OKUMA VTM 1200YB Mill-Turn 5-AXIS

OSP-P300S-H

TURN-POST

By

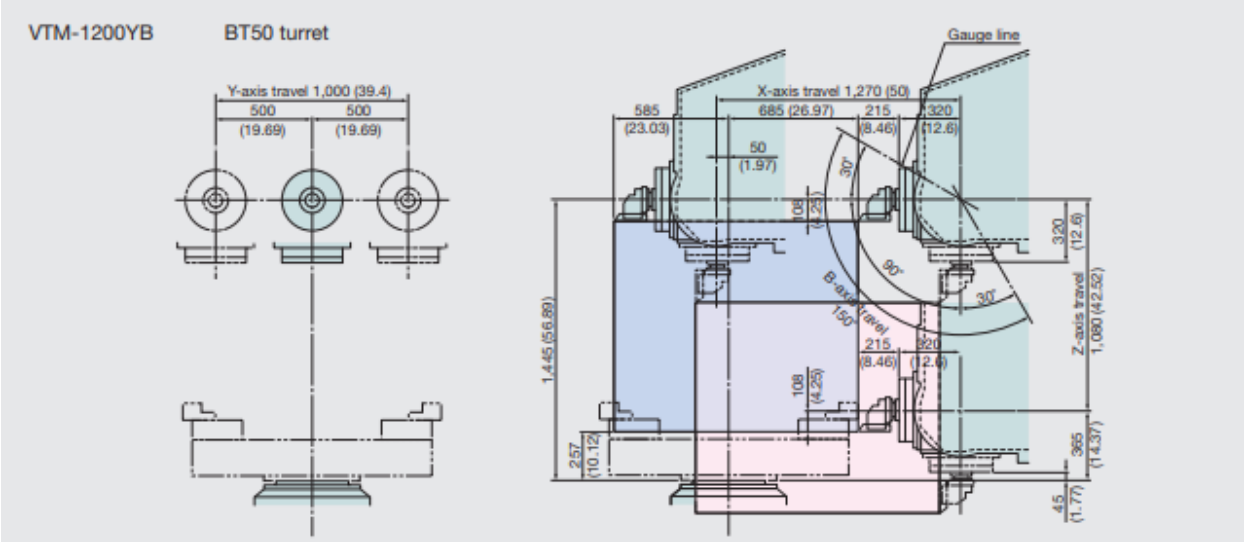
James

02/11/2020

I. Machine Specifications

MAX CUTTING DIAMETER	in	Ø49.21
MAX CUTTING LENGTH (HEIGHT)	in	42.52
X-AXIS TRAVEL	in	50
Y-AXIS TRAVEL	in	39.37 (-19.69 to +19.69) [-24.4 to +24.41]
Z-AXIS TRAVEL	in	42.52 [60.24]
RAPID TRAVERSE - X:Y:Z	fpm	105
RAPID TRAVERSE - C:B	min⁻¹	20:19.5
TABLE SPINDLE SPEED	min⁻¹	5~500 [4~400]
TABLE SPINDLE POWER	hp	40/30
MILLING SPINDLE SPEED	min⁻¹	40~10,000
MILLING SPINDLE POWER	hp	50/40/30 (3 min/30 min/cont)

