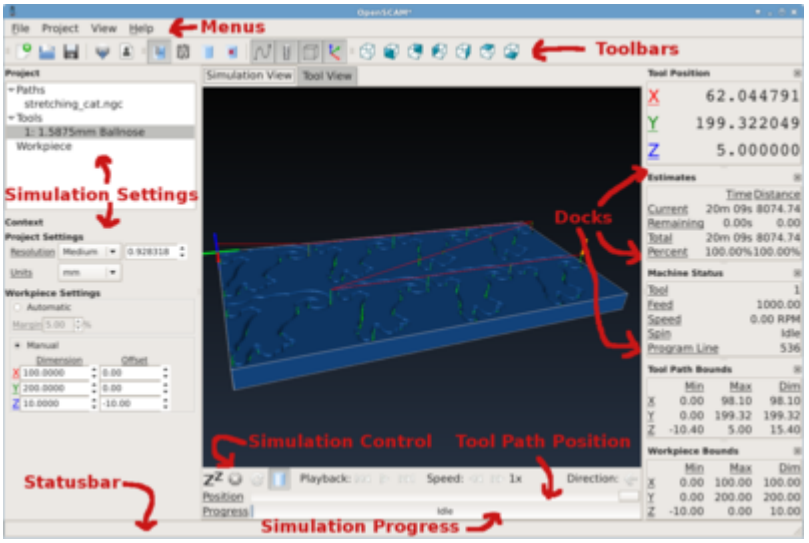
CAMotics

camotics.org/manual.html

Joseph Coffland

Overview

This manual describes how to use the various parts of the CAMotics software GUI. The GUI consists of a set of menus, toolbars, docks, the simulation view and the tool view. These parts are described in detail in the later sections. The graphic below highlights these parts.



Tooltips

All of the parts of the GUI have tooltips which you can view by holding the mouse pointer over component for a few seconds. Tooltips explain the component's purpose.

Projects

CAMotics project files keep track of the three main parts of the simulation:

1. Tool Paths (NC files).
2. Tool tables.
3. Workpiece descriptions.

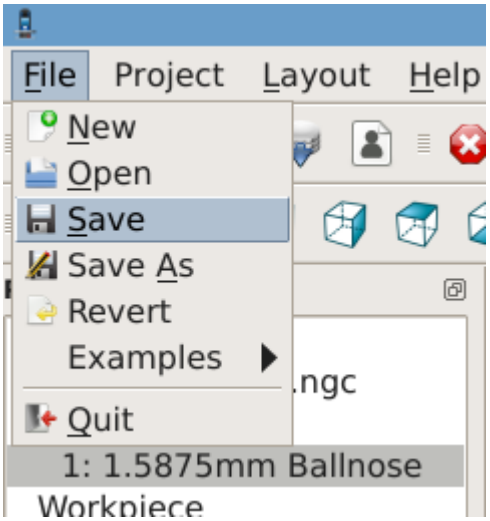
Project files are stored as XML. These files contain links to NC files, so they must be kept together.

If you open NC files directly in CAMotics, i.e. with out a project file, CAMotics will create a default workpiece and tool table automatically. If you make changes you will want to save

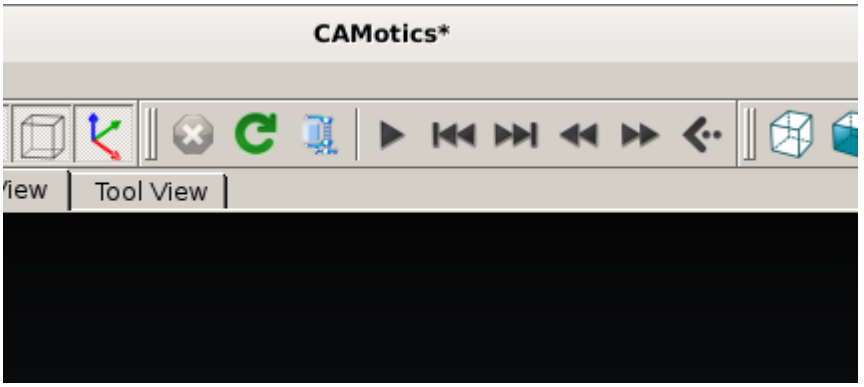
the .xml project file and open it, instead of the NC file, in future runs.

