

CIMCO NEWS - JULY 2009

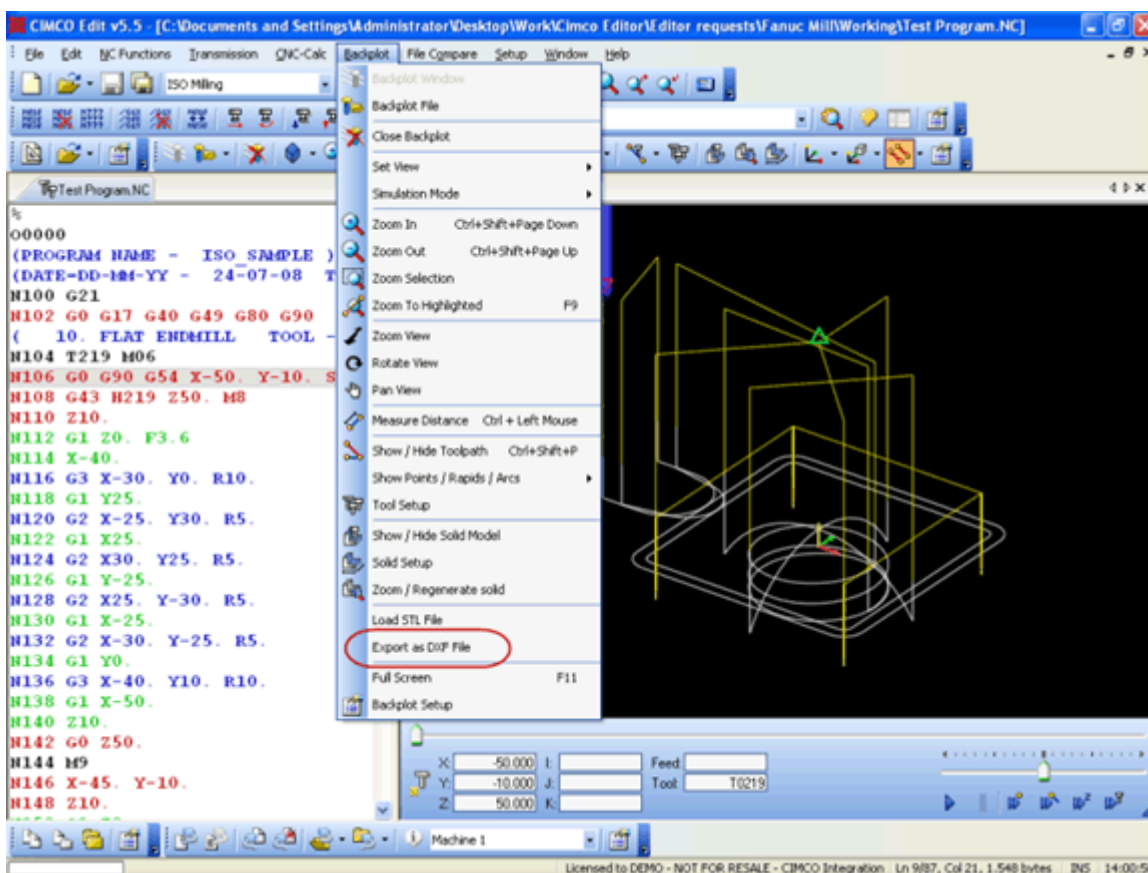
Welcome to this issue of CIMCO News. We hope to keep you up-to-date with some of the new features and also to give you technical information on how to use CIMCO products.

Contents:

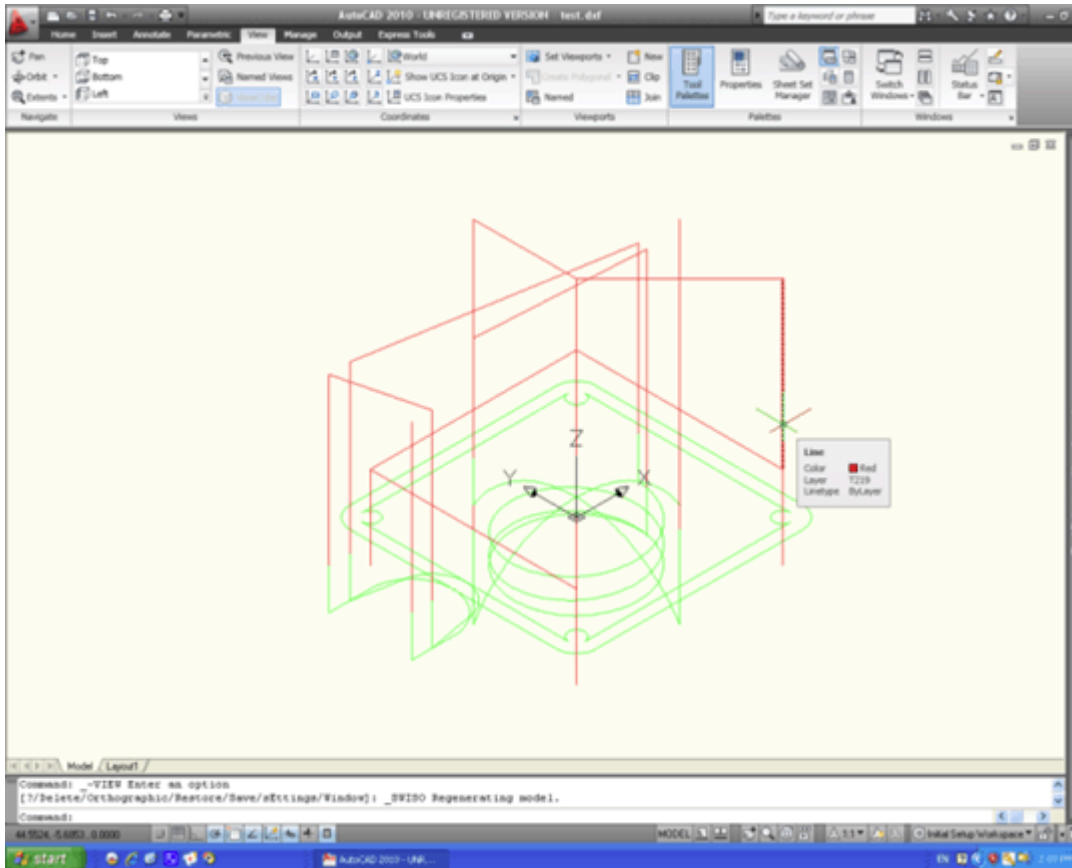
- Beta Export to DXF available
- Haas Milling and Turning macro files added
- New Fancu Turning convertors (Type A, B and C G Codes)
- New sample files
- New UPPER CASE option
- Fancu DPRNT commands added by DNC-Max
- CNC-Calc Speed/Feed calculator and Thread milling

EXPORT TO DXF

In the latest version of CIMCO Edit you can save a NC backplot as a DXF file. This is under Backplot – Export as DXF file. This is only shown on reseller keys and is Beta at the moment. Each Tool is saved to a separate layer and rapids are shown in red and feed moves are green. Any helix moves in the NC are broken to lines. There are a few problems with arcs going the wrong way and also in the wrong place when G18 is active. Please test this new function and report any problems or suggestions you may have.

