Advantech SE Technical Share Document

Date	2019 / 02 / 18	**Related Product**	WebAccess/CNC, M2I/CNC
Category	■ FAQ □ SOP □ Driver Tech Note		
Abstract	The quick start for NC program edit		
Keyword	WebAccess/CNC, M2I/CNC, Heidenhain iTNC 530 CNC Controller		
Related OS	Windows 7		

Revision History

Date	Version	Author	Reviewer	Description
2019/02/14	V1.0	JosephSun		

■ Problem Description & Architecture:

The document will show you How to create a simple NC program that can run a few minutes (hours) for further application in WebAccess/CNC & M2I/CNC.

■ Brief Solution - Step by Step:

Step1: Create a new NC program file. (Please add .H filename extension when you create)

-> Select  Programming and Editing key

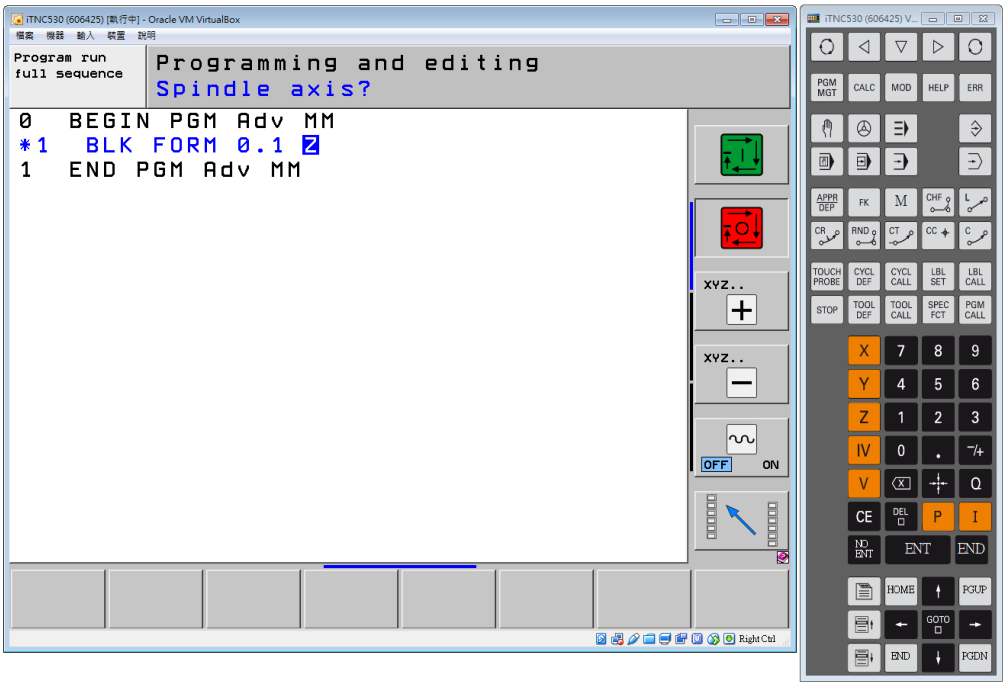
-> 

Select/Delet Program & File key

-> 

NEW FILE key, Please add .H filename extension when you create a new file.

-> To select the unit of measure, press the “MM” key. The TNC switches the screen layout and initiates the dialog for defining the **BLK FORM** as following picture.

