

# HB-20 NX POST Development User Guide

Version 1.0.0

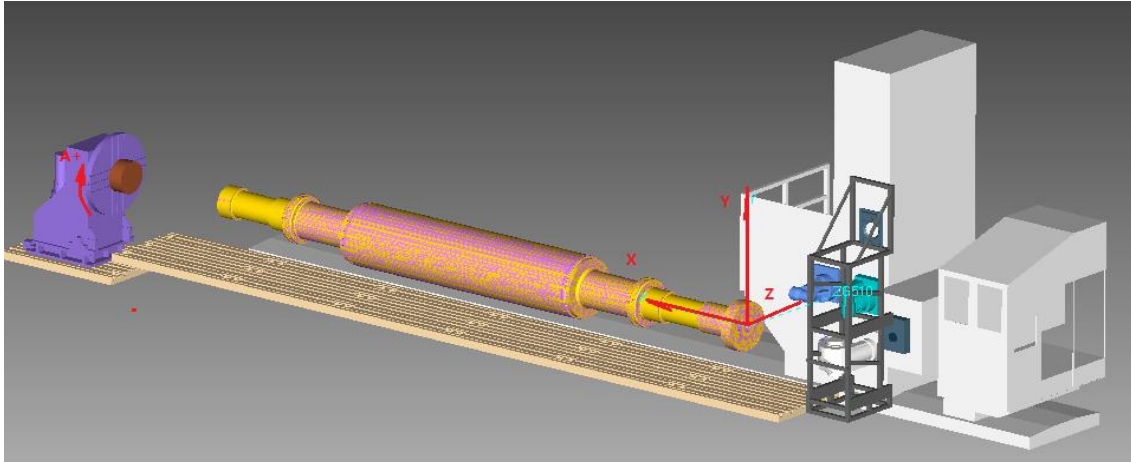
Machine Type	HBM
NC Controller	Siemens 840D
X-Axis Limit	?
Y-Axis Limit	?
Z-Axis Limit	?
A-Axis Limit	360 (0.001 resolution)

By  
James

**NX CAD CAM Automation Service (NCCAS)**

Email: [support@nxcadcam.com](mailto:support@nxcadcam.com) Web: [www.nxcadcam.com](http://www.nxcadcam.com)

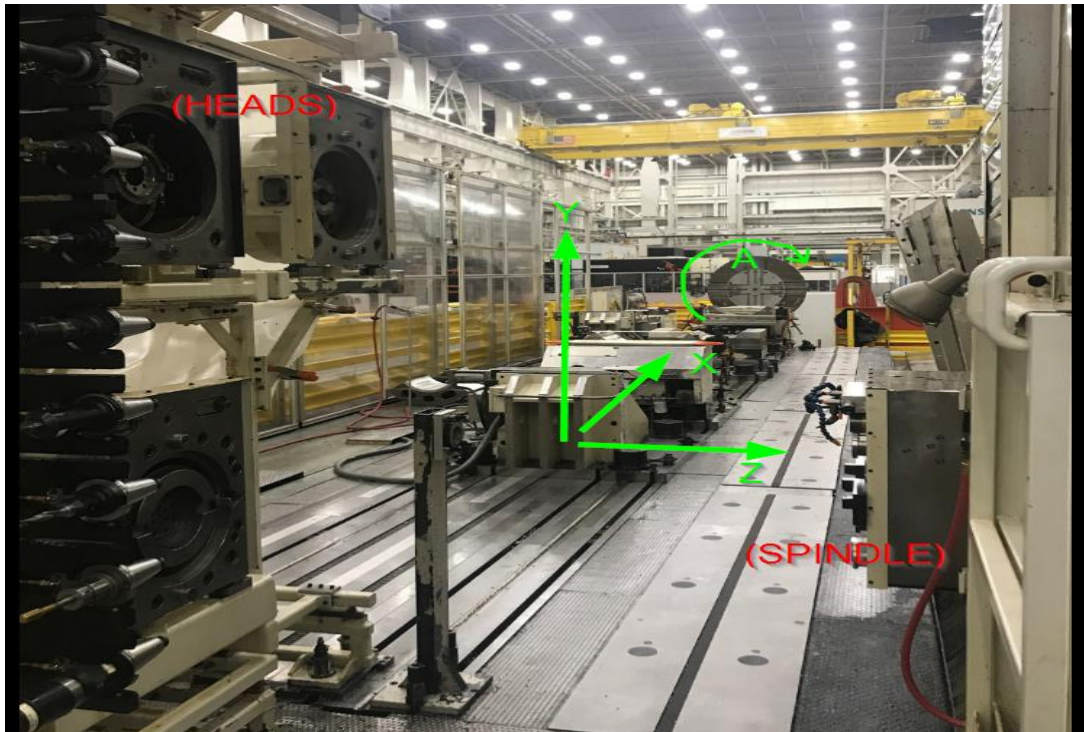
# Machine Specification:



## I. Straight Head

With straight head, spindle in Zm axis, it could be:

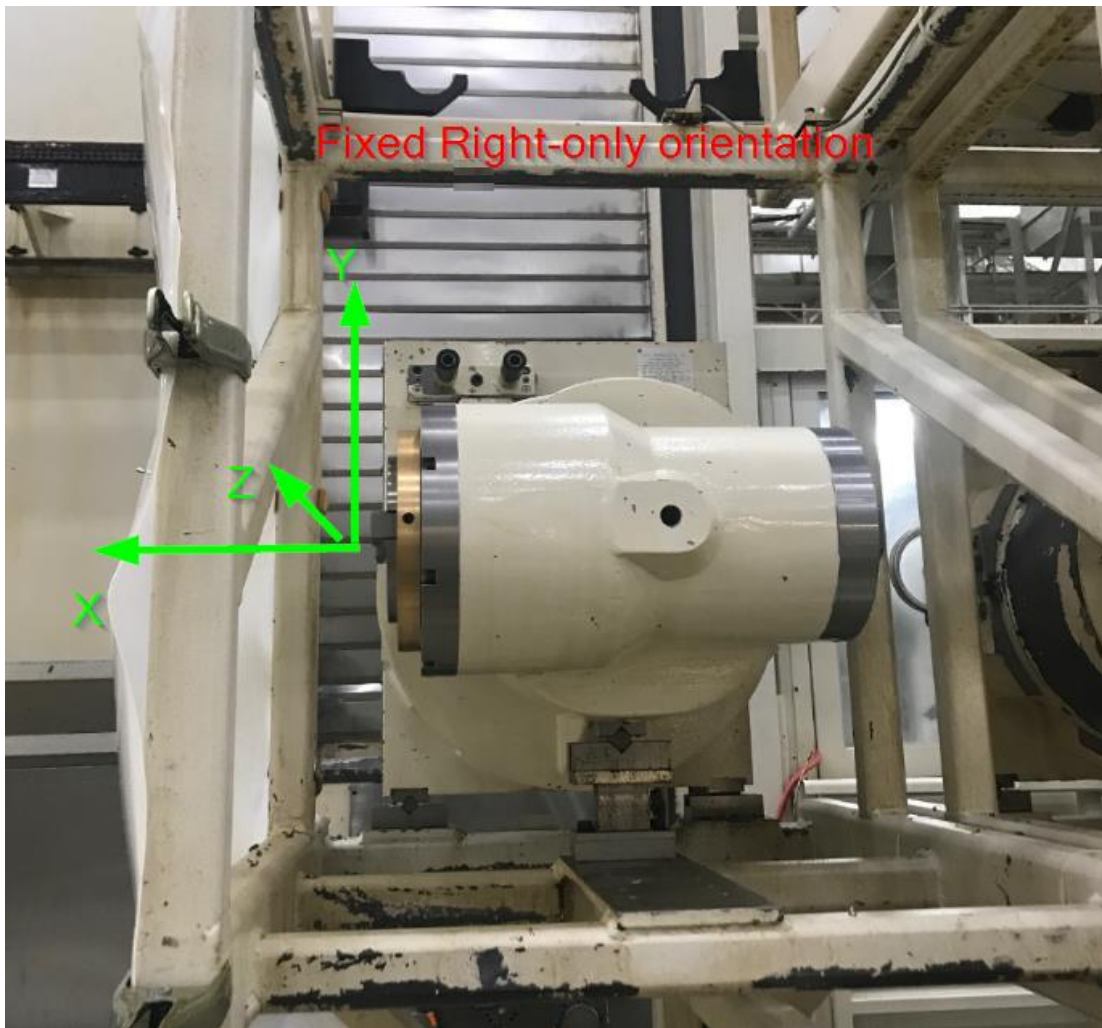
- 1) 3-axis machine without rotary A table;
- 2) 4-Axis machine with rotary A table.



## II. Fixed Right-Angle Head

The right-angle head is fixed orientation to the right. It could be:

- 1) 3-axis machine without rotary A table;
- 2) 4-Axis machine with rotary A table.

