

OKUMA VTM 1200YB Mill-Turn 5-AXIS OSP-P300S-H

TURN-POST

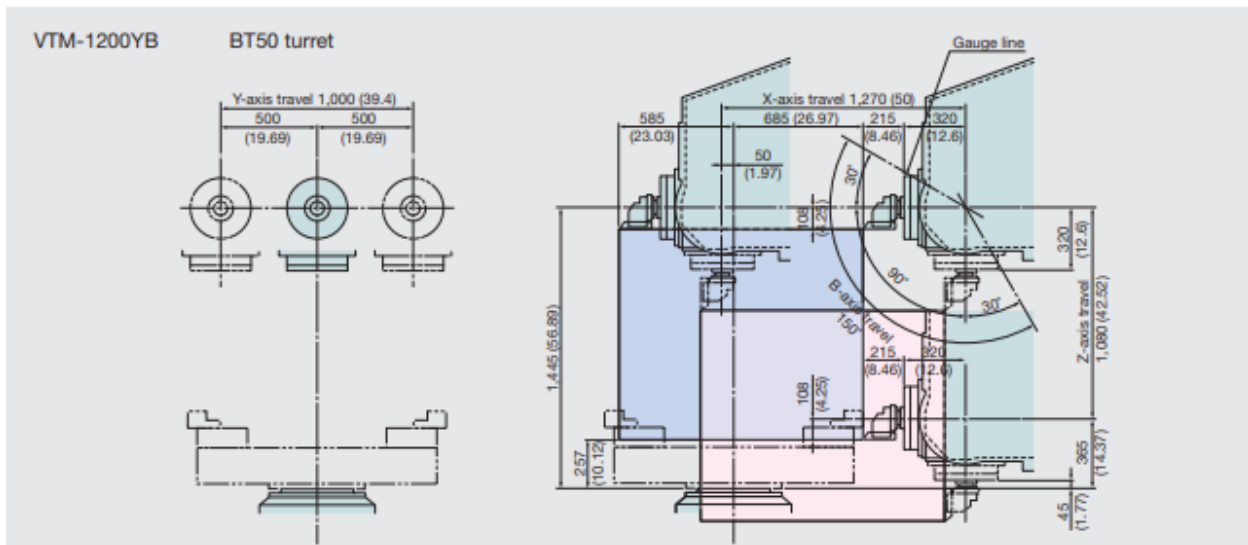


By
James
02/11/2020

I. Machine Specifications

MAX CUTTING DIAMETER	in	049.21
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)

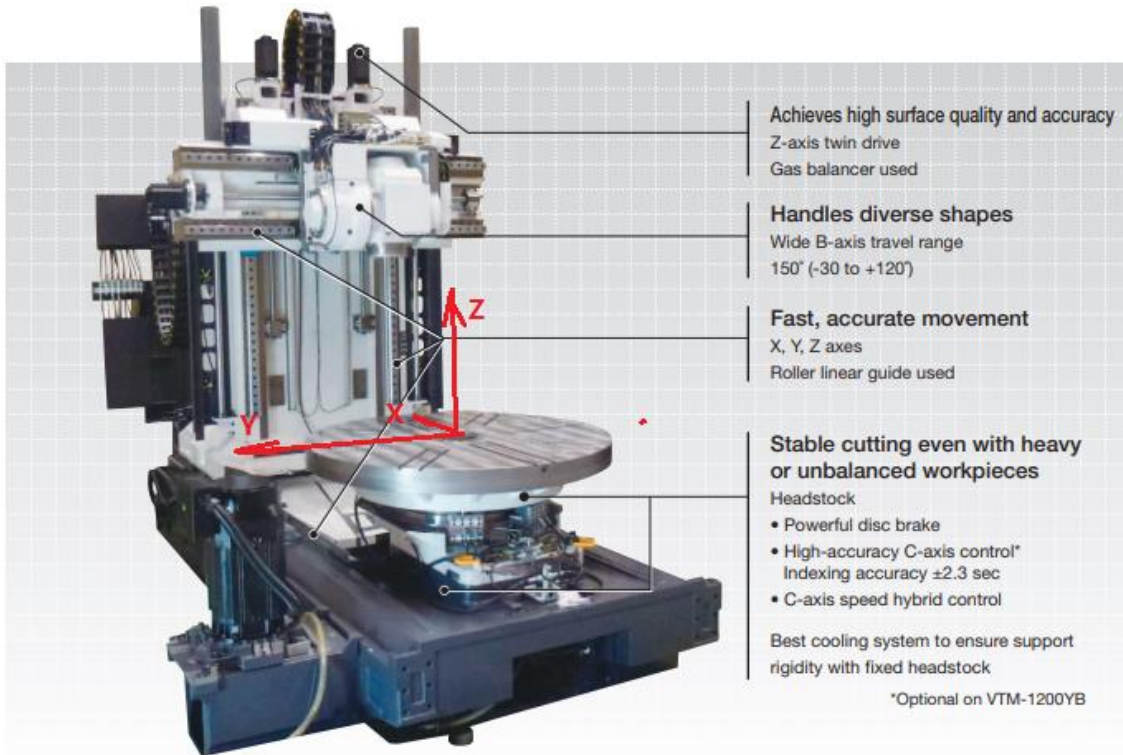
B Axis 150 Degree (-30 ~ +120) 0.001
C Axis 360 Degree 0.001