New, Open, Close, Save & Save As

The main operations on project files are new, open, close, save, and save as. These operations can all be accessed through the File menu.



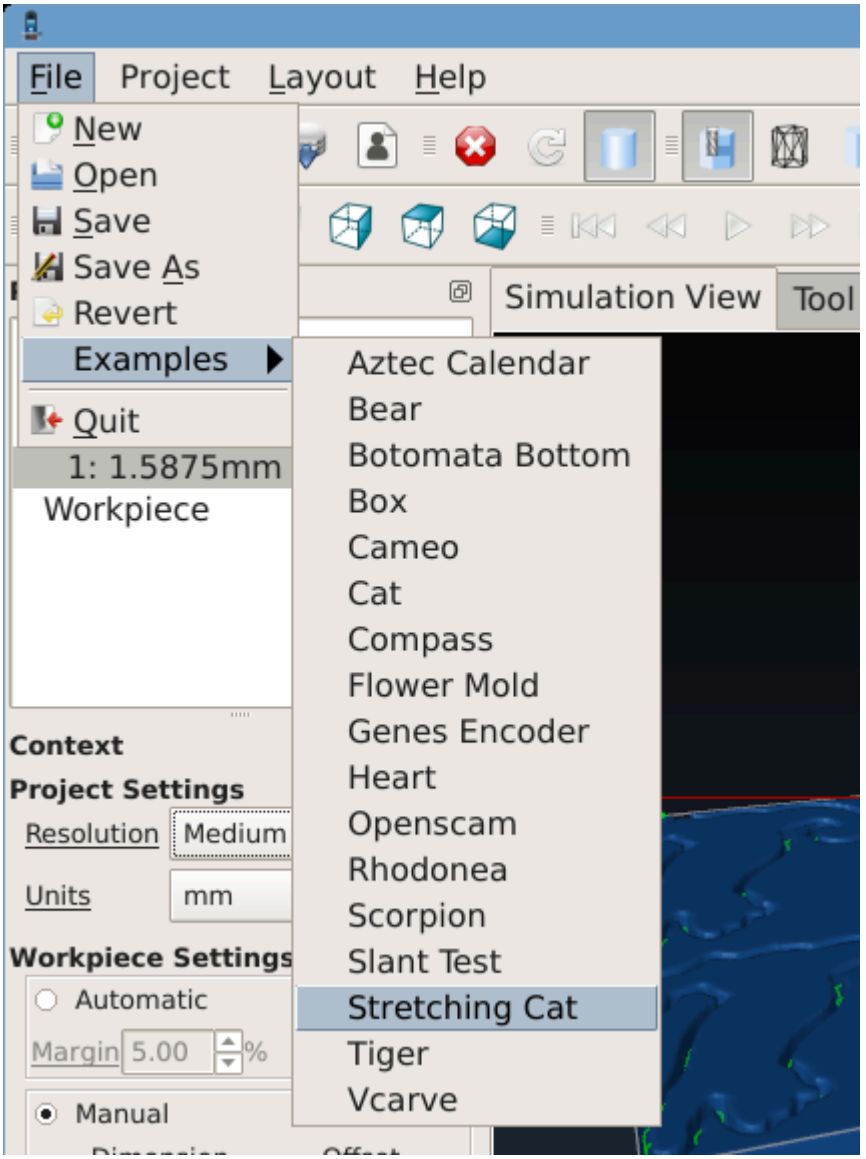
Detecting Modifications



CAMotics will automatically detect any changes to the project. When it does this a '*' will appear in the window's title bar. Any attempt to close the current project, either by exiting the program, or creating or opening a new project will bring up a dialog which will ask you to either save the current project, discard the changes and proceed with the requested operation or cancel.