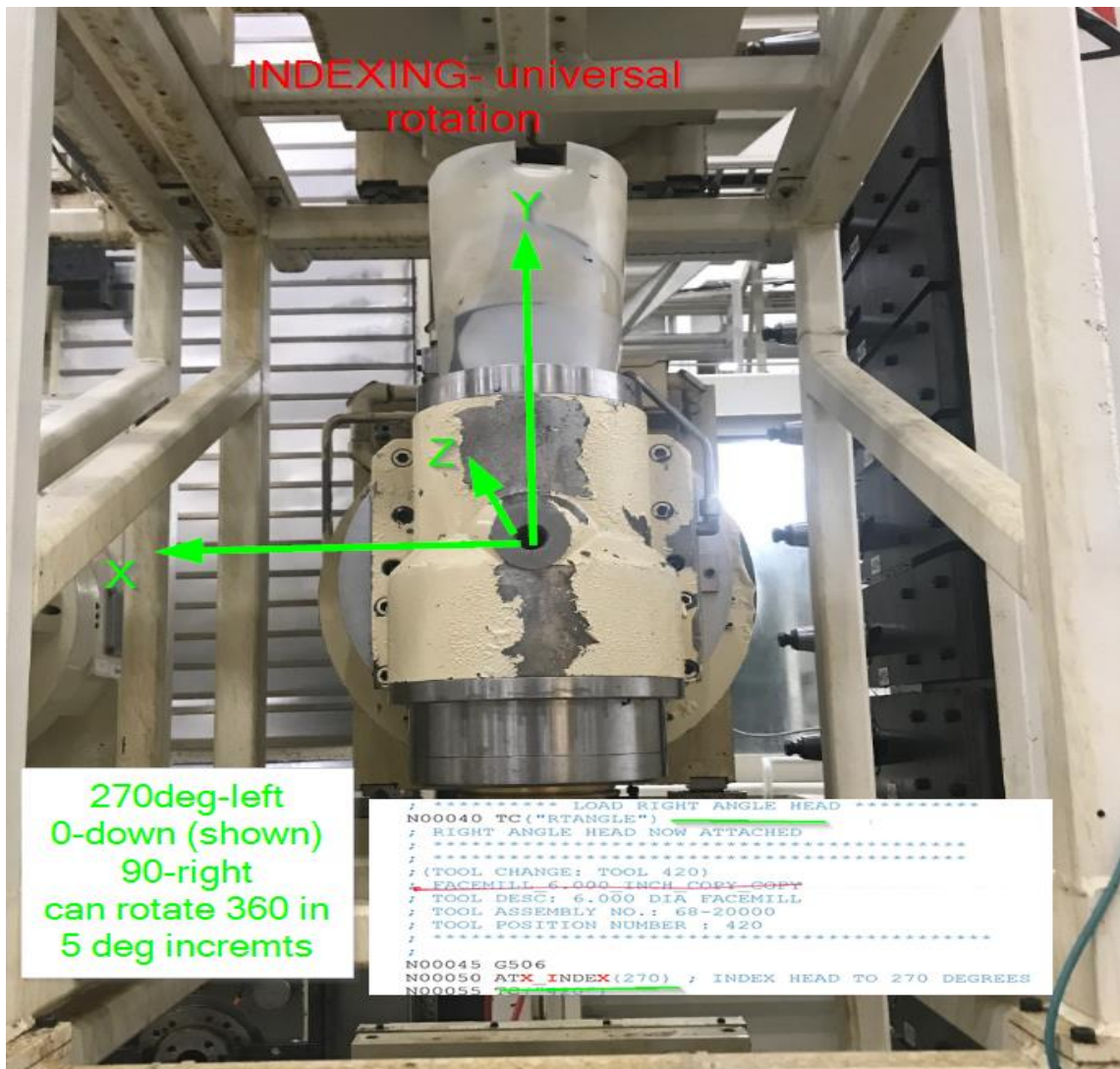


### III. Indexing Universal Head

The universal head has the incremental indexing of 5.0 degree.

- 0 degree → Down
- 90 degree → Right
- 270 degree → Left

This could be configured as a 5-Axis machine, with indexing resolution of 5.0 degree.