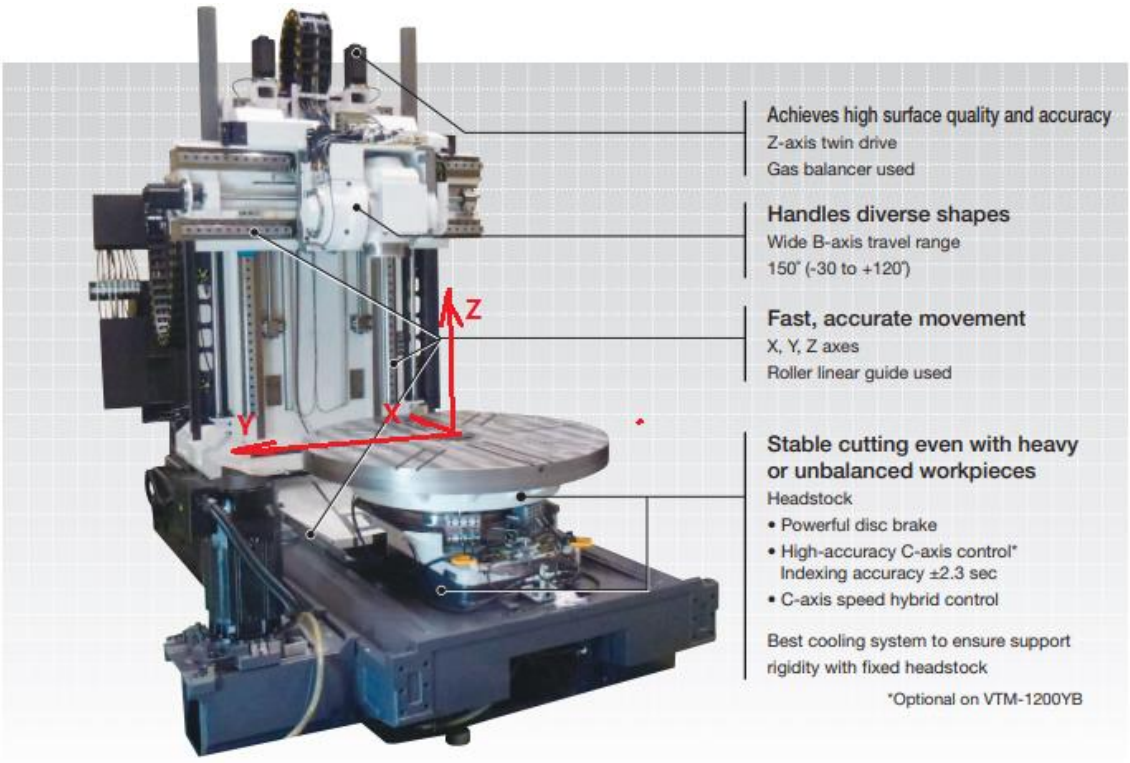
B Axis 150 Degree (-30 ~ +120) 0.001
C Axis 360 Degree 0.001



II. Machine MCS setup.

The MCS set up is as shown below, when spindle is vertical, B is 0.0, B travel limits is between -30 and +120 degree, as shown below.

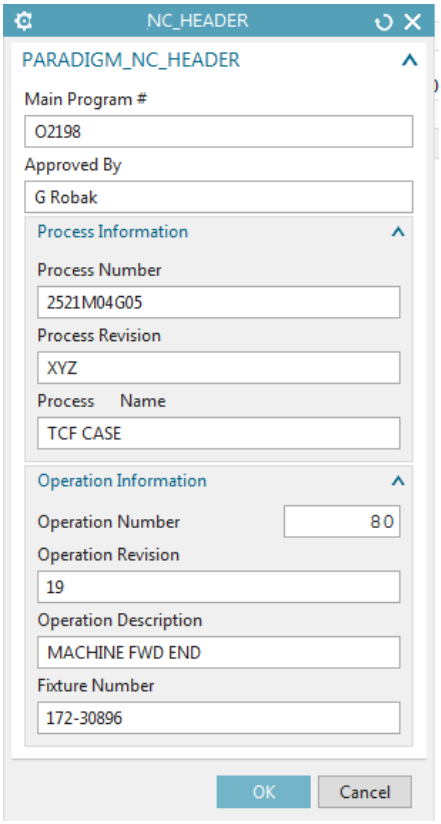
To make it simple, we made the B range to be **0 ~ 120**, because for -30 degree orientation, we can do that with 30 degree orientation, together with the C axis rotation. If later we do need that -30 degree orientation, we can add an UDE to change that.



- **B = 0, the spindle is in vertical position.**
- **B = 90, the spindle is in horizontal position.**

III. Program Header UDE

The program header UDE will be the same as the Parpas program header UDE. Turn Post and Mill Post are using the same Header UDE.

