ROMI®

PROGRAMMING AND OPERATION MANUAL ROMI LINE GL170G V1.0 CNC FANUC 0I-TD

T85994A

INDÚSTRIAS ROMI S/A

TRADING DIVISION
Rua Corioliano, 710 Lapa
05047-900 São Paulo - SP- Brazil
Phone + 55 11 3670-0110 Fax: 3865-9510
Website: www.romi.com.br

HEADQUARTERS
Avenida Pérola Byington, 56, Centro 13453-900
Santa Bárbara D'Oeste – SP – Brazil
Phone +55 19 3455-9000 Fax: 3455-2499
Email: aplicação@romi.com

TABLE OF CONTENTS

I - PROGRAMMING

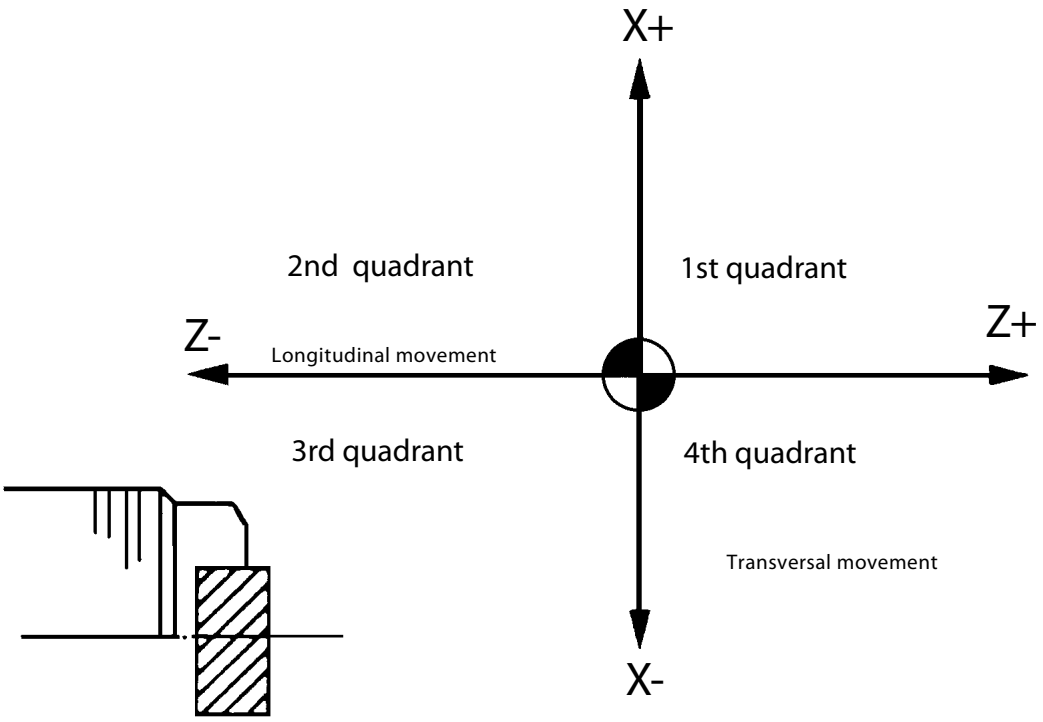
PART II - OPERATION

PART I

PROGRAMMING

1 - COORDINATE SYSTEM

All part geometry is transmitted to command with the aid of a cartesian coordinates system.



The coordinates system is defined in the plane created by the crossing of one line parallel to longitudinal movement (Z), with a line parallel to transversal movement (X).

All movement of tool end is described in this plan XZ, in relation to a pre-defined source (X0, Z0). Please remember X is always diameter measuring.

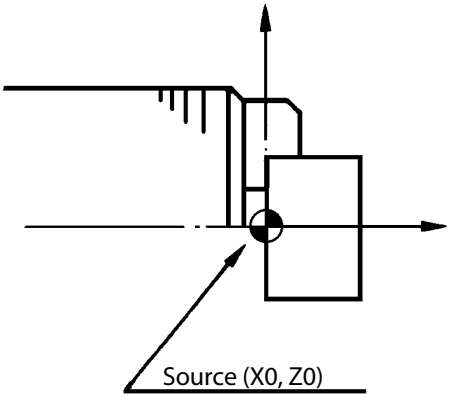
OBSERVATION: *The positive or negative signal introduced in the dimension to be programmed is given by quadrant where the tool is located.*

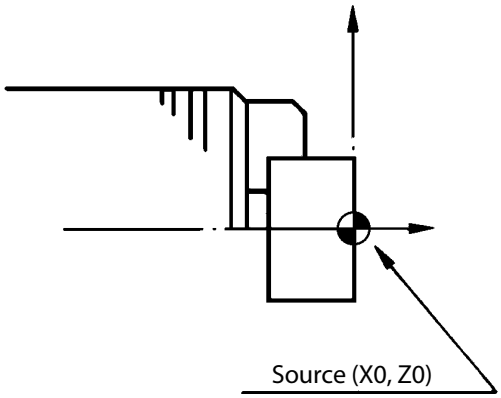
1.1- ABSOLUTE COORDINATE SYSTEM

In this system, the source is established in function of the part to be made, that is, we can establish it at any point of the space to make programming easier. This process is named “Zero-part”.

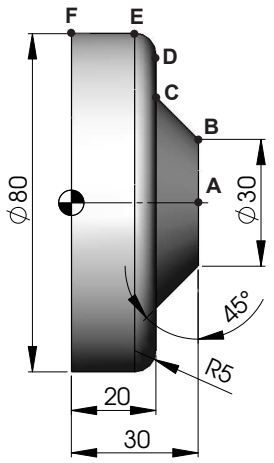
As we saw, the system source was fixed as the points X0, Z0. X0 point is defined by the Center line of the spindle. The Z0 point is defined by any line perpendicular to Center line of the spindle.

During programming, usually the source (X0, Z0) is pre-defined in the part bottom (jaw support) ot at the part face, according illustration below:



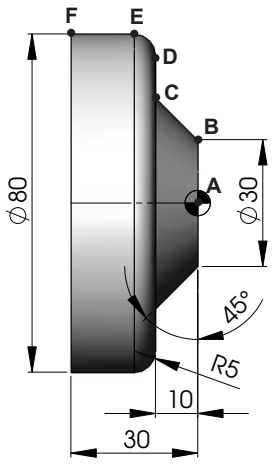


PROGRAMMING EXAMPLE:



SOURCE AT PART BOTTOM:

ABSOLUTE COORDINATE		
POINT	SHAFT	
	X	Z
A	0	30
B	30	30
C	50	20
D	70	20
E	80	15
F	80	0



SOURCE AT PART FACE:

ABSOLUTE COORDINATE		
POINT	SHAFT	
	X	Z
A	0	0
B	30	0
C	50	-10
D	70	-10
E	80	-15
F	80	-30

1.2- INCREMENTAL COORDINATE SYSTEM

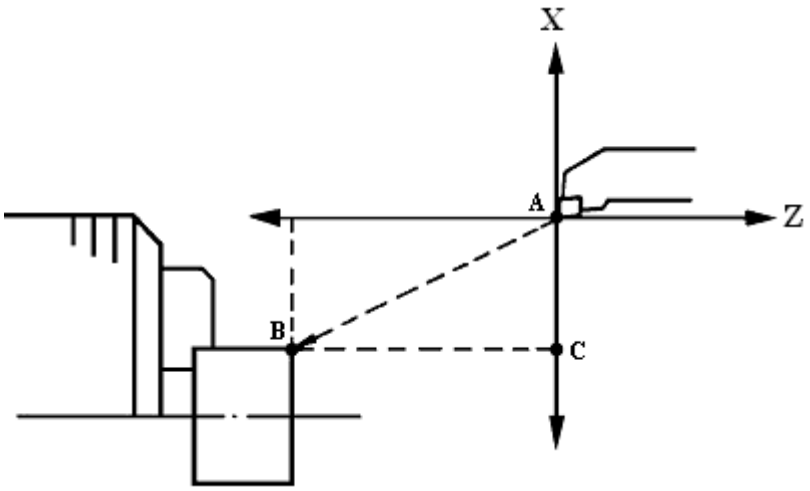
The source of this system is established for each tool movement.

After any displacement, a new source will occur, that is, for any point reached by the tool, the coordinates source will pass to be the reached point.

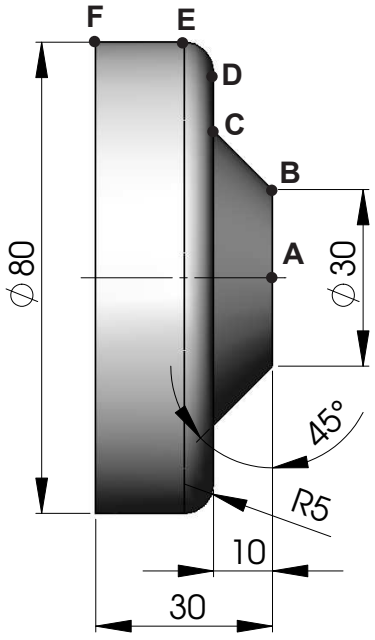
All measurings are done between the distance to be displaced.

If the tool displaces from a point A to B (any two points), the coordinates to be programmed will be distances between two points, measurements (designed) in X and Z.

Please observe the Point A is displacement source for point B, and B will be the source for a displacement up to point C, and so on.



PROGRAMMING EXAMPLE:



MOVEMENT		INCREMENTAL COORDINATES	
START	TARGET	SHAFT	
FROM	TO	X	Z
A	B	30	0
B	C	20	-10
C	D	20	0
D	E	10	-5
E	F	0	-15

2 - TYPES OF FUNCTION

2.1 - POSITIONING FUNCTIONS

FUNCTION X: Position at transversal shaft (absolute)

Format: X +-5.3 (milimeter)

FUNCTION X: Position at longitudinal shaft (absolute)

Format: Z +-5.3 (milimeter)

FUNCTION U: Displacement at transversal shaft (incremental)

Format: U+-5.3 (milimeter)

FUNCTION Z: Displacement at longitudinal shaft (incremental)

Format: Z+-5.3 (milimeter)

2.2 - SPECIAL CODES

2.2.1 - Code: N

Application: To identify blocks.

Each information block can be identified by function “N”, followed up to 4 digits, that the command launches automatically in the program, by keeping an increasing from 10 to 10.

Example:

```
N10  ...;
N20  ...;
N30 ...;
```

OBSERVATION: *In order to able / disable this function, it is necessary:*
- To actuate “MDI” key.
- To actuate “OFFSET SETTING” key.
- To actuate [DEFIN] softkey.
- To position cursor at “NO. SEQUENCIA”
- To type “0” to disable or “1” (one) to able.
- To actute “INPUT” key,

2.2.2 - Code: O

Application: To identify programs

Each program or sub-program at command memory is identified by a single number “O” composed by up to 4 digits, it can float from 0000 to 9999.

OBSERVATION: *All programs in the range 8000 to 9999 are protected, so the user has Access only to Edit programs in the range 0000 to 7999.*

2.2.3 - Code: Bar (/)

Application: To inhibit blocks running.

We use the Function Bar (/) when it is necessary to inhibit blocks running in program, not changing programming.

If the character “/” is typed in front of some blocks, they will be ignored by command, since the operator has been selected the option BLOCK DELETE in command panel.

If the option BLOCK DELETE is not selected, the command will execute normally the blocks, including thos with the character “/”.

2.2.4 - Code: F

Application: to determine infeed speed

The infeed speed is an importante data for machining, and it is obtained considering material, tool and operation to be done.

Usually, for CNC lathes, we define infeed using mm/revolution (function G95), but it is also used as mm/min (function G94).

2.2.5 - Code: T

Application: to select tool

The function T is used to select tool, it tells the machine its zeroing (PRE-SET), insert radius, cutting direction and correctors.

The code “T” shall be followed for at most four digits in its programming, being there are two ways to define the application of these digits, depending on the value inserted in parameter 50021.

a) Parameter 5002.1 = 0

b) Parameter 5002.1 = 1

3 - PREPARATORY FUNCTIONS

Application: This group of functions, also named “G codes”, defines what the machine do, preparing it to execute one type of operation, or to receive a determined information.

The functions can be **MODAL** or **NON-MODAL**.

MODAL: They are functions which, once programmed, stay in command memory, being valid for further blocks, unless if modificed by other function or by the same.

NON-MODAL: They are functions which, always when requested, shall be programmed, that is, they are valid only in the block which contain it.

LISTO F PREPARATORY FUNCTIONS			
G code	Function	Modal	Non--Modal
G00	Positioning (quick infeed)	X	
G01	Linear interpolation (programmed infeed)	X	
G02	Circular interpolation (clockwise direction)	X	
G03	Circular interpolation (counterclockwise direction)	X	
G04	Permanence time (Dwell)		X
G07.1	Cylindrical interpolation	X	
G10	Activates toll life management	X	
G11	Cancells toll life management	X	
G20	Programming in inches (in)	X	
G21	Programming in millimeter (mm)	X	
G12.1	Activates polar coordinates	X	
G13.1	Deactivates polar coordinates	X	
G28	Returns shafts to reference position		X
G33	Interpolation with thread (Step-to-step thread)	X	
G37	Automatic offset of tool wear		X
G40	Cancells radius compensation	X	
G41	Activates radius compensation (left tool)	X	
G42	Activates radius compensation (right tool)	X	
G53	Cancells zero-part coordinates (activates zero-machine)		X
G54	Activates zero-part 1 coordinates system	X	
G55	Activates zero-part 2 coordinates system	X	
G56	Activates zero-part 3 coordinates system	X	
G57	Activates zero-part 4 coordinates system	X	
G58	Activates zero-part 5 coordinates system	X	
G59	Activates zero-part 6 coordinates system	X	
G63	Semiautomatic zeroing (using TOOL EYE)		X
G65	Macro B Call		X

LISTO F PREPARATORY FUNCTIONS			
G code	Function	Modal	Non--Modal
G66	Macro B modal Call	X	
G70	Finishing cycle	X	
G71	Longitudinal roughing cycle		X
G72	Transversal roughing cycle		X
G73	Roughing cycle parallel to profile		X
G74	Longitudinal roughing cycle or axial dirlling		X
G75	Facing or grooving cycle		X
G76	Automatic threading cycle		X
G77	Longitudinal or taper roughing cycle	X	
G78	Semiautomatic threading cycle	X	
G79	Transversal or taper roughing cycle	X	
G80	Cancells drilling cycles	X	
G83	Axial drilling cycle	X	
G84	Threading cycle with axial male	X	
G86	Axial drilling cycle	X	
G87	Radial drilling cycle	X	
G88	Radial threading cycle	X	
G90	Absolute Coordinates System	X	
G91	Incremental Coordinates System	X	
G92	To determine new sourcer or maximum revolution (RPM)	X	
G94	Infeed, millimeter/inches per minute	X	
G95	Infeed, millimeter/inches per revolution	X	
G96	Activates cutting speed (m/min)	X	
G97	Cancells cutting speed (programming in RPM)	X	

4 - INTERPOLATION FUNCTIONS

4.1 - FUNCTION: G00

Application: Quick positioning (approach and return).

The shafts moves to the programmed target using the most infeed speed available in the machine:

Syntaxis:
G0 X__ Z__

Where:

X = Coordinate to be reached (values in diameter)

Z = Coordinate to be reached

The function G0 is Modal, and it cancells functions G1, G2, G3.

OBSERVATION: *At GL line, the quick displacement speed is 24m/min, in "X" and 30m/min in "Z", and it is processed in first at 45° up to one of the programmed targets "X" or "Z", and after this to displace in a single shaft to the wanted final point.*

4.2 - FUNCTION: G01

Application: Linear Interpolation (straigh line with programmed infeed)

With this function, we can obtain straight movements with any angle, calculated by the coordinates and with na infeed (F) pre-determined by programmer.

Syntaxis:
G1 X__ Z__ F__

Where:

X = coordinate to be reached (values in diameter)

Z = coordinate to be reached

F = Work infeed (mm/rev.)

OBSERVATION: *The Function G1 is Modal, and it cancells the functions G0, G2, G3.*

4.3 - FUNCTION: G02 E G03

Application: Circular interpolation (radius)

Both G2 or F3 executes machining operations of pre-defined arcs by a proper and simultaneous movement of the shafts.

Syntaxis:
G2/G3 X__ Z__ R__ (F__)
or
G2/G3 X__ Z__ I__ K__ (F__)

Where:

X (U) = final position of the arc

Z (W) = final position of the arc

I = Incremental distance at “X” between start point of the arc and its center (in radius).

K = incremental distance at “Z” between start point of the arc and its Center

R = Radius value

(F) = Infeed value

OBSERVATION: *At the programming of na arc, we shall observe the following rules:*
• The start point of the arc is the start position of the tool.
•The circular interpolation G02 or G03 (clockwise / counterclockwise) direction is programmed.
•Together the interpolation direction, we program the coordinates of the final point of the arc with X and Y.
•Together the arc direction and final coordinates, the R function (radius value) or so, functions I and K (coordinates fo the Center of the arc) are programmed.

4.3.1 - Function: R

Application: Arc defined by radius.

It is possible to program “circular interpolation” up to 180 degrees by the function R, detailing radius value always with plus signal.

4.3.2 - Function: I and K

Application: Arc defined by polar Center.

The functions I and K defines the position of arc Center, where:

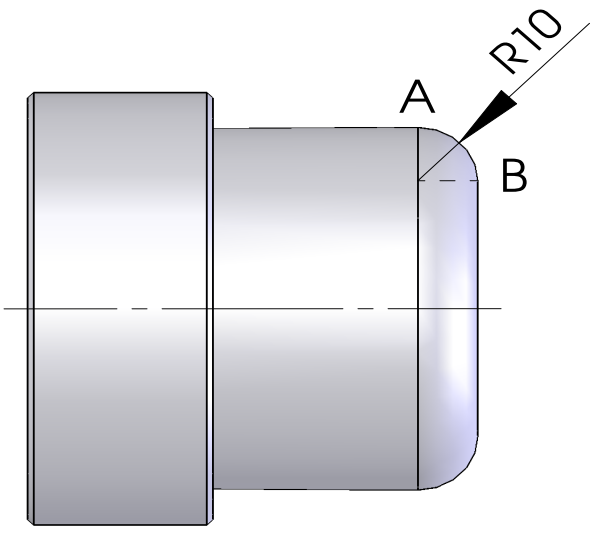
I is parallel to shaft X. K is parallel to shaft Z.

NOTES:

- *I and K functions are programmed by taking as reference the distance from the start point of the tool to arc Center, giving the corresponding signal to the movement.*
- *The function “I” shall be programmed in radius.*

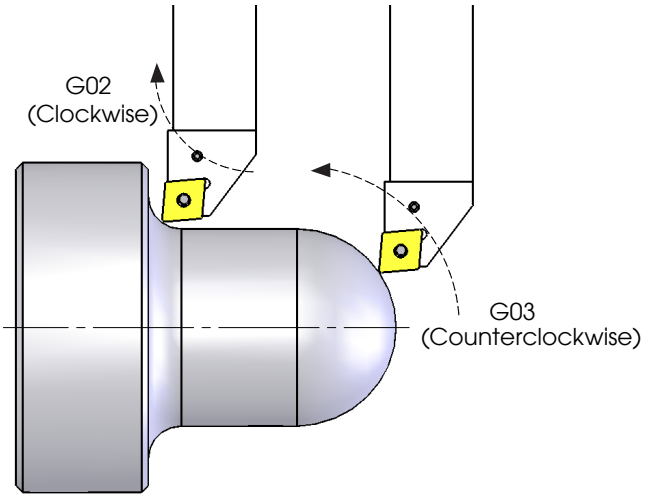
EXAMPLE:

DIRECTION A-B: I-10 K0
DIRECTION B-A: I0 K-10

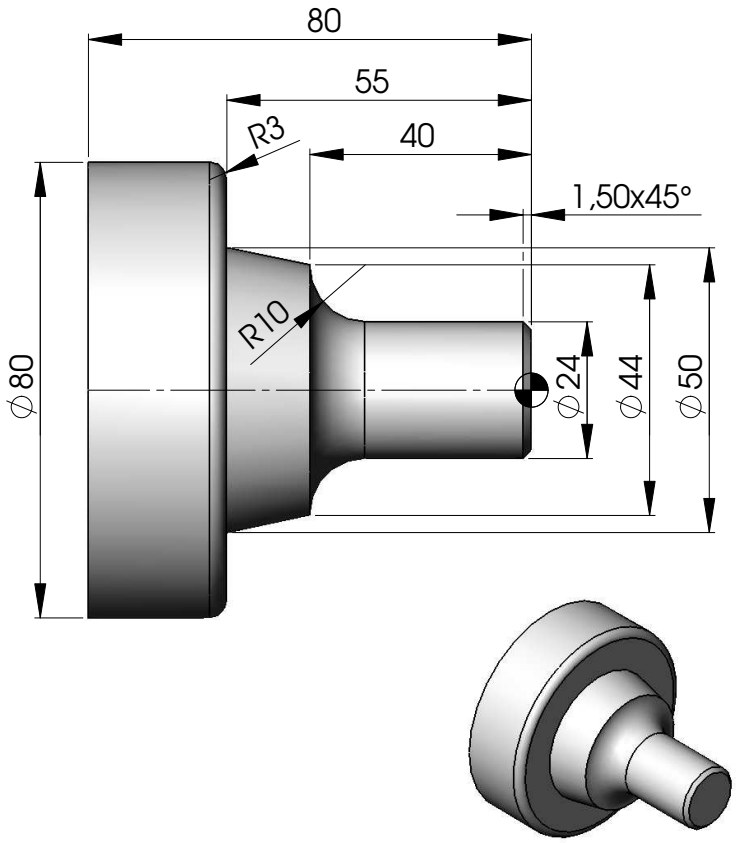


The direction for the arc machining execution defines if it is clockwise or counterclockwise, according charts below:

REAR TURRET (Positive Quadrant)



PROGRAMMING EXAMPLE



:
:
N30 G0 X21 Z2;
N40 G1 Z0 F.25;
N50 X24 Z-1.5;
N60 Z-30;
N70 G2 X44 Z-40 R10;
or
N70 G2 X44 Z-40 I10 K0;
N80 G1 X50 Z-55;
N90 X74;
N100 G3 X80 Z-58 R3;
or
N100 G3 X80 Z-58 I0 K-3;
N110 G1 Z-80;

OBSERVATION: *The functions G2 and G3 are Modal, so they cancel functions G0 and G1.*

4.4 - FUNCTION: “,R” / “,C”

Application: Corner rounding / beveling

Functions “,R” and “,C” are used to round / bevel corners. These functions shall be inserted in the programming block from the point of intersection between two straight lines.

Syntaxis:

G01 X__ Z__, R__

G01 X__ Z__

or

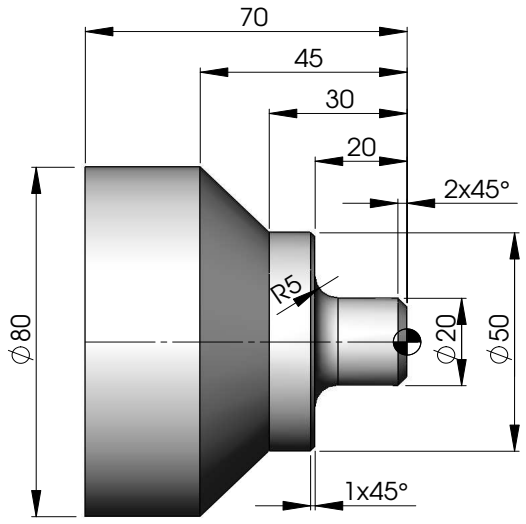
G01 X__ Z__, C__

G01 X__ Z__

Where:

,R = Value or rounding radius
,C = Bevel value

PROGRAMMING EXAMPLE WITH CORNER ROUNDING




```
:
:
:
N60 G00 X14 Z2
N70 G42
N80 G01 Z0 F.2
N90 G01 X20,C2
N100 Z-20,R5
N110 X50,C1
N120 Z-30
N130 X80 Z-45
N140 X84
N150 G40
N160 G0 X90
N170 Z300 T00
:
:
```

4.5 - FUNCTION: G33

Application: Step-by-Step threading

Function G33 executes the threading at shaft X and Z, where each deep is programmed clearly in separated block.

There are possibilities to open screws in internal or external diameters, being they parallel or taper screws, simple or multiple inlets, progressive, etc.

Function G33 demands:

X = Final diameter of threading

Z = Final position of thread length.

Q = angle of spindle for thread inlet (millesimal of degree).

R = Incremental taper angle value in shaft “X” (radius/negative for external and positive for internal)

F = thread pitch

OBSERVATIONS:

• There is no need to repeat the pitch value (F) in further blocks of G33.

•It is recommended to left during approach a minimum clearange around twice the thread pitch in shaft “Z”.

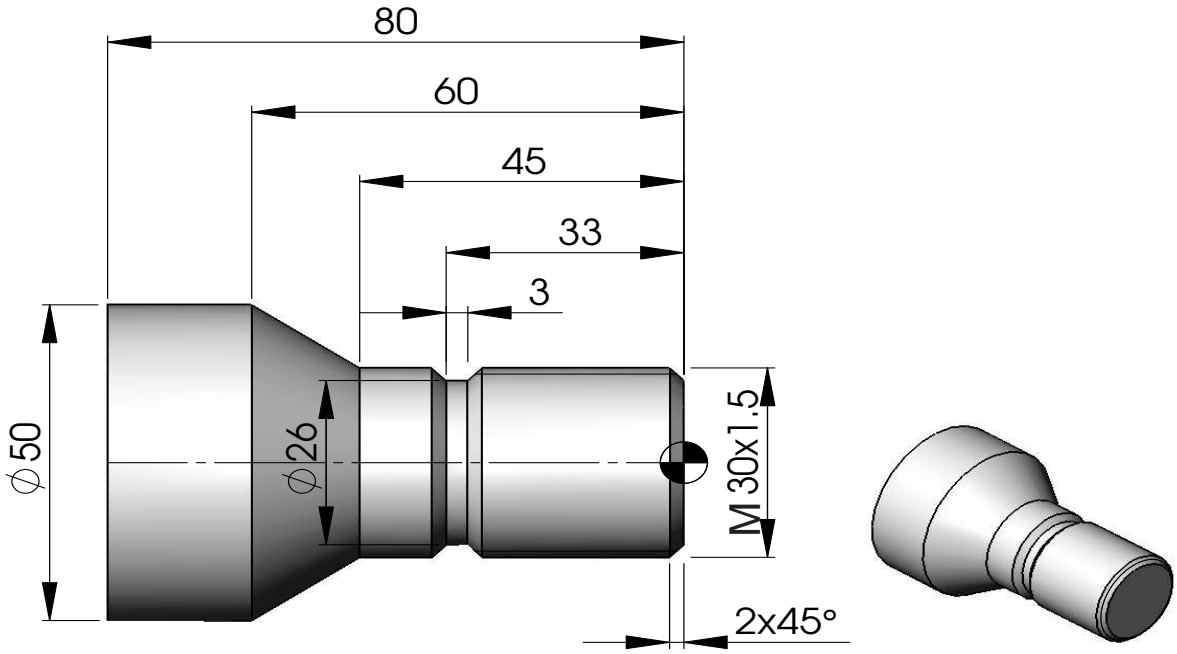
•The function G33 is modal.

