PARPAS THS-120-X-TILT
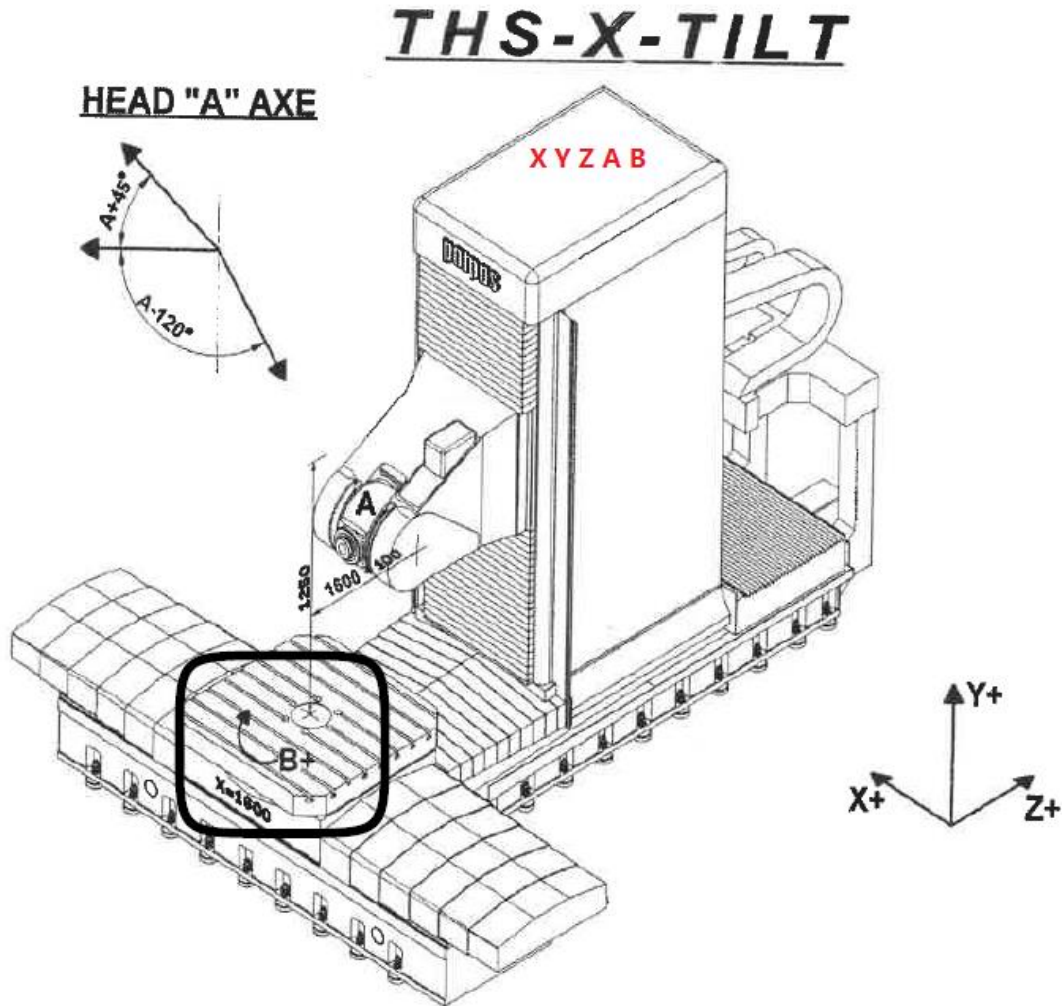## FANUC 31i
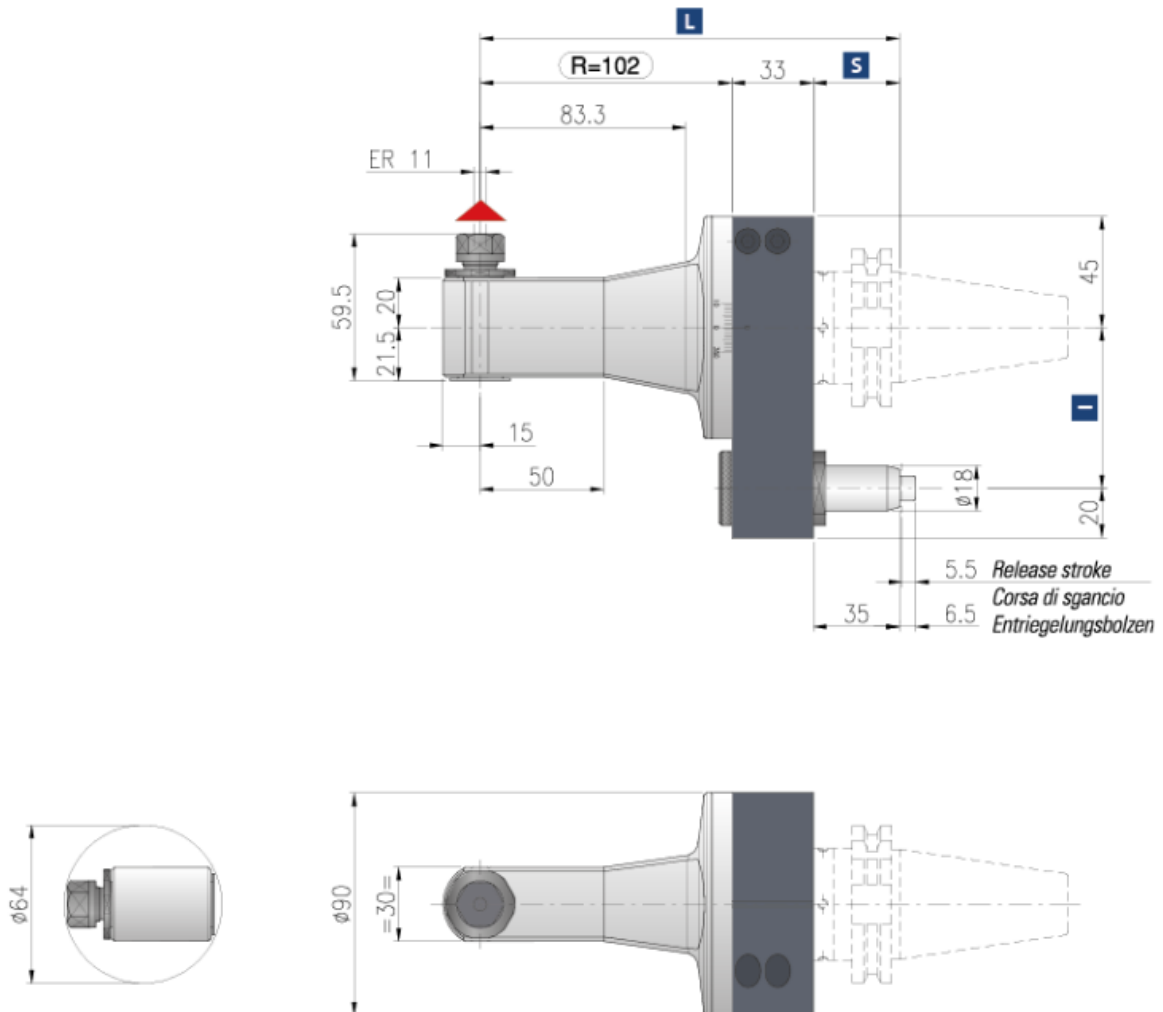## Right Angle Head