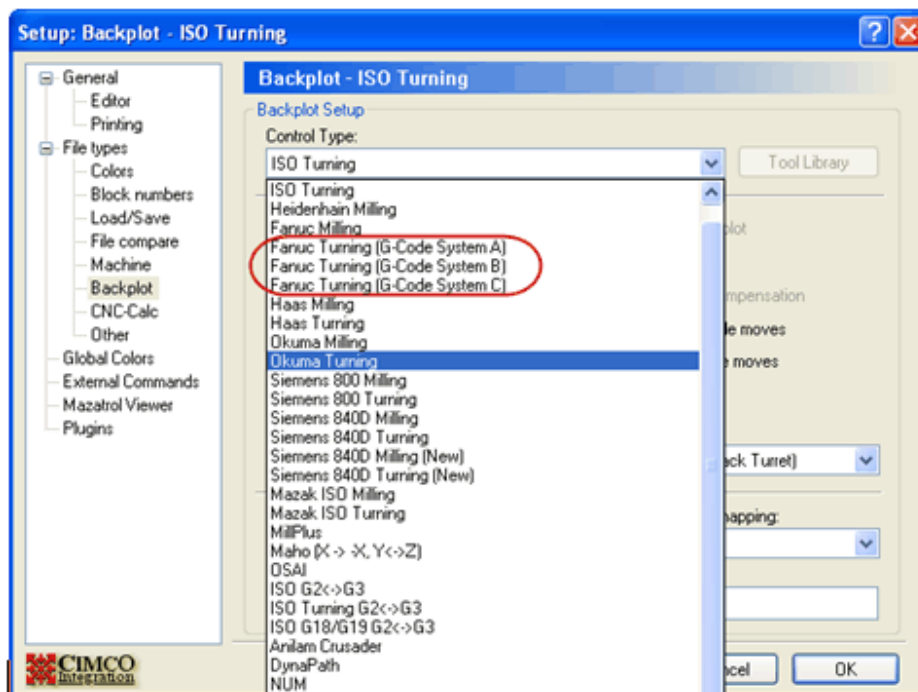


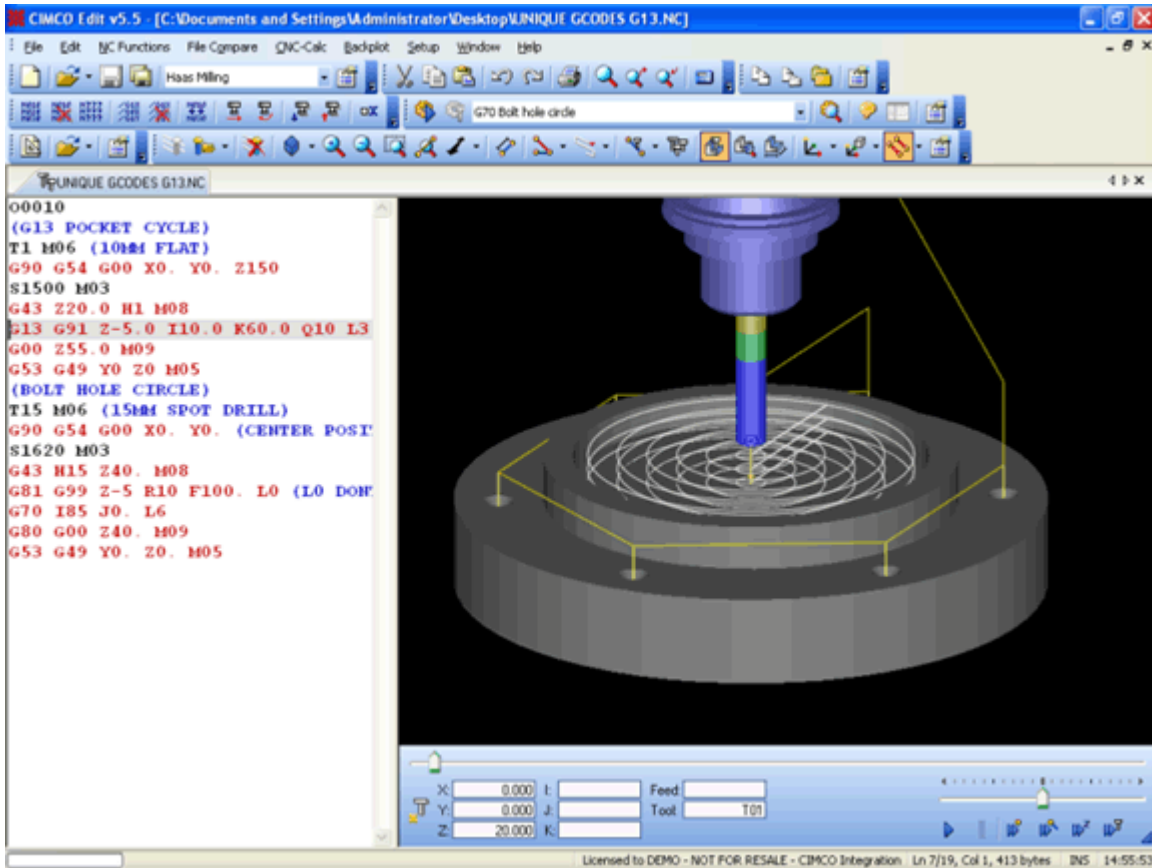
DXF file opened in Autocad



HAAS MILLING AND TURNING MACRO FILES INCLUDED WITH INSTALL

The Haas macro files include the unique Haas G codes. The backplot convertors have been rewritten to support these G codes.