In case of step-by-step threading programming, we shall use the function G97, in order to RPM stay fixed.

During the running of threading function, the maximum revolution of spindle shall not overpass the value determined by the following equation:

$$RPMmax = \frac{5000}{PASSO}$$

PROGRAMMING EXAMPLE:



O0330 (THREAD CYCLE);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00
N30 T0101 (ROSCA M30X1.5);
N40 G54;
N50 G97 S1000 M3;
N60 G0 X35;
N70 Z3;
N80 X29.35;
N90 G33 Z-31.5 F1.5;
N100 G0 X35;
N110 Z3;
N120 X28.95;
N130 G33 Z-31.5;
N140 G0 X35;
N150 Z3;
N160 X28.55;
N170 G33 Z-31.5;
N180 G0 X35;
N190 Z3;
N200 X28.15;
N210 G33 Z-31.5;
N220 G0 X35;
N230 Z3;
N240 X28.05;
N250 G33 Z-31.5;
N260 G0 X35;
N270 G54 G0 Z300 T00;
N280 M30;

CALCULATIONS:

1º) Thread height (P):
P = (0.65 x pitch)
P = (0.65 x 1.5)
P = 0.975

2º) Final diameter (X):
X = Start diameter - (P x 2)
X = 30 - (0.975 x 2)
X = 28.05

5 - DWELL TIME (DWELL)

5.1 - FUNCTION: G04

Application: Dwell time

Between one displacement and other of the tool, we can program a determined dwell time for it. The function G4 executes a dwell, which duration is defined by a related value “P”, “U” or “X” , which defines the time in seconds.

The function G04 demands:

G04 X__; (seconds)
or
G04 U__; (seconds)
or
G04 P__; (millesimal seconds)

EXAMPLE: (TIME: 1.5 SECOND)

```
G04 X1.5;
G04 U1.5;
G04 P1500;
```

6 - TOOL RADIUS OFFSET

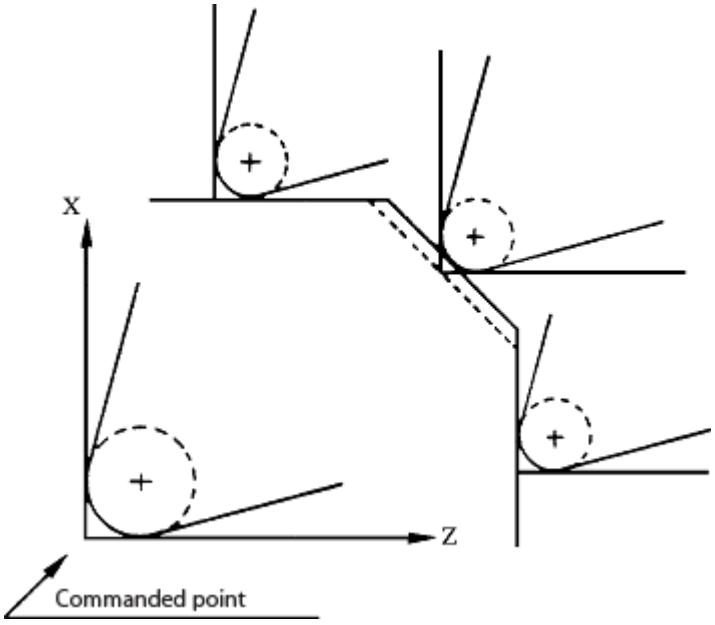
6.1 - FUNCTION: G40

Application: It cancels radius offset

The function G40 shall be programmed to cancel functions previously requested, as G41 and G42. This function, when requested, can use the further block to de-offset insert radius programmed in the page “GEOMETRIA DE FERRAMENTAS” (Tools geometry), using work advance (G1).

Function G40 is a Modal code, and it is active when the command is turned on.

The commanded point for work is found in the vertex between shaft X and Z.



6.2 - FUNCTION: G41

Application: It activaes radius offset (left)

Function G41 selects the offset value of tool end radius, being at left of the part to be machined, view in relation to cut stroke direction.

Tool end geomety and the manner it was informed are defined by “T” code”, Page “Tools Geometry”.

Function G41 is Modal, so it cancels G40.

6.3 - FUNCTION: G42

Application: It activates radius offset (right)

This function forces na offset similar to Function G41, except offset direction is rigth, view in relation to cut stroke direction;

Function G42 is Modal, so cancels G40.

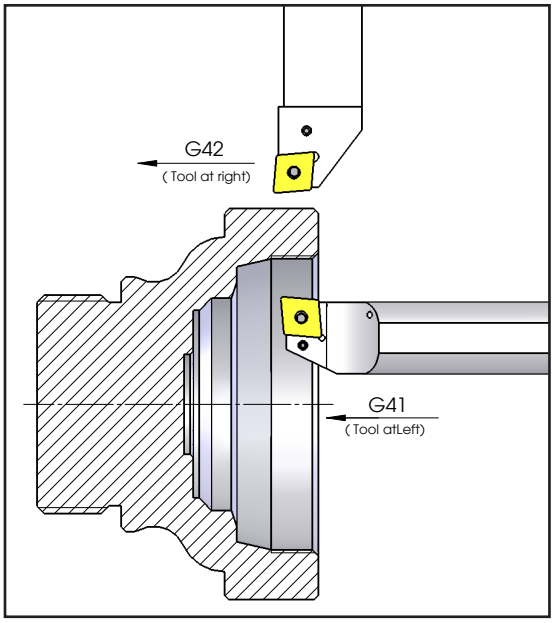
Function “T” shall be used in the “**GEOMETRY**” page, by giving the cutting side of the tool.

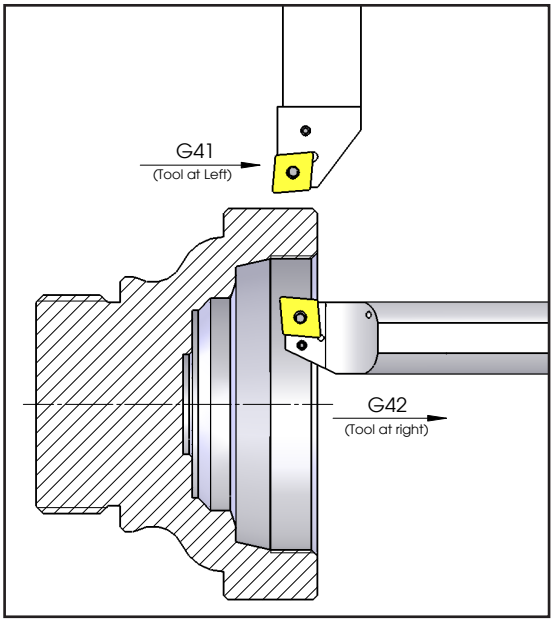
OBSERVATIONS:

• During the radius compensation, the programmed displacements shall be always higher than the insert radius (pad).

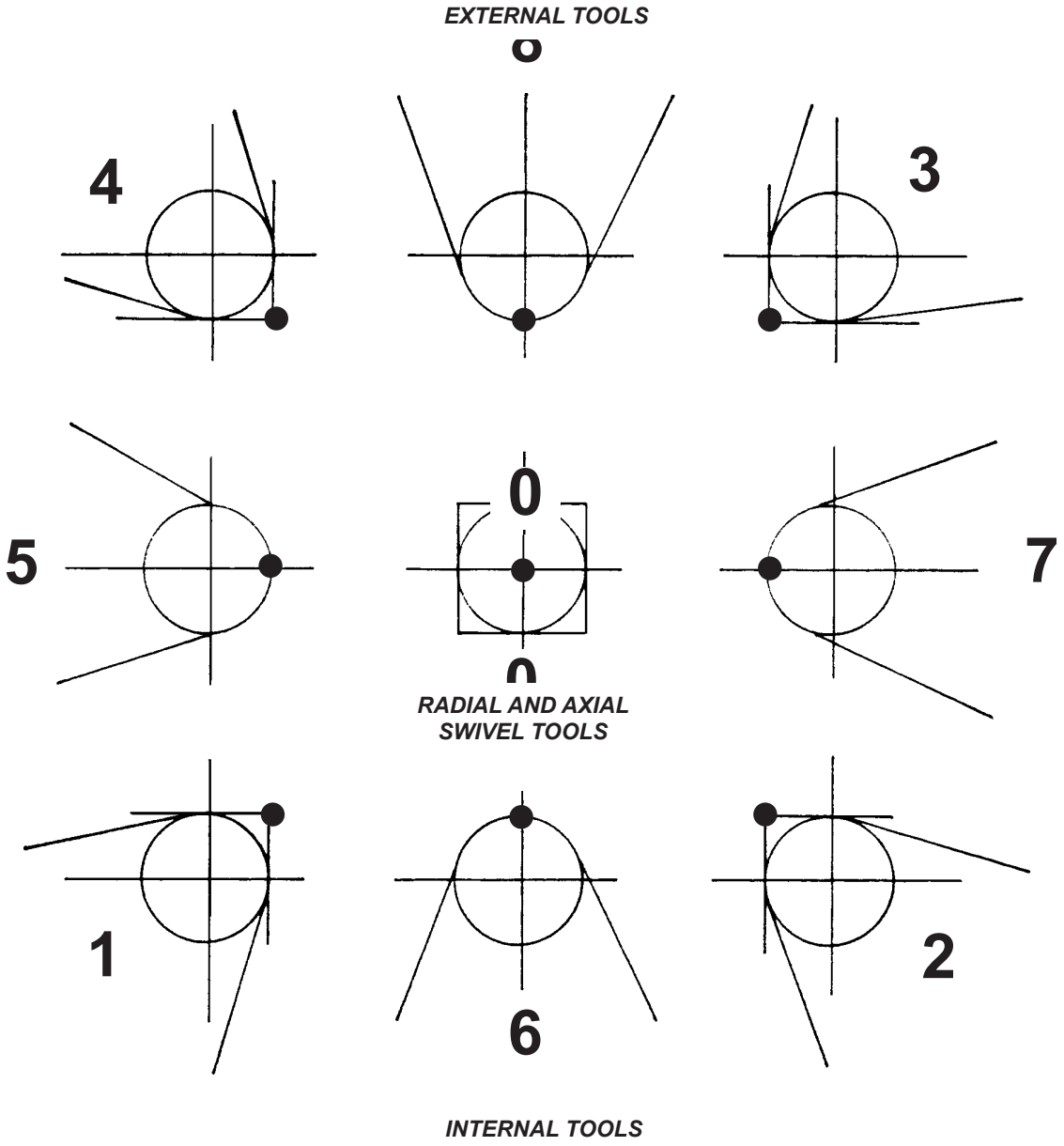
• The tool shall not be in contact with the material to be machined, when offset functiosn were activated in program.

CODES FOR TOOL RADIUS OFFSET: POSITIVE QUADRANT



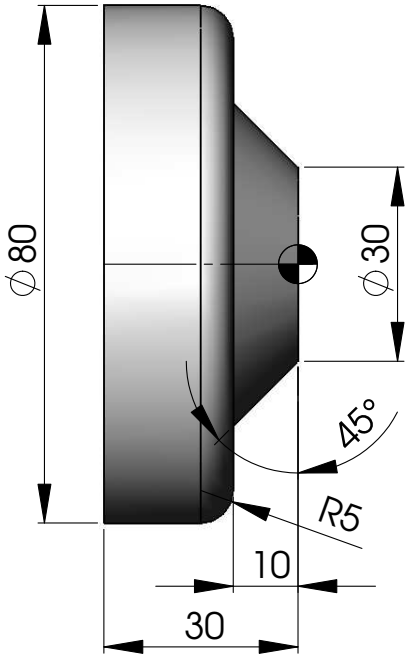


6.4 - TOOL QUADRANTS FOR RADIUS OFFSET



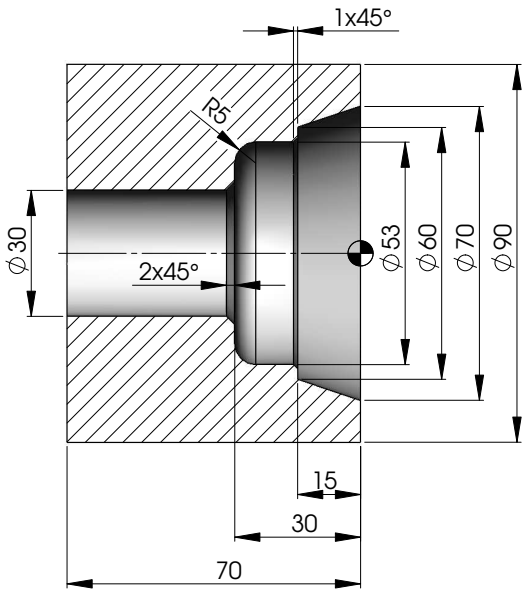
6.5 - EXAMPLES OF PROGRAM WITH RADIUS OFFSET:

Example 1: External machining




```
:
:
:
N60 G00 X34 Z0
N70 G01 X-2 F.2
N80 G00 X27 Z2
N90 G42
N100 G01 X27 Z0 F.2
N110 X30
N120 X50 Z-10
N130 X70
N140 G03 X80 Z-15 R5
N150 G01 X80 Z-17
N160 X84
N170 G40
N180 G00 X90
N190 G54 G0 Z300 T00
:
:
```

Example 2: Internal machining




```
:
:
N60 G01 X74 Z2
N70 G41
N80 G01 Z0 F.2
N90 X70
N100 X60 Z-15
N110 X53, C1
N120 Z-30, R5
N130 X30, C2
N140 Z-72
N150 X27
N160 G40
N170 G00 X25
N180 G54 G00 Z300 T00
:
:
```

7 - SIMPLE CYCLES

7.1 - FUNCTION: G78

Application: Semiautomatic threading cycle

The function G78 demands:

G78 X__ Z__ (R__) (Q____) F__; Where:

X = Threading diameter

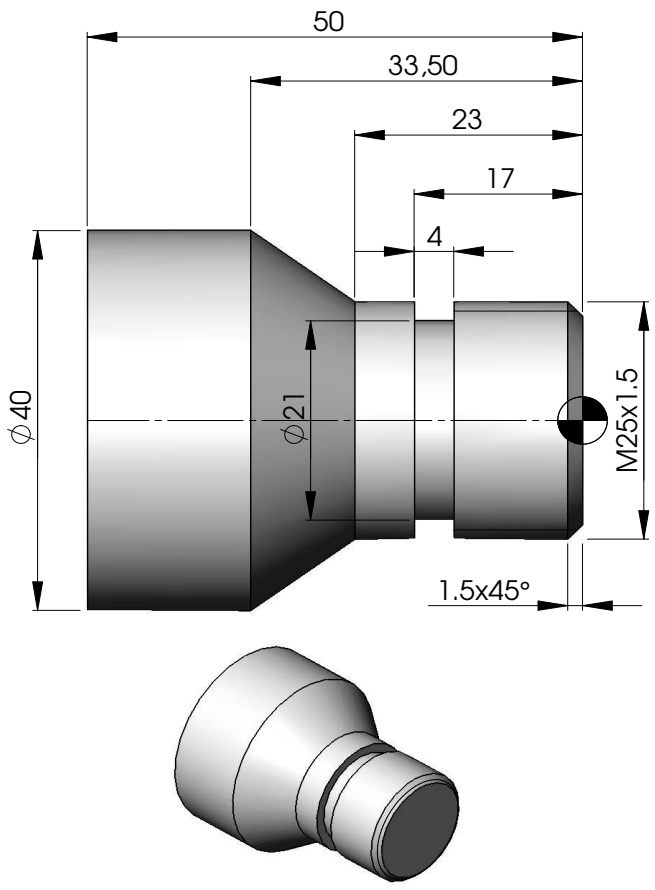
Z = Final threading position

R = Incremental tapering value at shaft “X” (taper thread)

Q = Angle of spindle for thread inlet (millesime degree)

F = Thread pitch

Example 1: Thread M25x1.5



O1000 (THREAD CYCLE);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T0303 (ROSCA M25X1.5);
N40 G54;
N50 G97 S1500 M3;
N60 G0 X30;
N70 Z3;
N80 G78 X24.2 Z-15 F1.5;
N90 X23.6;
N100 X23.2;
N110 X23.05;
N120 G0 X35 ;
N130 G54 G0 Z300 T00;
N140 M30;

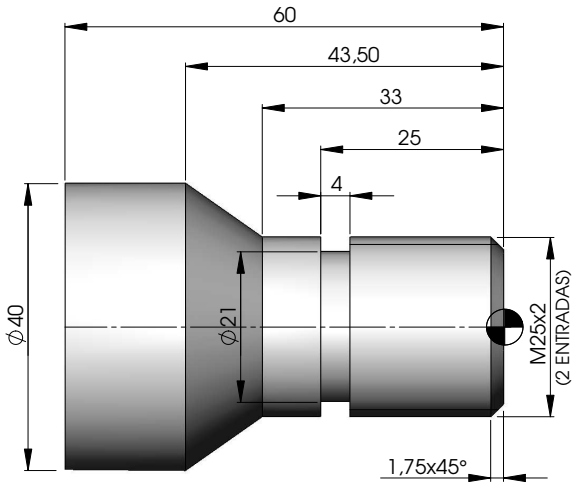
DEPTHS IN EXAMPLE:
1ST PASS = 0.8MM
2ND PASS = 0.6MM
3RD PASS = 0.4MM
4TH PASS = 0.15MM

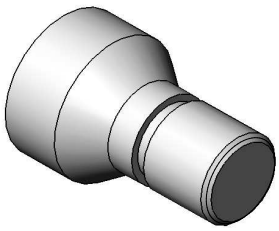
Calculations

1st) Thread height (P):
P = (0.65 x pitch)
P = (0.65 x 1.5)
P = 0.975

2nd) Final diameter (X):
X = Start diameter - (P x 2)
X = 25 - (0.975 x 2)
X = 23.05

Example 2: Thread: M25x2 (2 inlets)





```
O1000 (THREAD CYCLE);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T0303 (THREAD M25X2);
N40 G54;
N50 G97 S1500 M3;
N60 G0 X28;
N70 Z8;
N80 G78 X24 Z-23 Q0 F4; (1ª ENTRADA)
N90 X23.2 Q0;
N100 X22.6 Q0;
N110 X22.4 Q0;
N120 G78 X24 Z-23 Q180000 F4 (2ª ENTRADA)
N130 X23.2 Q180000;
N140 X22.6 Q180000;
N150 X22.4 Q180000;
N160 G0 X28
N170 G54 G0 Z300 T00;
N180 M30;
```

DEPTHS IN EXAMPLE:
1ST PASS = 1.0MM
2ND PASS = 0.8MM
3RD PASS = 0.6MM
4TH PASS = 0.2MM

CALCULATIONS:

1st) Thread height (P)
P = (0.65 x pitch) P = (0.65 x 2)
P = 1.3

2nd) Final diameter (X):
X = Start diameter - (P x 2)
X = 25 - (1.3 x 2)
X = 22.4

3rd) Programmed pitch:
F = Nominal pitch x no of inlets
F = 2 x 2
F = 4

OBSERVATION: *In the case of threading cycle programming, the function G97 shall be used, in order to RPM keeps fixed. During the performing of threading function, the maximum revolution of spindle shall not overpass the value determined by the following equation:*

$$RPMmax = \frac{5000}{PASSO}$$

8 - CYCLES OF MULTIPLE REPETITIONS

8.1 - FUNCTION: G70

Application: Finishing cycle.

This cycle is used after the application of roughing cycles G71, G72 and G73, in order to give final finishing of the part, whithout the programmer needs to repeat the whole sequence of the profile to be done.

The function G70 demands:

G70 P__ Q__; Where:

P = block number which defines profile start

Q = block number which defines profile end

The functions F, S and T, specified in blocks G71, G72 and G73 has no effect, but those specified between the block of profile start (P) and profile end (Q) are valid during the usage of code G70.

OBSERVATION: *After the running of the cycle G70, the tool automatically returns to the point used for positioning.*

8.2 - FUNCTION: G71

Application: Automatic cycle of longitudinal roughing

The function G71 shall be programmed in two subsequent blocks, due to the value related to cutting depth and metal excess for finishing at transversal and longitudinal shafts are informed by functions “U” and “W”, respectively.

The function G71, in 1st block, demands:

G71 U__ R__; Where:

U = Value of cutting depth during cycle (radius)

R = Value of spacing at transversal shaft for return to start Z (radius)

Function G71, in 2nd block, demands:

G71 P__ Q__ U__ W__ F__ ; Where:

P = Block number, which defines profile start.

Q = block number, which defines profile end

U = Metal excess for finishing at shaft “Z” (positive for metal excess at right and negative for machining at left).

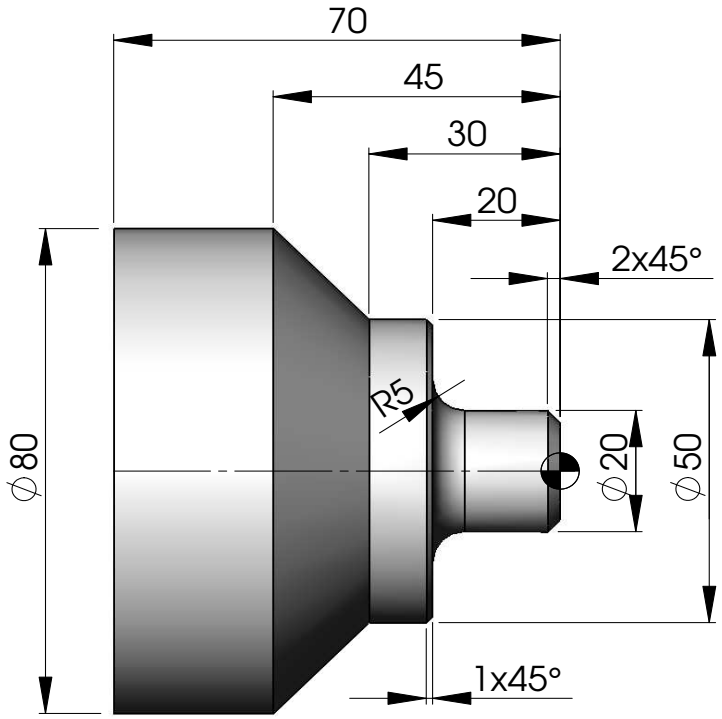
W = Metal excess for finishing at shaft “Z” (positive for metal excess at right of profile or negative for metal excess at left of profile.

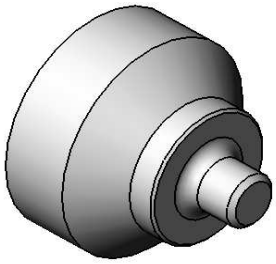
F = Work infeed

OBSERVATIONS: *After the cycle running, the tool automatically returns to positioned point.*

The coordinate programmed in “X” before the cycle start isthat the machine understands as being the raw material, that is, the machine uses the positioning diameter to start machining increasing.

During profile programming, it is not allowed the programming of commands fot tool end radius offset (G40, G41, G42).

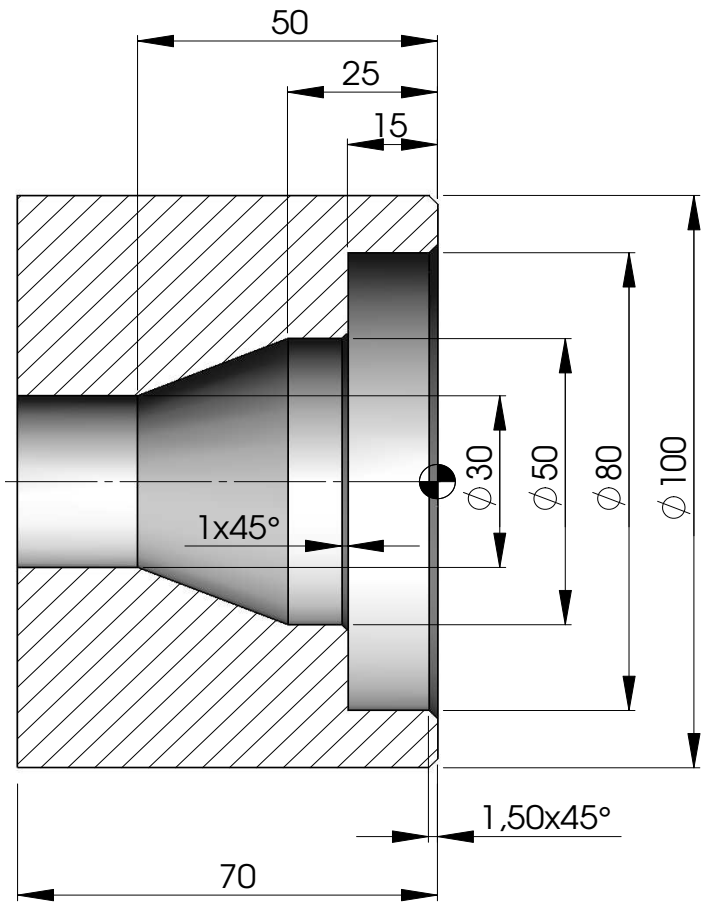


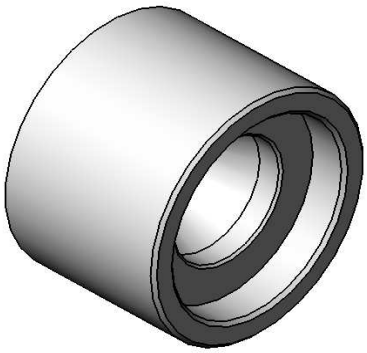


```
O0001 (LONGIT. ROUGHING);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T0101 (EXT.ROUGHING);
N40 G54;
N50 G96 S200;
N60 G92 S2500 M4;
N70 G0 X80;
N80 Z2;
N90 G71 U2.5 R2;
N100 G71 P110 Q190 U1 W.3 F.25;
N110 G0 X16;
N120 G1 Z0 F.2;
N130 X20 Z-2;
N140 Z-15;
N150 G2 X30 Z-20 R5;
N160 G1 X48;
N170 X50 Z-21;
N180 Z-30;
N190 X80 Z-45;
N200 G0 Z100;
N210 T0303 (ACAB. EXT.);
N220 G54;
N230 G96 S200;
N240 G92 S2500 M4;
N250 G0 X80
N260 Z2;
N270 G42
N280 G70 P110 Q190;
N290 G40
N300 G54 G0 Z300 T00;
N310 M30;
```

Cutting depth = 2.5 mm
Infeed = 0,25 mm/rot

OBSERVATION: *In example, it was considered roughing and finishing would be done by different tools.*





```
O0001 (LONGITUDINAL
ROUGHING);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00
N30 T1010 (INT. ROUGHING);
N40 G54;
N50 G96 S200;
N60 G92 S2500 M4;
N70 G0 X25;
N80 Z2;
N90 G71 U3 R2;
N100 G71 P110 Q180 U-1. W.3 F.3;
N110 G0 X83;
N120 G1 Z0 F.2;
N130 X80 Z-1.5;
N140 Z-15;
N150 X50 ,C1;
N160 Z-25;
N170 X30 Z-50;
N180 Z-71;
N190 G41
N200 G70 P110 Q180;
N210 G40
N220 G54 G0 Z300 T00;
N230 M30;
```

Cutting depth = 3 mm
Infeed = 0,3 mm/rot

OBSERVATION: *In example, it was considered roughing and finishing would be done by different tools.*

8.3 - FUNCTION: G72

Application: Automatic cycle of transversal roughing

Function G72 shall be programmed in two subseqent blocks, due to the value reçated to cutting depth and metal excess for finishing at longitudinal shaft are informed by function “W”.