## IV. Kaiser Universal Indexing Head

The universal head has 0.1 indexing resolution, and it could be indexed in 360 degree. Check and confirm the tool orientation:

- 0 Degree
- 90 Degree
- 180 degree
- 270 degree



- Firstly, confirm the tool orientation;
- This could be configured as a 5-Axis machine, with the tool spindle index of

Sample code as shown below –

```
; *****
;
; ( M101 )
N05570 G512
N05575 G75 Z0
N05580 KAISER_INDEX(180.0,"309")
; *****
; SPOT M8 HOLES ON TE BLOWER HUB FACE
; .5 MM DEEP
; *****
; .75 DIA 90 DEG SPOT DRILL
N05585 G512
N05590 M03 S500
N05595 G0 X-25.4 Y23.244
N05600 Z339.3406
N05605 X25.4
N05610 Z-40.6594
; REFERENCE PLANE = -40.65942
; CYCLE = DRILL
; DEPTH = .5
; CLEARANCE PLANE = 5
; RETRACT PLANE = 25.4
N05615 G0 F63.
N05620 MCALL CYCLE81 (25.4,0,5, ,.5)
N05625 Y23.244 Z-40.6594
N05630 Y-23.244 Z-40.6594
```

## Tool Head & Tool Number

For each tool head, we have the specific tool numbers, as shown below:

**1. Straight Spindle--**  
**Tool#1-299**

**2. Kaiser Head – (small indexing head-not yet tested)**  
**Tool#300 -400**

**3. Indexing Head**  
**Tool#401-699**

**4. Fix Rt. Angle**  
**Tool # 700+**

So, the postprocessor will decide which spindle head to use based on the tool number.

We don't need another UDE to select the tool head. Please pay extra attention to the tool number.

## NX Post Development and Modification

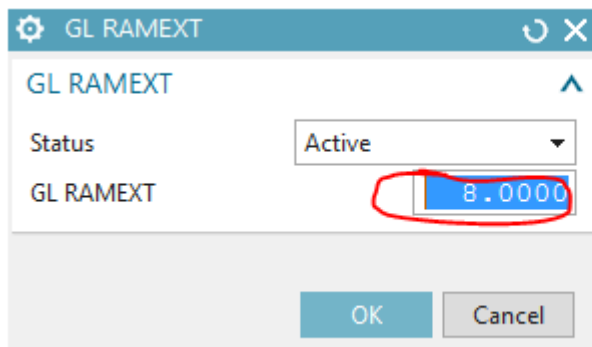
Based on the different machine configuration, we will utilize the current postprocessor and modify it for a 3-Axis postprocessor. And then the next step will develop a new postprocessor for 4&5 axis postprocessor.

### I. 3Axis Postprocessor Modification

Utilize the existing 3Axis postprocessor, optimize the output to meet the manufacturing needs.

## Change Records -20181220

- (1) Change the output digits of 4 decimals if inch output.
- (2) Added program unit at the program header.
- (3) Changed the programmer name to upper case.
- (4) Added “M01” at the start of the first tool operation.
- (5) Fixed the “MMPM =...” or “IPM = ...”
- (6) Remove the “Orient value = ..” for bore operation.
- (7) Fixe the GL\_RAMEXT output in Inch or Metrix, the original value must be set in the same unit as the part file unit.



**this value needs to be the same unit as the part, then the post will check the part unit against the output unit, then convert if not the same.**

- (8) To output the operator message just after the operation name, you need to insert a “Tool Change Marker” UDE after the operator message UDE.



(9) This change request is not implemented.

```
N00060 TRANS Z=RAMEXT
MSG{"T"<<$P_TOOLNO<<" IN SPINDLE:LENGTH="<<$P_TOOLL[1]<<" CUTTER COMP="<<$P_TOOLR)
N00065 M00
N00070 G94 M05
; **** ROTATE NOW ****
N00075 M01
N00080 M05
N00085 G01 A=ACP{.000} F300
N00090 STOPRE
N00095 M01
; *****
N00100 G505
N00105 M40
```

THIS OUTPUTS AUTOMATICALLY WHICH IS GOOD - CAN YOU CREATE UDE TO DEACTIVATE WHEN NOT NEEDED.  
- AS IN THIS CASE WHERE THERE'S NO PART ROTATION



This is not changed, because the logic is good:

(1) If it is the first operation, it will always rotate, even if it is 0.0, which is good to make sure the rotary table index to A0.0.

(2) If the current A value is different from the previous operation, this rotate will be output.

(3) If the current A value is the same as the previous operation, this rotation won't be output.

(10) The helix output for the Helical engage move is fixed, now helical is output.

