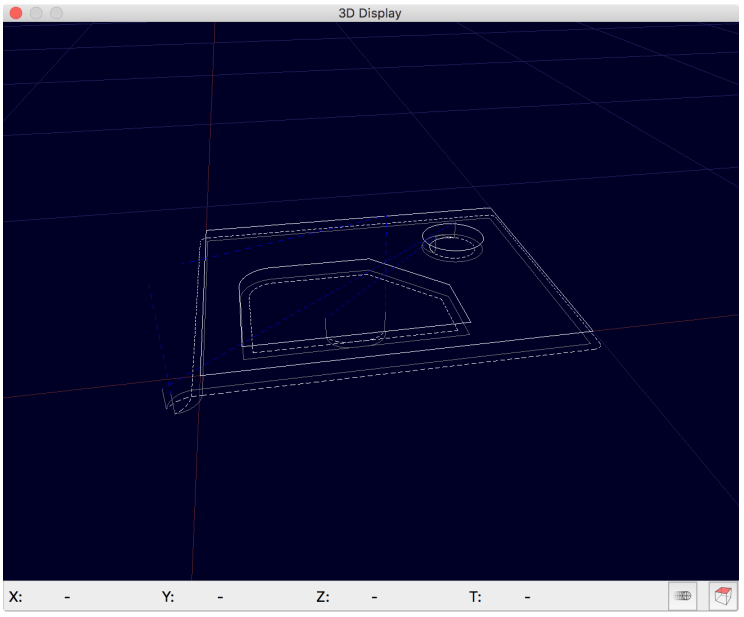
This function toggles the visibility of the layer *CAM,* the layer onto which all toolpaths are generated.

Simulation

1. The generated toolpaths can be simulated using the simulation Widgets *(CAM > Show Simulation Panel)*:

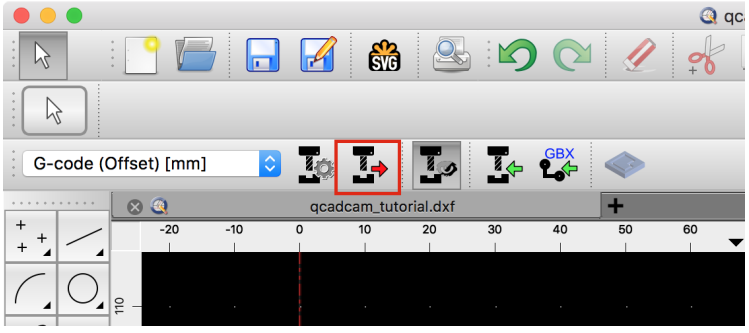


2. 3D movements can sometimes be better observed using the 3D view *(Cam > Show 3D Simulation View)*:



Generating G-Code

1. If the simulation appears to be satisfactory, the corresponding G-code can be generated using *CAM > CAM Export (V2)*:



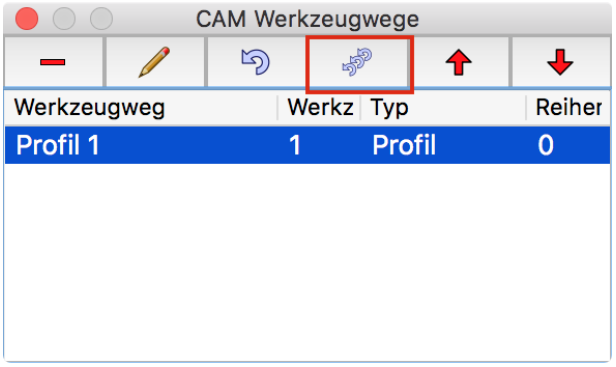
2. When generating G-Code for the first time, you are prompted for a filename for the output file. All future exports will overwrite that file without warning! The output file can be changed anytime in the configuration *CAM > CAM configuration...*.

Adjusting the Geometry

If the geometry is adjusted, there are two ways to update the toolpaths that were generated from the old geometry.

Regenerating Toolpaths

If a polyline has changed, the tool paths can be easily regenerated using *CAM > Regenerate All Toolpaths:*



Regenerating a Toolpath From Selection

To add new entities to an existing toolpath or base an existing toolpath on new entities, the tool path can be regenerated based on the current selection.

1. Click the toolpath from the toolpath list.
2. Select the entities to use in the graphics view.
3. Right-click on the tool path in the toolpath list and choose *Update Toolpath from Selection:*