Function G72 in 1st) block demands:

G72 W__ R__; Where:

W = cutting depth during the cycle

R = value of spacing at longitudinal shaft for return to start “X”

Function G72 in 2nd) block demands:

G72 P__ Q__ U__ W__ F__ ; Where:

P = Number of block which defines profile start

Q = Number of block which defines profile end

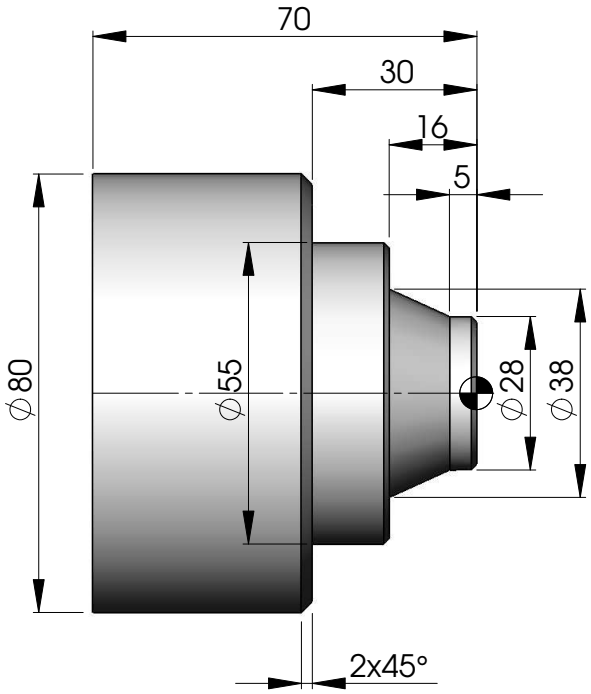
U = Metal excess for finishing at shaft “X” (positive for external or negative for internal/ diameter)

W = Metal excess for finishing at shaft “Z” (positive for metal excess at right of profile or negative for metal excess at left of profile.

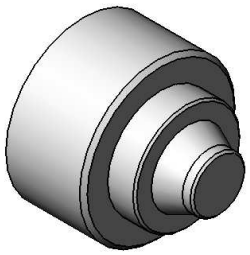
F = Work infeed

NOTA: *After the cycle running, the tool automatically returns to positioned point.*

IMPORTANT*: THE PROGRAMMING OF PART FINISHING PROFILE SHALL BE DEFINED FOR LEFT TO RIGHT.*



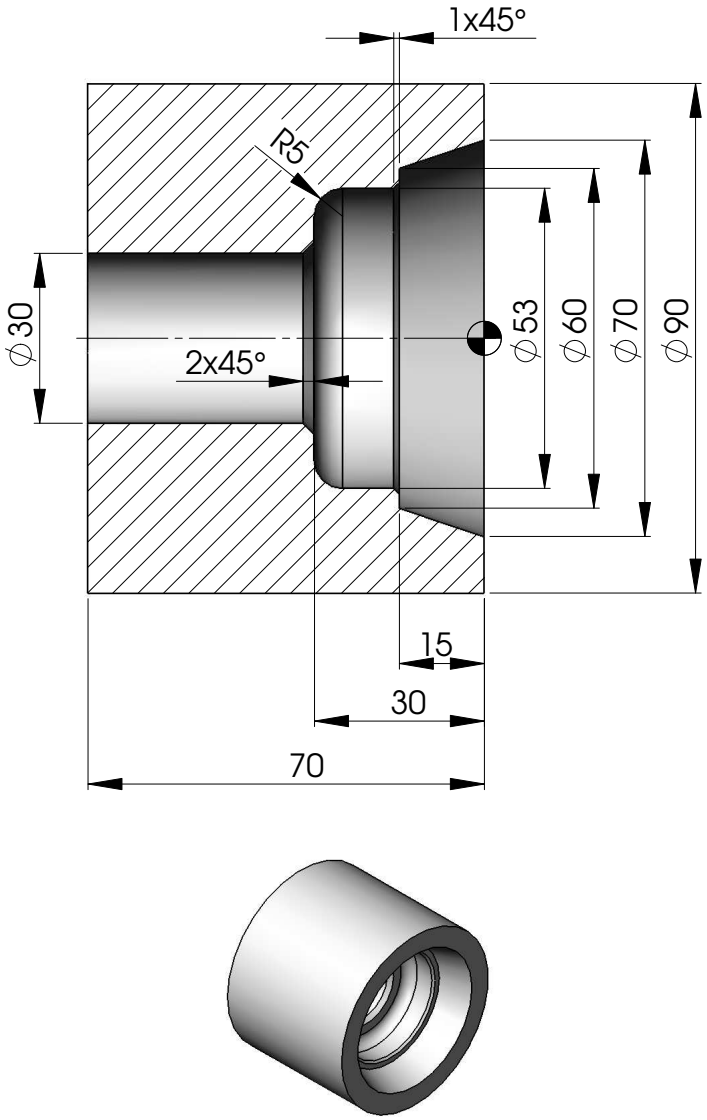
Chanfrar cantos não indicados com 1x45°



```
O0072 (TRANSV. ROUGHING CYCLE);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T1010 (EXTERNAL ROUGHING);
N40 G54;
N50 G96 S200;
N60 G92 S3500 M4;
N70 G0 X84;
N80 Z1;
N90 G72 W2 R1.;
N100 G72 P110 Q190 U1 W.3 F.25;
N110 G0 Z-32;
N120 G1 X80 F.18;
N130 X76 Z-30;
N140 X55;
N150 Z-16 ,C1;
N160 X38;
N170 X28 Z-5;
N180 Z-1;
N190 X26 Z0
N200 G41;
N210 G70 P100 Q180;
N220 G40
N230 G54 G0 Z300 T00;
N240 M30;
```

Cutting depth = 2 mm
Infeed = 0,25 mm/rot

OBSERVATION: *In the example, roughing and finishing were considered as done with same tool.*




```
O0072 (TRANSVERSAL
ROUGHING );
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00
N30 T1111 (INT. ROUGHING);
N40 G54;
N50 G96 S240;
N60 G92 S4500 M4;
N70 G0 X28;
N80 Z1;
N90 G72 W2.5 R1.5;
N100 G72 P110 Q170 U-1 W.3 F.3;
N110 G0 Z-32;
N120 G1 X30 F.2;
N130 X34 Z-30;
N140 X53 ,R5;
N150 Z-15 ,C1;
N160 X60;
N170 X70 Z0;
N180 G42;
N190 G70 P110 Q170;
N200 G40
N210 G54 G0 Z300 T00;
N220 M30;
```

Cutting depth = 2,5 mm
Infeed = 0,3 mm/rot

OBSERVATION: *In the example, roughing and finishing were considered as done with same tool.*

8.4 - FUNCTION: G73

Application: Automatic cycle of roughing parallel to final profile.

Cycle G73 allows the complete rough machining of a part, using only two programming blocks.

Function G73 is specific for cast and forged materials, because the tool always follows a stroke parallel to defined profile.

Function G73 demands:

G73 U__ W__ R__; Where:

U = direction and amount of material to be removed at shaft “X” per pass (radius).

W = Directiona and amount of material to be removed at shaft “Z” per pass.

R = Number of passes per roughing.

Formular for “U” and “W” values calcularion:

$$U = \frac{\text{Material excess at “X” (radius) - Metal excess for finishing at “X” (radius)}}{\text{Number of passes (R)}}$$

$$W = \frac{\text{Material excess at “Z” - Metal excess for finishing at “Z”}}{\text{Number of passes (R)}}$$

G73 P__ Q__ U__ W__ F__; Where:

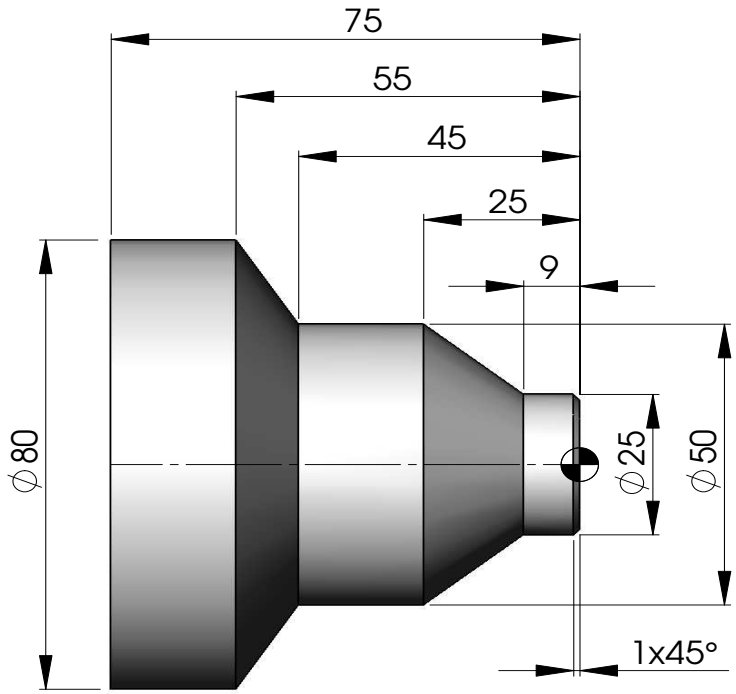
P = number of block which defines profile start

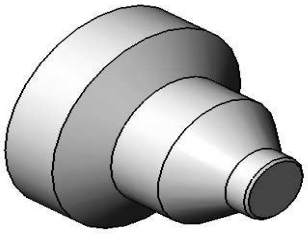
Q = number of block which defines profile end

U = metal excess for finishing at shaft “X” (positive for external and negative for internal / diameter)

W = metal excess for finishing at shaft “Z” (positive for metal excess at right of profile or negative for metal excess at left of profile)

F = Work infeed

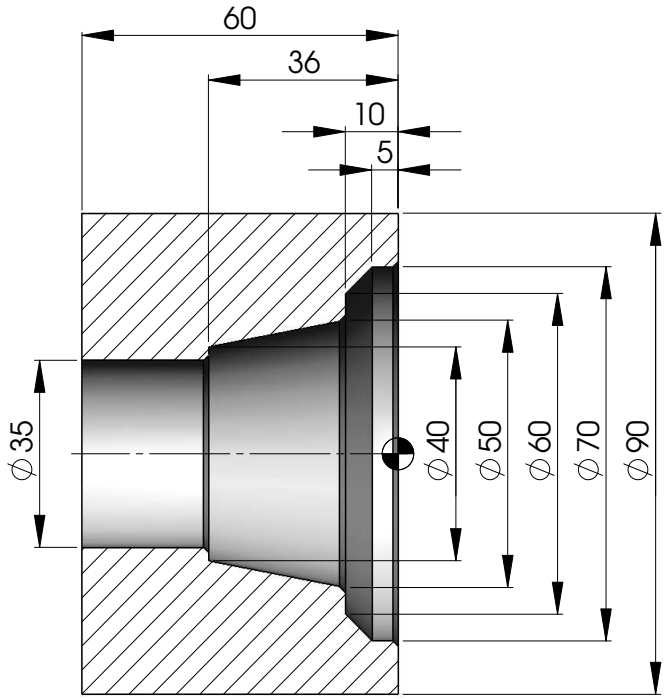




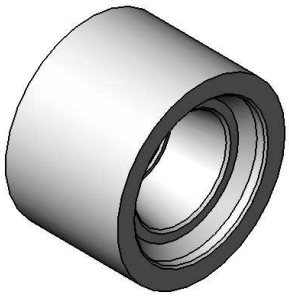
```
O0100 (PARALLELO ROUGHING);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T1111 (EXTERNAL ROUGHING);
N40 G54;
N50 G96 S240;
N60 G92 S4500 M4;
N70 G0 X90;
N80 Z5;
N90 G73 U2 W1.35 R2;
N100 G73 P110 Q170 U2 W.3 F.2;
N110 G0 X23 Z2;
N120 G1 Z0 F.18;
N130 X25 Z-1;
N140 Z-9;
N150 X50 Z-25;
N160 Z-45;
N170 X80 Z-55;
N180 G42;
N190 G70 P110 Q170;
N200 G40
N210 G54 G0 Z300 T00;
N220 M30;
```

In the exemple, we considered:
Roughing with 2 passes
Material excess “X” = 10mm (Ø)
Material excess “Z” = 3mm
Material excess for finishing “X” = 2mm (Ø)
Material excess for “Z” = 0.3mm
Infeed = 0.2mm/rev.

OBSERVATION: *In the example, we considered roughing and finishing as done by using the same tool.*



Chanfrar cantos com 1x45°



```
O0100 (PARALLELO ROUGHING);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T0606 (INT. ROUGHING);
N40 G54;
N50 G96 S215;
N60 G92 S5500 M4;
N70 G0 X27;
N80 Z6;
N90 G73 U-1 W1.2 R3;
N100 G73 P110 Q190 U-2 W.4 F.3;
N110 G0 X72 Z2;
N120 G1 Z0 F.2;
N130 X70 Z-1;
N140 Z-5;
N150 X60 Z-10;
N160 X50 ,C1;
N170 X40 Z-36;
N180 X35 ,C1;
N190 Z-61;
N200 G42;
N210 G70 P110 Q190;
N220 G40
N230 G54 G0 Z300 T00;
N240 M30;
```

In the exemple, we considered:
Roughing with 3 passes
Material excess “X” = 8mm (Ø)
Material excess “Z” = 4mm
Material excess for finishing “X” = 2mm (Ø)
Material excess for “Z” = 0.4mm
Infeed = 0.3mm/rev.

OBSERVATION: *In the example, we considered roughing and finishing as done by using the same tool.*

8.5 - FUNCTION: G74

8.5.1 – Drilling cycle

Function G74, as drilling cycle, demands:

G74 R__;
G74 Z__ Q__ F__; Where:

R = incremental return for chip break at drilling cycle

Z = Final position (absolute)

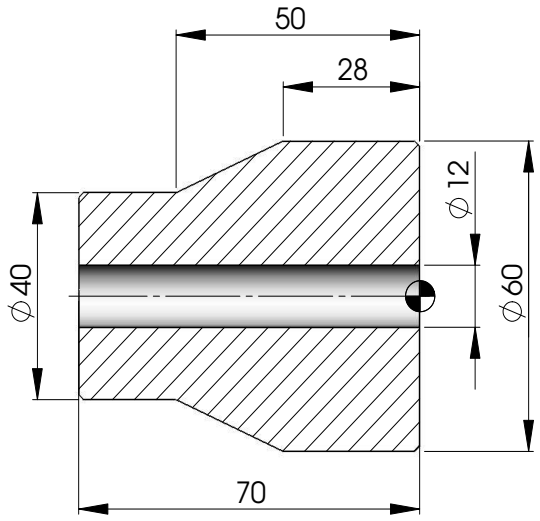
Q = increasing value at drilling cycle (millesime of millimeter)

F = Work infeed

NOTES:

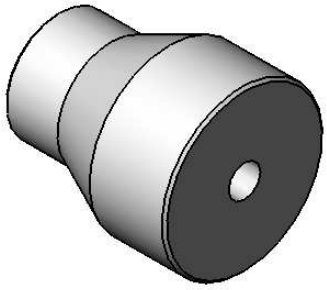
** After cycle running, the tool automatically returns to positioned point.*

** When using cycle G74 as drilling cycle, we can not inform functions "X" and "U" in the block.*



O0005 (DRILLING CYCLE);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T0505 (DRILL D12);
N40 G54;
N50 G97 S1200 M3;
N60 G0 X0;
N70 Z5;
N80 G74 R2;
N90 G74 Z-74 Q15000 F.12;
N100 G54 G0 Z300 T00;
N110 M30;

Drilling increasing = 15 mm
Infeed = 0,12 mm/rot



8.5.2 - Turning cycle.

Function G64 as turning cycle demands:

G74 X__ Z__ P__ Q__ R__ F__; Where:

X = Final turning diameter

Z= Final position (absolute)

P = Cutting depth (radius/ millesime of millimeter)

Q = Cutting length (incremental millesime of millimeter)

R = Spacing value at transversal shaft (radius)

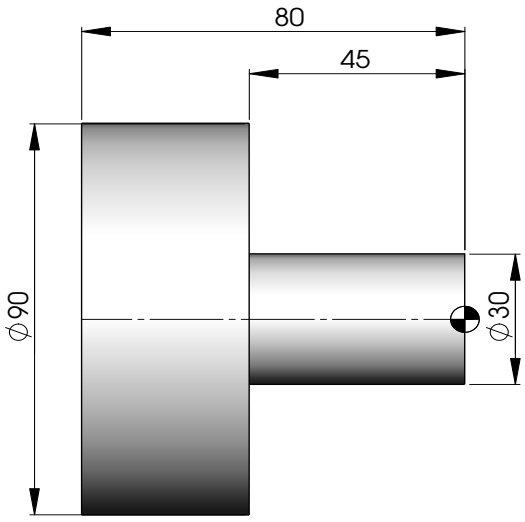
F = Work infeed

NOTES:

** After the cycle running, the tool automatically returns to positioned point.*

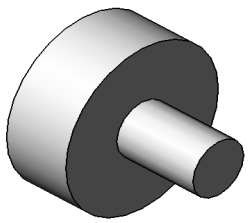
** For the running of this cycle, the tool shall be positioned at the diameter of first pass.*

** For the machining of the whole cutting length in one pass only (no chip break for shaft "Z"), we shall program the function "Q" more than cutting lenght.*




```
O0200 (ROUGHING CYCLE);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T0202 (ROUGHING);
N40 G54;
N50 G96 S250;
N60 G92 S3500 M4;
N70 G0 X84;
N80 Z2;
N90 G74 X30 Z-45 P3000 Q48000 R1 F.2;
N100 G54 G0 Z300 T00;
N110 M30;
```

Cutting depth = 3 mm
Infeed = 0,2 mm/rot



8.6 - FUNCTION: G75

8.6.1 - Grooves cycle.

The function G75, as grooves cycle, demands:

G75 R__;
G75 X__ Z__ P__ Q__ F__; Where:

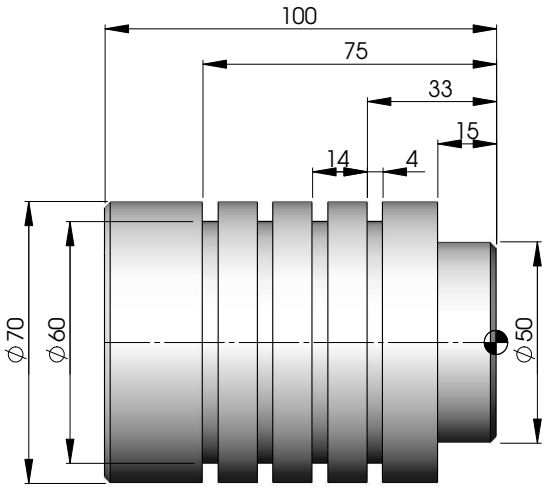
R = incremental return for chip break (radius)

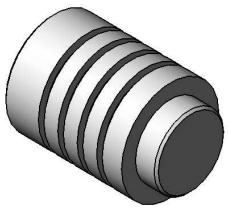
X = Final diameter of the groove

Z = Final position (absolute)

P = Cutting increasing (radius / millesime of millimeter)

Q = distance between grooves (incremental/ millesime of millimeter)





O0100 (GROOVES CYCLE);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T0202 (GROOVES);
N40 G54;
N50 G96 S130;
N60 G92 S4500 M4;
N70 G0 X75;
N80 Z-33;
N90 G75 R2;
N100 G75 X60 Z-75 P3000 Q14000 F.2;
N110 G54 G0 Z300 T00;
N120 M30;

Infeed = 0,2 mm/rot

8.6.2 - Facing cycle.

The function G75, as facing cycle, demands:

G75 X__ Z__ P__ Q__ R__ F__; Where:

X = Final diameter of facing

Z = Final position (absolute)

P = Cutting increasing in shaft “X” (radius/ millesime of millimeter)

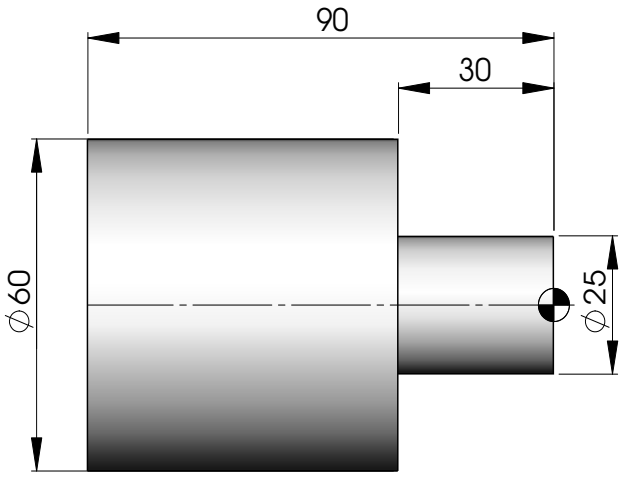
Q = Cutting depth per pass at shaft “Z” (millesime of millimeter)

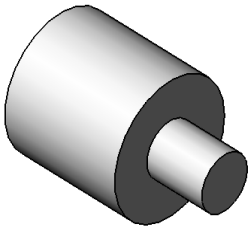
R = Spacing at longitudinal shaft for return to start “X” (Radius)

F = Programmed infeed

NOTES:

** For the cycle running, the tool shall be positioned at the length of the first pass..*
** After this cycle performing, the tool automatically returns to positioned point.*
** For the machining of the whole cutting length in one pass only (no chip break for shaft “Z”), we shall program the function “P” more than cutting lenght.*





```
O1000 (FACING CYCLE);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T0707 (DESB.);
N40 G54;
N50 G96 S210;
N60 G92 S3500 M4;
N70  G0 X64;
N80 Z-2;
N90  G75 X25 Z-30 P20000 Q2000 R1 F.2;
N100  G54 G0 Z300 T00;
N110 M30;
```

Cutting depth = 2 mm
Infeed = 0,2 mm/rot

8.7 - FUNCTION: G76

Application: Cycle of automatic threading

Function G76 demands:

G76 P (m) (s) (a) Q__ R__; Where:

m = repetitions number of last pass

s = angular exit of the thread $= \frac{r \times 10}{pass}$, where r = length of angular exit

a = tool angle (0°, 29°, 30°, 55° and 60°)

Q = minimum cutting depth (radius / millesime of millimeter)

R = depth of last pass (radius)

G76 X__ (U__) Z__ (W__) R__ P__ Q__ F__; Where:

X = Final threading diameter

U = incremental distance from positioned diameter to final thread diameter (diameter)

Z = final threading length

W = incremental distance of positioned point (start “Z”) up to final coordinate at longitudinal shaft (Final “Z”).

R = value of incremental taper angle at shaft “X” (radius/ negative for external and positive for internal).

P = height of thread (radius / millesime of millimeter)

Q = depth of first pass (radius / millesime of millimeter)

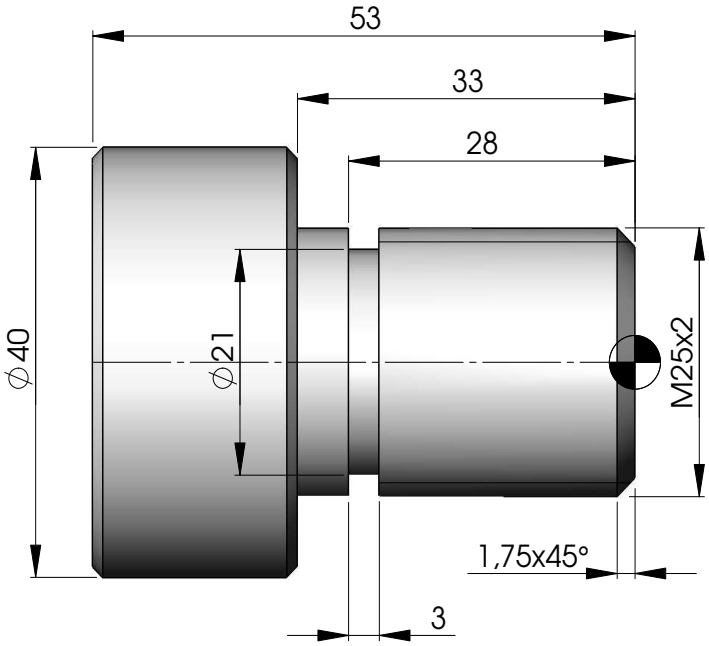
F = Thread pitch

OBSERVATION: *For the threading cycle programming, we shall use function G97, in order to RPM stay fixed.*

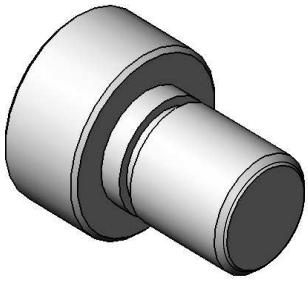
During the running of threading function, the maximum revolution of the spindle shall not overpass the value determined in the following equation:

$$RPMmax = \frac{5000}{PASSO}$$

EXAMPLE 1: Thread M25x2




```
O0330 (THREAD CYCLE);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T0101 (ROSCA M25X2);
N40 G54;
N50 G97 S1000 M3;
N60  G00 X29;
N70 Z4;
N80 G76 P010060 Q100 R0.1;
N90 G76 X22.4 Z-26.5 P1300 Q392 F2;
N100 G54 G0 Z300 T00;
N110 M30;
```



CALCULATIONS:

1st) Thread height (P):
P = (0.65 x pitch)
P = (0.65 x 2)
P = 1.3

2nd) Final diamenter (X):
X = Start diamenter - (P x 2)
X = 25 - (1.3 x 2)
X = 22.4

3rd) Depth of first pass (Q):

$$Q = \frac{P}{\sqrt{N.\ Passes}}$$

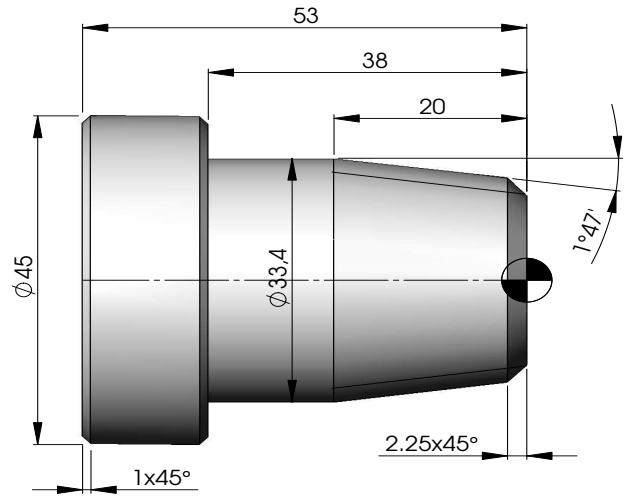
OBS.: In example, a calculation for 11 passes

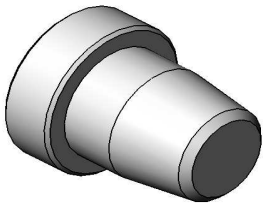
$$Q = \frac{1.3}{\sqrt{11}}$$

Q= 0.392

TAPER THREAD:

EXAMPLE 2: Taper thread NPT 11.5 threads/in.
(Slope: 1 degree 47 min)





```
O1000 (THREAD CYCLE);
N10 G21 G40 G90 G95;
N20 G54 G0 Z300 T00;
N30 T0202 (ROSCA NPT);
N40 G54;
N50 G97 S1200 M3;
N60 G0 X35;
N70 Z5;
N80 G76 P011560 Q150 R0.12;
N90 G76 X29.574 Z-20 P1913 Q479
R-0.778 F2.209;
N100 G54 G0 Z300 T00;
N110 M30;
```

CALCULATIONS:

1º) Pitch (F):
F=25.4 : 11.5
F=2.209

3º) Final diameter (X):
X = start diameter - (P x 2)
X = 33.4 - (1.913 x 2)
X = 29.574

5º) slope angle converting:

1° — 60'
A° — 47'

60 x A = 47 x 1
A =47 / 60
A = 0.783°

So, 1°47' = 1.783°

2º)Thread height (P):
P = (0.866 x pitch)
P = (0.866 x 2.209)
P = 1.913

4º) Depth of first pass (Q):

$$Q = \frac{P}{\sqrt{Nº\ PASSES}}$$

EXAMPLE: 16 passadas.

$$Q = \frac{1.913}{\sqrt{16}}$$

Q = 0.479

6º) Incremental taper angle at Shaft "X" (R):
tan α = Opposite cathetus / Adjacent cathetus
tan 1.783° = R / 25
R = tan 1.783° x 25
R = 0.778

9 - DRILLING CYCLES

9.1- FUNCTION : G80

Application: It cancels cycles from series G80
This function is used to cancel cycles from series G80, that is, from G83 to G85.