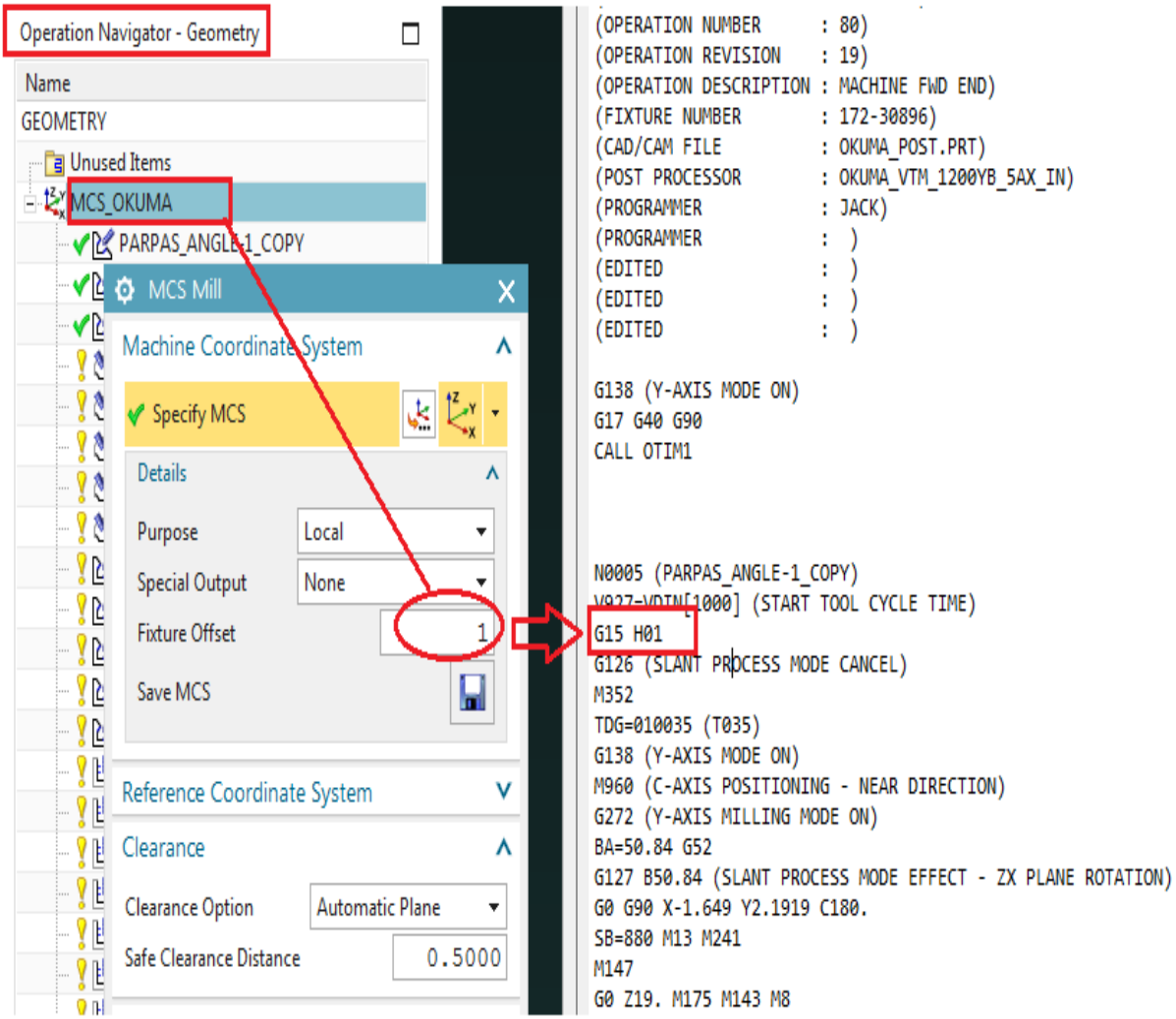


Output as shown below:

```
(MAIN PROGRAM NUMBER   : O2198)
(CREATED ON            : 02/21/19  AT 20:31:10)
(PROGRAMMED UNIT       : INCH  )
(APPROVAL              : G ROBAK)
(PROCESS NUMBER        : 2521M04G05)
(PROCESS REVISION      : XYZ)
(PROCESS NAME          : TCF CASE)
(OPERATION NUMBER      : 80)
(OPERATION REVISION    : 19)
(OPERATION DESCRIPTION : MACHINE FWD END)
(FIXTURE NUMBER        : 172-30896)
(CAD/CAM FILE          : OKUMA_POST.PRT)
(POST PROCESSOR        : OKUMA_VTM_1200YB_5AX_IN)
(PROGRAMMER            : JACK)
(PROGRAMMER            :  )
(EDITED                :  )
(EDITED                :  )
(EDITED                :  )
```

IV. Select work Offset

Okuma use G15 Hx to specify the work offset, the “x” value here will be the local fixture offset value set in the MCS details.



