

## D2nc Installation instructions

The installation process of D2nc consists of running an installation program and following the on screen prompts. The first time you install D2nc you should reboot your system. For install patches or upgrade releases it should not be necessary to reboot unless otherwise instructed.

D2nc is a stand alone program which can be run from a D2nc icon placed on the desktop or launched using a button within Mach3.

One of the options presented during the install is installing the D2nc Mach3 screen sets. Selecting this option will install screensets into the Mach3 directory to allow D2nc to be launched from within Mach3.

Launch the install program and follow the installation program prompts.

A D2nc program icon will be placed on the desktop and a program group created. D2nc will be installed into the directory C:\D2nc

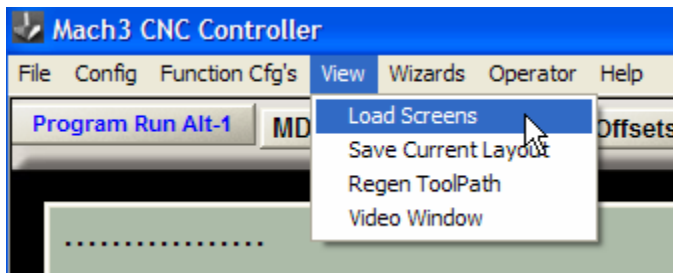
Two modified mill screen sets, 1024d2nc\_200.set and 1024d2nc\_184.set, are placed in the C:\Mach3 directory. These are the standard 1024.set screensets which have been modified by adding a single button to the "Program Run" screen. The button is labeled D2nc.



*The following screen shots are from Mach3 version 1.90.075 using the new menu layouts but the same settings although slightly different are available in earlier versions.*

To add the button to Mach3 you need to change the standard screen set to the modified one.

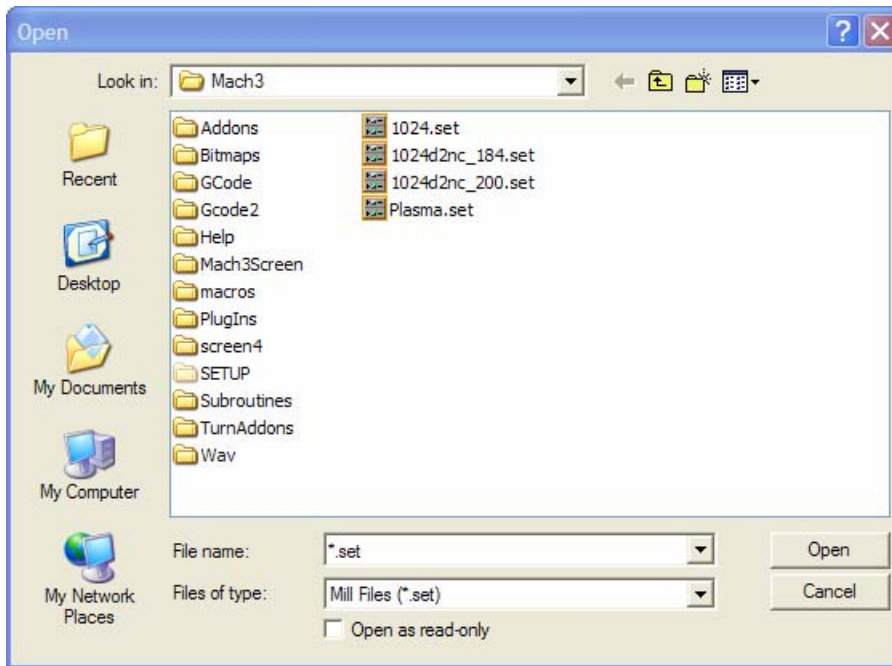
From the Mach3 menu select View -> Load Screens



If you're running Mach3 2.0.x, select the 1024d2nc\_200.set file so its name appears in the File name box.

If you're running Mach3 1.84, select the 1024d2nc184.set file name instead.

Click the Open button.

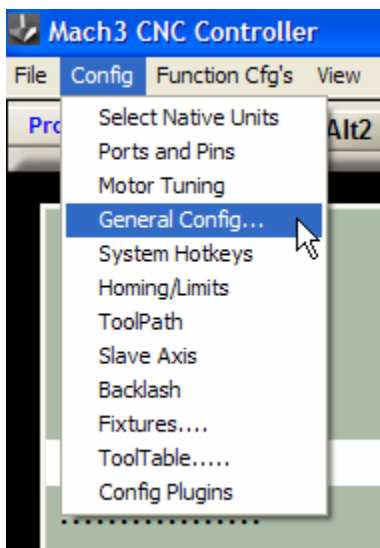


Mach3 will now have the D2nc program button ready for use.

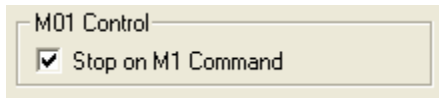
Additionally you should check the following settings in Mach3.

These are found on the Mach3 menu:

Config -> General Config...



If you plan on using D2nc's position/pause mode which allows for using the quill for drilling and tapping on mill/drills, then you must check "Stop on M1 Command".



To make Mach3 stop and wait when a tool number changes, set the "Stop spindle. Wait for Cycle Start" Tool Change setting.