9.2- FUNCTION : G83

Application: Drilling cycle

This cycle allows to perform holes with chip break with or without return to start point after each drilling increasing. We also can program a dwell time at the end point of the drilling, as we will see:

G83 Z__ Q___ (P__) (R__) F__: where;

Z = Final position of the hole (absolute)

Q = Increasing value (incremental / millesimal)

P = dwell time at the end of each increasing (millesime of second)

R = reference plan for drilling start (incremental)

F = Infeed

OBSERVATIONS: *After the running of the cycle, the tool returns to start point.*

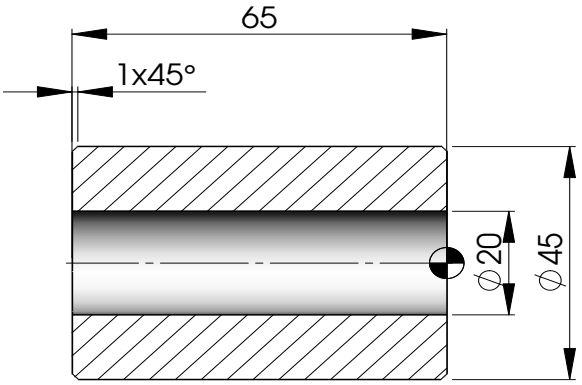
• If "R" is not programmed, drilling start will be the point "Z" of approach.

#5101.2 = 0 does the cycle in "chip break" mode.
#5101.2 = 1 does the cycle in "chip discharge" mode.

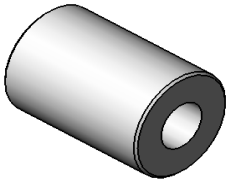
• If we use the drilling cycle with chip break (#5101.2 = 0) the return value shall be informed in the parameter 5114 - Value in "mm".

• If we use a drilling cycle with chip discharge (#5101.2 = 1),, the approach value after the return shall be informed in parameter 5115 – Value in "mm".

EXAMPLE :



:
N190 T0202 (DRILL);
N200 G54;
N210 G97 S1500 M3;
N220 G0 X0;
N230 Z3 ;
N240 G83 Z-68 Q15000 P1500 R -2 F0.12 ;
N250 G80 ;
:



9.3 - FUNCTION : G84

9.3.1 - Threading cycle with floating tapping

This cycle allows to open screws with tapping, using floating support. In order to do this, we shall program:

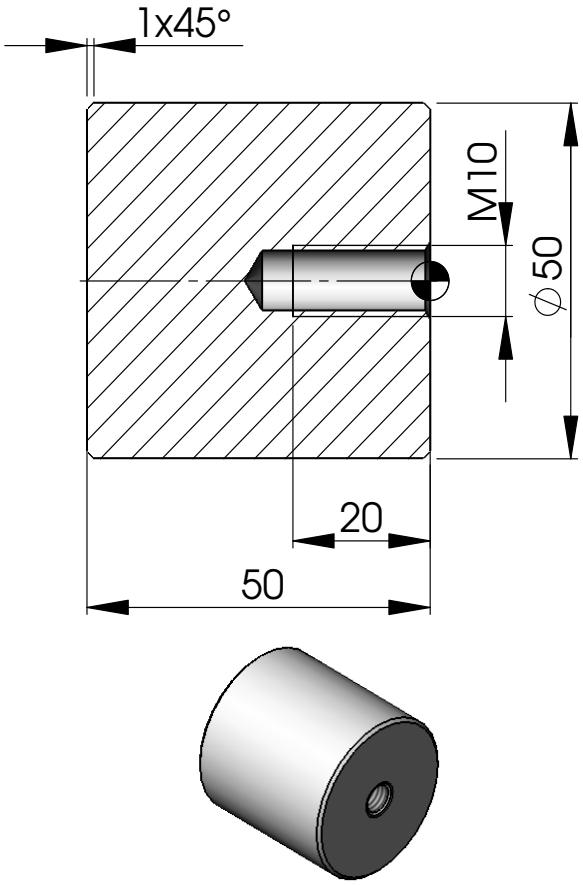
G97 S500 **M3**
G84 Z__ F__ , Where:

M3 = it indicates the revolution direction (M3 – right thread and M4 = left thread)

Z = final position of thread

F = Thread pitch

EXAMPLE :



:
N100 T0505 (FLOATING TAPPING);
N110 G54;
N120 G97 S500 M3;
N130 G0 X0;
N140 Z4;
N150 G84 Z-20 F1.5;
N160 G80;
N170 G54 G0 Z300 T00;
:

OBSERVATION: To make left thread, we shall change the code from M3 to M4.

9.3.2 - Threading cycle with rigid tapping

This cycle allows to open tapping threading, using rigid fixing, that is, no floating support. In order to do this, we shall program:

G97 S500 **M3**
M29
G84 Z__ F__, Where:

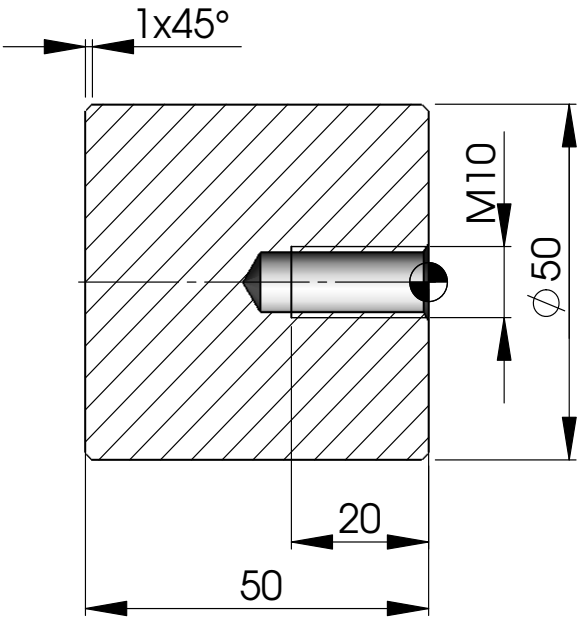
M3 = indicates revolution direction (M3 = right thread and M4 = left thread)

M29 = it activares threading with rigid tapping

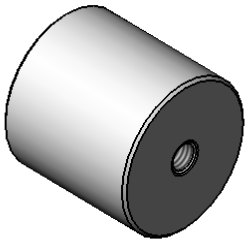
Z = final position of the thread

F = Thread pitch

EXAMPLE :



:
N100 T0606 (RIGID TAPPING);
N110 G54;
N120 G97 S500 M3;
N130 G0 X0;
N140 Z4;
N150 M29;
N160 G84 Z-20 F1.5;
N170 G80;
N180 G54 G0 Z300 T00;
:



OBSERVATION: To make left thread, we shall change code from M3 to M4.

9.4 - FUNCTION : G85

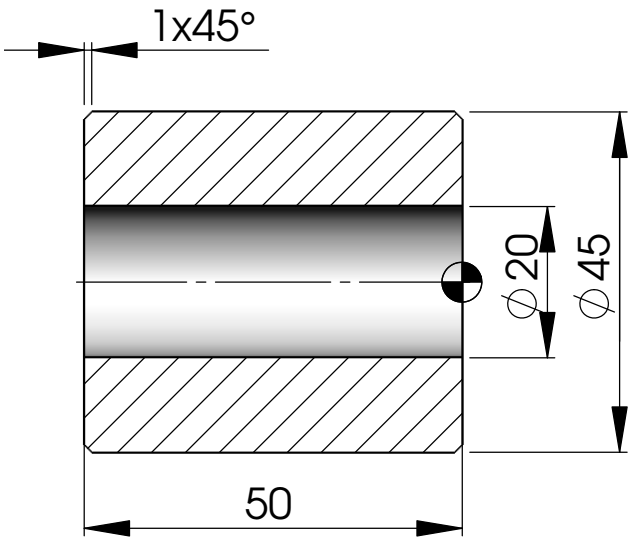
Application: Drilling cycle

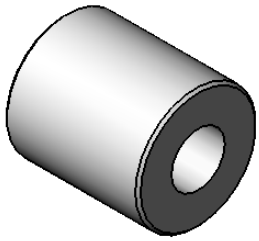
G85 Z__ F__, where

Z = Final position

F = Infeed

EXAMPLE:





```
.
.
N100 T0808 (DRILLING);
N110 G54;
N120 G97 S750 M3;
N130 G0 X0;
N140 Z2 ;
N150 G85 Z-55 F0.5 ;
N160 G80 ;
N170 G54 G0 Z300 T00;
.
.
```

OBSERVATION: Exit infeed is the Double of the programmed for machining.

10 - OTHER PREPARATORY FUNCTIONS

10.1 - FUNCTION: G20

Application: Reference measuring unit – Inch

This function prepares the command to compute all data inputs in inches.

OBSERVATIONS:
- Function G20 is modal, and it cancels functions G21.
- When changing measuring system from millimeter (G21) to inch (G20), the messages will be shown NEW COORDINATE SYSTEM ENABLED – TO CONVER EXTERNA COORDINATE AND G54 TO G59" and "NON REFERENCED SHAFTS" will be shown. So, we shall reference individually the machine shafts, according procedure below:

1- To press "HOME" key.
2- To press "CYCLE START" key once for each shaft to be referenced.

10.2 - FUNCTION: G21

Application: Measuring unit reference – International System.

This function prepares the command to compute all data inputs, in millimeter.

OBSERVATIONS:
- Function G21 is modal, and it cancels function G20.
- When changing measuring system from inches (G20) to millimeter (G21), the message: "NEW COORDINATE SYSTEM ENABLED – TO CONVER EXTERNA COORDINATE AND G54 TO G59" and "NON REFERENCED SHAFTS" will be shown. So, we shall reference individualy the machine shafts, according the procedure below:

1- To press key "Home".
2- To press the key "CYCLE START" one time for each shaft to be referenced.

10.3 - FUNCTION: G28

Application: It returns shafts for machine reference.

When we want to return the tool for the position "machine reference", we shall program:

EXAMPLE: **G28 U0 W0;**

10.4 - FUNCTION: G90

Application: Absolute coordinate system

This code prepares the machine to execute operations in absolute coordinate, with a pré-fixed source for programming.

OBSERVATION: Function G90 is model, and it cancels function G91.

10.5 - FUNCTION: G91

Application: Incremental coordinate system

This code prepares the machine to execute all operations in incremental coordinate. So, all measurings are done between the distance to be displaced, that is, the source of the coordinates from any point will be the point previous to displacement.

OBSERVATION: Function G91 is modal, and it cancels function G90.

10.6 - FUNCTION: G92

Application: It establishes revolution limit (RPM) / Establishes new source

Function **G92**, together code **S____**(4 digits) is used to limit maximum revolution of spindle (RPM). Usually, this function is programmed in the block next to the function G96; which is used to program cutting speed.

Example: **G92 S2500 M4**; (it limits spinde revolution at 2500 RPM).

Function **G92** also shall be used to establish a new source of coordinate system. In order to do this, it shall be programmed in a block together one or more machine shafts.

Example: **G92 Z0**; (it establishes a new source of coordinates system, fixing the current position as “Z0”).

OBSERVATIONS:

- Function G92 is modal.
- To cancel G92, when used to establish a new source of coordinates system, a new G92 shall be programmed, fixing a new source, or to program function G92.1 Z0, returning, so, the zero point to original position.

10.7 - FUNCTION: G94

Application: It establishes infeed x / minute

This function prepares the command to compute all infeeds in inches / **minutes** (**G20**) or millimeters / minutes (**G21**).

OBSERVATION: Function G94 is modal, and it cancels function G95.

10.8 - FUNCTION: G95

Application: It establishes infeed x /revolution:

This function prepares the command to compute all infeeds in inches / **revolution** (**G20**) or millimeter / revolution (**G21**).

OBSERVATION: Function G95 is modal, and it cancels function G94.

10.9 - FUNCTION: G96

Application: It establishes programming in fixed cutting speed.

Function G96 selects programming mode in fixed cutting speed, where RPM calculation is programmed by function “S”.

Maximum RPM reached by fixed cutting speed can be limited by the programming of function G92.

Example: **G96 S200**; (cutting speed at 200m/min)

OBSERVATION: Function G96 is modal, and cancels function G97.

10.10 - FUNCTION: G97

Application: It establishes programming in RPM

Function G97 is used to program a fixed revolution of the spindle (RPM), with the AID of function S and using a format (S4).

Example: **N70 G97 S2500 M3**; (revolution at 2500 RPM)

RPM range can be done by using the “Splindle Revolution Selector”, and it can float from 50% to 120% or programmed revolution.

OBSERVATION: Function G97 is modal, and it cancels function G96.

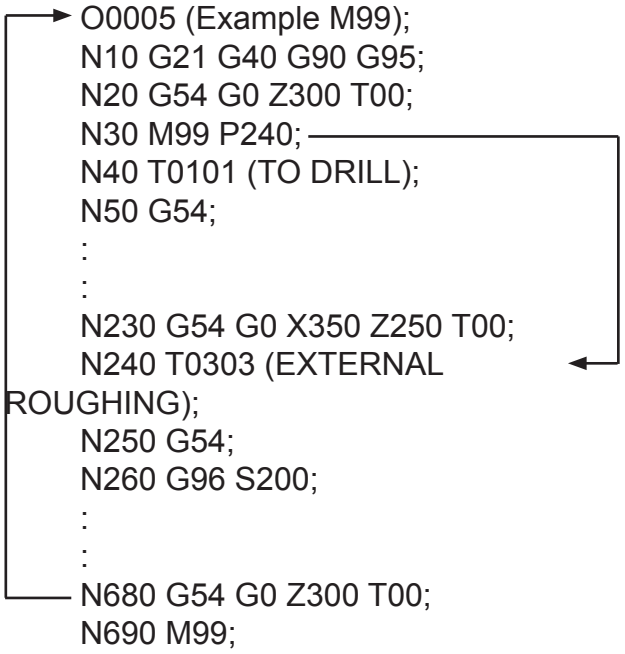
11 - UNCONDITIONAL DEVIATION

Function: M99

The programming of function M99 with function “P”, followed by block number, forces the command to advance / to return programming for the block indicated by “P”.

When function M99 replaces M30 in main program, the program will be executed next in “looping”.

EXAMPLE:



12 - CHAMADA E RETORNO DE SUBPROGRAMA

Funções: M98/M99

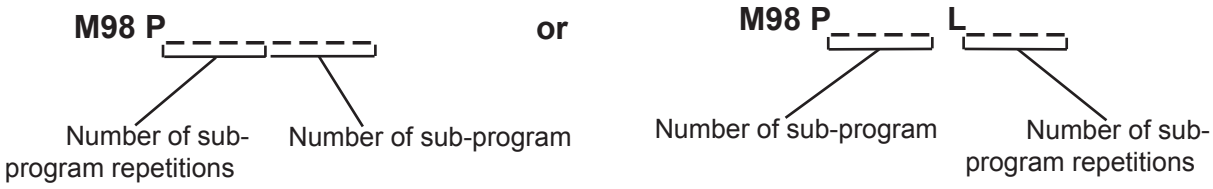
Many times, during the machining, a determined operations sequence appears, and we can use the sub-program call resource by the function **M98**.

The block containing function M98 shall also contain the sub-progam number by the function “P”. Ex.: **M98 P1001**.

The sub-program, by its time, shall contain refered number at the program start by the function “O” and by finishing with function M99.

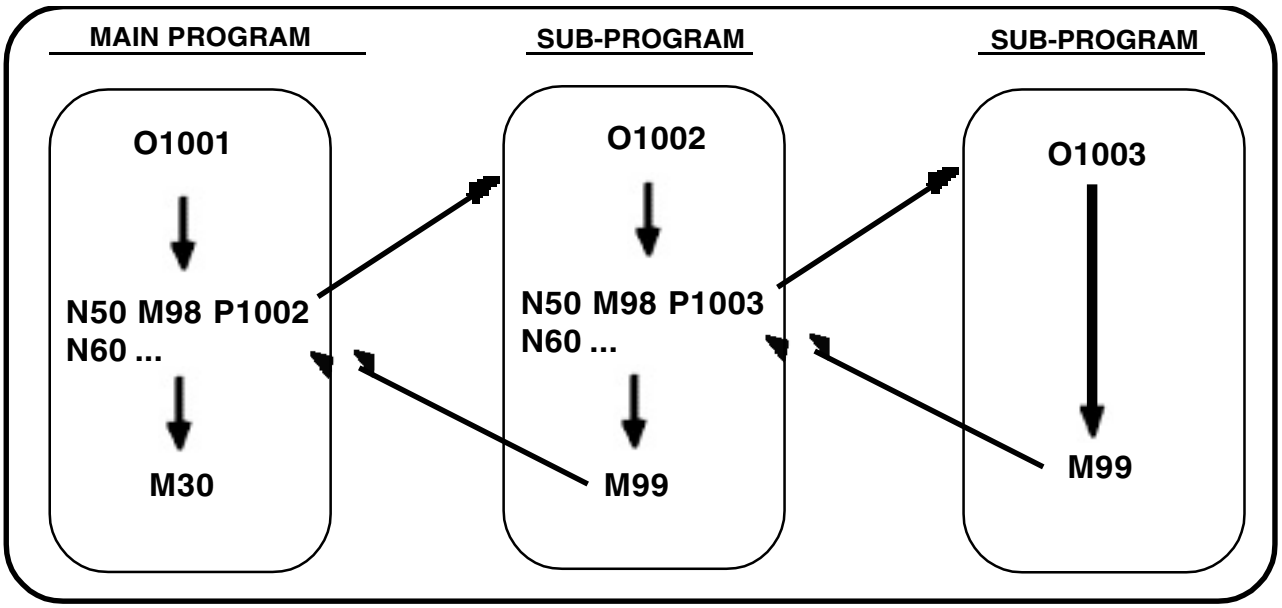
NOTE: The number of sub-program is the same found at the CNC command directory.

The formats for a sub-program call are the following:



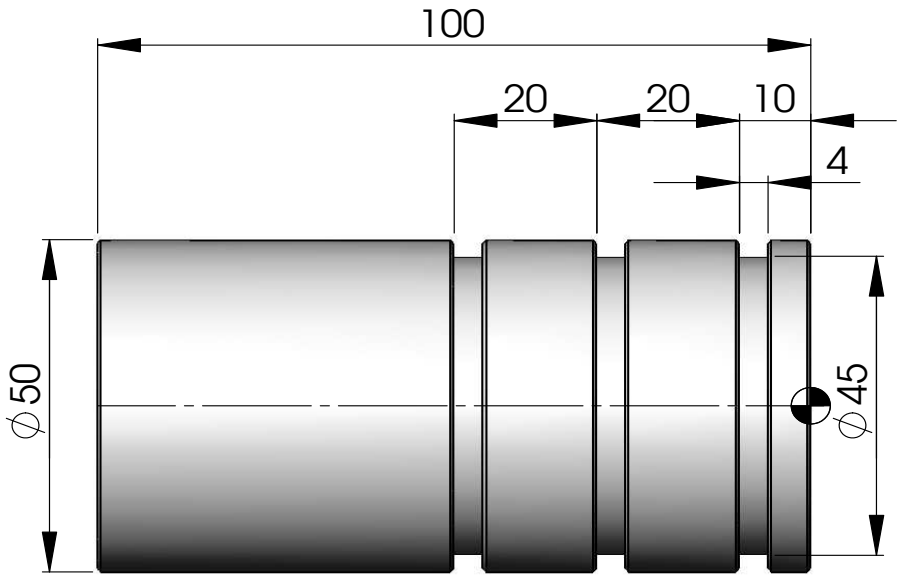
When the sub-program finishes its operations, control returns to main program.

EXAMPLE:

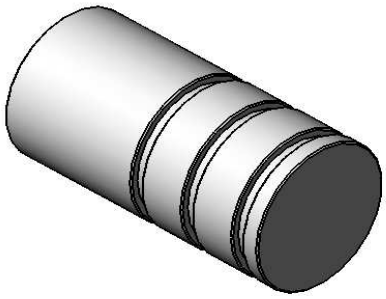


OBSERVATION: *It the number of repetitions is ommited, the command will execute sub-program one time.*

EXAMPLE: INCREMENTAL PROGRAMMING AND SUB-PROGRAM



To bevel corners with 0.5 x 45°



Main program (O0001)

```
O0001 (MAIN - PART)
N10 G21 G40 G90 G95;
.
.
.
N90 G0 X55 Z10;
N100 M98 P30002;
(N100 M98 P2 L3;)
N110 G54 G0 Z300 T00;
N120 M30;
```

Sub-program (O0002)

```
O0002 (PART SUB-PROGRAM)
N10  G0 W-20;
N20  G1 X45 F.15;
N30  G0 X55;
N40  W-0.5;
N50  G1 X50;
N60  X49 W0.5;
N70  G0 X55;
N80  W0.5;
N90  G1 X50;
N100  X49 W-0.5;
N110  G0 X55;
N120  M99;
```

13 - SPECIAL FUNCTIONS

13.1 - FUNCTION : G10

Application: Manager of Tool Life

This function allows to monotor time or frequence (number of parts) of a determined tool, objectifying its replacement for the wanted.

For the determination of monitoring mode, the parameter 6800#2 (LTM) shall be changed to:

0 = manages by **AMOUNT OF PARTS;**
1 = manages by machining **TIME** (minutes).

A program containing monitoring data shall be done, in order to be loaded in the life Page of the tool.

Tool groups for distinct operations shall be created.

Example :

```
O0010;
N10 G10 L3; (Activates the manager)
N20 P01 L20; (P01 = Group number, L20 = Time/ min. Or amount of parts)
N30 T0202;
N40 T0404; (Loads tools in group 01)
N50 T0606;
N60 G11; (Cancels manager)
N70 M30;
```

Example of machining program:

```
O0011 ;
N10 G21 G40 G90 G95 ;
N20 G54 G0 Z300 T00 ;
N30 T0101 ;
:
N100 T0199; (01 = Group number, 99 = Activats tools management)
N110 G96 S220 ;
:
N220 T0188; (Cancels management)
N230 T0505 ;
N240 G96 S180 ;
:
N350 M30 ;
```

OBSERVATION:
The command automatically will use the tools described in the group, being that when the life time of the last tool ends, a message will be shown, requesting the reloading of tools life. In order to reload, the procedure below shall be done:
- To set “OFFSET SETTING” KEY.
- To set softkey [►] until it shows [TOOLLF]
- To set softkey [TOOLLF]
- To set softkey [OPRT]
- To set softkey [LIMPAR] (delete)
- To set softkey [EXEC]

13.2 - FUNCTION “G64”

Application: Angular positioning of the spindle

By this function, following argument C (specified in degrees), we can program a determined angle for the positioning of spindle.

Before the activation of function “G64 C...”, we shall program function “M19”, which is responsible to guide spindle.

EXAMPLE:
:
M19;
G64 C0; (the spindle positions at zero degree)
:

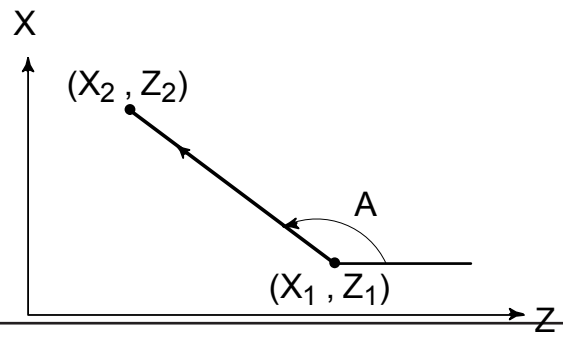
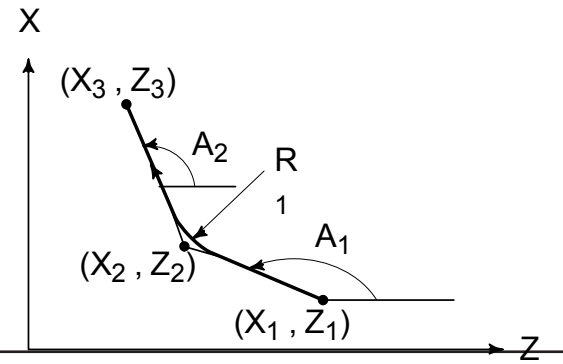
OBSERVATION:
This function only is available at machines from GL Line which not offer swivel tool resource, that is, machines version T. In order to know how to guide spindle in machines with swivel tools, Part II of this manual shall be consulted (Level II Programming).

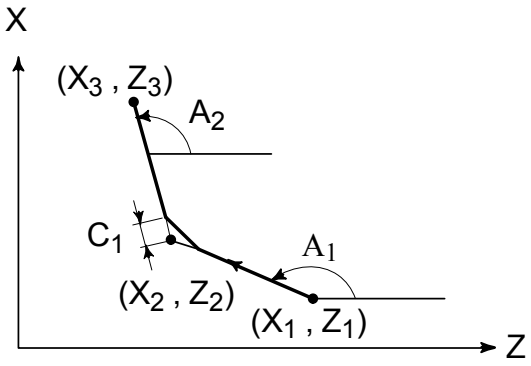
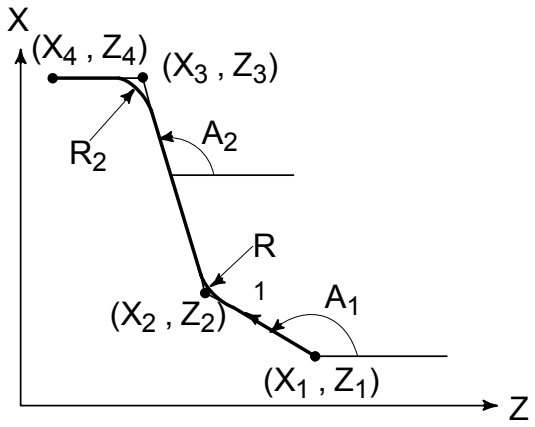
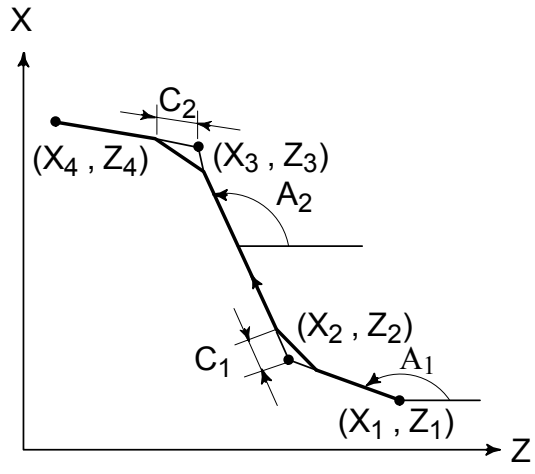
13.3 - DIRECT INPUT OF VALUES FOR BEVELS AND CONCORDANCES

Straight lines angles, bevel values and rounding values of corners and other values can be programmed, introducing it directly in coordinates.