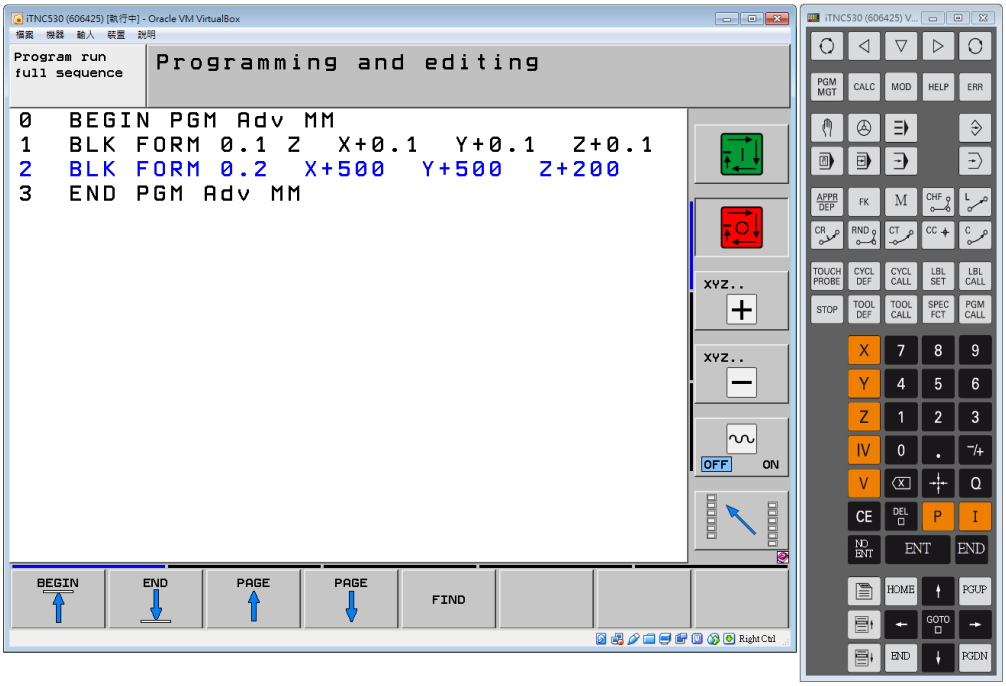


Step2: Edit NC program with iTNC530 Virtual Keyboard.

-> ***WORKING SPINDLE AXIS*** Click ENT key. To comfirm Z axis as spindle axis.

-> ***DEF BLK FORM: MIN-CORNER*** Give a MIN value for X, Y, Z axis with ENT key.

-> ***DEF BLK FORM: MAX-CORNER*** Give a MAX value for X, Y, Z axis with ENT key.



-> ***DEF TOOL NUM & CONFIG*** Select TOOL CALL Key. Give a tool number & Spindle speed value with

ENT key. After that, click NO ENT key Ignore the dialog question, until the program-block window display as the following line:

3 TOOL CALL 7 Z S500

-> *MISCELLANEOUS FUNCTION M?* Select L Key. click NO ENT key Ignore the dialog question, until M function can be configure:

4 L M3

-> *LBL SET* Select LBL SET Key. click NO ENT key until M function can be configure:

5 LBL 1

-> *Dialog / COORDINATES? / FEED RATE F=?* Select L Key. Enter the target coordinate for the X axis, and go to the next question with → key:

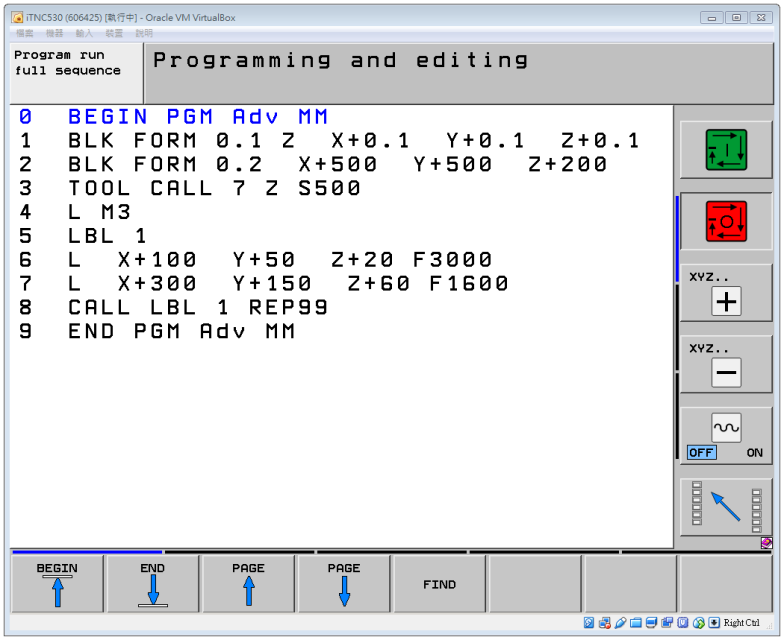
6 L X+100 Y+50 Z+20 F3000

-> *Dialog / COORDINATES? / FEED RATE F=?* Select L Key. Enter the target coordinate for the X axis, and go to the next question with → key:

7 L X+300 Y+250 Z+60 F1600

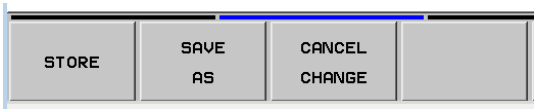
-> *LBL CALL* Select LBL CALL Key to set the times need to repeat of the program with click → key:

8 LBL 1 REP99



Step3: Save NC program & Run the program.

