2.8.6 CIRCULAR PATH MODE ON/OFF ON TOOL RADIUS COMPENSATION C (M97, M96)†

M code	Meaning
M96	Tool radius compensation circular path on.
M97	Tool radius compensation circular path off. (Execution of intersection point)

Note: When power is applied or the control is reset, the control is in the state of M code marked with ◥.

- In the G41 or G42 cutter radius compensation mode, when M96 is given, the tool moves along a circular path around a corner with an angle of 180° or larger. In the M97 mode, the tool does not move along a circular path at the corner, but moves along two intersecting straight lines intersecting at a calculated intersecting point shifted from the programmed contour by the tool radius.

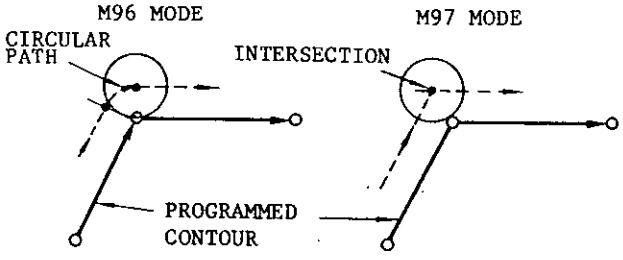

Fig. 2.10

- M96 and M97 are modal. When the power is turned on, M96 takes effect.

- M96 and M97 are effective on the following move command blocks.

```
G01  X··· Y··· F ;        }
                          }  Effective from
(G01) X··· Y··· M96 ;     }  the corner of
                          }  these 2 blocks

G01 X··· Y··· F··· ;      }
M96 (or M97) ;            }  Effective from
(G01) X··· Y··· ;         }  the corner of
                          }  these 2 blocks
```

2.8.7 SUBROUTINE PROGRAM (M98, M99)

With this function, call of subroutine programs which have been numbered and stored in advance is made and executed as many times as desired.

- The following M codes are used for this function.

Table 2.26

M code	Meaning
M98	Call of subroutine program
M99	Subroutine program end

- Call of subroutine program (M98)

M98 P··· L··· ;

With this command, call of the subroutine program with the number specified after P is made and is executed number of times specified after L. When no L code is programmed, the subroutine is executed once.
Subroutine programs can be nested up to 4 times.

- Format of subroutine program (M99)

Subroutine programs are written in the following format, and are stored in the part program memory in advance.

```
O ··· ;                    ← Program No.
····· ;
····· ;
····· ;
M99 ;                      ← Subroutine program end.
```

- Automatic return command from subroutine program

M99 ;

At the end of subroutine programs, M99 is written in a block of its own. When M99 is commanded in the subroutine program which has been called by M98, the execution of the main program is automatically restarted at the block immediately following the M98 block.