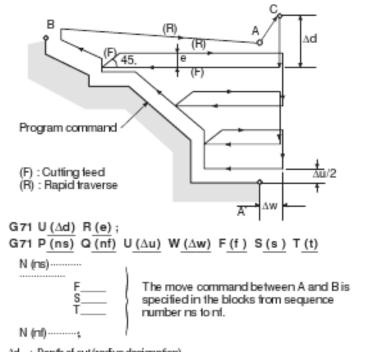
# ROUGH TURNING CYCLE - G71



∆d : Depth of cut (radius designation)

Designate without sign. The outling direction depends on the direction AA'. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5132), and the parameter is changed by the program command.

e : Escapingamount

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5133), and the parameter is changed by the program command.

ns : Sequence number of the first block for the program of finishing shape.

of : Sequence number of the last block for the program of finishing shape.

Distance and direction of finishing allowance in X direction (diameter / radius designation).

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

f.s.t : Any F., S, or T function contained in blocks as to of in the cycle is ignored, and the F. S, or T function in this G71 block is effective.

Note that point C is +X and +Z from stock

### G7 1 ROUGH TURNING CYCLE

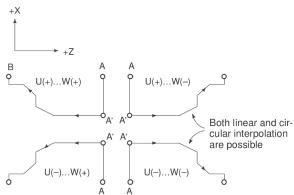
#### NOTE

- 1 While both Δd and Δu, are specified by address U, the meanings of them are determined by the presence of addresses P and Q.
- 2 The cycle machining is performed by G71 command with P and Q specification.

F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G71 block or the previous block are effective.

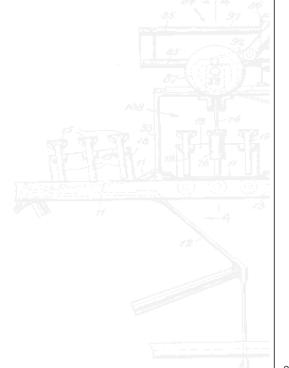
When constant surface speed control is enabled, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G71 block or the previous block is effective.

The following four cutting patterns are considered. All of these cutting cycles are made paralleled to Z axis and the sign of  $\Delta u$  and  $\Delta w$  are as follows:



The tool path between A and A' is specified in the block with sequence number "ns" including G00 or G01, and in this block, a move command in the Z axis cannot be specified. The tool path between A' and B must be steadily increasing or decreasing pattern in both X and Z axis. When the tool path between A and A' is programmed by G00/G01, cutting along AA' is performed in G00/G01 mode respectively.

3 The subprogram cannot be called from the block between sequence number "ns" and "nf".



# G71 ROUGH TURNING CYCLE

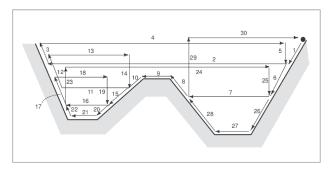


Fig. 13.2.1 (f) Cutting path in stock removal in facing

The offset of the tool tip radius is not added to finishing allowances  $\Delta u$  and  $\Delta w$ . In turning, the offset of the tool tip radius is assumed to be zero.

W=0 must be specified; otherwise, the tool tip may cut into one wall side. For the first block of a repetitive portion, two axes X(U) and Z(W) must be specified. When Z motion is not performed, W0 is also specified.

When only one axis is specified in the first block of a repetitive portion -- Type I

When two axes are specified in the first block of a repetitive portion
-- Type II

When the first block does not include Z motion and type IIis to be used, W0 must be specified.

(Example)

TYPEI TYPEII

G71 V10.0 R5.0; G71 V10.0 R5.0;

G71 P100 Q200....; G71 P100 Q200......;

N100X (U)\_\_; N100X (U)\_\_ Z(W)\_\_;

: : :
: N200.......; N200.......;

# G72 ROUGH FACING