Viewing the Examples

The File menu contains a submenu of example projects which you can try out to get a better feel for what CAMotics can do.

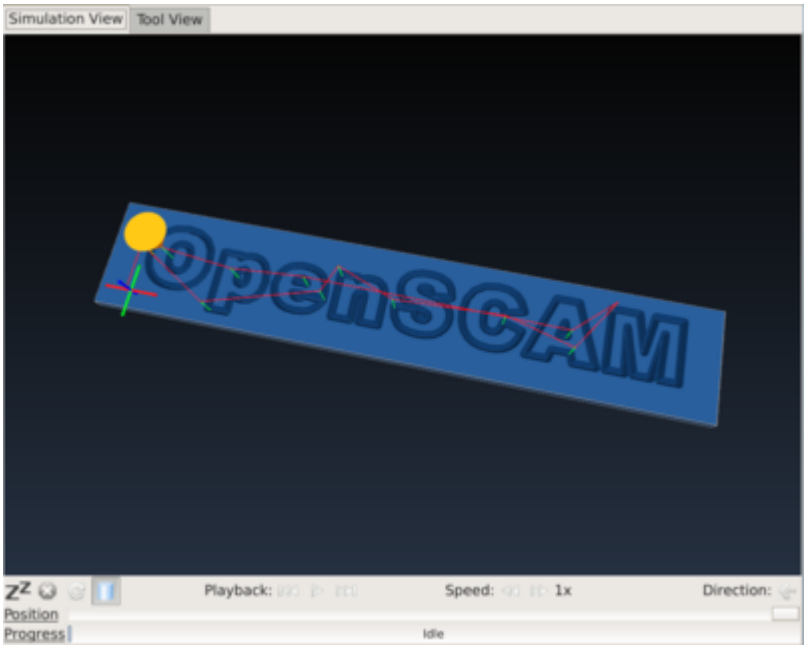


Units

Projects may use imperial or metric units. Tools units may be configured independently.

Simulation

This section describes how to configure and control simulations.



Starting, Stopping & Smoothing



Simulations can be stopped, started or restarted via the buttons at the bottom right of the simulation view. Any time you change the simulation's configuration you will need to rerun to view the changes. You can stop or restart a simulation at anytime. The status icon in the bottom left tells you if a simulation is active.

When you first open a project the simulation will run automatically. Simulations will also start automatically when one of the NC files changes on disk. You can abort a simulation by clicking the stop button.

If you enable smoothing (on by default), an extra smoothing operation will be performed at the end of the simulation. Smoothing does not affect accuracy but can improve the visualization.

Progressbar

The progressbar can be found at the bottom of the Simulation View tab. It displays an estimate of the current simulation's progress.

Adjusting the Resolution

The simulator can run at different resolutions. Higher resolutions improve accuracy but increase simulation time. CAMotics automatically chooses a simulation resolution based

on the workpiece volume. You can select a higher or lower resolution using the resolution combobox.

It is also possible to manually set the resolution by editing the resolution spinbox. The resolution value specifies the size of the simulation grid in millimeters. For example, if your workpiece is 100x100x10 mm and you select a resolution of 0.1 mm then CAMotics will probe 100,000,000 points in 3D space. As the simulation value decreases, simulation complexity increases exponentially so don't choose resolution values which are too small.

Note, the resolution value is not affected by the project units setting. It is always in mm.

Zoom, Translate & Rotate

The simulation view can be changed by placing the mouse pointer in the `Simulation View` and performing the following actions.

- Zoom in and out using the mouse wheel.
- Translate by holding down either the middle or right mouse button and dragging.
- Rotate by holding down the left mouse button and dragging.

Also see Perspective Snapping.

Perspective Snapping

The simulation view can be rotated, translated or zoomed using the mouse. The `perspective` toolbar, or alternately the `View` menu can be used to snap to particular views. The view defaults to the isometric perspective. Other perspectives are front, back, left, right, top and bottom.