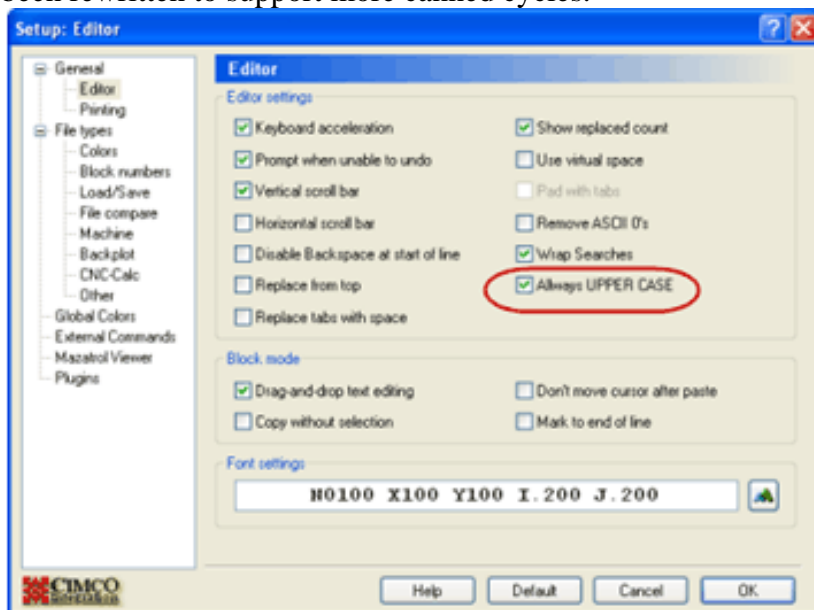




We intend to include other manufactures macro files as we update the backplot convertors.

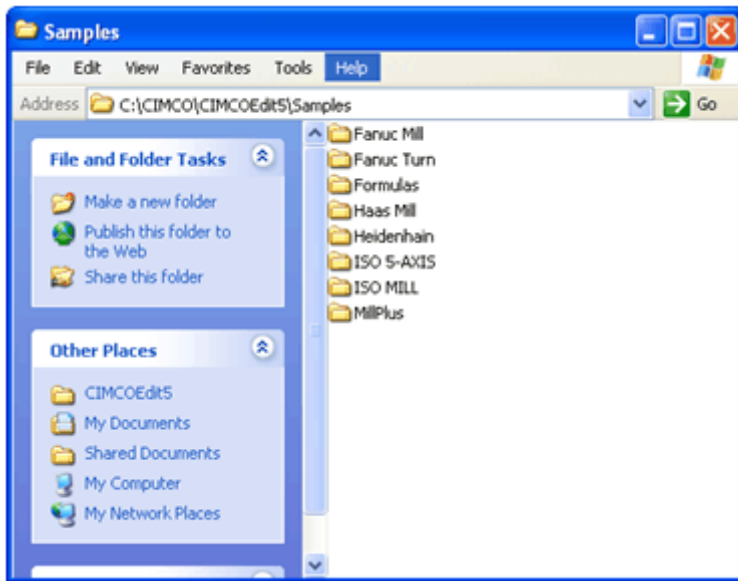
NEW FANUC TURNING CONVERTORS

Fanuc controls have 3 options for the G code sets used on the control. Most common is System A (G90 is simple rough cycle). System B (G90 is absolute position) is the next most common. The convertors have been rewritten to support more canned cycles.



SAMPLE FILES

New sample files for Haas Milling and Fanuc Turning have been added to
C:\CIMCO\CIMCOEdit5\Samples



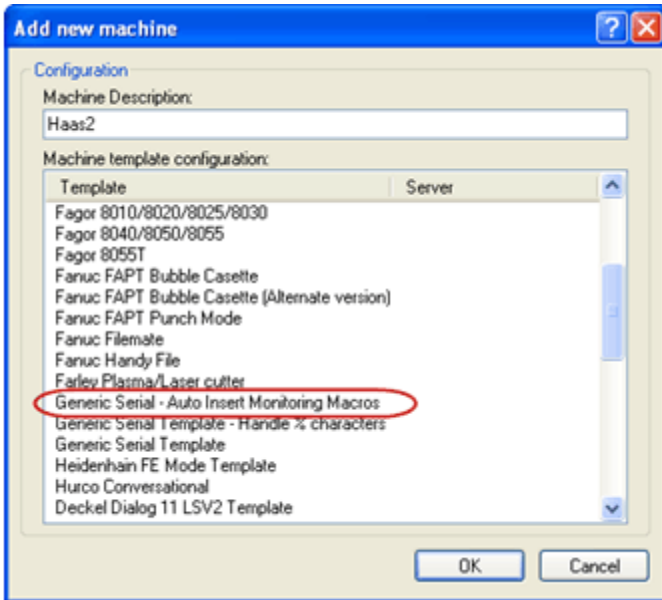
NEW UPPER CASE OPTION

New option for always writing in UPPER CASE. Don't need to press Caps Lock anymore. If you have any files with lower case letters you can always press Ctrl + U to convert to UPPER CASE.