Sintaxes:
A___; (where “A” is equivalent to displacement angle)
,R___; (where “R” is equivalent to corner rounging value)
,C___; (where “C” is equivalent to corner beveling value)

OBSERVATIONS:
- Codes G02, G03, G90, G94 shall not be programmed together the direct introduction of angular values, bevels and rounding.
- Corner rounding shall not be inserted in a thread opening block.
- Na alarm occurs if intersection point is calculated if the angle created by two lines is between +/- 1º.

COMMANDS	TOOL MOVEMENT
X_2____(Z_2____) A__;	
X2___ Z2___ ,R1___; **X3___ Z3___;** **OR** **A1___ ,R1___;** **X3___ Z3___ A2___;**	

COMMANDS	TOOL MOVEMENT
X2___ Z2___ ,C1___; **X3___ Z3___;** **OR** **A1___ ,C1___;** **X3___ Z3___ A2**	
X2___ Z2___ ,R1___; **X3___ Z3___ ,R2___;** **X4___ Z4____;** **OR** **A1___ ,R1___;** **X3___ Z3___ A2___ ,R2___;** **X4___ Z4____;**	
X2___ Z2___ ,C1___; **X3___ Z3___ ,C2___;** **X4___ Z4___;** **OR** **A1___ ,C1___;** **X3___ Z3___ A2___ ,R2___;** **X4___ Z4___;**	

13.4 - FUNCTION G65

Application: MACRO B

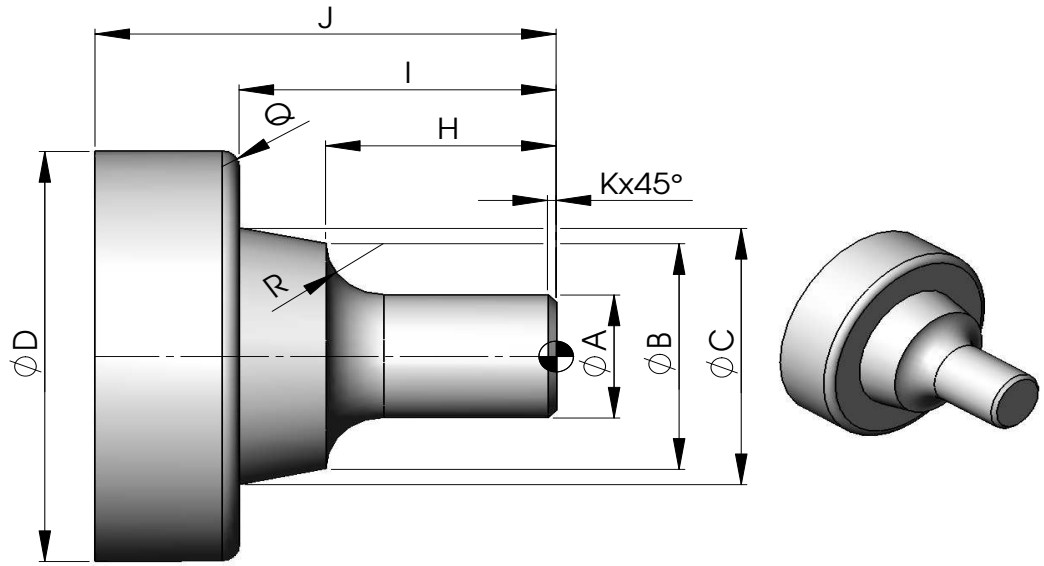
We can use this function when we want to elaborate programs, which parts to be manufactured presents equal geometric shapes, but with different dimensions, that is, in case of part families. So, we shall elaborate a program, defining the process to be used for machining, with dimensions quantities represented by variables, according table.

Arguments and variables table **MACRO B :**

ARGUMENT ADDRESS	CORRESPONDING VARIABLE
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

This program will be called by other, where function G65 shall be programmed, followed by function P, by defining the number of program, containing the machining process, and also the variavles addresses, represented by table letters with its respective dimensional values.

PROGRAMMING EXAMPLE:




```
O0001 (MAIN PROGRAM);
G21 G40 G90 G95;
G54 G0 Z300 T00;
T0101 (USIN. EXT);
G54;
G96 S200;
G92 S3500 M4;
G65 A24 B44 C50 D80 H25 I40 J80 K1.5 Q3 R10 F0.2 P100;
G54 G0 Z300 T00;
M30;
```

Ø A = 24 mm
Ø B = 44 mm
Ø C = 50 mm
Ø D = 80 mm

H = 40 mm
I = 55 mm
J = 80 mm

Q = 3 mm
R = 10 mm
K = 1 mm

```
O0100 (MACRO);
G0 X[#1-[2*[#6]]] Z2;
G42;
G1 Z0 F[#9];
X[#1] Z[-#6];
Z[-#11+#18];
G2 X[#2] Z[-#11] R[#18];
G1 X[#3] Z[-#4];
X[#7-[2*[#17]]];
G3 X[#7] Z[-#4-#17] R[#17];
G1 Z[-#5]
G40;
U2;
M99;
```

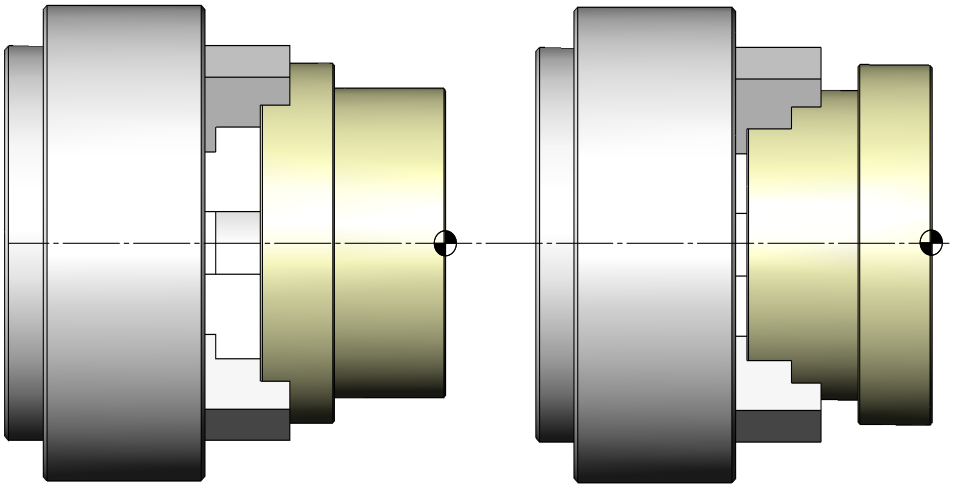
13.5 – WORK REFERENCE (G54 A G59)

Work Reference, also known as Zero-Part, corresponds to the point which serves as source for absolute coordinates system, that is, it is the part point referenced as “X0” and “Z0”.

In some cases, more than one work reference is used in a same program, with the purpose to make easier the programming of some parts. Example: in case of programming the machining of two sides of a part in a same program, it is recommended to use two zero-part, in order to the programmer must not be worried with somes elements, as metal excess at both sides of material, different jaw backs, etc.

NOTE: *At the machines “GL Line”, up to six zero-part can be referenced, which shall be done manually during machine’s preparation process. They are: G54, G55, G56, G57, G58 and G59.*

EXAMPLE:



G54 (FIRST RABBET) **G55 (SECOND RABBET)**

The values of family G54 shall be typed in the Page “OFFSET SETTING” by the softkey “TRAB”.

14 - MISCELLANEOUS OR AUXILIARY FUNCTIONS

Auxiliary functions comprehends machine resources not covered by previous functions.

NOTES: Machines from "GL Line" can be set by many ways (with/ without counter point, with/ without parts catcher, with/ without automatic door, etc.) and due to this, not all functions below described are enabled at all machines.

- Functions with "**INSTANT REPLY"** shall be carefully programmed, because the machine not expects any confirmation form Ladder to go on with program running.

FUNCTION: M00
Application: Program Stop.

This code causes instant stop of program running, including CUT coolant and spindle.

FUNCTION: M01
Application: optional stop of the program.

This function causes interruption at program running only if the Button "OPTIONAL STOP", located at the machine's operation panel is pressed. So, function M01 pass to be equivalent to the function M00, keeping program running in normal way.

When stop occurs by this code, we shall press the Button "CYCLE START", in order to continue program running.

FUNCTION: M02
Application: program end with no return to beggining.

This function is used to indicate the end of program existing at command memory, and also used when we work with tapes joined in "link" format.

FUNCTION: M03
Application: turns left spindle in clockwise direction.

This function turbs spindle in clockwise direction, adopting as reference for the turn direction the rear position of the plate.

Function **M03** is cancelled by: **M00, M01, M02, M04, M05** and **M30.**

FUNCTION: M04
Application: turns left spindle in counterclockwise direction.

This function turbs spindle in counterclockwise direction, adopting as reference for the turn direction the rear position of the plate.

Function **M04** is cancelled by: **M00, M01, M02, M04, M05** and **M30.**

FUNCTION: M05
Application: It turns spindle off and deactivates low and high torque brakes from left head.

This function is used to turn off spindle revolution, cancelling functions **M03** or **M04**, and to deactivate low and high torque brakes, cancelling functions **M85** and **M86**, respectively.

Function **M05** is already active when starting program.

FUNCTION: M07
Application: turns on high pressure cut coolant.

FUNCTION: M08
Application: turns on cut coolant.

FUNCTION: M09
Application: Turns off cut coolant.

FUNCTION: M18
Application: Turns off spindle guiding.

FUNCTION: M19
Application: guides the spindle.

FUNCTION: M20
Application: turns on bar's Power source.

FUNCTION: M21
Application: turns off bar's power source.

FUNCTION: M24
Application: Opens plate of the spindle.

FUNCTION: M25
Application: Closes plate of spindle

FUNCTION: M28
Application: Turns off rigid tapping.

FUNCTION: M29
Application: turns on rigid tapping.

FUNCTION: M30
Application: End of program with return to start..

FUNCTION: M34
Application: selects pressure level 1 for the plate.

FUNCTION: M35
Application: selects pressure level 2 for the plate.

FUNCTION: M36
Application: opens to automatic door.

FUNCTION: M37
Application: Closes automatic door.

FUNCTION: M38
Application: Advances parts catcher device.

FUNCTION: M39
Application: Returns parts catcher device.

FUNCTION: M40
Application: selects to fix by internal for left plate.

FUNCTION: M41
Application: selects to fix by external for left plate.

FUNCTION: M42
Application: turns on air for left plate cleaning.

FUNCTION: M43
Application: turns off air for left plate cleaning.

FUNCTION: M45
Application: turns on protections cleaning.

FUNCTION: M46
Application: turns off protections cleaning.

FUNCTION: M47
Application: turns on chips conveyor.

FUNCTION: M48
Application: turns off chips conveyor

FUNCTION: M76
Application: activates parts counter

FUNCTION: M78
Application: turns on mist exhauster

FUNCTION: M79
Application: turns off mist exhauster

FUNCTION: M80
Application: conditional jump

FUNCTION: M93
Application: enables programs running via PCMCIA card.

FUNCTION: M94
Application: disables programs running via PCMCIA card.

FUNCTION: M99
Application: restarts program running / unconditional jump.

FUNCTION: M105
Application: turns off spindle with instant replay.

FUNCTION: M124
Application: opens spindle plate with instant replay.

FUNCTION: M125
Application: closes spindle plate with instant reply.

FUNCTION: M128
Application: Allows to turb spindle with open plate.

FUNCTION: M136
Application: opens automatic door with instant replay.

FUNCTION: M137
Application: closes automatic door with instant replay.

FUNCTION: M138
Application: advances parts catcher device with instant replay.

FUNCTION: M139
Application: returns parts catcher device with instant replay.

FUNCTION: M142
Application: turns on air for plate cleaning with instant replay.

FUNCTION: M143
Application: turns off air for left plate cleaning with instant replay.

FUNCTION: M176
Application: turns on air for left plate cleaning with instant replay.

FUNCTION: M177
Application: turns off air for left plate cleaning with instant replay.

FUNCTION: M224
Application: Unlocks plate's "opening and closing" movement.

FUNCTION: M225
Application: Blocks plate's "opening and closing" movement.

FUNCTION: M300
Application: releases turret turn

15 - SEQUENCE FOR MANUSCRIPT PROGRAMMING

The program needs to be aware of all parameters involved in the process, and to obtain a proper solution for the machining of each type of part. It shall also analyze all machine resources, which shall be demanded on the part running.

15.1 - PART DRAWING STUDY: FINAL AND RAW

The programmer shall be able to compare drawing (finished part) with the dimension wanted in machining with Numerical Control machine.

It is also necessary an analysis on the part running feasibility, considering dimensions requested, metal excess existing from previous step. Tools needed, part fixing, etc.

15.2 - PROCESS TO USE

It is necessary to have a definition of machining steps for each part to be executed, establishing the proper fixing system related to machining.

15.3 - TOOL SET FOCUSED ON CNC

The choice of tool set is very important, and also its arrangement on turret. It is necessary the tools are placed in a way with no interference between itself and the remaining of the machine. A good programs depends on very much of the choice of proper tools and its fixing in a proper arrangement.

15.4 - KNOW-HOW OF MACHINE'S PHYSICAL PARAMETERS AND COMMAND PROGRAMMING SYSTEMS

The programmer shall also know how to fit operations in a way to use all resources from machine and from command, always objectifying to minimize time and operation phases, and also to ensure product quality.

15.5 - DEFINITION IN FUNCTION OF MATERIAL, CUTTING PARAMETERS, AS INFEED, SPEED, ETC.

In function of material to be machined, and also in function of tool used and in function of the operation to be done, the programmer shall establosh cutting speed, infeeds and Power requested from machine. Calculations necessary to obtain those parameters are the following:

16 - CALCULATIONS

16.1 - CUTTING SPEED (VC)

Depending on material to be machined, cutting speed is an important and necessary data.

Cutting speed is a quantity directly proportional to diameter and revolution of spindle, given by formula:

$$VC = \frac{Ø_P \times 3{,}14 \times N}{1000}$$

Where:

VC = Cutting speed (m/min)

$Ø_P$ = Part diameter (mm)

N = Revolution of spindle (rpm)

16.2 - REVOLUTION (N)

At the determination of cutting speed for a determined tool do it at machining, revolution is given by formula:

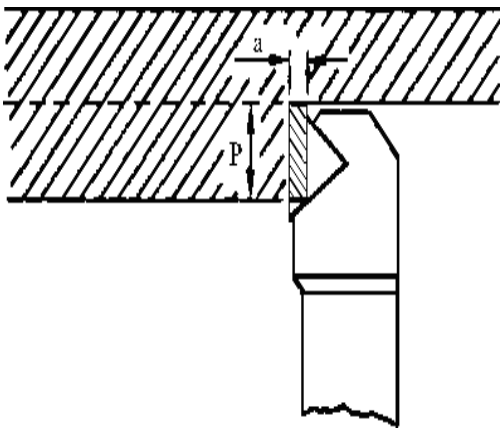
$$N = \frac{VC \times 100}{3{,}14 \times Ø}$$

16.3 - CUTTING POWER (NC)

In order to avoid some problems during machining, as motor overload and so stop of spindle during operation, it is necessary to make a previous calculation of Power to be used, which is given by formula:

$$NC = \frac{KS \times FN \times AP \times VC}{4500 \times \eta} \quad (CV)$$

CUTTING ÁREA FOR 90 DEGREE TOOLS



Ks = Specific cutting pressure
Ap = Cutting speed
Ks = Specific cutting pressure
Ap = cutting depth
Fn = Infeed
Vc = Cutting speed
η = Performance:

LINE E	=	0,9
LINE GL / GLM	=	0,9
CENTER	=	0,8

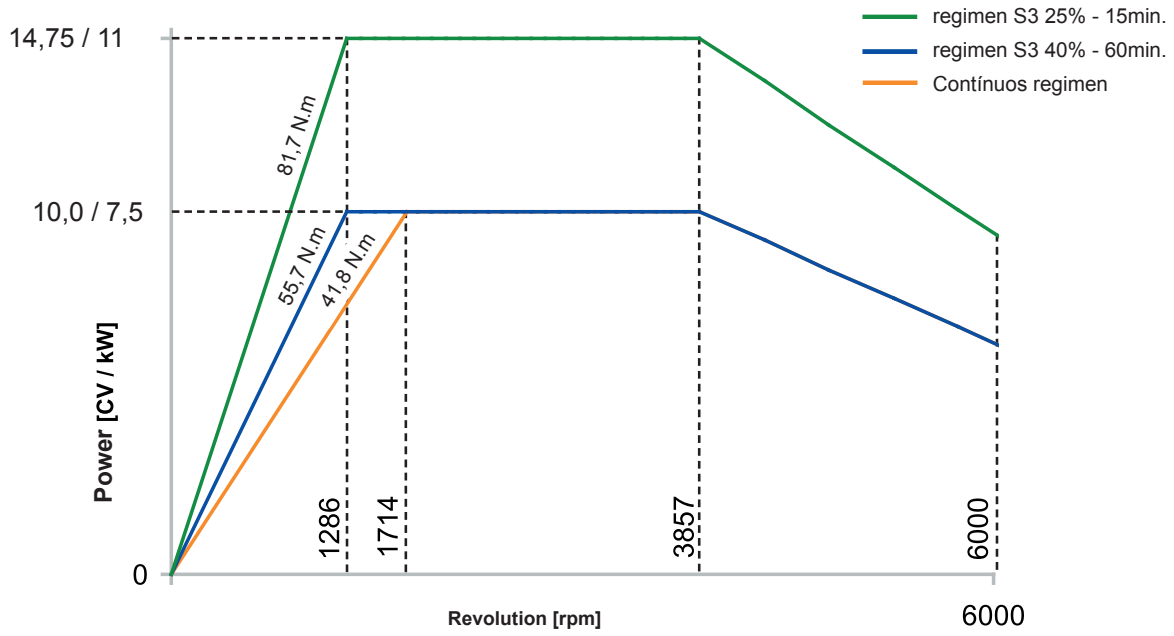
ORIENTATION VALUES FOR SPECIFIC CUTTING PRESSURE (KS)

MATERIAL	PULL STRENGTH Kgf/mm² BRINELL HARDNESS		"KS" IN KG/MM² INFEED IN MM/REV			
	Kgf/mm²	HB	0,1	0,2	0,4	0,8
SAE 1010 to 1025	Up to 50	Up to 140	360	260	190	136
SAE 1030 to 1035	50 to 60	140 to 167	400	290	210	152
SAE 1040 to 1045	60 to 70	167 to 192	420	300	220	156
SAE 1065	75 to 85	207 to 235	440	315	230	164
SAE 1095	85 to 100	235 to 278	460	330	240	172
Soft Cast Steel	30 to 50	96 to 138	320	230	170	124
Mid Cast Steel	50 to 70	138 to 192	360	260	190	136
Hard Cast Steel	Above 70	Above 192	390	286	205	150
Mn-Steel Cr-Ni Steel	70 to 85	192 to 235	470	340	245	176
Cr-Mo Steel	85 to 100	235 to 278	500	360	260	185
Soft Alloy Steel	100 to 140	278 to 388	530	380	275	200
Hard Alloy Steel	140 to 180	388 to 500	570	410	300	215
Stainless Steel	60 to 70	167 to 192	520	375	270	192
Tool Steel	150 to 180	415 to 500	570	410	300	215
Hard Manganese Steel			660	480	360	262
Soft soft		Up to 200	190	136	100	72
Mid Soft		200 to 250	290	208	150	108
Hard Soft		250 to 400	320	230	170	120
Hardened Soft			240	175	125	92
Aluminum		40	130	90	65	48
Cooper			210	152	110	80
Cooper with alloy			190	136	100	72
Brass		80 to120	160	115	85	60
Red Brass			140	100	70	62
Cast Brass			340	245	180	128

17 - POWER GRAPHIC

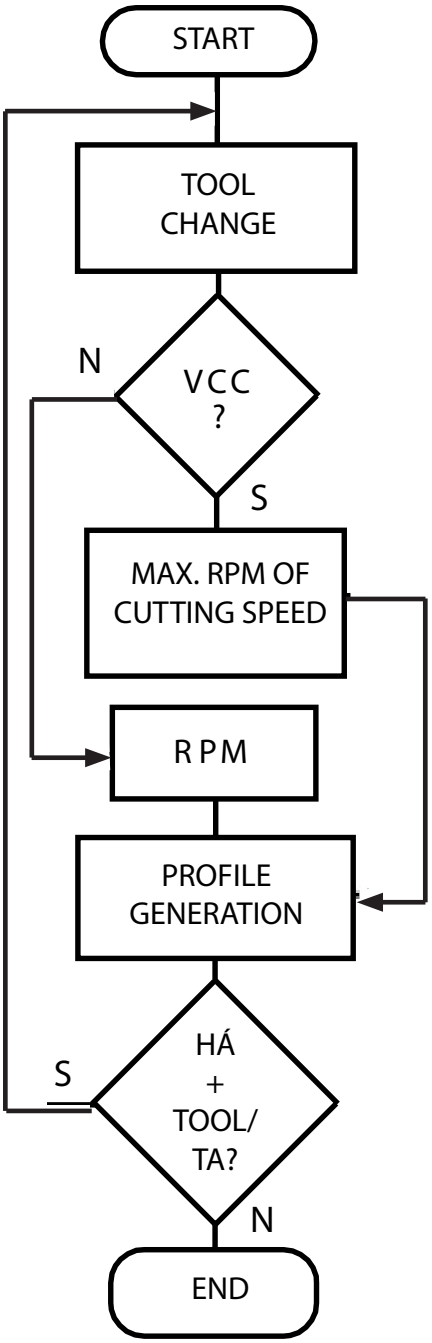
GL170 GANG TOOLS

Head – 6,000 rpm
Nose ASA A2-5” – hole Ø58mm



18- PROGRAMMING FLOWCHART

Flowchart of machines from G / GL line:



***START**
O_ _ _ _ (COMMENT); -program number
G21 G40 G90 G95; - Safety block

***TOOL CHANGE**
G5_G0 Z_ _ _ T00; - Activates the point zero-part and displaces to the point of tool change at shaft Z
T_ _ _ _;number of wanted tool
G5_; - Definition of zero-part (G54 to G59)

***VCC PROGRAMMING**
G97 S_ _ _ _M_ _;- defines RPM and turns on spindle (M03 or M04)

***PROFILE GENERATION**
(Instructions according programmer creativity)

***END OF PROGRAM**
G5_G00 Z_ _ _T00; - point of tool change
M30/M99; - end of program (M30) or restart program (M99)

19 - GANG OF TOOLS

19.1- INTRODUCTION

The Gang Table of Tools is a table, transversal to longitudinal shaft of the machine (“Z” shaft), where working tools are arranged. Fig. 1.

This setting allows a quicker turning, reducing passive time (idle displacements), due to there is no need of large displacements for tool change nor time used with turret turn.

(Obs.: For Gang table of tools it is not possible to use mobile head).

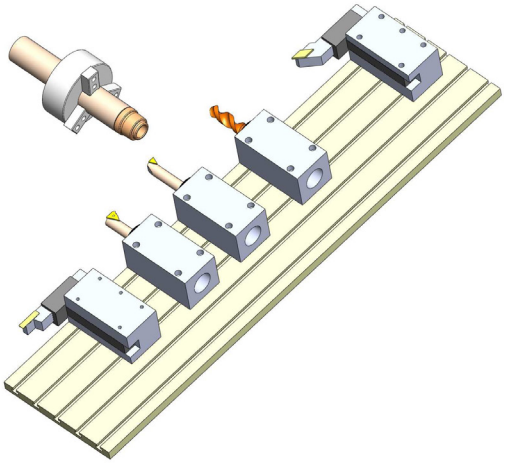


Figure 1

19.2- TOOLS ASSEMBLY

Tools assembly at GANG TABLE shall be done carefully, taking care with possible interferences on part, material or collet plate / chuck. **Fig 2** and **Fig. 3**.

Correct Position

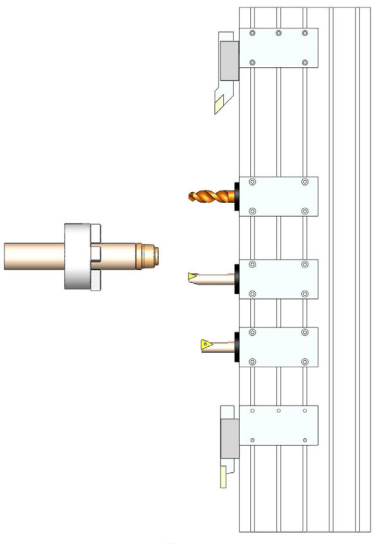


Figure 2

Position with collision

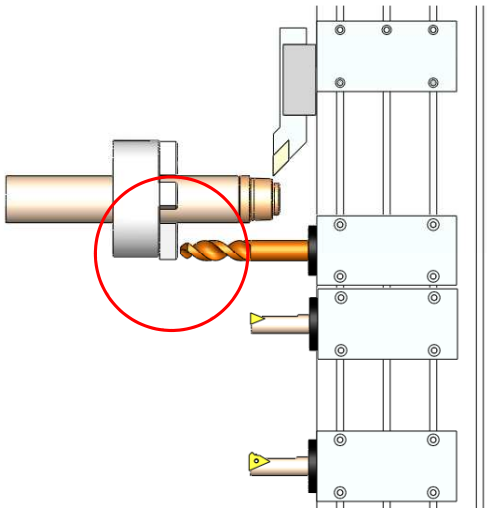


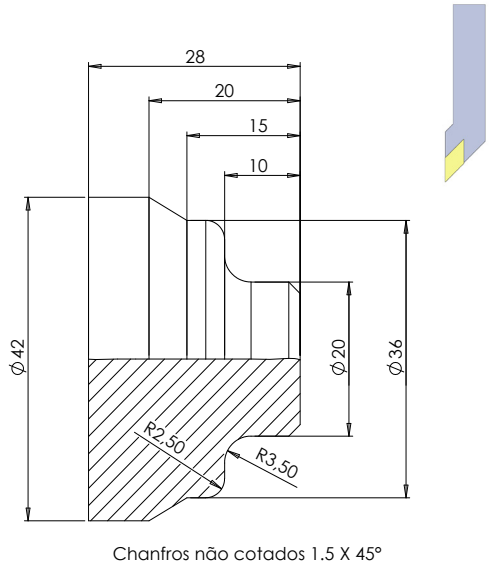
Figure 3

20 - PROGRAMMING WITH GANG TOOLS

At the GANG TABLE we can work with tools both with quadrant at positive shaft “X”, or negative quadrant, so we must pay attention in relation to coordinates of “X” shaft (positive or negative), radius direction (G2/G3), cutting side of tool, code for tool radius offset and metal excess for roughing cycles.