Changing View Options

You can control what is visible in the simulation window either using the `view` toolbar or the `View` menu.

The first set of view options let you select which surface is viewed. You can view either then cut workpiece, if it's available, the wire view of the cut workpiece, the uncut workpiece or no surface. The remaining options allow you to turn off or on the tool path, tool workpiece bounds or axis indicator.

Partial Simulations

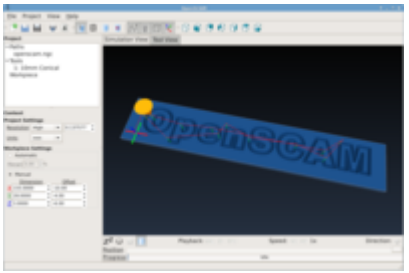
Although CAMotics is not yet able to show you the cutting process in real-time (#2), it is possible to view the cut workpiece at any point in time by setting the tool path position, using the slider bar, and then rerunning the simulation by click the rerun button.

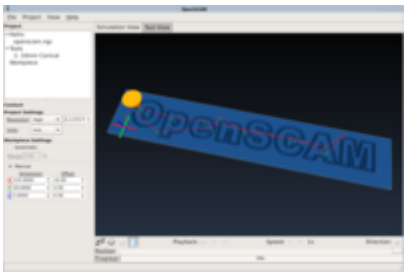
Also see the section on Tool Path Playback.

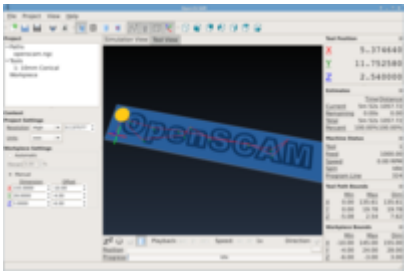
Layout

CAMotics's layout can be easily configured in several ways. You can enable or disable any of the toolbars or informational docks from the `View -> Toolbars` or `View -> Docks` menus.

In addition, there are several built-in layouts which can be selected from the View menu.







Moving Docks & Toolbars

Docks and toolbars can be dragged to different locations by holding down the left mouse button on either the toolbar handle on the left or the dock titlebar on the top of the dock.

Fullscreen

It's possible to switch to a fullscreen view either by pressing F11 or from the `View` menu.

Saving the Layout

The layout is automatically saved when you exit the program and restored on the next run.

Tool Path Playback

Although CAMotics is not yet able to show real-time simulations (#2) it is possible to playback the tool path moves and to do Partial Simulations.

Tool Path Position

The tool path position can be set manually using the position slider found at the bottom of the `Simulation View`. The Position dock will tell you where the tool is in 3D space. The Estimates dock will give the current distance traveled and estimates of the time. The Machine Status dock will tell you the current tool, feed, spindle speed and direction as well as the current line in the program.

Playback Control

The `Playback` buttons allow you to start and stop playback or jump to the end or begining of the simulation.

The `Speed` buttons can increase or decrease the playback speed. A value of `1x` is real-time.

The `Direction` button tells in which direction the simulation is running and allow you to change the direction.

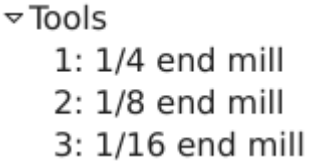
NC Files

NC files, or Numerical Control files, contain tool path information. These may either be GCode files or TPL (Tool Path Language) files. GCode files usually have a `.nc`, `.ngc` or `.gcode` file extension. TPL file names should end in `.tpl`.

NC files may be added either via the `Project` menu or by right clicking on the `Project` dock. To remove an NC file right click on it's name in the `Project` dock and select `Remove File`.