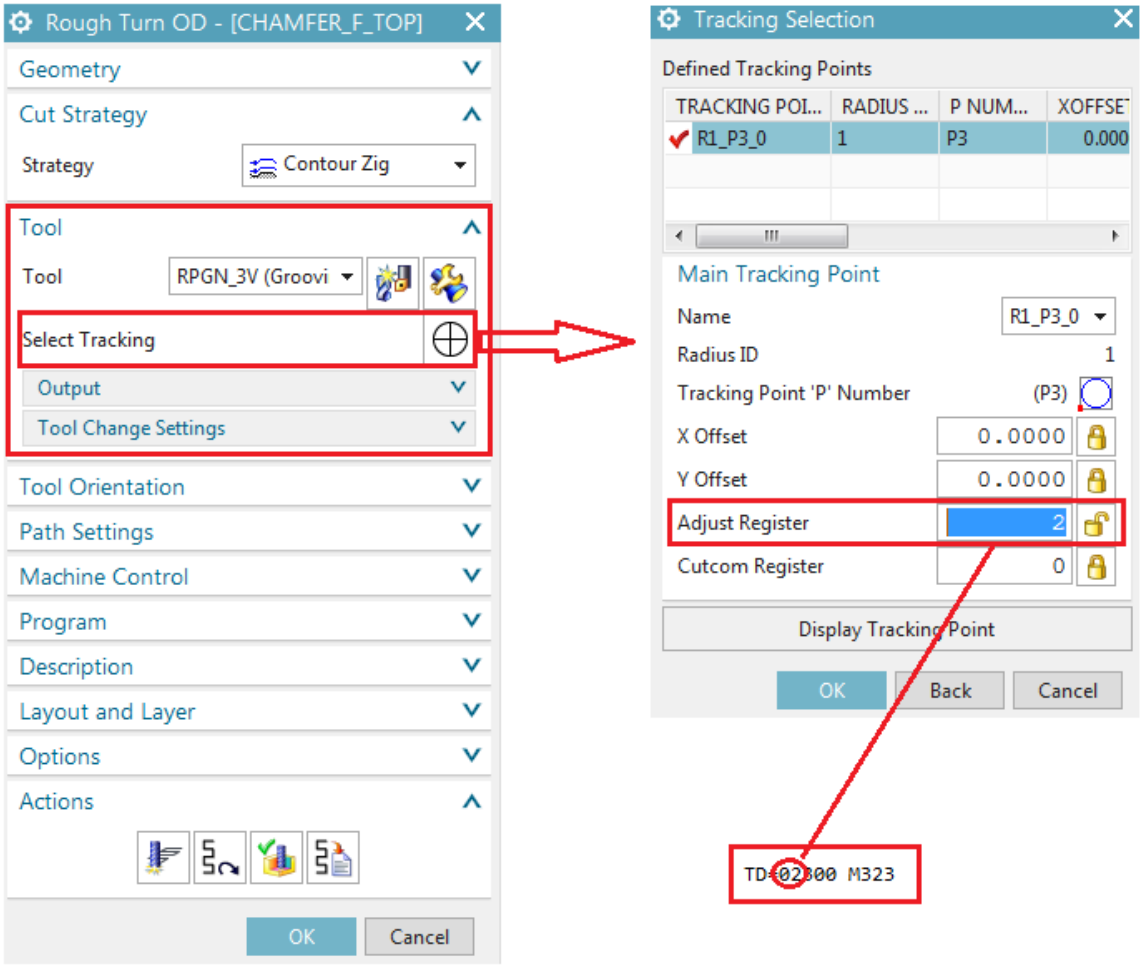
V. Tool Number and Position Number

TD command

It is possible to set the nose R compensation value for each position number in the Tool Data Setting screen.
The nose R compensation value is determined by specifying the position number.