Programming examples below:

20.1 PROGRAM WITH TOOL AT POSITIVE QUADRANT (X+)



Chanfros não cotados 1.5 X 45°

Materail: Steel 1045 – Bar Ø 1’3/4” (44.5mm).

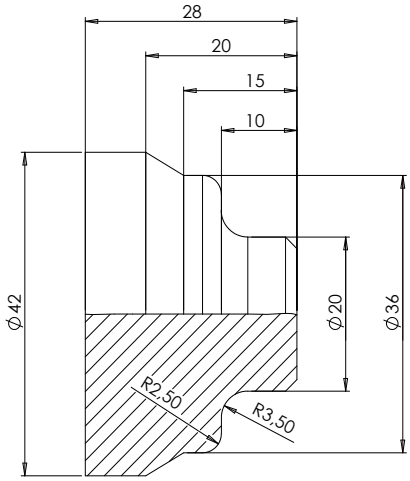
T0101 – Roughing and external finishing

Cut side of tool: **3**

Revolution direction: **M4**

```
%
O0020 (EXAMPLE X+);
N10 G21 G40 G90 G95;
N20 T0101 (ROUGHING AND EXT. FIN);
N30 G54 G0 Z50;
N40 G96 S200;
N50 G92 S3000 M4;
N60 G0 X46 M8;
N70 Z0;
N80 G1 X-2 F.15;
N90 G0 X45 Z2;
N100 G71 U2.5 R1;
N110 G71 P120 Q210 U1 W.2 F.3;
N120 G0 X15;
N130 X20 ,C1.5;
N140 Z-6.5;
N150 G2 X27 Z-10 R3.5;
N160 G1 X31;
N170 G3 X36 Z-12.5 R2.5;
N180 G1 Z-15;
N190 X42 Z-20;
N200 Z-28;
N210 X45;
N220 G96 S220;
N230 G42;
N240 G70 P120 Q210 F.15;
N250 G40;
N260 G0 Z50 M9;
N270 M30;
%
```

20.2 PROGRAM WITH TOOL AT NEGATIVE QUADRANT (X-)



Chanfros não cotados 1.5 X 45°

Materail: Steel 1045 – Bar Ø 1’3/4” (44.5mm).

T0101 – Roughing and external finishing

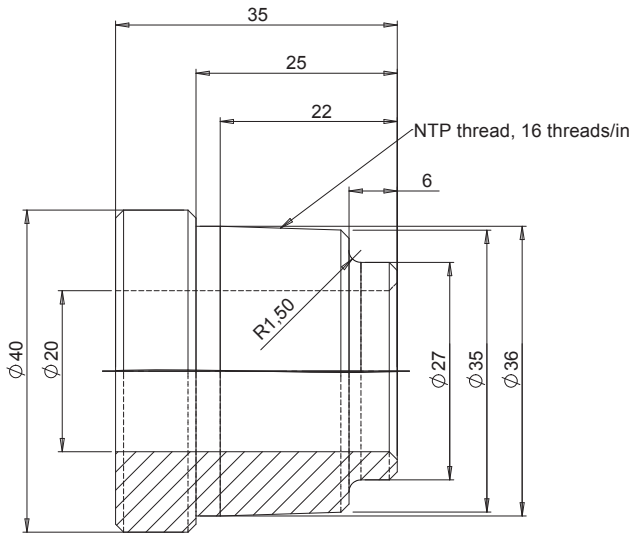
Cut side of tool: **2**

Revolution direction: **M3**

```
%
O0020 (EXAMPLE X-);
N10 G21 G40 G90 G95;
N20 T0101 (ROUGHING AND EXT. FIN);
N30 G54 G0 Z50;
N40 G96 S200;
N50 G92 S3000 M3;
N60 G0 X-46 M8;
N70 Z0;
N80 G1 X2 F.15;
N90 G0 X-45 Z2;
N100 G71 U2.5 R1;
N110 G71 P120 Q210 U-1 W.2 F.3;
N120 G0 X-15;
N130 X-20 ,C1.5;
N140 Z-6.5;
N150 G3 X-27 Z-10 R3.5;
N160 G1 X-31;
N170 G2 X-36 Z-12.5 R2.5;
N180 G1 Z-15;
N190 X-42 Z-20;
N200 Z-28;
N210 X-45;
N220 G96 S220;
N230 G41;
N240 G70 P120 Q210 F.15;
N250 G40;
N260 G0 Z50 M9;
N270 M30;
%
```

20.3- PROGRAMMING EXAMPLE WITH GANG TOOLS

Part: Connection.
Material: Brass Bar, Ø 41mm



Bevels are mot marked, 1 x 45°

LAY-OUT :

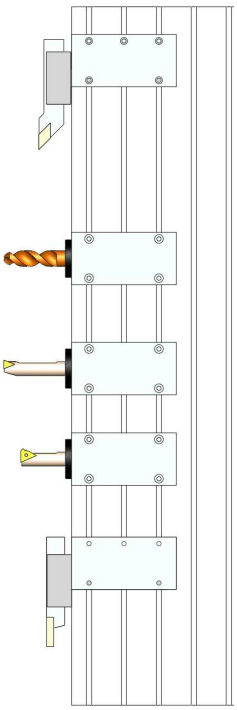
T0101 - Roughing and external finishing

T0202 - Drill Ø 18mm

T0303 - Internal finishing

T0404 - External thread

T0505 - Cut



PROGRAMMING:

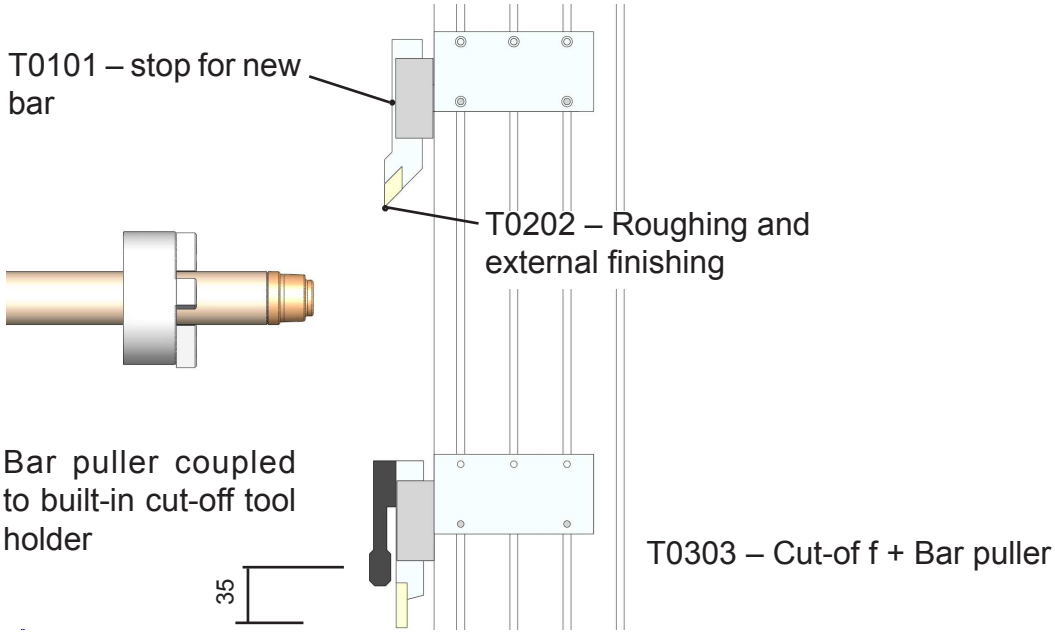
```
%
O0010 (CONNECTION – GANG TOOLS);
N10 G21 G40 G90 G95;
N20 T0101 (ROUGHING AND EXTERNAL
FINISHING);
N30 G54 G0 Z50 (SAFETY POSITION);
N40 G96 S220;
N50 G92 S3000 M4;
N60 G0 X44 M8;
N70 Z0;
N80 G1 X-2 F.15;
N90 G0 X41 Z2;
N100 G71 U2 R1;
N110 G17 P120 Q210 U1 W0.2 F.3;
N120 G0 X23;
N130 G1 Z0;
N140 X27 ,C1;
N150 Z-6 ,R2;
N160 X35 ,C1;
N170 X36 Z-22;
N180 Z-25;
N190 X40 ,C1;
N200 Z-38;
N210 X41;
N220 G96 S250;
N230 G92 S3000 M4;
N240 G42;
N250 G70 P120 Q210 F.15;
N260 G40;
N270 G0 Z50;
N280 T0202 (BROCA DIAM. 18MM);
N290 G54;
N300 G97 S1300 M3;
N310 G0 X0 M8;
N320 Z3;
N330 G74 R1;
N340 G74 Z-40 R2 Q6 F.1;
N350 G0 Z50;
N360 T0303 (INTERNAL FINISHING);
N370 G54;
N380 G96 S180;
N390 G92 S3000 M4;
N400 G0 X24 M8;
N410 Z2;
N420 G41 G1 Z0 F.15;
N430 X20 ,C1;
N440 Z-36;
N450 G40 X18;
N460 G0 Z50;
N470 T0404 (EXTERNAL NTP THREAD);
N480 G54;
N490 G97 S1600 M3;
N500 G0 X-40 M8;
N510 Z0;
N520 G76 P011060 Q100 R0.08;
N530 G76 X-33.31 Z-22 P1345 Q359
R-0.685 F1.587;
N540 G0 Z50;
N550 T0505 (BUILT-IN CUTT-OFF TOOL
HOLDER 3MM - CUTTING);
N560 G54;
N570 G96 S150;
N580 G92 S3000 M3;
N590 G0 X42 M8;
N600 Z-38.1;
N610 G1 X25 F.1;
N620 G0 X42;
N630 Z-37;
N640 G1 X40;
N650 X38 Z-38;
N660 X23;
N670 G0 X24;
N680 G97 S600;
N690 G1 X19;
N700 G0 Z50 M9;
N710 M30;
%
```

20.4 - BAR PULLER.

In case of machines not equipped with the optional bar feeder, we can program an automatic feeding by a mechanical bar puller.

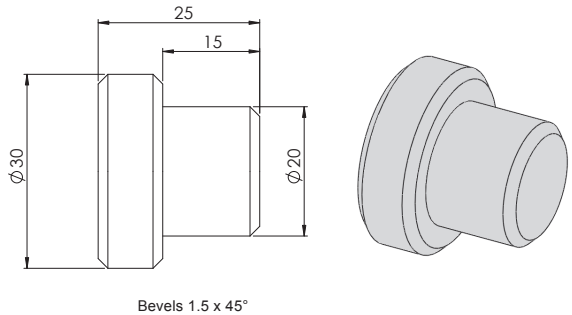
Bar puller can be at a gang position (as a tool) or be coupled at its own Built-in cut-off tool holder.

LAYOUT OF GANG WITH BAR PULLER COUPLED TO BUILT-IN CUTT-OFF TOOL HOLDER.



In case of next example, we will use the “body” of roughing tool as STOP, which shall be used in the moment of a new tool insertion.

EXAMPLE: TO PROGRAM BAR PULLER TO MAKE 10 PARTS.



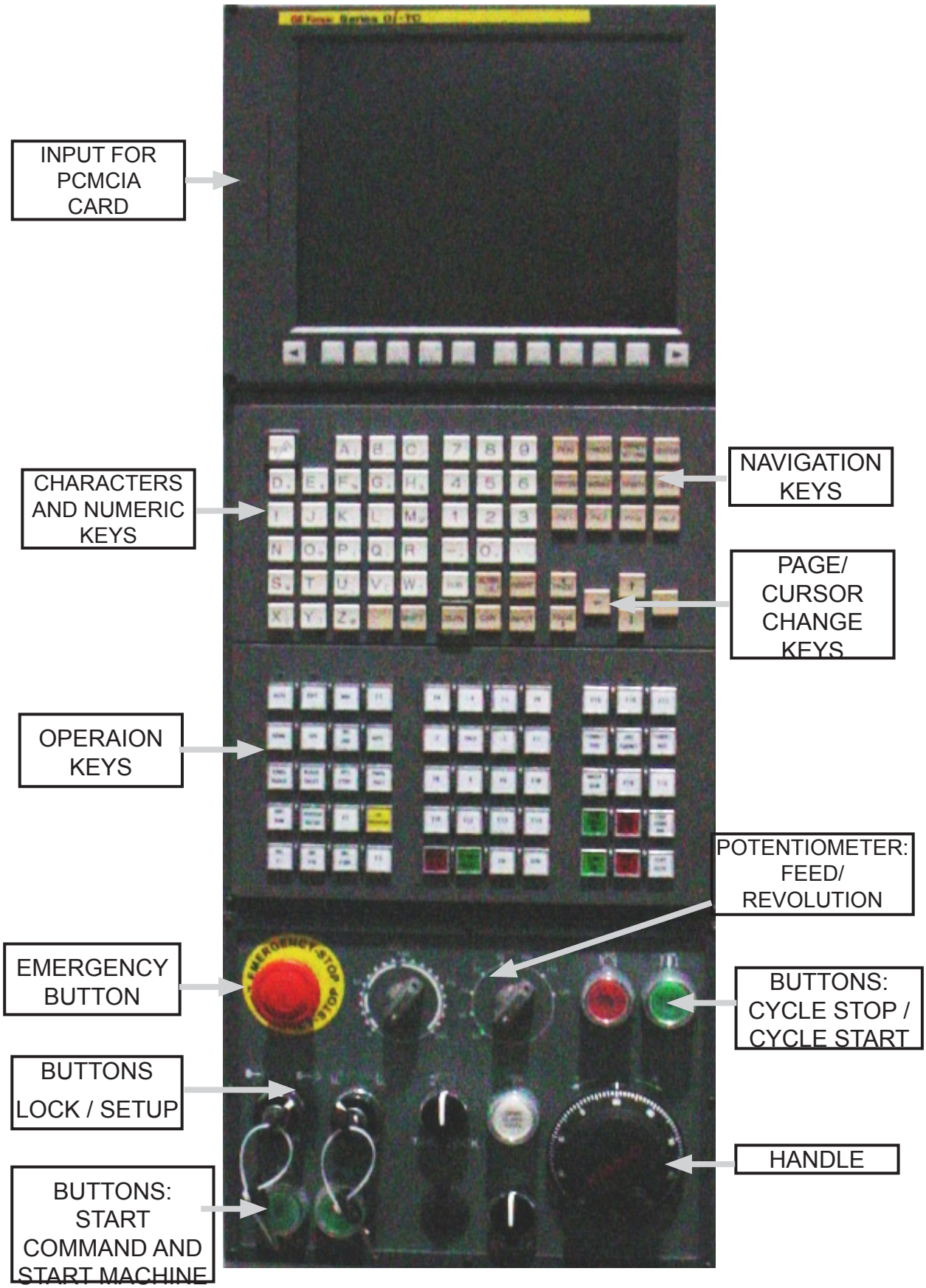
```
%
O0200(MAIN PROGRAM)
N10 G21 G40 G90 G95
N20 T0101 (STOP)
N30 G54 G0 Z60
N40 G0 X0
N50 Z0.5
N60 M00 (TO TOUCH BAR AT STOP)
N70 G54 G0 Z60
N80 M98 P0003 L10
N90 G54 G0 Z60
N100 M30
%

%
O0003(SUB-MACHINING)
N10  T0202 (ROUGHING AND EXTERNAL
FINISHING)
N20  G96 S200
N30  G92 S3000 M4
N40  G0 X40
N50  Z0
N60  G1 X-2 F0.2
N70  G0 X40 Z2
N80  G71 U3 R1
N90  G71 P100 Q160 U1 W0.2 F0.3
N100  G0 X15
N110  G1 Z0
N120  X20 ,C1.5
N130  Z-15
N140 X30 ,C1.5
N150 Z-28
N160 X38
N170 G96 S240          (FINISHING CUT-OFF SPEED)
N180 G42
N190 G70 P100 Q160 F0.2
N200 G40
N210 G54 G0 Z60
N220 T0301             (CUT-OFF AND PULLER)
N230 G96 S130
N240 G92 S3000 M3
N250 G0 X40
N260 Z-28
N270 G1 X25 F0.1
N280 G0 X32
N290 Z-26.5
N300 G1 X30
N310 X27 Z-28
N320 X8
N330 G0 X10
N340 G97 S800
N350 G1 X-2 F0.08
N360 W1                (BACK 1MM AT SHAFT “Z”)
N370 M5                (TURN REVOLUTION OFF)
N390 G94 G1 X-70 F800 (AV IN MM/MIN)
N400 M24               (OPENS THE PLATE)
N410 G4 X1             (DWELL TIME)
N420 G1 Z1.5           (DRAGS THE BAR)
N430 M25               (CLOSES THE PLATE)
N440 G4 X1
N450 G1 X40
N460 G95
N470 G54 G0 Z60
N480 M99
%
```

PART II

OPERATION

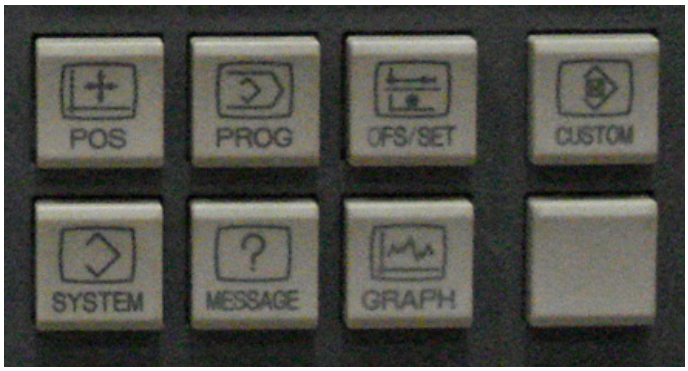
1 - COMMAND PANEL

1.1 - COMMAND PANEL – CNC'S GE FANUC 01-TD



1.2 - COMMAND PANEL - MDI UNIT

1.2.1 – Navigation keys



KEYS	FUNCTIONS
POS	**POSITION:** Access to the Page of shafts position (Relative / Absolute / Machine
PROGRAM	**PROGRAM**: Access to programming page.
OFS/SET	**OFS/SET:** Access to tools brokers screen and definitions Page.
CUSTOM	**CUSTOM:** Access to differentiating scree (Macro Romi).
SYSTEM	**SYSTEM:** Access to screen of system definitions.
MESSAGE	**MESSAGE:** Access to alarms, message and alarm history screen.
GRAPH	**GRAPH:** Access to graphic simulation screen.
HELP	**HELP:** Access to help screen, as: Tool set machine operation, MDI Keys operation, or details of na alarm which occured in CNC.
PK 1	**PK1 to PK4:** Spare function Keys for special applications
◁ ▭	**SOFTKEY:** Software Keys. Miscellaneous functions, according options shown in lower part of the screen.

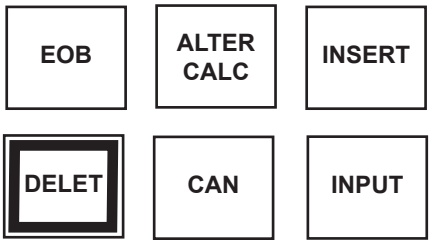
1.2.2 – Character and numeric keys

Details of character and numeric keys

KEYS	FUNCTIONS
RESET	**RESET:** This key has many functions. Among others, to zero CNC, to cancel na alarm, to interrupt a cycle program, etc.
SHIFT	**SHIFT:** This key is responsible to able the second function of alphabetical, numeric characters and other characters.
7	**LETTERS / NUMBERS / OTHER CHARACTERS:** Keys to introduce alphabetical, numerical characters and other characters,

1.2.3 – Edition Keys

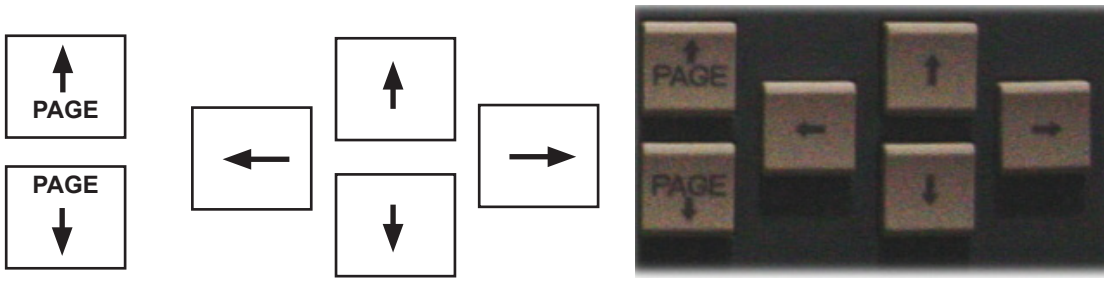
Details of edition keys



KEYS	FUNCTIONS
EOB	**EOB:** Program block end key. This key shall be pressed at the end of each program line, identifying to CNC the end of programming block.
ALTER	**ALTE CALC:** Key to change any character used in program.
INSERT	**INSERT:** Key to introduce character in program.
DELET	**DELET:** Key to delete character in program.
CAN	**CAN:** Key to delete last character or symbol, which was introduced in program. Example: when input screen shows “N10X100Z_” and CAN key is pressed, "Z" will be deleted, and "N10X100_" will be shown.
INPUT	**INPUT:** Key responsible to offset data or record, which were typed, and to be show at screen. The key [INPU] of soft has the same function of INPUT key, it produces the same result when pressed.

1.2.4 – Page / cursor change keys

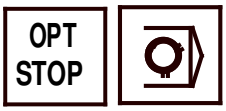
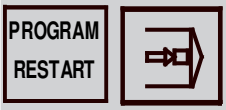
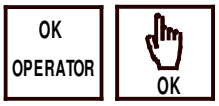
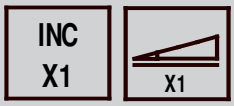
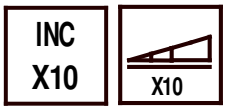
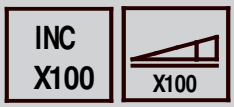
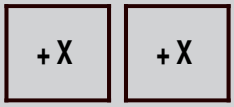
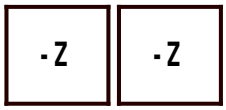
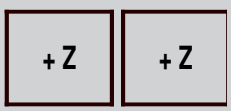
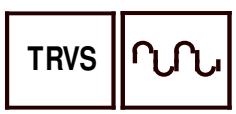
Details of Page / cursor change Keys.

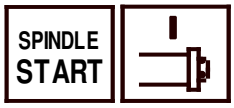
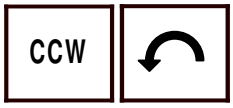
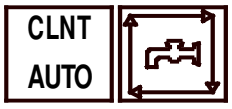
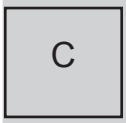


KEYS	FUNCTIONS
PAGE PAGE	**PAGE DOWN PAGE UP:** Keys responsible to change screens for next or previous Page.
	CURSOR MOVEMENT: Keys responsible to move cursor (right, left, up and down).

1.3 - OPERATION PANEL – MDI UNIT

KEYS	FUNCTIONS
AUTO	**AUTO:** AUTOMATIC MODE ENABLING KEY
EDIT	**EDIT:** Access to program edition enabling key
MDI	**MDI:** Enabling key for data input mode, allowing to insert and to execute manually one or more data blocks.
HOME	**HOME:** Enabling key for machine reference mode.
JOG	**JOG:** Enabling key for shaft movement by directional Keys (-x, +x, -z, etc).
INC JOG	**INC JOG:** Not used
MPG	**MPG:** Handle mode enabling key
SINGL BLOCK	**SINGL BLOCK:** Enabling / disabling key for block-to-block program running

KEYS	FUNCTIONS
	**BLOCK DELETE**: Enabling / disabling key for block deletion (any block preced with bar (/) is deleted).
	**OPT STOP:** Enabling / disabling key for optional stop of program.
	**PRG TEST:** Enabling / disabling key for program test with no machine movement (simulation).
	**DRY RUN:** Enabling / disabling key for program test.
	**PROGRAM RESTART:** Key with function under development.
	**OK OPERATOR:** Operator's confirmation key for messages which appears at CNC screen.
	**INC X1:** Enablig key for increasing in order of 0.001mm or 0.00001" for mpg operation (electronic handle).
	**INC X10:** Enablig key for increasing in order of 0.01mm or 0.0001" for mpg operation (electronic handle).
	**INC X100:** Enablig key for increasing in order of 0.1mm or 0.01" for mpg operation (electronic handle).
	**-X:** Directional key for shaft X movement at negative direction.
	**+X:** Directional key for shaft X movement at positive direction.
	**-Z:** Directional key for shaft Z movement at negative direction.
	**+Z:** Directional key for shaft Z movement at positive direction.
	**TRVS:** Enabling key for infeed at quick mode. In order to obtain result, it shall be pressed at the same time with directional key, from the shaft we want to move fast.

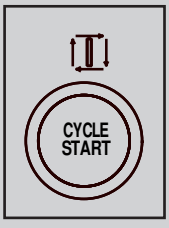
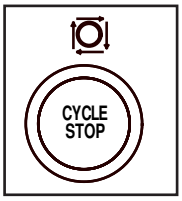
KEYS	FUNCTIONS
	**SPINDLE STOP:** Disabling key for spindle revolution (left and right).
	**SPINDLE START:** Enabling key for spindle revolution (left and right).
	**CW:** Enabling key for clockwise spindle rotation (left or right).
	**CWW:** Enabling key for counterclockwise spindle rotation (left or right).
	**CHIP CONVEYOR FW:** Enablig key for chip conveyor movement.
	**CHIP CONVEYOR STOP:** Enabling key for chip conveyor movement.
	**CHIP CONVEYOR VER:** Enabling key for reverse movement of chip conveyor.
	**CLNT ON:** Enabling key for coolant system.
	**CLNT OFF:** Disabling key for coolant system.
	**CLNT AUTO:** Enabling key for coolant system by program.
	**FUNCTION KEYS:** Spare Keys for special executions (F1; F21)
	**DNC:** Key used to execut “ON LINE” programs, that is, to execute programs from drivers “external" to machine.
	**C:** Key which indicates C shaft movement.

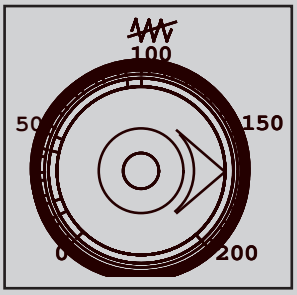
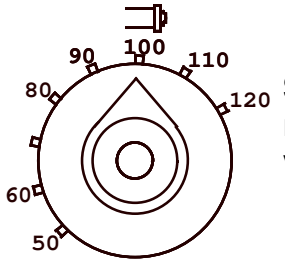
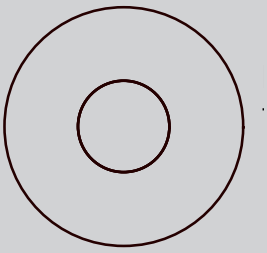
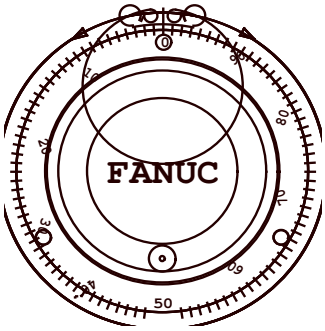
KEYS	FUNCTIONS
SPIN DLE	**SPINDLE:** Key used to indicate spindle movement.