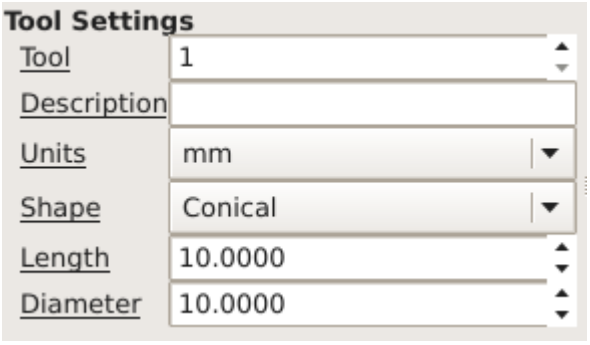
If CAMotics detect changes to an NC file it will automatically restart the simulation.

Tools



The tool table keeps track of the tools used to cut the workpiece. These tools are indexed by numbers starting from 1. You can add remove and change the parameters of the tools in the tool table. Rerun the simulation to see the effect of your changes.

Any tools referenced by the tool path which are not in the tool table will be automatically created by the simulator. The default tool is probably not what you want so you should add the tools you need, edit their parameters and rerun the simulation.

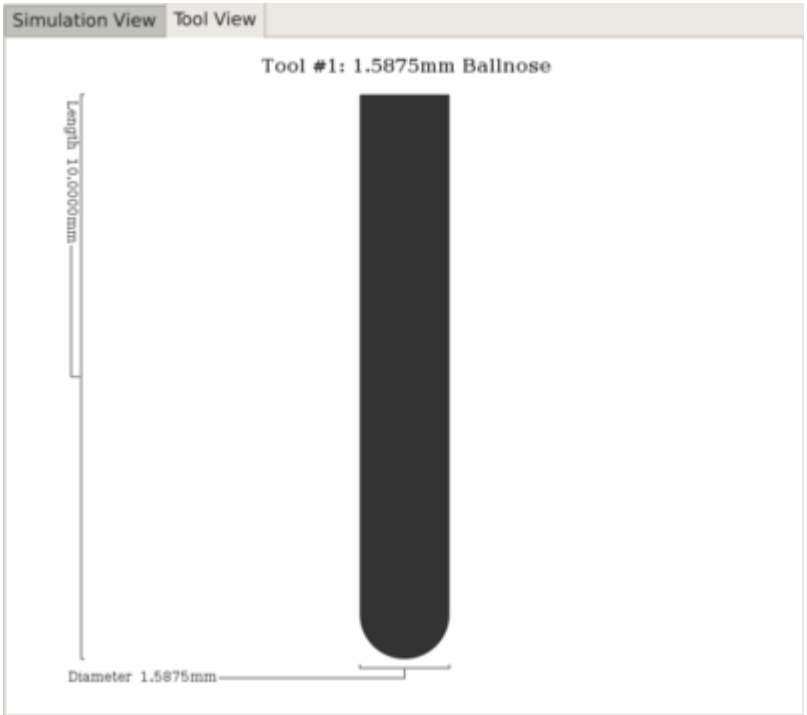


Adding & Removing

New tools can be added either via the `Project` menu or by right clicking on the `Project` dock and select `Add Tool`.

To remove a tool right click on the tool and select `Remove Tool`.

Tool View



The `Tool View` shows a diagram of the currently selected tool. It automatically updates as you change tool parameters. Note, that the view automatically zooms to fit the tool so

increasing tool length may appear to shrink the tool's diameter but it is only the relative diameter that is shrinking.

Changing Tool Order

Tools are always sorted in numerical order. You can change a tool's order by changing it's number. The user interface will not allow you to create two tools with the same number. Make sure all of the tools referenced by your tool paths are defined in the tool table.

Description

CAMotics will give each tool a default description based on it's shape and diameter. You can change this by editing the `Description` field.

Units

Tools can be specified in either millimeters or inches regardless of the project units setting.