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 : 02198)
(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.

The screenshot displays the 'Specify MCS' dialog box in the NX CAD/CAM software. The 'Machine Coordinate System' section is active, and the 'Fixture Offset' field is set to '1'. A red circle highlights the '1' in the 'Fixture Offset' field, and a red arrow points from it to the 'G15 H01' code in the NC program. The 'Operation Navigator - Geometry' tree on the left shows 'MCS_OKUMA' selected. The NC code on the right includes the following lines:

```
(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 : )

G138 (Y-AXIS MODE ON)
G17 G40 G90
CALL OTIM1

N0005 (PARPAS_ANGLE-1_COPY)
V027=VDIN[1000] (START TOOL CYCLE TIME)
G15 H01
G126 (SLANT PROCESS MODE CANCEL)
M352
TDG=010035 (T035)
G138 (Y-AXIS MODE ON)
M960 (C-AXIS POSITIONING - NEAR DIRECTION)
G272 (Y-AXIS MILLING MODE ON)
BA=50.84 G52
G127 B50.84 (SLANT PROCESS MODE EFFECT - ZX PLANE ROTATION)
G0 G90 X-1.649 Y2.1919 C180.
SB=880 M13 M241
M147
G0 Z19. M175 M143 M8
```

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

TD=01 0001 M323 (M423)

*** 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.

The image shows two screenshots from the NX CAD/CAM software. The left screenshot is the 'Rough Turn OD - [CHAMFER_F_TOP]' operation dialog. The 'Tool' section is expanded, showing 'RPGN_3V (Groovi)' as the selected tool. The 'Select Tracking' button, represented by a circle with a plus sign, is highlighted with a red box. A red arrow points from this button to the right screenshot. The right screenshot is the 'Tracking Selection' dialog. It shows a table of 'Defined Tracking Points' with one entry: 'R1_P3_0' with a radius of 1 and position number 'P3'. Below the table, the 'Main Tracking Point' settings are shown. The 'Adjust Register' field is highlighted with a red box and contains the value '2'. A red arrow points from this field to a red box at the bottom of the image containing the command 'TD=02000 M323'.

VI. Tool holder angle output.

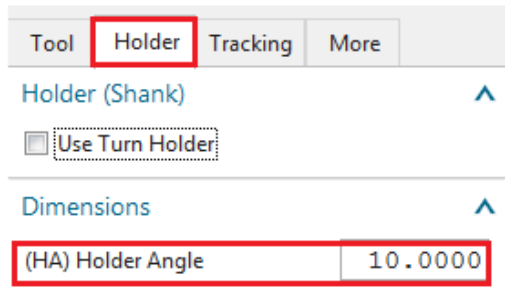
In many cases, we need to index the spindle angle so as to cut somewhere more easy, as shown below:



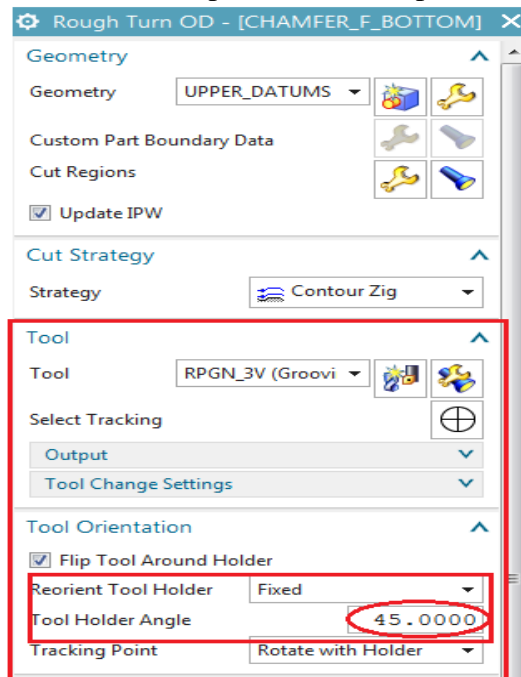
In this case, we need to output the spindle angle as “M66 BA=45.0 G52”.