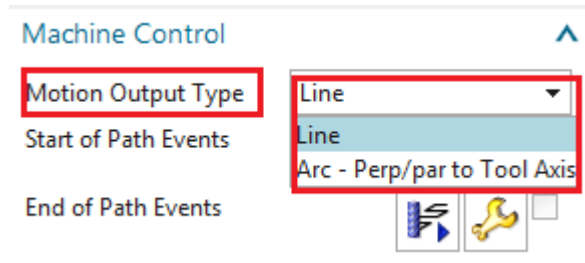


## Update on 2019-01-10

### (1) About the helical move output I J and K.

Since the tool path is a helical move, we can output it as a helical move or output it as linear.

If you want to output as linear, change the cycle output option as linear. If this option is not available in the operation dialog, you can always customize the operation dialog to display it.



By default, it should be output as helical move, there will have XYZ and IJK. So the following output is good, we need to check if the machine could accept it.

- The K has been removed as it may not be necessary.

```

N00280 Z.35
N00285 G94 G03 X1.97 Y0.0 Z.1 I.985 J0.0 K.0796 F5.58
N00290 G03 X1.97 Y0.0 Z-.9 I-1.97 J0.0 K.1592
N00295 G03 X1.97 Y0.0 Z-1.9 I-1.97 J0.0 K.1592
N00300 G03 X1.97 Y0.0 Z-2.9 I-1.97 J0.0 K.1592
N00305 G03 X1.97 Y0.0 Z-3.9 I-1.97 J0.0 K.1592
N00310 G03 X1.97 Y0.0 Z-4.9 I-1.97 J0.0 K.1592
N00315 G03 X1.97 Y0.0 Z-5.9 I-1.97 J0.0 K.1592
N00320 G03 X1.97 Y0.0 Z-6.9 I-1.97 J0.0 K.1592
N00325 G03 X1.97 Y0.0 Z-7.9 I-1.97 J0.0 K.1592
N00330 G03 X1.97 Y0.0 Z-8.9 I-1.97 J0.0 K.1592
N00335 G03 X1.97 Y0.0 Z-9.9 I-1.97 J0.0 K.1592
N00340 G03 X.6088 Y1.8736 Z-10.1 I-1.97 J0.0 K.1592
N00345 G03 X0.0 Y0.0 Z-10.35 I-.3044 J-.9368 K.0796
N00350 G00 Z1.
    
```

?

NO K ON G17

(2) Corrected the Feed mode and output for Tapping operation.

```

N00960 M03 S125
N00965 G17
N00970 G00 X0.0 Y15.
N00975 Z1.
N00980 GO W--(RAMEXT)
; REFERENCE PLANE = 0
; CYCLE = DRILL_TAP
; DEPTH = 1.987
; IPM = 2.54
; CLEARANCE PLANE = 0.1
; RETRACT PLANE = 1
N00985 F12.5
N00990 M08
N00995 G95
N01000 MCALL CYCLE84(1.,0.,0.1,,1.987,0.,3, 2.54 0.,125.,125.,,0,0,0,0,0)
N01005 X0.0 Y15.
N01010 X0.0 Y-15.
N01015 MCALL

```

*Handwritten notes:*  
 IPR FOR TAPPING  
 MIPR  
 ← PROG'D .100 IPR  
 ↓ IPR

The new output is like this –

```

; REFERENCE PLANE = 0
; CYCLE = DRILL_TAP
; DEPTH = 1.987
; IPR = 0.10
; CLEARANCE PLANE = 0.1
; RETRACT PLANE = 1
N00105 F12.5
N00110 M08
N00115 G95
N00120 MCALL CYCLE84(1.,0.,0.1,,1.987,0.,3, 0.1,0.,125.,125.,,0,0,0,0,0)
N00125 X0.0 Y15.
N00130 X0.0 Y-15.
N00135 MCALL

```

## Updates on 2019-03-13

### (1) About the thread-Milling

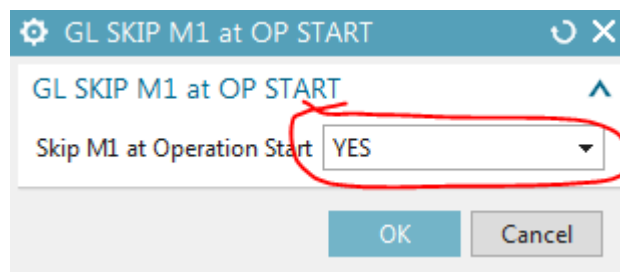
“THREADMILLING --- I NEEDED M01'S BETWEEN HOLE OUTPUT SO WE CAN PICK BACK UP AS NEEDED AND CHASE THREAD SIZES I HAD TO PROGRAM EACH HOLE IN SEPARATE OPS INORDER TO GET THE DESIRED OUTPUT. I'D PREFER TO SELECT ALL THE HOLES IN ONE OPERATION AND LET THE POST PUT M01'S AND SPINDLE CODES BETWEEN HOLES. DO YOU HAVE A FIX FOR THIS?”