By
James
02/15/2021

I.Machine Specifications

- **X -1600, Y-1250, Z-1600**
- **A (-120 ~ +45), B (-360 ~ 360).**
- **A0 → Spindle is aligned with Z axis.**

II. Right-Angle-Head Application.

- The head is an Alberti T90cn-0.5C, CAT 50.
- The distance you are calling R would calculate to be **38** and L is listed as **170**.
- For this application we will keep it simple. **Spindle at A0 and tool pointing up**. We will **be drilling only**. Let me try to get you the DL and R value.
- I have never programmed one so it will be learning experience on this

Solution I: G43 Tool Length Compensation—
■ Along Two or More Axis

Format

Type	Format	Description
Tool length compensation A	G43 Z_ H_ ; G44 Z_ H_ ;	G43 : Positive offset G44 : Negative offset G17 : XY plane selection G18 : ZX plane selection G19 : YZ plane selection α : Address of a specified axis H : Address for specifying the tool length compensation value X, Y, Z : Offset move command
Tool length compensation B	G17 G43 Z_ H_ ; G17 G44 Z_ H_ ; G18 G43 Y_ H_ ; G18 G44 Y_ H_ ; G19 G43 X_ H_ ; G19 G44 X_ H_ ;	
Tool length compensation C	G43 α_H_ ; G44 α_H_ ;	
Tool length compensation cancel	G49 ; or H0 ;	

Explanation
- Selection of tool length compensation

Select tool length compensation A, B, or C, by setting bits 0 (TLC) and 1 (TLB) of parameter No.5001 .

