1.29 Stock Removal in Turning (G73)

G73 command (stock removal in turning) processes the workpiece to specified shape, leaving a specified value of distance for finishing allowance. This cutting cycle needs to define the block path range of workpiece, the depth of each cut and both the distance and direction of finishing allowance.

1.29.1 Format

G73 U(Δd) R (e) H__
G73 P (ns) Q (nf) U(Δu) W(Δw) F S T

Δd: depth of each cut in X axis direction, it can be specified by the parameter#4013 -when this statement is not applied.

e: escaping amount, it can be specified by the parameter#4012 when this statement is not applied.

ns: sequence number of the first block for the program of stock removal in turning.

nf: sequence number of the last block for the program of stock removal in turning.

Δu: distance and direction of finishing allowance in X direction (diameter/radius designation)

Δw: distance and direction of finishing allowance in Z direction.

F: feedrate

T: number of the tools

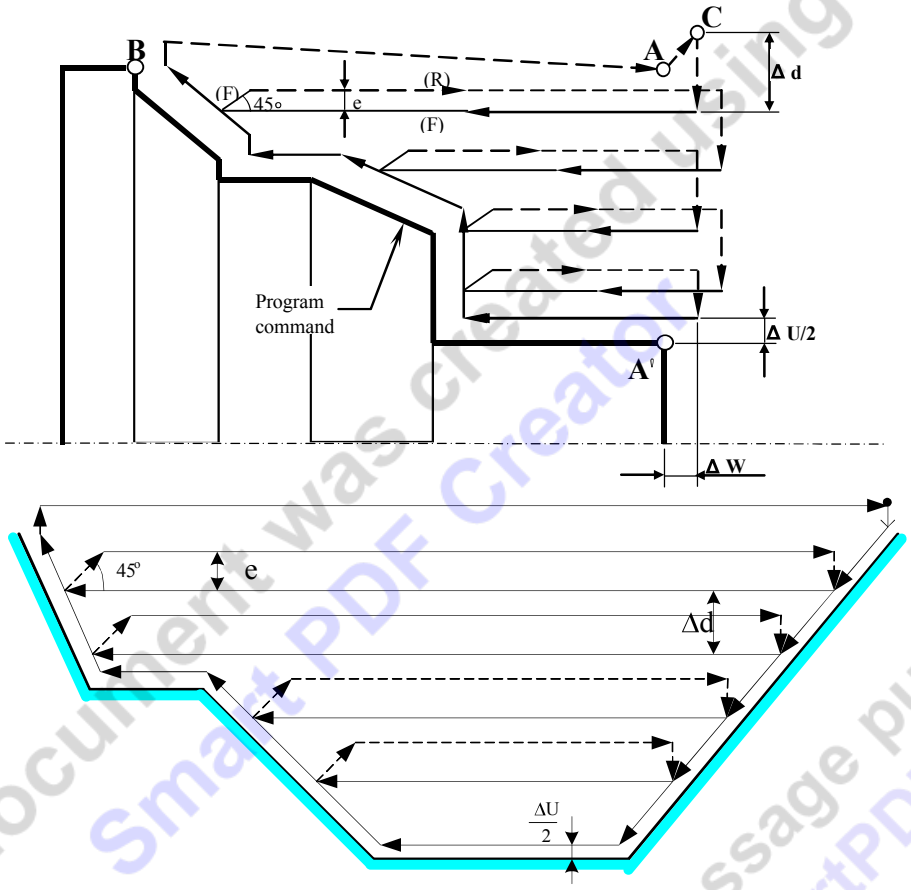
S: spindle rotate speed

H: cutting type. Tpye I set 0. Type II set 1.

If user dosen't set value in H, system will check the type automatically.

1.29.2 PIC

TYPE I: The figure must show monotone increase or decrease along both X and Z axes. Each block must satisfy that the amount of cut is always increase or decrease. Usually start cutting from end face.



1.29.3 Description:

Tool should be positioned to **point A(start point)** before cycle starts.

Tool offsets to point C by specified finishing allowance **(ΔU/2 for X axis, ΔW for Z axis)**.