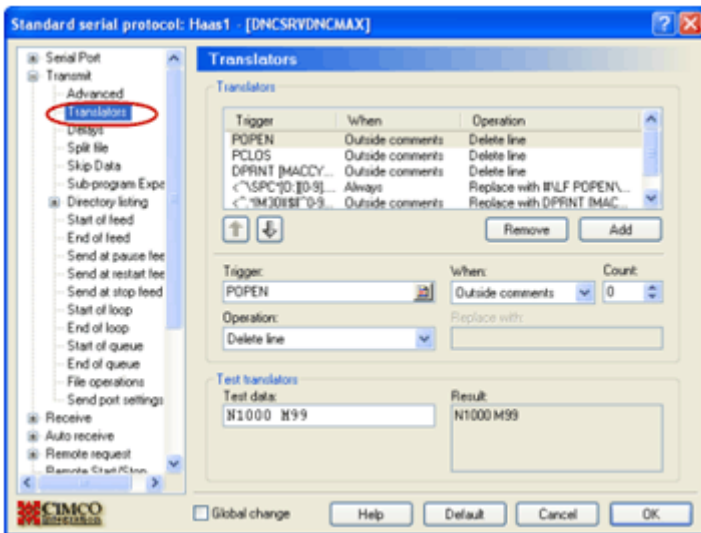
FANUC DPRNT COMMANDS ADDED BY DNC-MAX

Fanuc DPRNT commands can automatically be added by DNC-MAX when the program is sent to the controller. When the NC file is sent back from the controller the DPRNT commands are automatically removed. Use the Generic Serial - Auto Insert Monitoring Macros template.

Fanuc DPRNT is an option on most Fanuc controls and many other controls also have this function.



This template uses the translator feature in Transmit and Receive to add or remove the Fanuc commands



When this program (1000.nc) is sent to the CNC

```

%
O1000 (TEST)
G0
.
.
M30
%
```

DNC-Max will add the POPEN, DPRNT and PCLOS commands

```

%
O1000 (TEST)
POPEN
DPRNT[MACCYCLESTART1000]
G0
.
.
```

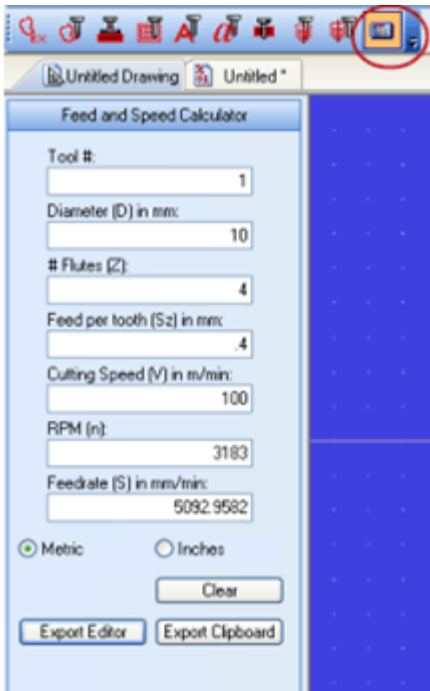
DPRNT[MACCYCLESTOP]
PLCOS
M30
%

When the program is run on the CNC the text inside the [] is output on the RS232 and this can be captured by DNC-Max to measure cycle times.

