INDEX

1. NEW FEATURES

1.1 DXF-to-ISO file converter

The WinDNC application may be used to convert Autocad generated files in DXF format into part-programs in ISO format for the 8055 CNC.

This feature may be used to generate a part-program from the part blueprint. The DXF format is a standard for exchanging graphic files.

The DXF file must consist of points, lines, arcs and circles. If the drawing uses polylines, they must be previously segmented.

When selecting this option using the "IMPORT" "DXF FILE" icons, a list is displayed with all the dxf files in the selected directory. Select the desired file from the list and press [ENTER].

After selecting the file, define the following configurable fields:

IMPORT AS Name of the converted file, by default it adds the extension PIM or PIT depending on whether it is in mill mode or lathe mode. The converted file stays in the work directory of the WINDNC.

UNITS Inches or mm

RADIUS/DIAMETER When selecting lathe.

AUTONUMBERING To get the line number with the block number. When selecting this field as "Yes", it will be necessary to fill out the fields for starting line number and line numbering step.

Main Plane Ordinate, Main Plane Abscissa and Perpendicular Axis To define the work plane and the perpendicular axis.

Layer, Priority and Offset The DXF files may have layers and each layer contains different heights of the drawing on the perpendicular axis.

All the layers together make up the whole drawing.

When importing the file, it will be possible to select which layers must be included into the part-program. By default, all the layers are included. To exclude any of the layers, select it with the mouse or the space bar.

Priority and offset of the layers: For each layer, it is necessary to define its priority and offset (height) on the perpendicular axis.

The priority defines the order in which the layers will be executed; i.e. the order in which they will be included in the part-program. Those with priority ·1" will be executed first and so on.

The offset (height) on the perpendicular axis, if the perpendicular axis has been defined, it permits executing each layer in the Z coordinate, or that of the desired corresponding perpendicular axis.

Once this data has been defined, press the "IMPORT" icon to generate the file in the part-program.

1.2 8025CNC-to-8055CNC program converter

The same way DXF files are imported, program files of the 8025 can also be converted into programs for the 8055 CNC.

When selecting this option using the "IMPORT" "8025 PROGRAMS" icons, a list is displayed with all the programs in the selected directory. Select the desired file from the list and press [ENTER].

After selecting the file, define the following configurable fields:

IMPORT AS Name of the converted file, by default it adds the extension PIM or PIT depending on whether it is in mill mode or lathe mode. The converted file stays in the work directory of the WINDNC.

TYPE Mill or lathe

NUMBER OF AXES OF THE SYSTEM

THIRD AXIS If lathe

FOURTH AXIS If lathe

INCOMPATIBLE AXIS If mill and more than 3 axes.

PLANE CONFIGURATION If lathe.

Offset for range of arithmetic parameters This offset is added to the numbering of the parameters of the 8025 to generate the 8055 program.

When done with the conversion, the screen shows the number of warnings and errors occurred; the warnings are incomplete translations or not entirely correct, the errors should not occur if the 8025 CNC program is correct.

The first line of the converted program shows as a comment the number of errors and warnings of the conversion.

When there are no errors in the conversion, a single window is displayed containing the program converted into the 8055 CNC.

When there are errors in the conversion, it offers the choice to view the source program in a window and it may be edited while another window below it shows the description of the errors and warnings. When selecting an error describing line at the bottom window, the top window of the programs points at the line containing that error or warning.

1.2.1 Data that remains not translated:

N1 P1=X P2=Y
N1 P1=0X P2=0Y

This type of assignments are not interpreted properly when a block contains more than one, this may be solved by writing one per block.

G76 X Y Z

Blocks G76 X+/-4.3 Y+/-4.3 Z+/-4.3 and G76 XP3 YP3 ZP3 have their equivalent at the 8055 CNC, but not the G76 X Y Z which would load the theoretical coordinates at the time.

Incompatible G and M functions.

