

## **Drip Feed/DNC from PCMCIA Card**

### **HYUNDAI-KIA**

#### **Application Notes**

Some of the Hyundai-Kia VX machines come with 256kByte of memory, when program is larger than that, operator can use drip feed (or DNC) function to run the NC code from PCMCIA card directly.

#### **Steps:**

1. Make sure the tools and fixture offsets are established.
2. Insert machine PCMCIA card with the program on the card slot to the left of the CNC screen, make sure the I/O is in Channel 4.  
Verify through [MDI MODE] > [OFS/SET – HardKey] > [SETTNG - SoftKey]
3. Go to [TAPE MODE] > [PROGRAM –HardKey] The use very left and right soft key to page around until you see [DNC-CD] on the screen button manual, push the soft key underneath to select it
4. On the screen you will see a list of programs in you PCMCIA card, you will also see a line “DNC FILE NAME:           ”. Type the line number for the desired program, (i.e. 3 for the third program down the list, O1234, this program can have any name), then push [DNC-ST] SoftKey to select that program. After which, you will see something like “DNC FILE NAME = O1234”, which means the program (e.g. line # 3 program name O1234) is selected.
5. Turn down the feed rate and rapid override; push the green cycle start to run the program.
6. Once Cycle Start is pushed, the program will appear on the screen. If the program is paused, (Single Block, M01, M00, or Feed Hold, a search for a N word can be done, allowing the operator to Program N words at safe starting positions within the program, so interrupted programs can easily be started at a chosen operation.

## Second way of using PCMCIA card to run big NC codes

Second way that uses a “Main-Program” program that resides in Memory to call a “Sub-Program” that resides on the card. Make sure I/O channel is set to 4.

Sub-programs can be created to be the entire program and contain all the information for part machining.

Sub-Programs can also contain only data for a single tool, allowing the Main-Program to call the same operation over multiple fixture offsets.

Main-program Example: The Calling of 2 sub-programs with different work offsets

```
%  
G0G90G40G17G80  
G54X0.Y0.  
M198 P1 (Run sub-program 1 with workoffset G54)  
M01  
M198 P2 (Run sub-program 2 with workoffset G54)  
G55X0.Y0.  
M198 P1 (Run sub-program 1 with workoffset G55)  
M01  
M198 P2 (Run sub-program 2 with workoffset G55)  
G56X0.Y0.  
M198 P1 (Run sub-program 1 with workoffset G56)  
M01  
M198 P2 (Run sub-program 2 with workoffset G56)  
. . .  
M30.  
%
```

The format of the **P** on the **M198** line depends on Parameter 3404.2 (SBP).

If **parameter 3404.2** is set to 1 then the **name of the file** is called. In this case the name **must** be a 5 character name with no extension

Example: For the Line the line **M198 P1** in the Main-Program to call the sub program on the card, the Sub-Program file must look like O0001, both on the PC and the CNC Control, i.e. **M198 P275** will call and run O0275 from the card.

If parameter 3404.2 is set to 0 then the Card File Number is called. In this case the **M198 P1** calls to the first file on the card. **M198 P102** calls to the file number 102 on the card.