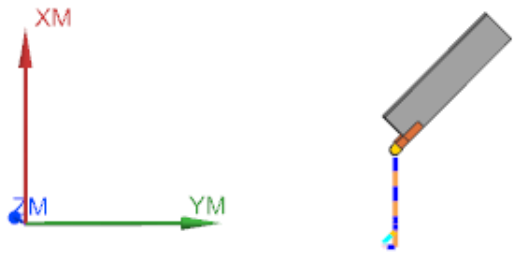
Command format

*** We will use the tool adjust register number as the position number. ➔ Set the tool adjust number at the operation dialog, which can overwrite the adjust number in the tool definition, so that we can define different position number for the same tool.

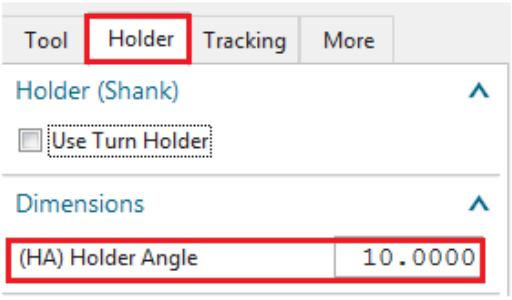


VI. Tool holder angle output.

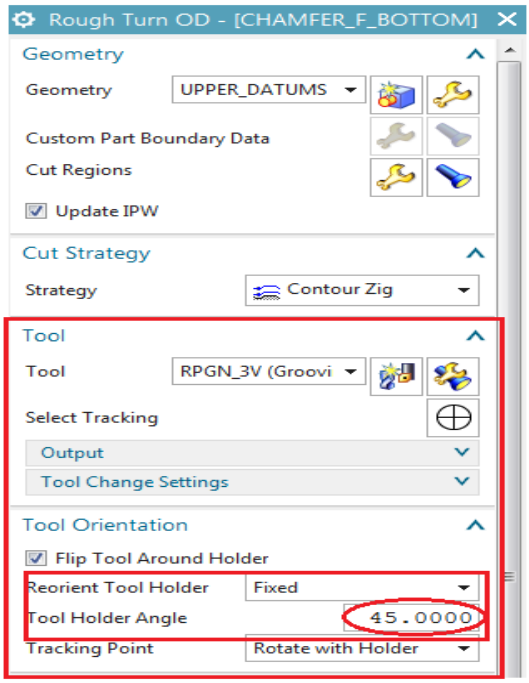
In many cases, we need to index the spindle angle so as to cut somewhere more easy, as shown below:



In this case, we need to output the spindle angle as "**M66 BA=45.0 G52**".
In NX, we could set the BA angle in 2 places:
(1) In tool definition, in this case, the tool is used for all at one orientation.



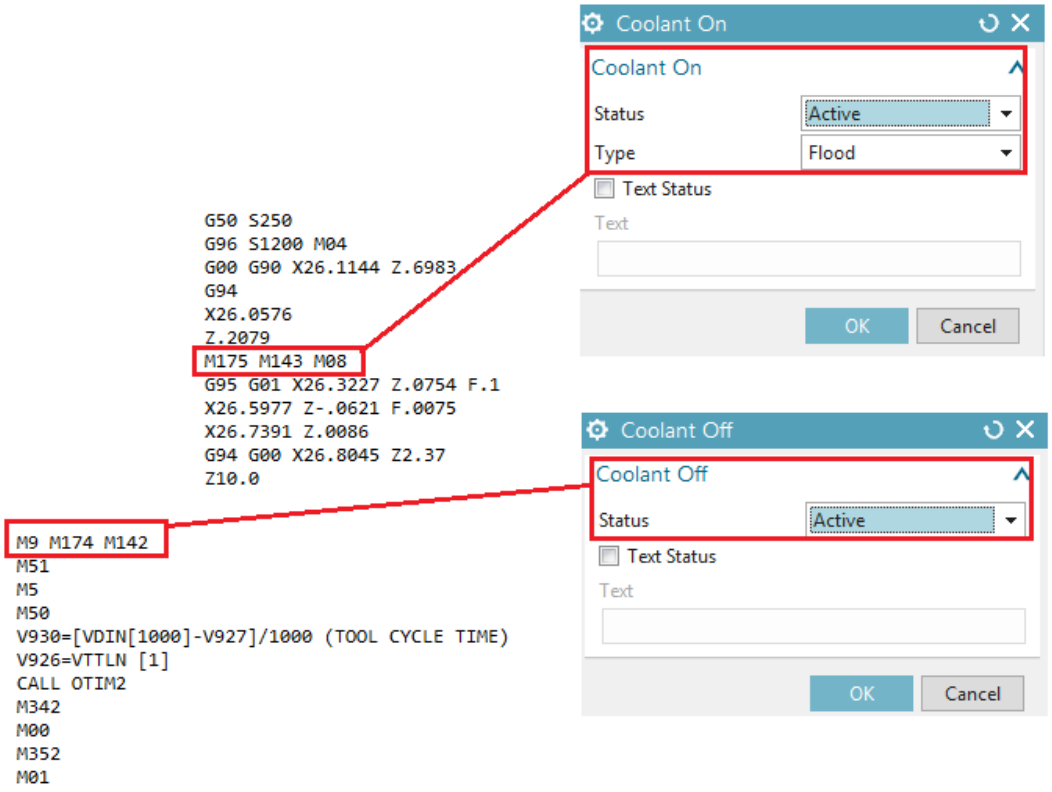
(2) In operation dialog's tool orientation, this tool could be used at different orientation as specified in the operation dialog.