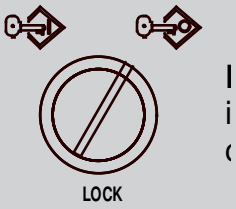
1.4 - ROMI'S OPERATION PANEL

1.4.1 CNC Ge FANUC 0i-TD

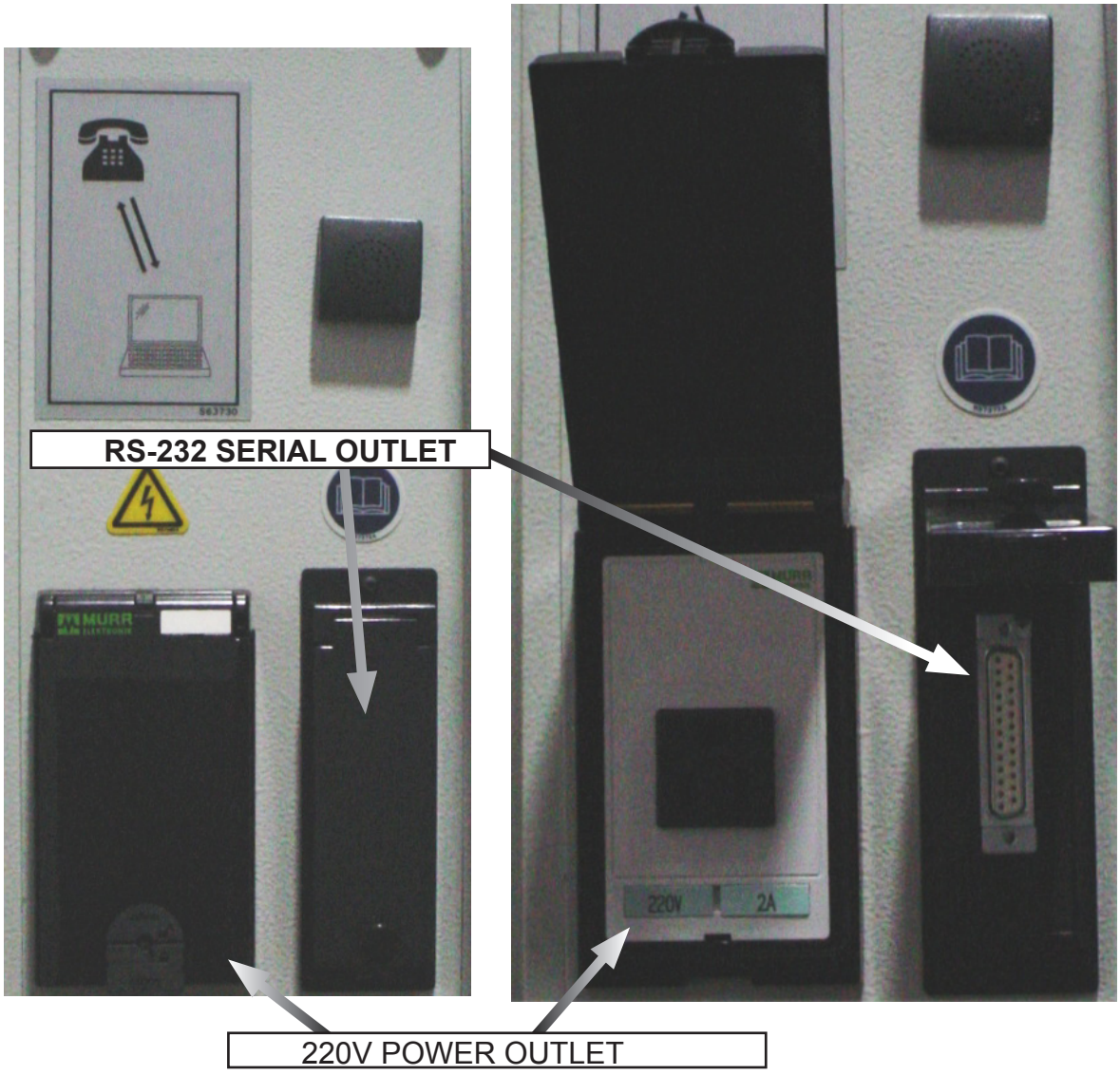
Buttons and Keys details:

KEYS	FUNCTIONS
	**CYCLE START:** Enabling Button for program running.
	**CYCLE STOP:** Disabling button for program running.

KEYS	FUNCTIONS
	**INFEED SELECTOR:** Turning key, which allows to change, in percentage, from 10% to 200% programmed infeed speed (F) and manual infeed. For JOG and quaick infeed, [G00], percentage step is 25%.
	**SPINDLE REVOLUTION SELECTOR:** Turning key, which allow to change, in percentage, from 50% to 120% spindle revolution speed (left or right).
	**EMERGENCY STOP:** Button which stops all machine functions, including shafts movement.
	**ELECTRONIC HANDLE:** Turning key which defines the direction of shaft movement. This function is possible when key MPG is enabled ony, together one of the increasing keys: x1 or x10 or x100.
	**OPEN CLOSE DOOR:** Button which opens or closes operator door.
	**SETUP:** Switching key, two (2) fixed positions, which allows work with open door, when enabled. For CE market, there are limits of shafts feed speeds and spindle turning speed (left and right). Key removal only at disabled position.

KEYS	FUNCTIONS
	**LOCK:** Switching key, 2 (two) fixed positions, which blocks program edition, when turned on. Key removal only at enabled position.
	**CHUCK:** Switching key, 2 (two) fixed positions, return to Center, for plate opening and closing.
	**CNC ON:** Button which turns CNC on.
	**MACHINE ON:** Button which turns machine on. (This Button shall only be pressed after you press CNC ON button).

1.5 - SERIAL OUTLET RS-232 AND POWER OUTLET



You must take care when connecting any instrument on power outlet. Do not connect any instrument other than those indicated for use.
Voltage is always present at the outlet when general switch is turned on.

Please certify the instrument connected to Power outlet not exceeds current released to outlet. Also certify if the instrument is adjusted for the proper voltage.

2 - INITIAL OPERATIONS

2.1 - TURN MACHINE ON

- Turno n general switch, by positioning handle at "**ON**".
- Press Button "**CNC ON**" (NC) located at machine panel (The comand will do a general checking, placing in video the message EMG ALM).
- To deactivate emergency Button.
- To close front door of the machine.
- To press Button "**MACHINE ON**"

2.2 - TO TURN MACHINE OFF

- To press emergency Button.
- To turn geneal switch off.

2.3 - TO MOVE SHAFTS IN CONTINOUS JOG

- To press "**JOG**" key.
- To press "**POS**" key.
- To press **[ABS]** softkey.
- To press Keys for shaft movement, **X+, X-, Z+ or Z-**. If you want quick displacement, please press wanted key and "TRVRS" at the same time.

OBSERVATION: *We can change shaft displacement speed by feed selector.*
*In the alarm "end of stroke" appears, the shafts shall be removed from the position of end of stroke and the key "**RESET**" shall be pressed to remove alarm.*
*In order to perform movements with open door, we shall press "**JOG**" key and place "**SETUP**" switch to work with open door.*

2.4 - TO MOVE SHAFTS USING ELECTRONIC HANDLE

- To press **MPG + "x1", "x10" or "x100"** to select speed, which corresponds to 1 millesime, 1 centesimal or 1 tenth part, respectively of each pulse generated by handle.
- To press "**POS**" key (to view movement).
- To press **[ABS]** softkey.
- To press **X+, X-, Z+** or **Z-** to select the shaft.
- To turb electronic Wheel (handle) in the wanted direction.

2.5 - TO OPERATE COMMAND BY MID (MANUAL INPUT OF DATA)

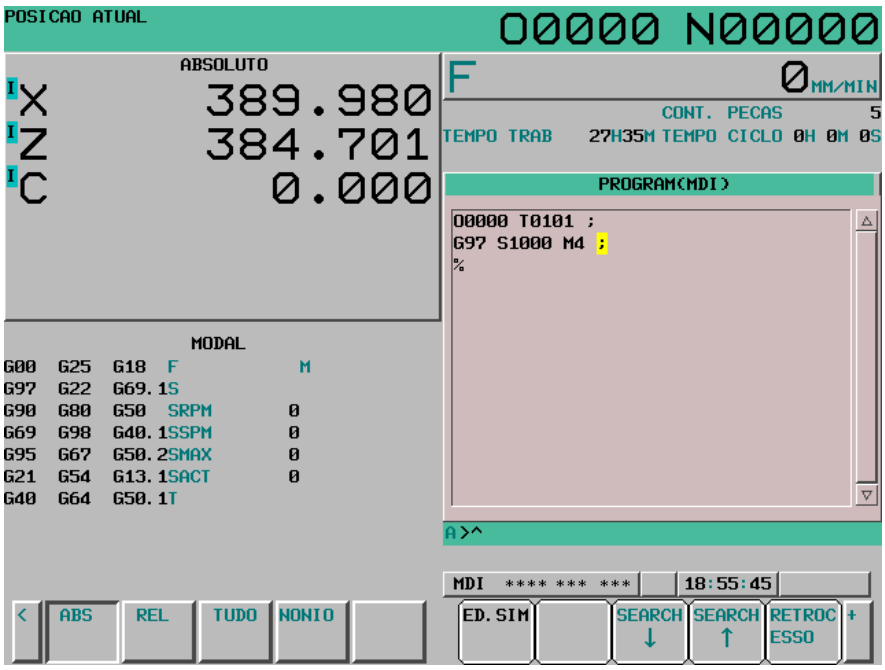
- To press "**MID**" key.
- To press "**PROG**" key.
- To press **[MD**I] softkey.
- To type instructions:
  Examples:

N10 T0101 "EOB" "INSERT" (to select tool 01).

N20 G97 S1000 M4 "EOB" "INSERT" (to turn on spindle at counterclockwise direction with 1000 RPM).

To press **"CYCLE START"** key.

OBSERVATION: *If "RESET" key is pressed, operation is cancelled.*



MDI Page

2.6 - TO MOVE SHAFTS WITH SPINDLE TURNED ON.

– Please execute operations described in items 2.6 (to turn spindle on) and 2.4 (to move shafts).

2.7 - TO MOVE SHAFTS WITH INCREMENTAL JOG

– To press "**POS**" key.
– To press **[ABS]** softkey.
– To press "**INC JOG**" softkey.
– To press the key corresponding to increasing value (in millesime of millimeter), **X1, X10, or X100.**
– To press shaft movement Keys: **X+, X-, Z+ or Z-.**

3 - PROGRAM EDITION

3.1 - TO CREATE A NEW PROGRAM

- To press “**EDIT”**.
- To press “**PROG”**.
- To press [**DIR]** softkey (to show directory screen).
- To press [**DEVICE CHANGE]** softkey.
- To press **[CNC MEM]** softkey.
- To type address “**O”**.
- To type program number. Example: **O0001.**
- To add “**INSERT”.**
- To type comment (program name) between brackets. Example: **(PECA 01).**
- To add “**EOB”**.
- To add “**INSERT”.**

3.2 - TO SELECT A PROGRAM EXISTING AT DIRECTORY

- To add “**EDIT**”.
- To add “**PROG”.**
- To press [**DIR]** softkey (to show directory screen).
- To press **[OPRT]** softkey.
- To press [**DEVICE CHANGE]** softkey.
- To press **[CNC MEM]** softkey.
- To type address “**O”**
- To type program number. Example: **O0001**
- To add [OSRH] or one cursor (←, ↑, →, or ↓).

NOTE: The program existing at the directory will appear for edition or checking.

3.3 - TO SEARCH A PROGRAM DATA

3.3.1 - To search a data by cursors (←, ↑, → or ↓)

a) Indirect search (address by address)

- To press cursors until select wanted address, being that::
  ← - To move cursor back
  → - To move cursor forward
  ↑ - To move cursor up
  ↓ - To move cursor down

b) Direct search (direct to address)

- To type wanted address. Example: “**T0505”** (to search tool 05).
- To add “↑” or “←” (if information is before last) or , “↓” or “→” (if information is after last).

3.3.2 - To search a data by “SRH” key

– to type wanted address. Example: “**S2000”** (to search information S2000).
– to add “**SRH ↑”** (if information is before last) or “**SRH ↓”** (If information is after last).

3.4 – TO INSERT DATA IN PROGRAM.

– To place cursor in a address Just before the information to be inserted.
– To type the address to be inserted. Example: **X**
– To type numeric data. Example: **10.**
– To add “**INSERT”**

EXAMPLE 1: To insert function “M8” in the block “N350 G0 X-30 Y-50;”:
– To place cursor at “Y-50”.
– To type *M8*.
– To press “*INSERT”*.
So, the block will be in the following syntaxis: “N350 G0 X-30 Y-50 M8”.

EXAMPLE 2: To insert identification “N105” in the following block “G0 X60 Y-20;”:
– To place cursor in the character of block end (“;”) fro previous block.
– To type *N105*.
– To press “*INSERT”*.
So, the block will be in the following syntaxis: “N105 G0 X60 Y-20”.

3.5 – TO CHANGE PROGRAM DATA

– To place cursor at the data to be changed.
– To place new data. Example: **X-25.**
– To add “**ALTER”**.

EXAMPLE: To change function “X-15” to “X-25” in following block: “N400 G0 X-15 Y-20;”:
– To place cursor at “X-15”.
– To type *X-25*.
– To add “**ALTER”.**
So, the block will be in the following syntax “N400 G0 X-25 Y-20”.

3.6 – TO DELETE DATA FROM PROGRAM

– To place cursor at the data to be deleted.
– To add “**DELETE”**.

3.7 – TO DELETE A PROGRAM BLOCK

– To place cursor at the beginning of the block to be deleted.
– To add “**EOB”**.
– To add “**DELETE”**.

3.8 – TO DELETE MANY BLOCKS FROM PROGRAM

– To place cursor at the first block to be deleted.
– To add softkey [►] until it shows softkey [SELECT]
– To add softkey ***[SELECT]***.
– To place cursor at the last block to be erased.
– To add softkey ***[CUT]***.

3.9 – TO DELETE A PROGRAM

– To add "**EDIT"**.
– To add "**PROG"**.
– To add softkey **[DIR]** (to show the listo f existing programs).
– To add softkey [**OPRT]**
– To add softkey [**DEVICE CHANGE]**
– To add sofkey **[CNC MEM]**
– To type address "**O**" and the number of the program to be deleted. Example: **O0001.**
– To add "**DELETE"**
– To add softkey ***[EXEC]***.

OBSERVATION*: This procedure shall be used with extreme care, because once deleted, a program shall not be recovered from machine's memory.*

3.10 - TO DELETE ALL PROGRAMS

– To add "**EDIT"**.
– To add "**PROG"**.
– To add softkey **[DIR]** (to show the list of existing programs).
– To add softkey **[OPRT]**
– To add softkey [**DEVICE CHANGE]**
– To add Softkey **[CNC MEM]**
– To add "**DELETE"**.
– To add softkey ***[EXEC]***.

OBSERVATION: *This procedure shall be used with extreme care, because once deleted, the programs shall not be recovered from machine's memory.*

3.11 - TO RENUMBER A PROGRAM

– To select the program to be renumbered.
– To place cursor at letter "**O"** (at the beginning of the program).
– To type address "**O**" and new number of the program. Example: **O1000**.
– To add **"ALTER".**

4 - DATA COMMUNICATION

4.1 - SPECIFICATION OF COMMUNICATION PORT

With the command FANUC 0i-TD, it is possible to communicate by two ports: Serial (RS 232) and PCMCIA. In order to specify which will be the communication port, the procedure below shall be done:

- To press "**MDI"** key.
- To press **"OFS/SET"** key.
- To press softkey [**SETTING]**.
- To place cursor at "**I/O CHANNEL"**.
- To type the number of the communication port, that is, 0, 1, 2 or 3 for serial communication (RS 232) or to type 4 for communication by PCMCIA.
- To press "**INPUT**" key.

4.2 - COMMUNICATION BY SERIAL PORT (RS 232)

Serial communication is that done between machine and peripheral (computer, drill, collector, etc.) by a serial port.

In order to do this, it is necessary to use a cable (please see "SPECIFICATION OF COMMUNICATION CABLE – Item 4.2.2), and, if the peripheral is a computer, a communication software.

There are many communication softwares, so, in this item only the setting and procedures related to machine will be described. For more details on communication software, please consult manufacturers.

4.2.1 - TO SET COMMUNICATION PARAMETERS

- To press "MDI" key.
- To press "SYSTEM" key.
- To press softkey [►] until it shows [ALL I/O]
- To press softkey [ALL I/O]
- To set transmission parameters according necessity.

Example:

I/O CHANNEL	1
DEVICE NUM.	0
BAUDRATE	9600
INPUT CODE	EIA/ISO
PUNCH CODE	ISO
STOP BIT	2BIT
FEED OUTPUT	NO FEED
NULL INPUT (EIA)	NO
EOB OUTPUT	LF
TV CHECK	OFF
TV CHECK (NOTES)	OFF

Setting Page of Data Transmission

OBSERVATION: *Computer and CNC shall be set in same way.*

NOTE: *With commands FANUC 0i-TD, transmission parameters: "DATA BITS" or "WORD SIZE" and "PARITY" are already set as: "7" and "PAR" (or "EVEN"), respectively.*

4.2.2 - Cable specification

The computer or external peripheral, which will communicate shall have a free serial port type DB 9 or DB 35. The connector type is not relevant, since perfectly fastened, with no risk of bad contact. The cable for connection shall obey the following configuration:

DB25 (SOCKET)		DB25 (PLUG)
1	SHIELD	1
2	TXD	3
3	RXD	2
4	DTR	5
5	GND	4
6	DSR	20
20	RTS	6
7	CTS	7

DB9 (SOCKET)		DB25 (PLUG)
1	SHIELD	1
2	RXD	2
3	TXD	3
4	DTR	6
5	GND	7
6	DSR	20
7	RTS	5
8	CTS	4

4.2.3 - To save a program

- To prepare peripheral (micro, drill, etc.) to receive data.
- To press “**EDIT**” key.
- To press “**PROG**” key.
- To press softkey [**DIR**].
- To type “**O**” and the number of the program.
- To press softkey **[F OUTPUT].**
- To press softkey **[EXECUTE]**.

OBSERVATION: *To save all programs from directory, we shall replace to type 0-999, instead one number of program.*

4.2.4 - To load a program

- To press “**EDIT**” key.
- To press “**PROG**” key.
- To press softkey [**DIR**].
- To type **“O”** and the number of new program to be stored.
- To press softkey [**F INPUT**].
- To press softkey [**EXECUTE**], (LSK will appear).
- To activate the peripheral (computer, reader, etc).

4.2.5 - To save tool correctors.

- To activate peripheral (computer, drill, etc).
- To press key “**EDIT**”.
- To press key “**OFS/SET**”(until you view [**OFFSET**])
- To press softkey [**OFFSET**].
- To press softkey [**OPRT**].
- To press softkey [**F OUTPUT**].
- To press softkey [**EXEC**].

4.2.6 - To load tools correctors

- To press key “**EDIT**”.
- To press key “**OFS/SET**”(until you view [OFFSET])
- To press softkey [**OFFSET**].
- To press softkey [**OPRT**].
- To press softkey [**F INPUT**].
- To press softkey [**EXEC**].
- To activate peripheral (computer, reader, etc).

4.3 – COMMUNICATION BY PCMCIA PORT

The term PCMCIA means Personal Computer Memory Card International Association, and it is an electrical and mechanical structure of a data storage system.

Machines from “GL / GLM Line” have a PCMCIA port located beside vídeo, which can be used to transfer different types of data, as: programs, machine parameters, tool correctors, etc. For the communication with this port, we can use two types of cards: PCMCIA and CompactFlash.

In case of PCMCIA card, due to it is from the same technology of the port, it can be directly coupled to the machine, with no fitting system. CompactFlash, due to is from a different technology, shall only be coupled by a electrical-mechanical adapter.

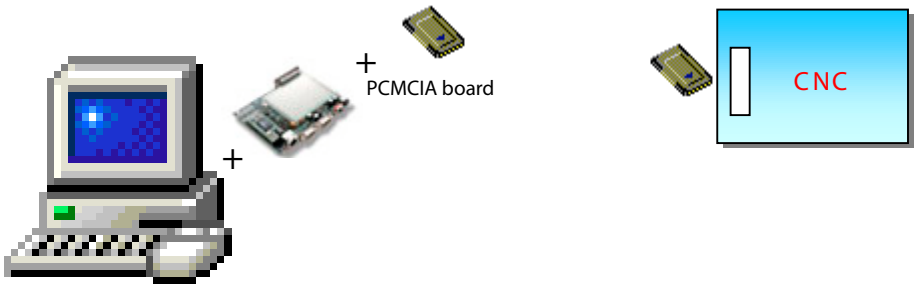
In order to read and to Record data in these cards, it is necessary to use computers equipped with respective drivers, which can be external or internal. Usually, at the use of PCMCIA cards, it is used internal drivers, and for CompactFlash, external, Which is usually connected to computer by USB port.

4.3.1 – RECOMMENDED HARDWARE FOR READING AND RECORDING:

a) PCMCIA card:

In order to read and to Record PCMCIA card, we recomend PC interface PCD-895A 00B1 KIT PCMCIA, from Advantech. In this case, PCMCIA modules shall be homologated by FANUC, if used in this CNC. This means that not all PCMCIA will work in CNC, mainly with FANUC platforms.

We also suggest the use of ATA Card, from AVED, already implemented in ROMI (AVED99604).

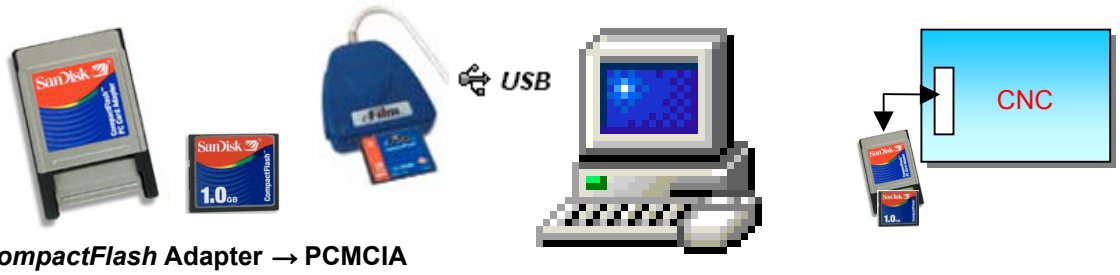


PC computer with PCMCIA interface

b) CompactFlash:

To read and to Record PCMCIA Card, we recommend interface eFilm Reader-12 USB POR T CompactFlash I/II Reader from Delkin Devices, which shall be connected at the USB port of the computer.

Please observe that, if CompactFlash is used, a CompactFlash Adapter is necessary, when connected to CNC device. This procedure is necessary, due to the own CompactFlash has not its own mechanical interface in PCMCIA standard. This adapter can be purchased at Computer Stores, but always mention you want to purchase a PCMCIA Adapter for CompactFlash Type I.



***CompactFlash* Adapter → PCMCIA**

PC Computer with USB interface and *CompactFlash*

In case of computers where USB port is not available, we can installa a USB controller mode, which is plugged at the own bus of the computer, making available the USB Port. Once this module is installed, we can configure PC according shown in figure above.

***IMPORTANT:** Due to incompatibilities between Windows and FANUC, it is necessary to format PCMCIA or CompactFlash device at the own CNC before you use it. This procedure shall be done one time only.*

4.3.2 – TO FORMAT MEMORY CARD:

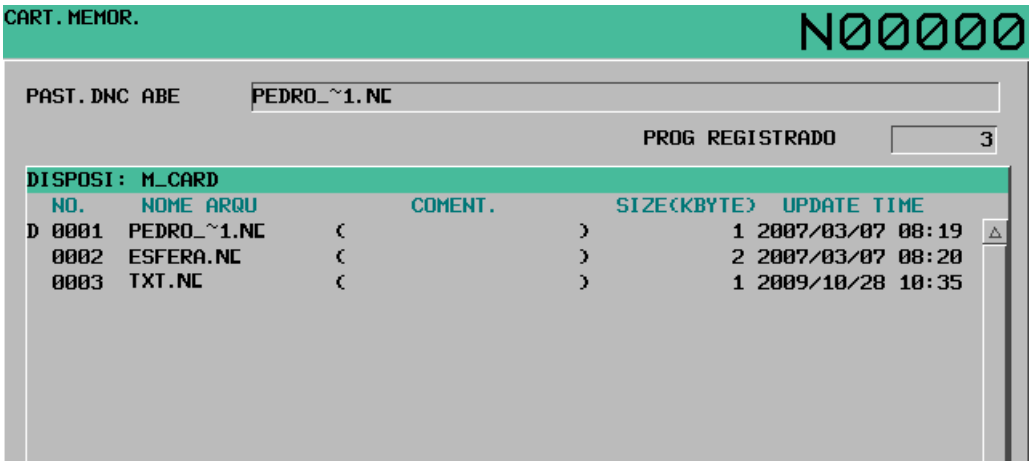
– To press key “**PROG**”
– To press key “**EDIT**”
– To press softkey [**DIR**]
– To press softkey [**OPRT**]
– To press softkey [**DEVICE CHANGE**]
– To press softkey [**MEMORY CARD**]
– To press softkey [**FDEL**]
– To type -9999
– To press softkey [**EXEC**]

NOTE: *Considering PCMCIA cards (memory cards) are sensitiv instruments, we recommend to take special cares on its handling and storage, as: to avoid shocks (fall), heat, moisture, do not disconnect during data communication, etc.*

4.3.3 - TO VIEW FILES FROM MEMORY CARD

With the command, there is a possibility to view files from memory card using the following procedure:

– To press key “**EDIT**”
– To press softkey [**DIR**]
– To press softkey [**OPRT**]
– To press softkey [**DEVICE CHANGE**]
– To press softkey [**MEMORY CARD**]



Directory Page of a Memory Card

OBSERVATION: *In case of many files in card, it shall be necessary to add Keys “PAGE UP” and “PAGE DOWN”, in order to view the other files.*

4.3.4 - To search a file

- To press key “**EDIT**”
- To press key “**PROG**”
- To press softkey [**DIR]**
- To press softkey [**OPRT**]
- To press softkey [**DEVICE CHANGE**]
- To press softkey [**MEMORY CARD**]
- To press softkey [**F SRH**]
- To type file number (left column). Ex.: **5**
- To press softkey [**F SET**]
- To press softkey [**EXEC**]

4.3.5 - To save a program at the memory card

- To set the value of communication channel = **4**
- To press key “**EDIT**”
- To press key “**PROG**”
- To press softkey [**DIR**]
- To press softkey [**OPRT**]
- To press softkey [**DEVICE CHANGE**]
- To press softkey [**CNC MEM**]
- To press softkey [**F OUTPUT**]
- To type a name for the file. Example: **TEST**
- To press softkey [**F NAME**]
- To type the number of the program that shall be sent. Ex. **1** (for program O0001)
- To press softkey [**O SET**]
- To press softkey [**EXEC**]

4.3.6 - To load a program from memory card

a) By file numner

- To press key “**EDIT**”
- To press key “**PROG**”
- To press softkey [**DIR**]
- To press softkey [**OPRT**]
- To press softkey [**DEVICE CHANGE**]
- To press softkey [**MEMORY CARD**]
- To press softkey [**F INPUT**]
- To type the number of file that shall be loaded (left column) Ex.: **5**
- To press softkey [**F SET**]
- To type the number of the program that shall be loaded. Ex.: **1** (program O0001)
- To press softkey [**O SET**]
- To press softkey [**EXEC**]

b) By file name

- To press key “**EDIT**”
- To press key “**PROG**”
- To press softkey [**DIR**]
- To press softkey [**OPRT**]
- To press softkey [**DEVICE CHANGE**]
- To press softkey [**MEMORY CARD**]
- To press softkey [**N INPUT**]
- To type the number of file that shall be loaded (left column) Ex.: **TEST**
- To press softkey [**F NAME**]
- To type the number of the program that shall be loaded. Ex.: **1** (program O0001)
- To press softkey [**O SET**]
- To press softkey [**EXEC**]

4.3.7 - To delete a file form memory card

- To press key “**EDIT**”
- To press key “**PROG**”
- To press softkey [**DIR**]
- To press softkey [**OPRT**]
- To press softkey [**DEVICE CHANGE**]
- To press softkey [**MEMORY CARD**]
- To press softkey [**F DEL**]
- To type the number of file that shall be deleted (left column) Ex.: **5**
- To press softkey [**F SET**]

5 - TEST OF PROGRAMS

5.1 - TO TEST PROGRAMS WITH NO PLATE TURNING AND NO SHAFT MOVEMENT

5.1.1 - Quick test

The purpose of this test is to check where syntaxis errors of program are (if any). To do this, you shall:

- To select program (item 3.2).
- To press key "**AUTO**"
- To press key "**PROG TEST**"
- To press key "**RESET**"
- To press key "**CYCLE START**".

