



Enabling Your Ideas

# Technical Document

TD\_30620

**Title:** M3 Machine Control Software Installation and Revision Information

**Date:** November 3, 2008

**Current Software Install Release:** rel1.9u

**Product Identification:** PCNC1100 Mill (30204)

**Subject:** Specification: This document is to track revision changes to M3 for all non-Series II (serial number 1325 and below), without Spindle Drive Upgrade. Series II Mills and non-Series II with spindle upgrade use different control software.

**Instructions:** Please note; if users are updating their control software from our website TORMACH.COM, and if an updated version of XMLTweak will be selected to be installed with this software, then it is necessary to delete the folder "C:\Program Files\XMLTweak\" prior to installation.

1. Download from <http://www.tormach.com/documents.htm>, open the "CAD, CAM & MACHINE SOFTWARE" section, and selecting **PCNC3Relx.xx.exe**.
2. Run the executable accepting the default installation directories.
3. Reboot the computer after installation.
4. If XML Tweak was selected to install, than review section 9 of the owner's manual to update the XML Tweak settings.

## Version & Release History:

*Changes made to M3 v1.9u (this is a later release from 1.9w listed below)*

- To enable PCNC owners to continue to use their mill on increasingly faster PC's, a configuration change needed to be made. Pulse width in the .xml file has been changed to 16000.

*Changes made to M3 v1.9w*

- Probe code changed to eliminate potential false readings from probe bounce.
- M87x macros improved to support multiple styles of tapping using reversing tapping heads.
- Duality Lathe support functions added.
- Removed spurious small motion on startup related to jog shuttle controller.

*Changes made to M3 v1.8*



- Jog/Shuttle controller plug-in updated to Rev 3.1a to fix situations on some USB ports where jogging continues even if none is commanded. This version also allows capture of diagnostics to debug USB problems.
- Screenset updated to Rev 7.02. This has a better algorithm on the Toolsetter Z probing screen to cater for contact bounces on some setters. These gave the error "Probe asked to move too far." There are a few minor cosmetic improvements in the layout on other screens.
- The running control software no longer has a redundant black "DOS window" visible in the task bar.

### *Changes made to M3 v1.7*

- This version relates to increasing the safety of running part programs whose workflow mixes using the tool table for some tool lengths and setting work offsets to set Z = 0.0 for others. This is the problem that was first reported by users running the FirstPart.nc sample program referred to in the manual. In essence the problem is solved by always grouping the commands T~ M06 G43 H~ on one line of the part program. SprutCAM does this already but updates are required to SheetCAM and TurboCAD/CAM post processors.
- The change for SheetCAM is in the release installer file (PCNCM3Rel1.7b). It consists of an updated version of SheetCAM (4.1.45 - for full details see [www.sheetcam.com](http://www.sheetcam.com)). An opportunity was taken with this release to update to the newest version of SheetCAM as well as supplying a modified post processor. This version includes bug fixes and significant new functionality - in particular the ability to offset the cutting of open paths (say machining one edge of stock). We have taken the opportunity to rationalize the post-processors that are installed by not installing those for other equipment. This should avoid the confusion caused by the old post processor being called Mach2.post. The PCNC110 postprocessor will generate code for both the Mach2 based control software and the current Mach3 control software. Post processors that are already installed on your machine (in C:\Program File\Sheetcam\Posts) will not be deleted automatically but you might wish to remove them manually. Please note that a recent Microsoft Windows XP update has introduced a bug which can cause SheetCAM to fail to load with an "Illegal system DLL relocation" error. The SheetCAM site has a link on its download page to the Microsoft hotfix for this.
- Related changes that are relevant to the toolchanging problem but not part of the installer file are:
  - (a) A revised copy of the sample program in FirstPart.zip
  - (b) The TCC post processor is in TormachCAM\_r3.zip
  - (c) The User manual is updated to revision C2-3

### *Changes made to M3 v1.6*

- Tapping M codes revised and improved - Screenset (Rev 6.23) and Profile (Rev 2.7) updated;
- XY Probe screen and sub-screen added with setup and reference functions - Screenset update;
- Z Probe screen added with digitization and automated tool length sensor functions - Screenset update;
- Jog/Shuttle controller operated improperly when Machine Coordinates were active. Jog/Shuttle can now be operated under both coordinate systems - Jog/Shuttle controller plug-in updated to Rev 2.0a;
- 4th axis feedrate did not take account of the values in the diameter correction DROs. Fixed by new Profile. 4th axis users - IMPORTANT - the correction of this bug will probably alter the feedrate on 4th axis with your existing part programs. In indexing operations this will not be significant but if you are doing multi-axis coordinated moves involving "A" please check the code (perhaps by air cutting) in case your settings now give excessive feeds. In general the larger the correction diameter, the lower the rate of rotation of the A-axis.

### *Changes made to M3 v1.5*

- Tool Length Touch button had a bug affixing Goto Tool Change position. Goto Tool Change now always works;



- Edit/Save button on the offsets screen had a bug affecting tool offset selection;
- Jog/Shuttle controller operated improperly under G91 mode. Jog/Shuttle can now be operated under all modes;
- Auto control would not run the spindle counterclockwise. Auto control fixed for both forward and reverse. This is corrected in the updated Profile (XML) file;
- Feedhold used during long arcs could cause the control software to get confused. Two important variants of this were two consecutive holds in one G02/03 would rewind the program and restart from the top and on resuming a held arc, spurious movement could take place in the Z direction. This is corrected by the release providing an updated version of the Mach3 program;
- The LEDs on the Diagnostics screen were not properly spaced out. This is corrected by the revised screen set;
- Tool and Fixture tables were overwritten on an upgrade. This is corrected by altered settings in the Installer software.

### *M3 initial release compared to M2*

- Larger tool path window and better tool path view controls;
- User definable G28 operation;
- Simple speed and feed calculator linked to the tool table;
- Support for Tormach electronic handwheel or "Shuttle" (PN 30616);
- Integrated CAM software included with the installation. Operator can now go directly to CAM from the machine control program instead of editing code with a text editor;
- Tools provided for integration of TurboCAD, SprutCAM, or other CAM program with the machine control software;
- Speed calibration is revised with a software based method. XMLTweak program allows more convenient speed calibration.