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.

```

G50 S250
G96 S1200 M04
G00 G90 X26.1144 Z.6983
G94
X26.0576
Z.2079
M175 M143 M08
G95 G01 X26.3227 Z.0754 F.1
X26.5977 Z-.0621 F.0075
X26.7391 Z.0086
G94 G00 X26.8045 Z2.37
Z10.0

M9 M174 M142
M51
M5
M50
V930=[VDIN[1000]-V927]/1000 (TOOL CYCLE TIME)
V926=VTTLN [1]
CALL OTIM2
M342
M00
M352
M01
    
```

Coolant On dialog box:

- Status: Active
- Type: Flood
- Text Status
- Text: [Empty]
- Buttons: OK, Cancel

Coolant Off dialog box:

- Status: Active
- Text Status
- Text: [Empty]
- Buttons: OK, Cancel

- When there is no tool change between operations, 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**”.

The screenshot shows a dialog box titled "OKUMA TOOL MEASURE_BREAK CHECK" with a gear icon and a close button. The dialog is divided into two sections: "G211" and "OTOFF".

G211 Section:

- TX= -1
- TZ= 1
- ADX= .375
- ADZ= -.1275
- V11= 0.
- V12= 0.01
- V13= 0.02

OTOFF Section:

ZOFF	0.1000
XOFF	0.2000
ROFF	0.3000
LMTR	0.4000
LMTX	0.5000
LMTZ	0.6000

TOOL BREAK DETECTION at LAST OPERATION

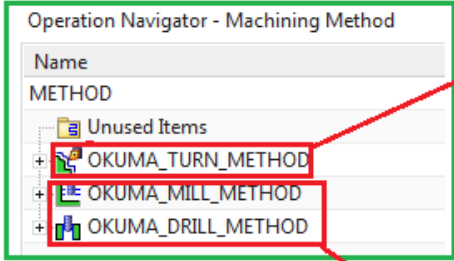
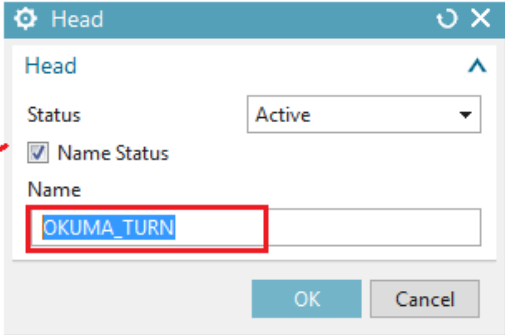
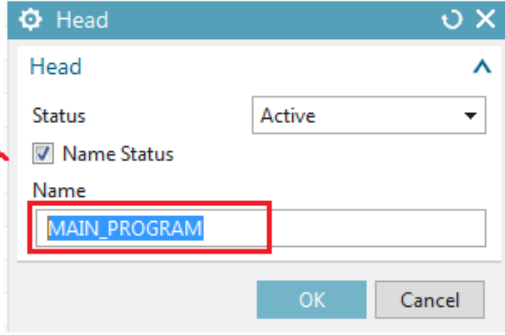
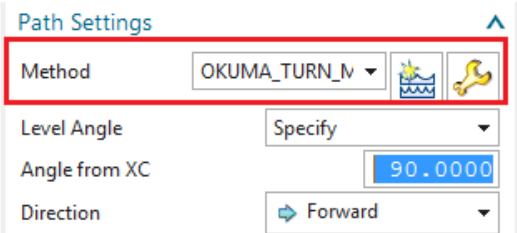
Buttons: OK, Cancel

Annotations:

- Red arrow pointing to the title bar: "Insert this UDE at the start event of the very first operation of that tool. Make sure the UDE is at the top of all UDEs."
- Red arrow pointing to the G211 section: "Input all variables for each tool as needed"
- Red arrow pointing to the OTOFF section: "Toggle on this option if you want to do tool break detection at the last operation of that tool"

IX. Combined Mill & Turn Post

- Recommend to post the Mill and Turn tool path separately.
- 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. Create 3 machining methods as shown above.
2. Right-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!