Code G172
G172 define the start point of working
G172 X... Y... Z... T1/... S... E... J...
X,Y,Z are the coordinates of the start point
T is the tool code (Only the second number indicate the tool e.g.: 1/1 is the first tool 1/2 is the second tool...)
S is the spindle speed (18=18000 rpm cw -18=18000 rpm ccw)
E is the speed entry of tool
J is the inclination of tool (always 90)

Code G101
G101 define a linear interpolation
G101 X... Y... Z... F...
X,Y,Z are the coordinates of the end point
F is the tool feed rate

Code G102
G102 define a circular cw interpolation
G102 X... Y... Z... I... J... F..
X,Y,Z are the coordinates of the end point
I,J are the coordinates of the center point (I=x ; J=y)
F is the tool feed rate

Code G103
G103 define a circular ccw interpolation
G103 X... Y... Z... I... J... F..
X,Y,Z are the coordinates of the end point
I,J are the coordinates of the center point (I=x ; J=y)
F is the tool feed rate

The circular interpolation works only into x y plane.

Code G40 G41 G42
They are the same than a normal ISO code

E.g.

% 1 WORK  (This is description of program; not important)
| 2 0  (This is "Piano6" always 0 )
| 3 10 (This is tools table)
| 4 3000.0 (This is max plane lenght always 3000)
| 5 160.0 (This is max plane height always 160)
| 6 800.0 (This is max plane width always 800)
| 7 0 0 0 0 (This is always 0 0 0 0 )
| :1 (Number of program)
G172 X100.00 Y100.00 Z10.00 T1/4 S18 E4 J90
G101 X400.00 Y100.00 F3
G102 X400.00 Y200.00 I 400.00 J150.00
G101 X100.00 Y200.00
G101 X 100.00 Y100.00
%