Select  Switch the soft-key rows key

-> 

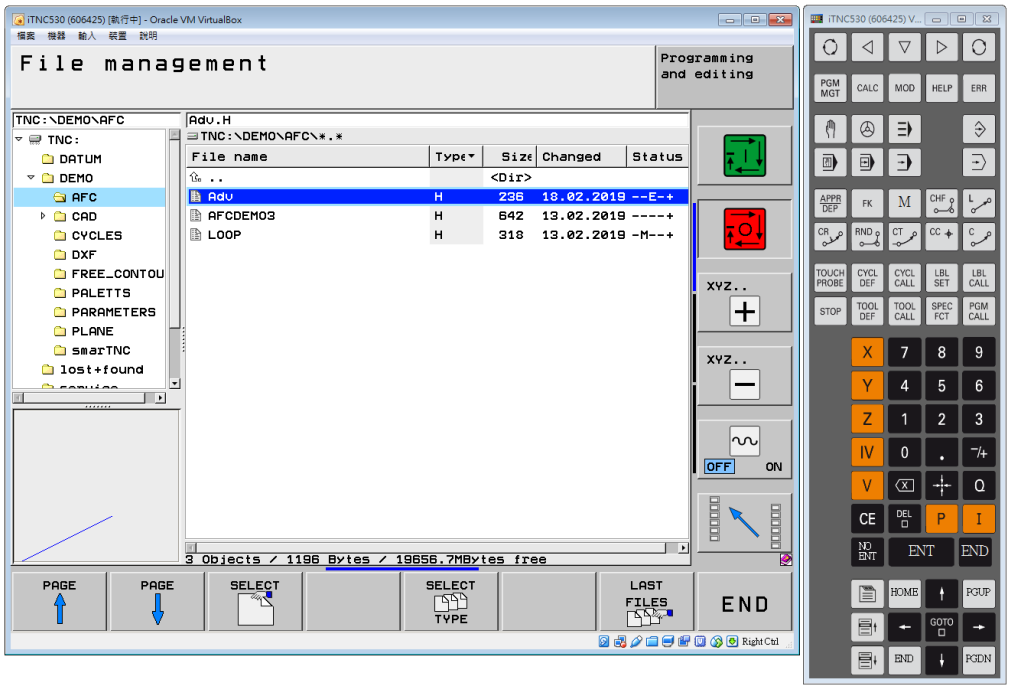
Store the program that you edit.

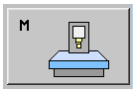
-> Select  Program Run with Full Sequence key

-> 

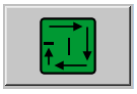
Select/Delet Program & File key

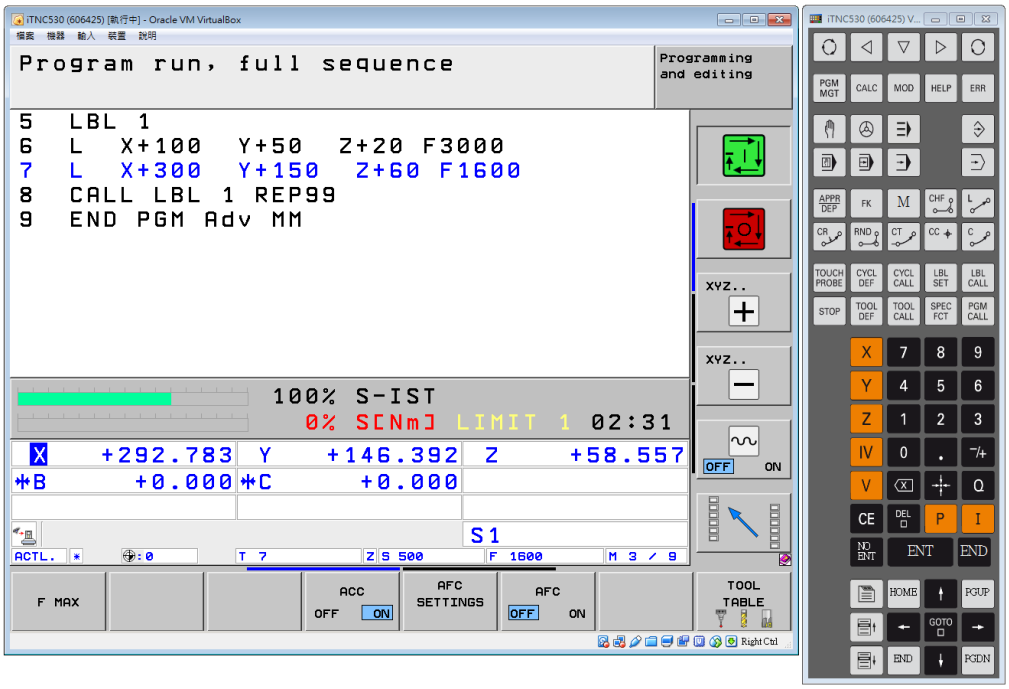
-> Find path: "**TNC:\DEMO\AFC*.***" and choose "Adv.H" file (For this example usage) as picture below



-> Click 

Machine Functions key

->  NC Start key for running the program that you selected.



■ **Reference:**

Heidenhain iTNC 530 User's Manual HEIDENHAIN