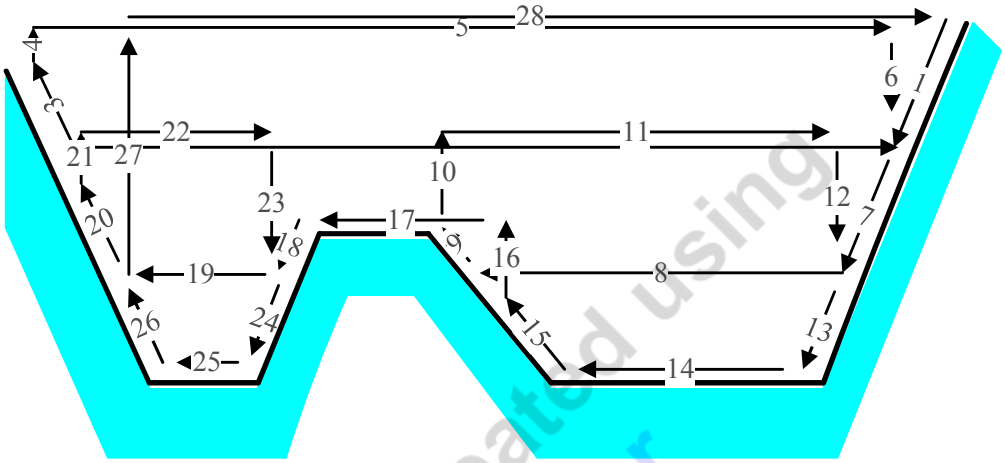
Tool moves Δd amount of distance in X axis direction. Tool begins to move to the endface of contour.

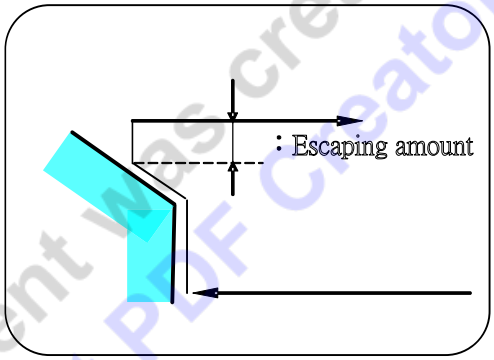
Tool escapes(retracts) **e** amount of distance in X axis direction, but moves by the direction of 45°.Tool then retracts in reversed Z axis feed direction to the point that parallels in X direction to the start point.

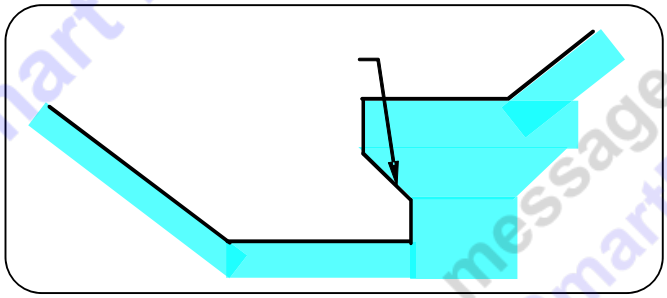
Move Δd amount of distance in X direction, continuing next cycle

1. In last cycle, tool cuts along contour **A'→ B** once
2. After finishing last cycle, tool positions to point A.

TYPE II: Usually be performs in the middle part of the workpiece. The figure need not show monotone increase or decrease in the direction of X axis. Only Z axis needs to satisfy the condition that cutting amount is always increasing or decreasing.







1.29.4 Notice

When **ns and nf** are not specified, specified **U** in **G73** block is depth of cut **Δd**. Otherwise, **U** is finishing allowance in X direction.

Contour path is described by the blocks **ns and nf**, passing through point A → A' → B. If Z coordinate of contour path is not monotone,

System will send out [MAR-002 the profile must be monotone along X, Z axis] alarm. If starting point (defined by the block before G73 command)

is lower than contour path, System will send out [MAR-005 the position higher than the cycle start point] alarm.

F, S or T function issued within block range of **ns→nf** will be ignored. The relevant F,S and T functions specified in G73 block are effective instead.

G00/G01 command in the G73 blocks will be used to perform linear cut to the workpiece.

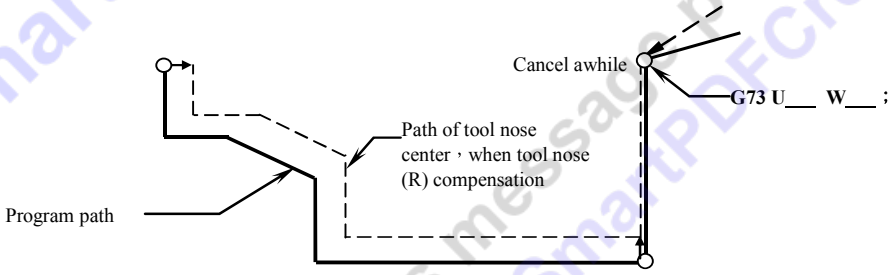
Using G73 command, If H is equal to 0, system will interpret as TYPE I. If H is equal to 1, system will interpret as TYPE II. If H is not specified,

system will diagnose automatically. If H variable is specified wrong, System will send out [MAR-018 ERROR INPUT OF G73/G74 H VALUE] alarm

Sub-program cannot be called during blocks **ns→nf**.

All tool nose compensation commands will be disabled when G73 is in the block. However, the compensation value will be added to the finishing allowance.

When H value is not specified in G73, and the first block contains only movements along X axis, system will take TYPE-I as default.



Direction of finishing allowance: the direction depends as figures shown below, passing through point A→A'→B.