Shape

CAMotics support the following tool shapes:

- Cylinderical
- Conical
- Ballnose
- Spheroid
- Snubnose

Dimensions

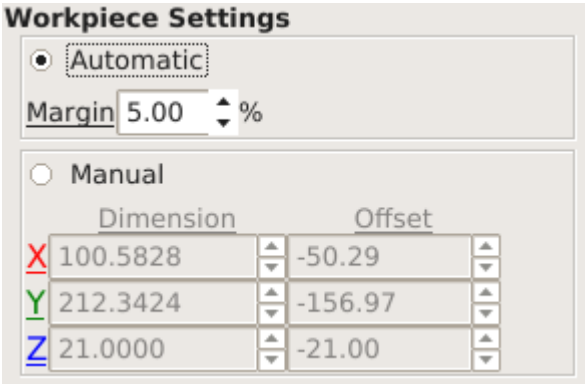
Tool `Length` and `Diameter` can be set. These specify the cutting area only and do not include the tool shaft. Snubnose tools have an additional parameter `Snub Diameter`.

Workpiece

The workpiece defines the original uncut material. Currently CAMotics only supports cuboid workpieces. The workpiece may be defined either manually or automatically.

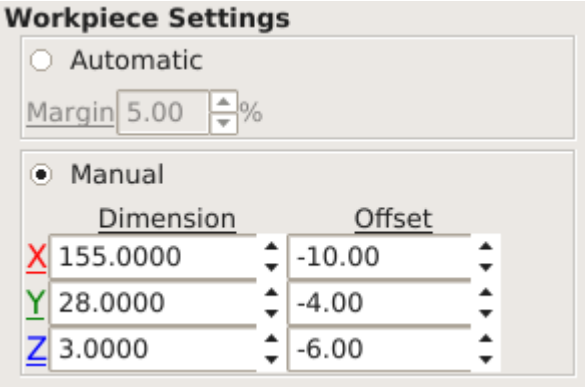


Automatic



By default CAMotics will automatically create a workpiece for you based on the tool path and a percent margin. The `Margin` extends in all directions except the positive Z axis direction. There is no margin in the positive Z direction because it is assumed this is the zero point for your cutting operation.

Manual



Often the automatic workpiece is close but not perfect. You can fine tune the workpiece dimensions by switching to `Manual` mode.

The X, Y and Z `Dimensions` specify the length, width and height of the workpiece. `Offset` specifies it's offset from the origin (0, 0, 0).

View

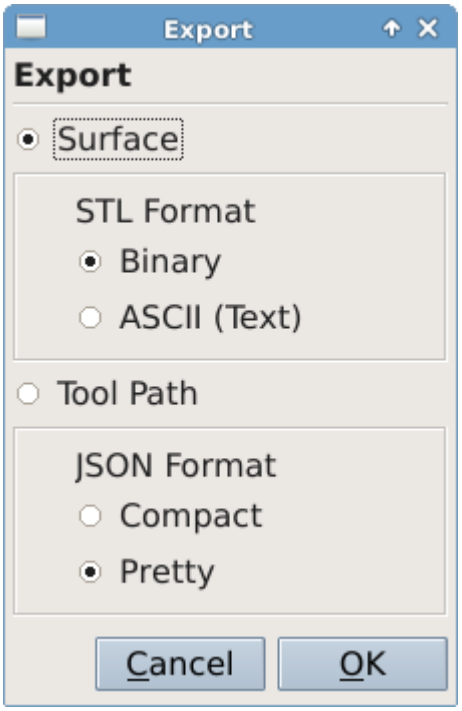
You can view the workpiece by selecting the workpiece view from the `View` toolbar or `View` menu.

When the workpiece view is selected you can see changes to the workpiece dimensions immediately but you must rerun the simulation to see the effect on the final cut workpiece. Be sure to select the cut workpiece view when your done.

You can also more details about the workpiece and tool path dimensions in the Bounds Docks.

Exports & Snapshots

It is possible to save a 3D model of the cut surface, the tool path, or a snapshot of the `Simulation View` or `Tool View` by selecting options either from the `Project` menu or `Project` toolbar.



Surface

The cut surface can be exported to either an ASCII or binary format STL file. STL files can be imported in to many other tools for further analysis such as MeshLab.

Select `Project -> Export` from the menu or click on the export button on the `Project` toolbar. Then select `Surface` and the STL format of choice. Finally click the `Ok` button,

enter a file name and click the `Save` button.