- ⇒ I understood what you want, and have tried many methods, but I could not find a solution for that yet. Because the tool path is listed as many helix moves, it is not like a canned cycle, it is hard to separate each of those hole.
- ⇒ I will keep working on that, and hopefully to find a solution in the future. I believe there is a way to do that, I will need more time to work on that.

### (2) About “Skip M1 at OP start”.

- ⇒ Add the following ude definition to the ude.cdl.

```
EVENT GL_SKIP_M1
{
  POST_EVENT "GL_SKIP_M1"
  UI_LABEL "GL SKIP M1 at OP START"
  CATEGORY MILL DRILL
  PARAM gl_skip_m1
  {
    TYPE o
    DEFVAL "YES"
    OPTIONS "YES","NO"
    UI_LABEL "Skip M1 at Operation Start"
  }
}
```

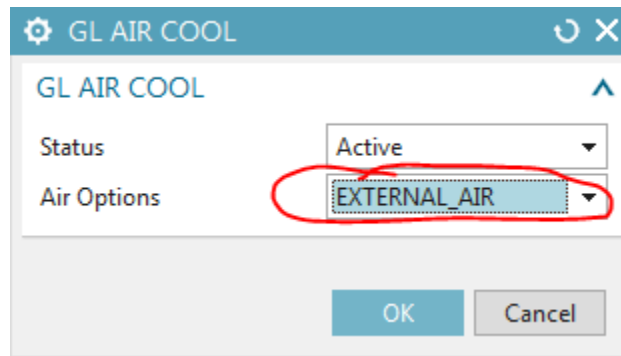


**(3) Remove Feed for Tapping operation.**

⇒ Done.

**(4) Mist M50**

⇒ Fixed.

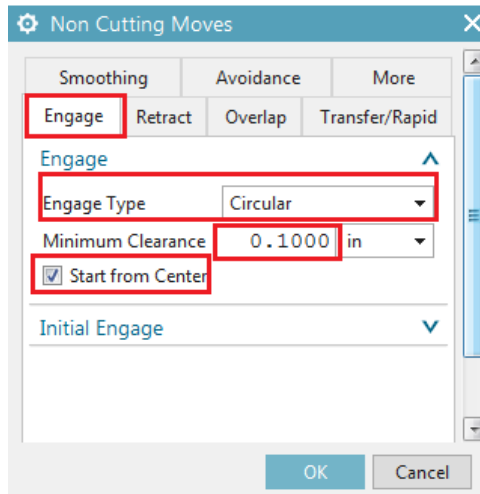


## HELICALBPORE OUTPUT CUSTOMIZATION

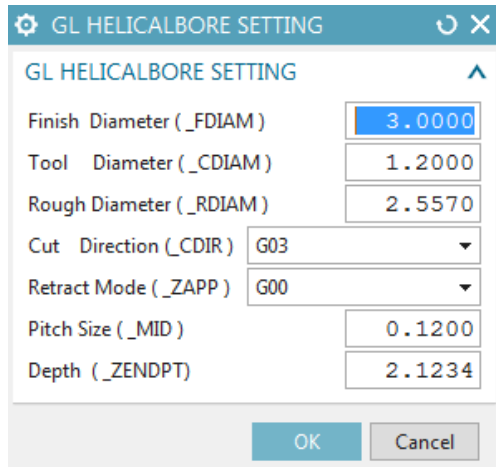
- (1) The operation must be “HOLE\_MILLING” operation type.

You can create the operation from “Mill Planar” → “Hole Milling” or from “Hole Making” → “Hole Milling”.

- (2) The non-cutting engage type must set to circular and toggle on start from center. → The Postprocessor will find the start point from the engage move position.



- (3) Insert the “GL HelicalBore Setting” UDE at the start event of the operation. → Input all the parameters as needed.



**For more information**

**Please contact **NCCAS** (NX CAD/CAM Automation Services)**

**[support@nxcadcam.com](mailto:support@nxcadcam.com)**

**THANKS!**