**** The second option will overwrite the first option.

VII. Coolant On/OFF

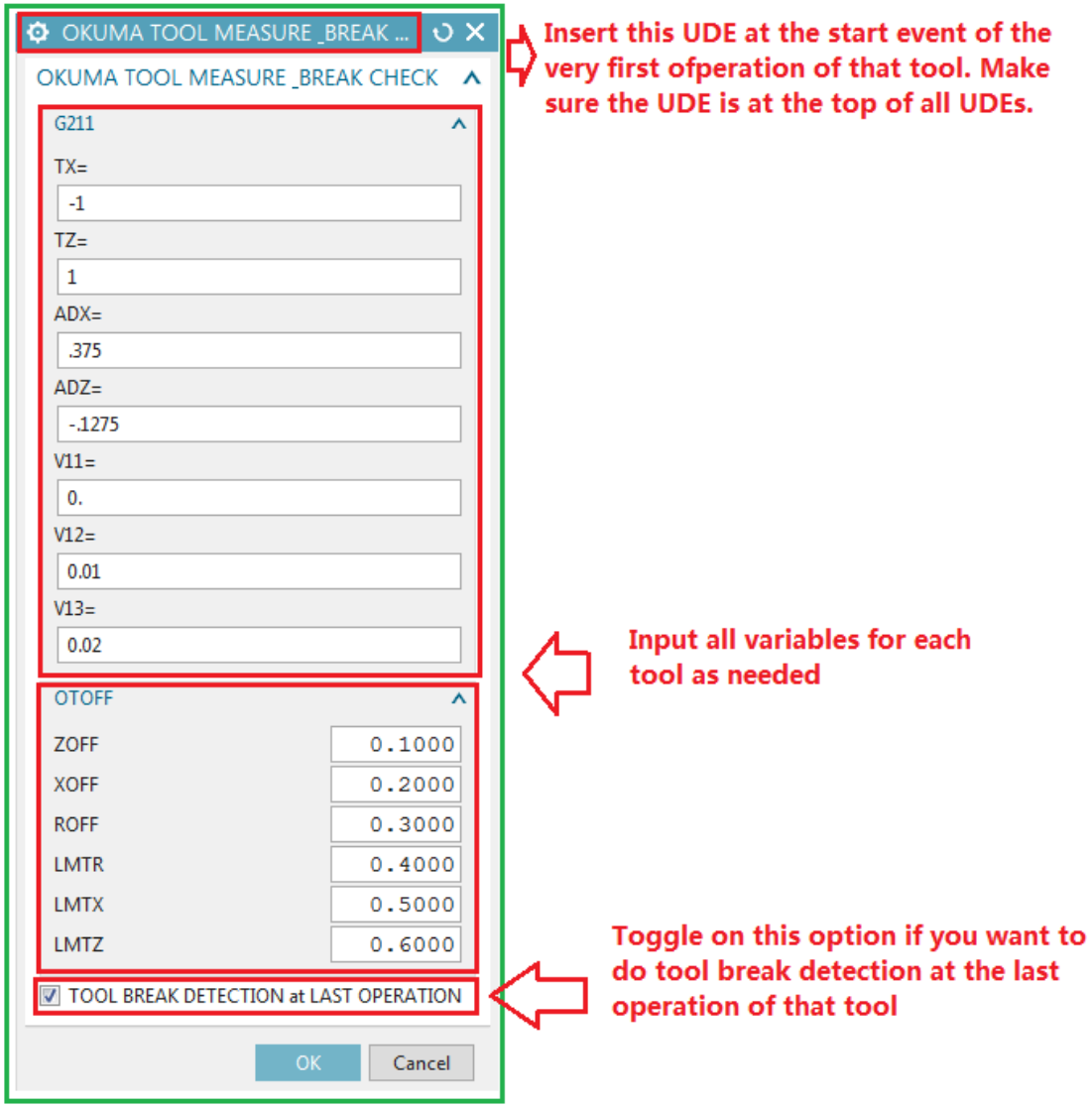
The coolant on/off is controlled together with the coolant_on/coolant_off udes. When coolant_on ude is specified, you can set the status to any status (on/flood/mist…). ➔ M175, M143 and M08 will be turned on all together.