Tool Path

A description of the tool path can be exported to a JSON file by following the same procedure for exporting a surface, selecting `Tool Path` in the `Export` dialog. `Compact` format creates smaller files but `Pretty` format is easier to read.

The JSON file format consists of a list of linear tool path moves. Non-linear moves, such as arcs, are broken in to several linear moves which approximate the non-linear move. Each move can contain the following move entries:

- `X`, `Y` & `Z` axes
- `A`, `B` & `C` axes (Currently unused by CAMotics)
- `U`, `V` & `W` axes (Currently unused by CAMotics)
- `type` either `RAPID`, `CUTTING`, `PROBE`, or `DRILL`
- `line` the active program line
- `tool` the active tool number
- `feed` the active feed rate
- `speed` the active spindle speed. Negative for counterclockwise.

Each move entry is only present if it's value has changed.

Snapshot

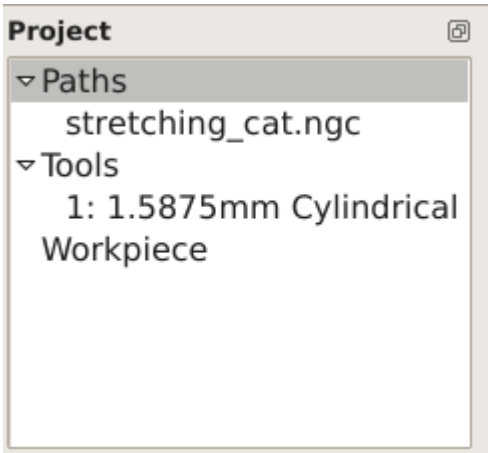
Snapshots or screenshots of the `Simulation View` or `Tool View` can be saved in several different image formats by either selecting `Project -> Snapshot` from the menu or clicking the shapshot button from the `Project` toolbar.

The image format is set by the file extension. The available formats can be seen in the file dialog's `Image files` filter. .png is a good image format choice.

Docks

Docks represent logic parts of the GUI. All but the `Project` and `Context` docks can be moved around by left clicking on the dock title bar and dragging or closed by either clicking the dock's close button in the top left corner or via the `View -> Docks` menu.

Project



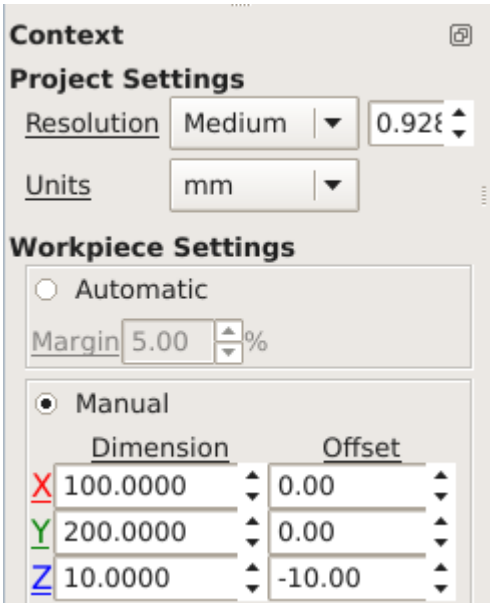
The project dock cannot be moved or closed. It contains the tree view of the project's three main components:

1. Tool paths (created from NC files)
2. Tool table
3. Workpiece

Context

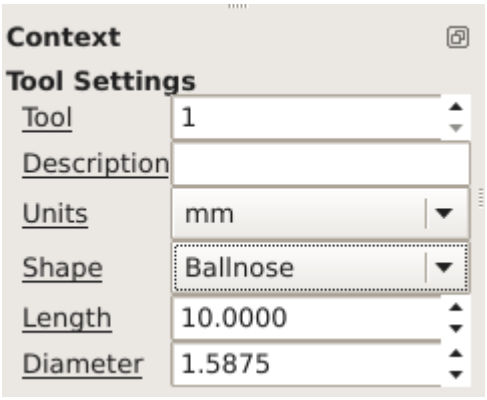
The context dock cannot be moved or closed. It changes depending on whether the `Simulation View` or `Tool View` is selected.

