

Release Notes Version 11.1:

This is a major release adding preliminary support for 5-axis machining.

To prevent the PDF documentation file from becoming large and unwieldy, the documentation has been divided into several smaller documents. PDF documents can be opened from the main menu help menu which further acts as an index. The html documentation of supported G and M codes has been removed from the CNC document in favor of the PDF versions.

RiceCNC has long supported up to nine independent axes, however, 5-axis machining generally refers to the ability of a machine and software to bring a cutting tool to any point on a object at any angle. There are excellent illustrated guides to 5-axis machining available on the internet, so only a brief introduction is provided here.

Five axis machining generally requires the machine to have three linear axes, usually the three orthogonal machine axes, plus two rotational axes to change the angle of the cutter relative to the work. The rotational axes may be fixed to the table, e.g., a cradle and turntable, or on the spindle axis, or a combination of the two.

As the relative angle of of the cutter is changed, the machine position must be moved to maintain the position of the cutter relative to the workpiece. Generally the computations to do this would be too complex for CNC programs. Therefore, 5-axis software needs to have some knowledge of the machine configuration, i.e., the kinematics.

This release adds vector and polar transforms to support machine kinematics. The transforms can describe the interactions of the five axes. Both vector and polar transforms can be configured interactively or by CNC programs. Refer to other guides for more information on this configuration.

The new vector and polar transforms duplicates and obsoletes much of the functionality provided in previous releases. Some of the obsoleted functionality has been removed, other functionality is deprecated by maintained for compatibility. Other functionality has been temporarily removed but will reappear in future releases in a different form.

In particular, the "Coordinate Transformation Matrix" has been removed effectively being replaced by vector rotations. Vector transforms are both more flexible and more intuitive to use. C codes for manipulating the CTM have been removed being replaced by G codes for configuring and rotating vectors.

The Path Wizards have also been removed for this release. The pocket machining wizard functionality may reappear in a different form, i.e., Bézier surfaces.

Vector transforms enable drill cycles to drill at any angle rather than being restricted to an orthogonally aligned axis. The orthogonal plane selection G codes remain supported for compatibility but are now deprecated.

The “Program Position Mode” has been removed effectively being replaced by vector transform positions. The new non-modal solution enables CNC programs to reference vector transforms and axis names directly in the same move or feed command.

Polar transform can calculate cartesian coordinates given a polar center and polar coordinates and vice versa. However, polar transforms can also generate circular arcs and are easier and more flexible to use than the circular arc G codes G02 and G03. These G codes remain supported for compatibility but are now deprecated. Polar transforms can generate circular arcs with any axis orientation.

Axis, vector and polar transform references can be freely mixed on move (G00) and feed (G01) commands. This added flexibility makes relative coordinate addressing mode more convenient and it is now the default.

The path view now displays a grid to indicate scaling and also displays Bézier control points and path direction.