- When there is no tool change betweenoperations, coolant status wont be changed.
- When there is a tool change on the next operation, all coolant will be turned off.

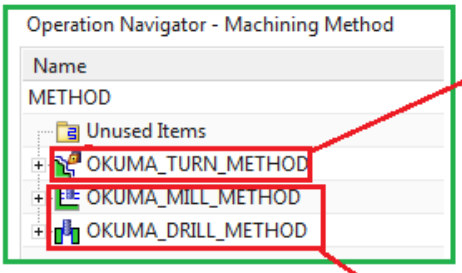
VIII. Tool Measure/Break Detection UDE (G211, G213 and OTOFF)

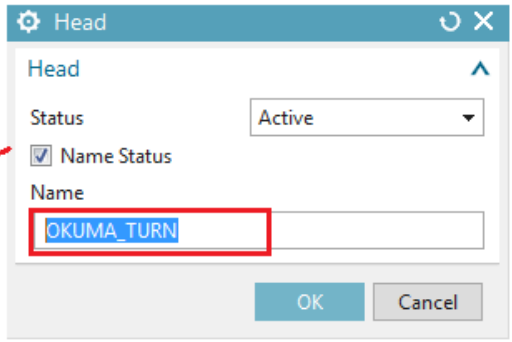
Tool Measure and Break detection UDEs are combined into one UDE, called "**OKUMA TOOL MEASURE BREAK CHECK**".

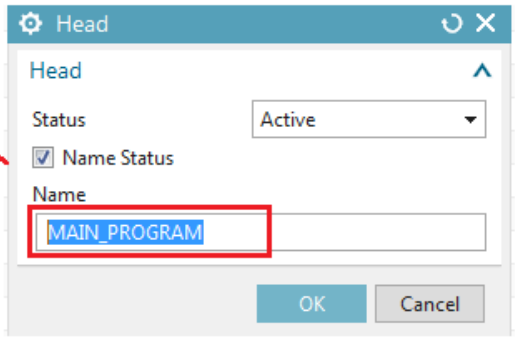


IX. Combined Mill & Turn Post

- Recommend to post the Mill and Turn tool path seperately.
- If there are Mill and Turn tool path mixed together, we can post them together.
- To post mixed mill and turn operation, we need to set the "Machining method" on each operation.





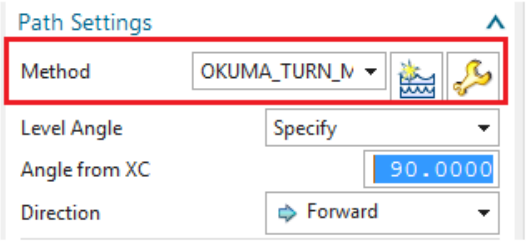


1. Creat 3 machining methods as shown above.

2. Righ-click on each method -> Object -> Start Event -> Insert "Head" UDE and input the name of the head.

3. For mill and drill use the same head name.

4. for each operation, select the right machining method.



For more information

Please contact NCCAS (NX CAD/CAM Automation Services)

support@nxcadcam.com

THANKS!