Simulation View



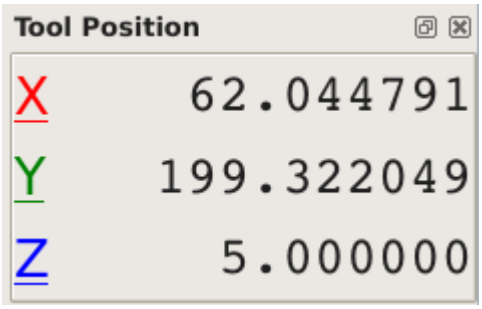
When the `Simulation View` is selected the `Context` dock displays project and workpiece settings.

Tool View



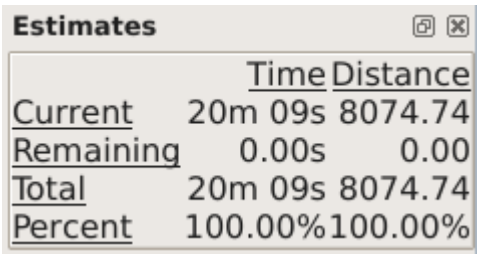
When the `Tool View` is selected the `Context` dock displays tool settings.

Position



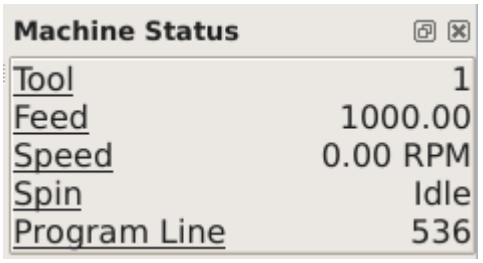
The `Position` dock shows the current tool position in the currently. selected project units. If you move the position slider at the bottom of the `Simulation View` or start tool path playback the tool position will likely change.

Estimates



The `Estimates` dock shows estimates of the current, remaining, total and percent cutting time and distance travelled by the tool in the currently selected project units.

Machine Status

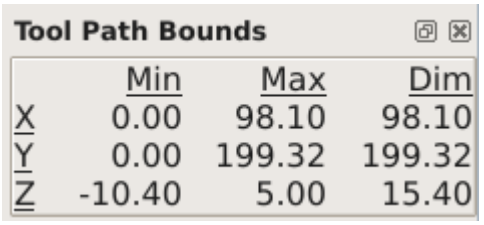


The `Machine Status` dock shows the selected tool, feed rate, spindle speed, spin direction and program line. These values change as the tool path position changes.

Bounds

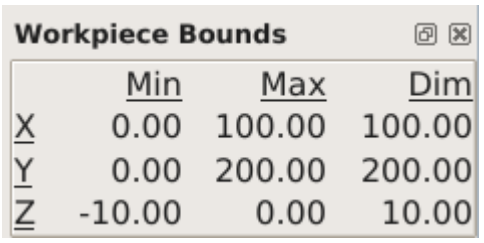
The bounds dialogs show the X, Y and Z boundaries and dimensions in the currently selected project units.

Tool Path



The bounds of the tool path.

Workpiece



The bounds of the currently defined workpiece. This may not be the same as the bounds of the currently visible cut workpiece if the workpiece has been changed since the simulation was last run.

Toolbars

The toolbars contain tool buttons which perform various functions when clicked. Toolbars can be moved by holding down the left mouse button on the toolbars handle (found on the left) and dragging. Toolbars may also be enabled or disabled via the `View -> Toolbars` menu.

Project

The `Project` toolbar allows quick access to project file operations new, open and save as well as export and snapshot.

View

The `View` toolbar allows you to select simulation view options to select the surface view or enable/disable the tool path, tool, workpiece bounds, or axes.

Perspective

The `Perspective` toolbar allows you to quickly change the simulation perspective to either isometric, front, back, left, right, top or bottom.