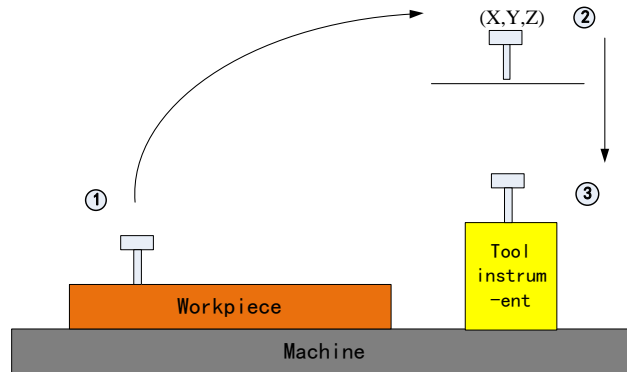# self-made tool auto-checking operating

Tool setting method 1 (parameter No. 68 = 1)

- ◆ **Ready to work**
    - ◆ system is in 【free】 state
    - ◆ The main interface switch to 【main page】
- ◆ **Instructions**

    1. Open the tool setting function, parameter 68 is set to 1, mode 1;

    2. Set the effective level of the knife counterparts, the default normally open, the relevant parameters No. 70;

    3. Set tool presetter mode, parameter No. 71, 1: fixed tool presetting tool 0: floating tool setter；

    4. Set the initial scan position of the tool presetter, as shown in Figure 2 above, the relevant parameters for the 72,73,74 number; (If the 71 parameter is 0, ignore this step);

    5. The first time the knife (measuring knife position);

    6. Move the tool Z to the workpiece surface (Z program zero point) as shown above ① and clear Z workpiece coordinate;

    7. Click the button $[2nd]$ and $[A\text{-}]$ to execute the current tool setter position measurement in sequence. The operation flow is as shown in the above figure, and ① moves ② position (the position set in step 4), and then ③ searches for the tool setting movement until it stops. (Execute probe.nc) **(If parameter 71 is 0, you need to manually move the tool to the tool presetter, and then perform the tool setting operation)**

    8. The current tool alignment tool position measurement is completed;

    9. If you do not change the workpiece zero conditions, the replacement of different length of the tool, you only need to repeat in any position Step 7 action, you can automatically complete the knife on the knife action;

    10. If the Z zero position changes need to repeat 6-9 steps can be;

- ◆ **note**
    - ◆ Step 4 of the knife on the scanner position parameters set for the mechanical coordinate position;
    - ◆ Step 6 Z Workpiece zero calibration and step 7 Initial tooling must be performed with the same tool; the tool can not be re-installed;
    - ◆ If the zero point of the Z-axis workpiece changes, you need to re-measure the knife position measurement;

Tool setting method 2 (parameter No. 68 = 2)

Custom features, temporarily do not use!

The principle of self-made tool auto-checking and macro program analysis

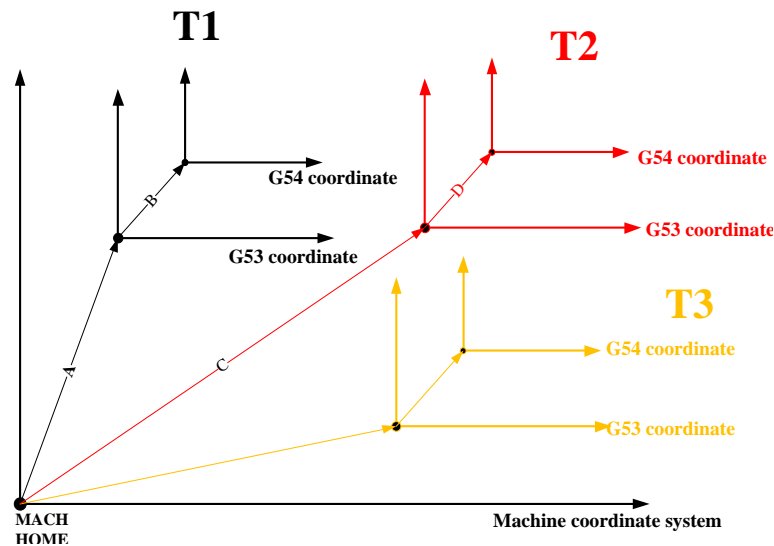## Tool setting method 1 (parameter No. 68 = 1)

Chart Description:

A and C: Zero point offset of G53 coordinate system (relative amount of G53 zero point to mechanical zero point);

B and D: G54 workpiece coordinate system zero offset (G54 workpiece zero to G53 zero relative amount)

T1, T2, T3: indicates the coordinate system under different tool lengths

1: After executing the reset operation, B and D are determined.

2: After the tool length is changed, the alignment of A and C will be caused after executing the tool alignment.

3: A and C changes will cause the G54 coordinate system to change the absolute position of the workpiece;

$$\text{Z Absolute position (G54) = Z Mechanical position (MAH) - A (C) - B (D) ;}$$

Macro (probe.nc) analysis

- Related Macro Address Description:

  #0- #49 for each subroutine internal private variables;

  #50- #499 is the system internal variable area 1;

  #500- #999 is the system parameter area;

  #1000- #1999 is the system internal variable area 2;

  # 400: G53 coordinate system Z zero point offset after execution of the tool presetting

  # 402: same as # 400;

  # 403: Coordinate system enable switch is automatically corrected after tool execution

  # 404: Tool nose point at the G53 Z coordinate of the tool presetter (record the Z position of

the initial tool setting)

#571:71- Initial tool's position；

#572:72--Initial probe position on X axis(MAC POS)；

#573:73--Initial probe position on Y axis(MAC POS)；

#574:74--Initial probe position on Z axis(MAC POS)；

#575:75--back distance after probe；

#578:78- Z axis lifting protection speed

864: X-axis machine position;

865: Y-axis machine position;

866: Z axis mechanical position;

870: Z axis G53 coordinate position

- probe.nc Macro analysis

```
G04P0;              ----- Pause 0s, read the current machine coordinate position correctly
                          for the subsequent program
M5;                 ----- Close the spindle
# 20 = # 864     ----- Read the current X machine coordinate position
# 21 = # 865     ----- Read the current Y machine coordinate position
# 22 = # 866     ----- Read the current Z machine coordinate position
IF # 571EQ0GOTO1;    ----- Floating tool presetter or fixed tool selection
(Fixed tool presetter mode, find the knife position XYZ need to feed)
# 1 = # 572- # 20
# 2 = # 573- # 21
# 3 = # 574- # 22
GOTO2
N1 # 1 = 0
# 2 = 0
# 3 = 0
N2G91G00Z # 3       ----- Move the machine tool to the tool presetter Z position
G91G00X # 1Y # 2    ----- Move the machine to the tool setting XY position
M101                ----- Perform exploration
G91G01Z-100F100     ----- Search distance and speed
M102                ----- End exploration action
G04P0;              ----- pause 0s
# 402 = # 400;      ----- G53 Coordinate system Z Zero offset average
# 403 = 1;          ----- Enable automatic correction of the coordinate system flag
# 404 = - # 870;    ----- Record the Z G53 position of the initial tool setting
G91G01Z # 575F # 578;    ----- The tool setting is completed and the Z axis is retracted
```