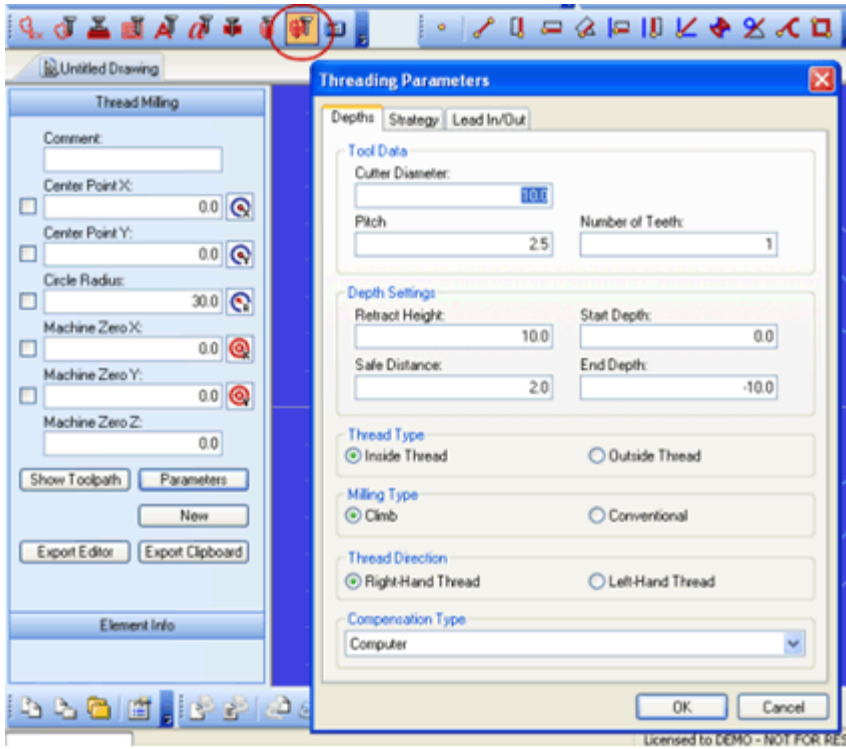
Tue 6/30/2009 5:02:48 AM	Puma 240MS	CYCLESTART	Cycle Start: 1-120-0060P1	1-120-0060P1
Tue 6/30/2009 5:27:32 AM	Puma 240MS	CYCLESTOP	Cycle Stop	1-120-0060P1
Tue 6/30/2009 5:30:02 AM	Puma 240MS	CYCLESTART	Cycle Start: 1-120-0060P1	1-120-0060P1
Tue 6/30/2009 5:54:46 AM	Puma 240MS	CYCLESTOP	Cycle Stop	1-120-0060P1
Tue 6/30/2009 6:13:13 AM	Puma 240MS	CYCLESTART	Cycle Start: 1-120-0060P1	1-120-0060P1

CNC-CALC FEED SPEED CALCULATOR AND THREAD MILLING

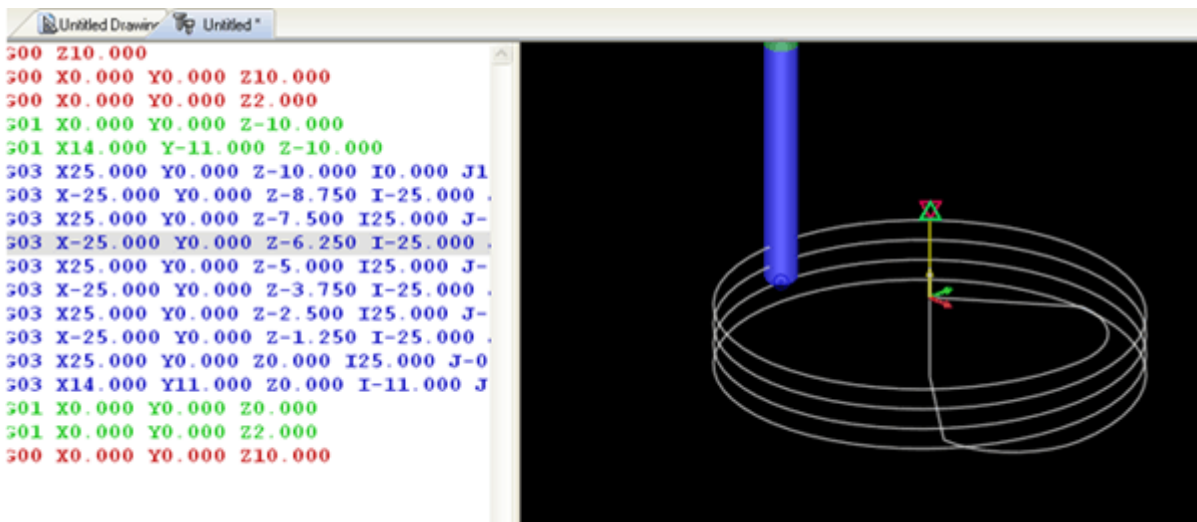
CNC-Calc can now calculate Spindle speed and Feedrate based upon cutting speed and feed per tooth



Thread Milling toolpaths can now be generated by CNC-Calc. Single and multiple teeth tools can be specified along with Inside/Outside threads and milling direction. There is an option to linearize helix moves to lines for older machines



NC code from thread Milling



CIMCO Integration is a registered trademark. Microsoft Windows, Windows XP and Windows Vista are registered trademarks of Microsoft Corporation. Other brand and product names are trademarks of their respective owners. Copyright (c) 2009 CIMCO Integration.