The 8025 CNC allows programming them in the same block and keeps the last one active, but the 8055 CNC issues an error message.

There are converters for the MILL and LATHE models, but not for the Jig Grinder, Punch, Laser or other model CNC's.

1.2.2 Notes about the conversion

Global parameters have been necessary to translate different blocks. The following parameters of the 8055 CNC have been used that do not exist at the 8025 CNC.

P298 - P299	as flags for conditional jumps.
P290 - P294	for G31-G32
P288 - P289	for G93
P295	to calculate the pitch in taper threads G33 (LATHE)

Parameters P100 and P101 of the 8025 CNC are converted into variables PARTC and FIRST.

At the 8025 CNC, the Polar origin becomes the center of the circle every time a G2 or a G3 is executed; thus the 8055 CNC should be set the in same way.

The numbers for the parametric subroutines are converted by adding 100 because at the 8055 CNC they are defined that way and the 8025 CNC may have subroutines from 0 to 99 without parameters as well as with them.

Function N4 G76 <content of the block to be created> is converted into (WRITE <content of the block to be created> It does not analyze the content nor assumes the language used to write it, it is simply copied as it is in the block.

Cycles in LATHE

They go from Gxx P0=.. P1=.. P2=.. to Gxx X.. Z.. Q..,

If the original block contains assignments to parameters that do not belong to the cycle, they are properly converted in a previous block because they may be relevant somewhere else in the program. This happens, for example, with parameter P12 of G66 without translation at the 8055 CNC.

The dwell units at the 8025 and 8055 CNC's are not the same (seconds and hundredths of a second). They are converted both in G4 and in the cycles.

In general, a warning will be issued every time something cannot be fully or partially translated. No translation errors should occur because it means that the block could not be analyzed, that it is not written in 8025 language. The main warnings come up in the following cases:

- G functions without equivalent at the 8055 CNC: G52, G65
- G functions nonexistent in one of the models: G10-G13, G14-G16, G17-G19, G43-G44, G51, G64, G66, G68, G69, G73, G79,G80, G98-G99.
- G functions existing in both models, but with different meanings: G53, G58, G59
- Auxiliary axes have been programmed without being defined: V, W, G77-G78, C
- M45 S and M45 K features in LATHE
- Operators without translation: F17, F18, F19, ...
- Arithmetic parameters within the P26-P99 range.

Function G80 will sometimes be translated into (MDOFF) when there is a previous (MCALL) call (translation of G79). (A warning is issued in case the assumption is wrong)

Taper thread G33 X Z I K in LATHE. There is only one L pitch at the 8055 that is the square root of the sum of the squares of I and K.

Programming G2, G3 with auxiliary axes requires a previous plane change for the 8055 CNC to accept the block and another change to leave things as they were. The information on the planes to be activated is obtained from the coordinates programmed in the block and from the incompatible axis; it could sometimes not be fully correct when the information is not complete. The same could happen when sorting and/or renaming the axes and centers according to the 8055 CNC criteria.

There are many instances where blocks are generated before and after the translated block, they all have the original's block number.

The comments of the 8025 CNC are converted into 8055 CNC messages, but they cannot blink.

All the operations affecting the jump flags generate assignments to parameters P298-P299 that replace them at the 8055 CNC.

The subroutines have been moved to the end of the program, after M30. Inside the program, they been replaced by the corresponding call.

An error is issued in:
G72 X K1.0
G2, G3 with C axis in LATHE

2 VERSION COMPATIBILITY

This version will be necessary for CNC versions V11.01, V12.01 and newer to work properly with the Windnc.

There are the following limitations with older WINDNC versions:

- At the CNC, using the explorer, it is NOT possible to see programs located in the root directory of DNC 1 or DNC 2 although it IS possible to see those located in directories that hang from those root directories.

- It is not possible to transfer OEM or hidden files (encrypted) from the CNC to the PC using CNC versions V11.01, V12.01 or newer.