Parameter No.5001		Type
Bit 1 (TLB)	Bit 0 (TLC)	
0	0	Tool length compensation A
1	0	Tool length compensation B
0/1	1	Tool length compensation C

- Performing tool length compensation along two or more axes

Tool length compensation B can be executed along two or more axes when the axes are specified in two or more blocks.

By setting bit 3 (TAL) of parameter No. 5001 to 1, tool length compensation C can also can be executed along two or more axes when the axes are specified in two or more blocks. If no axis is specified in the same block, the alarm PS0027, "NO AXES COMMANDED IN G43/G44" is issued. If two or more axes are specified in the same block, the alarm PS0336, "TOOL COMPENSATION COMMANDED MORE TWO AXES" is issued.

Example 1
When tool length compensation B is executed along the X-axis and Y-axis
G19 G43 H_ ;   Offset in X axis
G18 G43 H_ ;   Offset in Y axis
Example 2
When tool length compensation C is executed along the X-axis and Y-axis
G43 X_ H_ ;   Offset in X axis
G43 Y_ H_ ;   Offset in Y axis

Test Tasks:

(1) Check machine Parameter No.5001, bit 1 and bit 0, as shown in the table above. It should be 10 for Compensation B, and 01 for Compensation C.

(2) Set the tool length offset data in the machine controller. Set L in Z, R in Y.
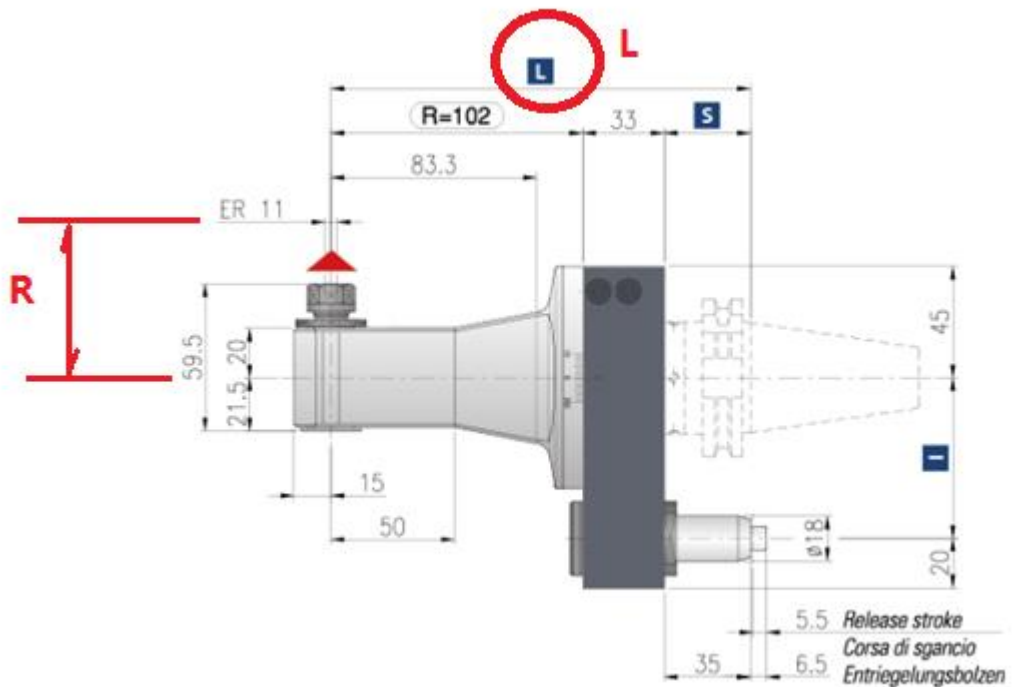
(3) Test the sample part to dry-run the output program with the latest postprocessor atatched.

What to check –

- Check if G43 compensation in Y and Z axis works.
- Check if the Y axis compensation R should be Positive or Negative.
- Check if the Drilling cycle works in G18 plane.

Solution II. Shift the work offset by "G10 L2 Px".

Solution III. Compensate the Y coordinate from postprocessing, linear output only.

For more information

Please contact NCCAS (NX CAD/CAM Automation Services)

support@nxcadcam.com

THANKS!