OBSERVATIONS:

• To correct program, please press "EDIT", so do the desired correction. To test it again, Just repeat the procedure described above.

• When simulation finishes, Button "PROG TEST" shall be deactivated.

5.1.2 - Graphic test

The purpose of this test is to check if part profile is correct, because by it we can observe the whole stroke the tool will develop during that machining. In order to do this test:

- To select program (item 3.2).
- To press key "**RESET**"
- To press key "**GRAPH**".

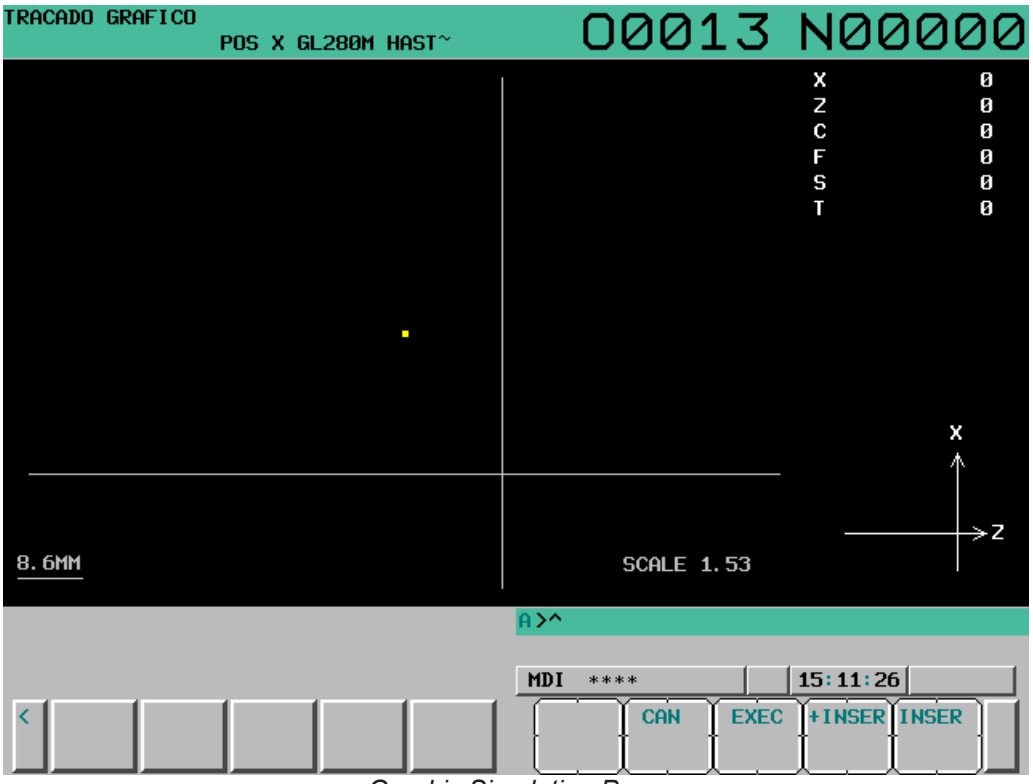
***OBSERVATION**: : If it is the first graphic simulation of the part, we shall inform the length and diameter values, in order to the command calculate graphic sclae, allowing a better viewing of it. To do this:*

- To press sofkey [**PARAMETER**]
- To fill the values "part length" and "Part diameter" in the fields "**WORK LENGTH**" and **"WORK DIAMETER"**, respectively. It is also necessary to add the key "INPUT" to introduce data.
- To press sofkey [◄]
- To press sofkey [**GRAPH**]
- To press key "**AUTO**"
- To press key "**PROG TEST**"
- To press key "**RESET**"
- To press key "**CYCLE START**".

NOTES:

1- If you want to enlarge the profile

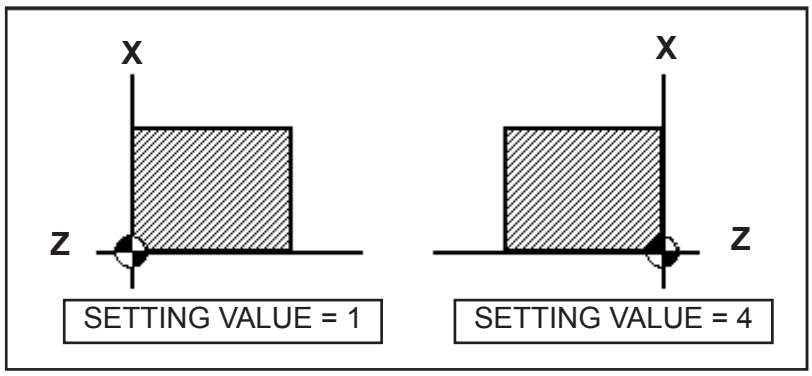
- To add softkey [**ZOOM**]
- To add softkey [**CENTER**]
- To approach cursor to part Center.
- To type the value to be enlarged. Ex.: **1**
- To add softkey [**+INPUT]** to enlarge the scale
- To add softkey [**EXEC**]
- To add tkey "**CYCLE START**"



Graphic Simulation Page

2- To change quadrant at graphic screen:

To change parameter "6510" to 1 or 4, being, bottom or face of the part, respectively. To change parameters, please consult chapter 14).



IMPORTANT: *When determining graphic simulation, Button "PROG TEST" shall be deactivated.*

5.2 - TO TEST PROGRAM (“DRY RUN”)

In mode “DRY RUN”, we can test all shaft displacements with feed more than programmed (5000 mm/min), a way to eliminate possible collisions or unnecessary displacements during program running in real time;

In order to use it, it is necessary:

- To select program (item 3.2).
- To press key “**PROG**”
- To press key “**RESET**”
- To press key “**AUTO**”
- To press Softkey **[ALL]** (To view all coordinates and distances to be displaced).
- To press key “**DRY RUN**”
- To press key “**SINGL BLOCK**” (To execute in block-to-block mode)
- To press key “**CYCLE START**” (At each touch, the command shall execute one programming line).

IMPORTANT: *Please submit the program under “DRY RUN” test with no part in the plate and only after zero tools and zero-part is defined.*
In this mode, shafts displacement, turret indexing and spindle turn occurs.

5.3 - TO INSERT BAR CODE (/) BEFORE FUNCTIONS M3 AND M4:

Application: To perform tests with no turn..

We use the function Barr (/) at the beginning of the blocks which contains commands “M3” and “M4”, in order to test with no turn to check the existence of casual interferences, collision problems, wrong position of tool”change, etc...

Beyond to insert the character “/” in program, it is necessary to execute the option BLOCK DELET. If this option is not selected, the command shall execute normally all blocks.

To test with no turn, it is necessary:

- To insert bar “/” function before blocks which contains command M3 or M4.
- To press key “**AUTO**”.
- To press key “**BLOCK DELET**”.
- To press key “**SINGLE BLOCK**”.
- To press key “**CYCLE START**”.

Example of program with function “/” (Bar):

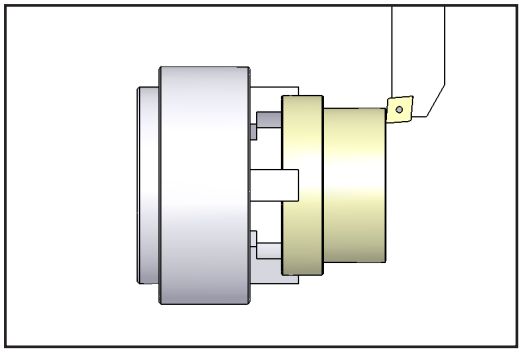
```
:
N30 T0101 (External Roughing);
N40 G54;
N50 G96 S200;
N60  / G92 S2500 M4;
:
```

6 - TOOL SETTING

Tool setting is a practical process, which purpose is to specify for the machine which are the lenghts of the tools. In the case of this modelo f machine, it is considered as preset the distance from the maximum machine coordinate up to the zero-part point.

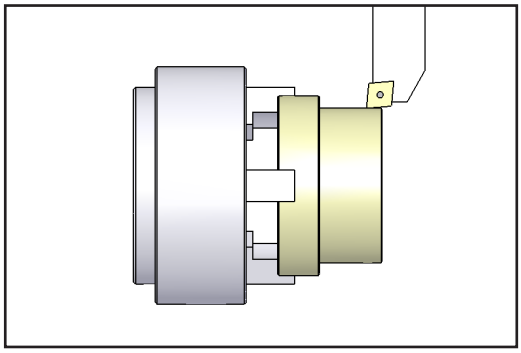
6.1 - MANUAL SETTING OF TOOLS

6.1.1 - Setting at shaft “Z”.



– Manually, plase touch tool at the face of material which will be used as reference.
– To press key “**OFS/SET**”.
– To press softkey [**OFFSET**].
– To press softkey [**GEOMETRY**].
– To position cursor at the column of shaft “**Z**” in the corresponding tool number using the Keys ↓ and ↑.
– To type “**Z0**”.
– To press softkey [**MEASUR**].

6.1.2 - Setting at shaft “X”.



– To measure the diameter of the part which will be used as reference
– Manually, plase touch tool at the measured diameter.
– To press key “**OFS/SET**”.
– To press softkey [**OFFSET**].
– To press softkey [**GEOMETRY**].
– To position cursor at the column “**X**” in the line of the tool which will be referenced.
– To type “**X**” and measured diameter. Example: **X50**.
– To press softkey [**MEASUR**] (value will be recorded).

6.1.3 - Tool Radius and Quadrant

After “X” and “Z” zeroing, the values of RADIUS and QUADRANT of the tools shall be informed, corresponding to fields “R” and “F”, respectively. For this:

– To press key “**OFS/SET**”.
– To press softkey [**OFFSET**].
– To press softkey [**GEOMETRY**].
– To place cursor at columns “**R**” and “**F**” and in the line corresponding to tool number.
– To type radius value or cut-off side of the tool, according item 6.4 in the programming parto f this manual.
– To press key “**INPUT**”.

7 - JAWS TURNING

This process is important when we want to fix a part and concentricity is necessary, due to it has already got some transformation process, or to obtain a proper fixing.

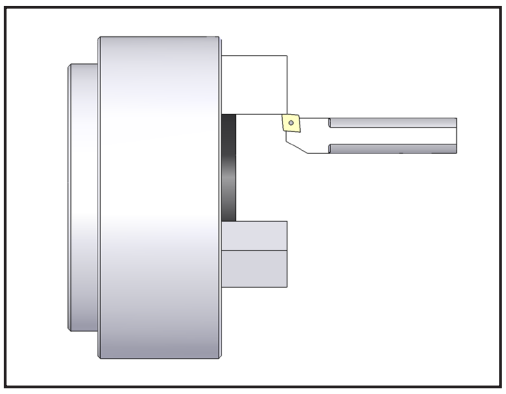
It is characterized by using a turnable jaw set (not hardened), which is prepared in a way that, with a small removal of material, shall be shaped according the diameter to be defined.

7.1 - HOW TO MACHINE JAWS

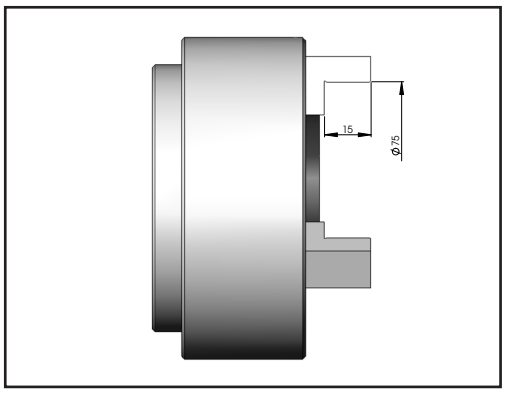
The metal excess to be removed shall be enough to establish a back for the part to be fixed. In example below, we can observe the preliminary diameter in 60.50 mm and shall be turned in 75.5mm x 15 mm. So, we shall have a wall in 7.5mm for back or limit.

For jaw machining, please follow the instructions according the drawing below.

Before machining



After machining



7.1.1 – MANUAL MACHINING

For jaw turning using electronic handle, we shall follow the following procedure:

- To índex turret, positioning the necessary internal tool:
  - To press key **"JOG"**
  - To press, at the same time, the keys "**JOG TURRET**" and "**TURRET POS**" or the keys "**JOG TURRET**" and "**TURRET NEG**" up to the turret be placed at the desired tool.

- To touch the tool at the jaw face:
  - To press key "**MPG**"
  - To press key "**x1**", "**x10**" or "**x100**"
  - To select the desired shaft (X or Z).
  - To turn handle up to it touch jaw face.

- To zero the relative coordinate from shaft Z::
  - To press key "**POS**"
  - To press sofkey [**RELATIVE**]
  - To type **Z0**
  - To press sofkey [**PRESET**]

- To return the tool:
  - To press key "**MPG**"
  - To press key "**x1**", "**x10**" or "**x100**"
  - To turn the handle, returning the tool from jaw

- To turn spindle on:
  - To press key "**MDI**"
  - To press key "**PROG**"
  - To press sofkey [**MDI**]
  - To type functions referring to the necessary revolution. Ex.: **G97 S800 M4**
  - To press key "**EOB**"
  - To press key "**CYCLE START**"

- To make a turning pass in jaws and return the tool only at shaft Z, that is, with no displacement of shaft X:
  - To press key "**MPG**"
  - To press key "**x1**", "**x10**" or "**x100**"
  - To select the necessary shaft
  - To turn the handle, approaching tool from jaw, lefting a clearance at shaft Z and at the position of the first passo n shaft X.
  - To turn the handle in the direction "**Z-**", turning the jaw up to a depth enough to measure jaw diameter.
  - To turn handle only in the direction "**Z+**" up to it gets out from jaw.

- To stop spindle:
    - To press key **“RESET”**

- To adjust the relative coordinate of shaft X:
    - To press key “**POS**”
    - To press sofkey [**RELATIVE**]
    - To measure the current diameter of the jaw. Example: **68 mm**.
    - To type X and measured diameter. Example: **X68**
    - To press sofkey [**PRESET**]

- To turn spindle again:
    - To press key “**MDI**”
    - To press key “**PRO**G”
    - To press sofkey [**MDI**]
    - To type functions referring to the revolution value. Example: **G97 S800 M4**
    - To press key “**EOB**”
    - To press key “**CYCLE START**”

- To access Page of relative coordinates
    - To press key “**POS**”
    - To press sofkey **[RELATIVE**]

- To turn jaws manually
    - To press key “**MPG**”
    - To press key “**x1**”, “**x10**” or **“x10**0”
    - To select the shaft (**X** or **Z**)
    - To turn the handle, turning jaws up to wanted dimensions

7.1.2 – To machine by program

- To press key “**MDI**”
- To press key “**PROG**”
- To press sofkey [**MDI**]
- To type the number of the tool. Example: **T0505.**
- To press key “**EOB**”
- To press key “**INSERT**”
- To press key “**CYCLE START**”
- To press key “**MPG**”
- To press key “**x10**”
- To press key “**X**” or “**Z**”
- By the handle, to touch tool at jaw face;
- To press key “**OFS/SET**”
- To press sofkey [►], until softkey [**WORK SHIFT**] appears.
- To press softkey [**WORK SHIFT**]
- To place the cursor at the Field “**Z**” in the column “**MEASURMENT**” and to type “**0**”
- To press key “**INPUT**”
- To place the tool at the start diameter of the jaw and press “**POS**”.
- To press sofkey [**ABS**]
- To note the value of shaft “**X**”
- To make the program (according example below).
- To test and execute the program
-

Program for jaws turning (example)

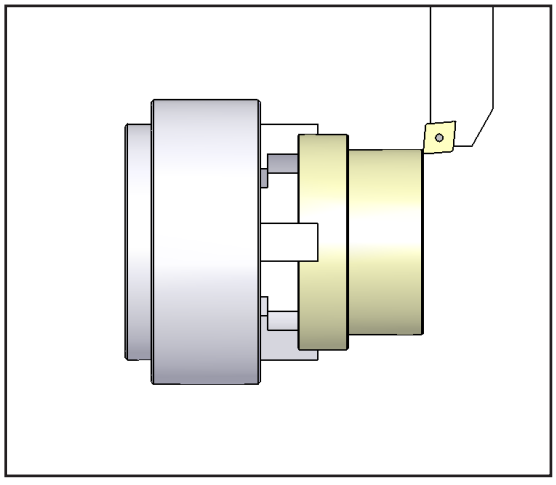
```
O0500 (JAWS TURNING);
G21 G40 G90 G95;
G54 G00 X350 Z250 T00;
T0505 (INTERNAL ROUGHING);
G54;
G96 S120;
G92 S1000 M03;
G00 X59 Z2;
G74 X75 Z-14.9 P2000 Q17000 R1 F.2;
G00 X77.5 Z2;
G01 Z0 F.16;
X75.5 Z-1;
Z-13;
X76.1 Z-15.;
X59;
G00 Z2;
G54 G00 X350 Z250 T00;
M30;
```

8 - DEFINITION OF WORK PIECE COORDINATE

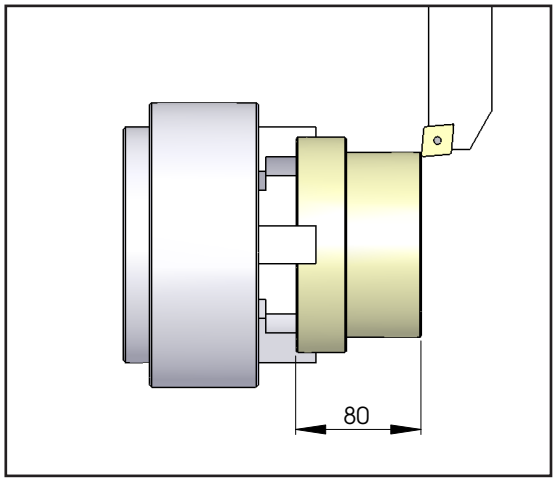
8.1 - USING “WORK SHIFT”

In order to define zero-part using “WORK SHIFT”, we shall obey the procedure below:

- To press key “**MDI**”
- To press key “**PROG**”
- To press sofkey [**MDI**]
- To type “T” and the number of tool used in this proces. Example: **T0404**.
- To press key “**EOB**”
- To press key “**INSERT**”
- To press key “**CYCLE START**”
- To add “**MPG**” + “**x1**” or “**x10**” or “**x100**”.
- To select the shaft by the keys “**X+**”, “**X-**”, “**Z+**” or “**Z-**” and to turn handle up to it touchs the tool at part face.
- To activate Page “**OFS/SET**”
- To press sofkey [►], until softkey [**WORK SHIFT**] appears.
- To press softkey [**WORK SHIFT**]
- To place cursor at Field “**Z**” at column “**MEDIDA**” (right of the screen).
- To type “**0**” (for zero-part in face) or part length (for zero-part in bottom, for example 80 mm)
- To press key “**INPUT”** – CNC shall calculate and define automatically the “**WORK SHIFT**” in field “**Z**” (left), so, the new “**ZERO PART**” is defined.



Zero-part at Face = 0



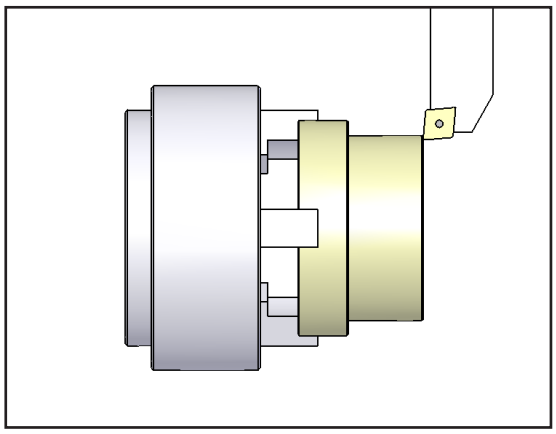
Zero-part at bottom = 80

***OBSERVATION:** For this procedure, we can use any tool, since previously referenced (preseted).*

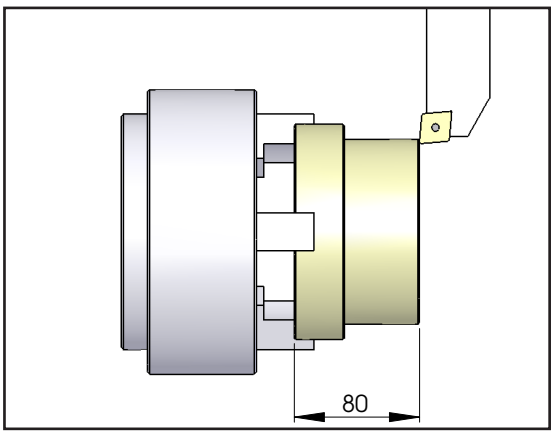
8.2 - USING A WORK COORDINATE SYSTEM (G54 TO G59)

In order to define zero-part using “WORK COORDINATE SYSTEM” (G54 to G59), we shall follow the procedure below:

- To press key “**MDI**”
- To press key “**PROG**”
- To press sofkey [**MDI**]
- To type “T” and the number of tool used in this proces. Example: **T0404**.
- To type the code referring to zero-part (**G54** to **G59**). Example: **G54**.
- To press key “**EOB**”
- To press key “**INSERT**”
- To press key “**CYCLE START**”
- To add “**MPG**” + “**x1**” or “**x10**” or “**x100**”.
- To select the shaft by the keys “**X+**”, “**X-**”, “**Z+**” or “**Z-**” and to turn handle up to it touchs the tool at part face.
- To press key “**OFS/SET**”
- To press sofkey [►] until softkey [**WORK**] is shown.
- To press sofkey [**WORK**]
- To place cursor in Field “**Z**” of zero-part (**G54** to **G59**).
- To type “**Z0**” or “**Z**” and the length of the part, according figures below.
- To press softkey [**MEASUR**] - the CNC will define zero-part value.



Zero-part at Face = Z0



Zero part at Bottom = Z80

8.3 - TO MAKE CORRECTION AT WORK COORDINATE SYSTEM (G54 TO G59)

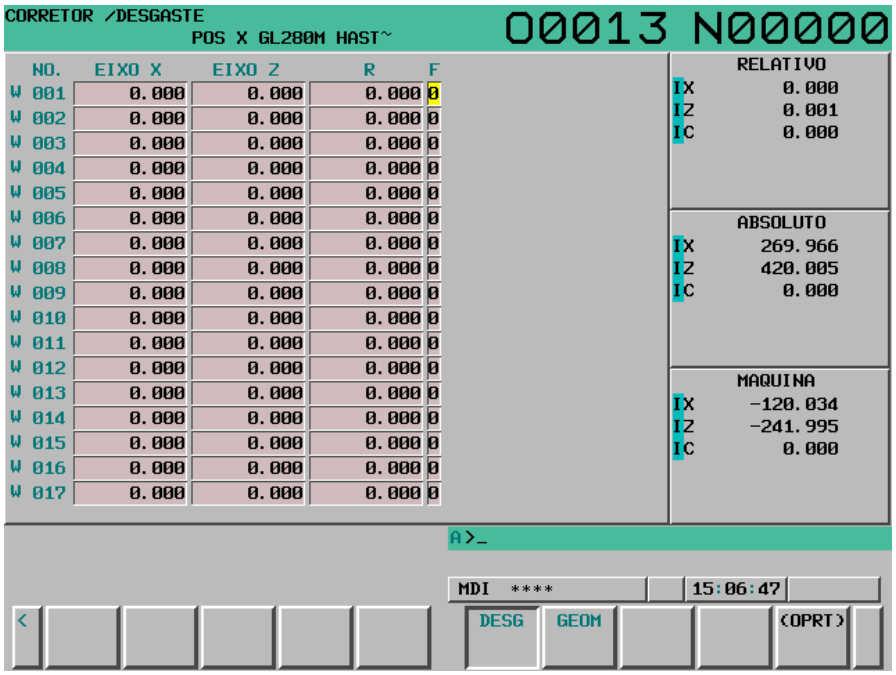
- To press key “**OFS/SET**”
- To press sofkey [**WORK**]
- To place cursor in Field “**Z**” of the zero-part (**G54** to **G59**)
- To type the correction value (**+/-**). Example: **0.5**
- To press sofkey [**+INPUT**]
- To press sofkey [**EXEC**]

9 - CORRECTION OF TOOL WEAR

9.1 - MANUAL MODE

All tool gets progressive wear when in friction with the material under removal. So, when is a tool intended for calibration, it is necessary to correct this wear, in order to keep qualitylevel of the product, in size aspect.

- To press key “**OFS/SET**”
- To press sofkey [►] until softkey [**OFFSET**] is shown.
- To press sofkey [**OFFSET**]
- To press sofkey [**WEAR**]
- To place the cursor on the tool and shaft using the Keys:↑ and ↑.
- To type the value to be corrected (**+/-**). Example: **0.05**
- To press sofkey [**+ INPUT**]
- To press sofkey [**EXEC**]



Page of Tool Wear Correction

10 - PART COUNTER

The parts counter function is activated by the code **M76**.

Example1:	Example2:
:	:
N120 M76	N120 M76
N130 M30	N130 M99

OBSERVATION: *Each time the program executes the function* ***M76****, it will sum 1 to part counter.*

10.1 - PART COUNTER VIEWER

Mode 1:

– To press key "**POS**" (The Field "**CONT. PECAS**" shall appear at the right bottom of the command.

Mode 2

– To press key "**OFS/SET**"
– To press sofkey [**SETTING**]
– To press key "**PAGE ↓**" until the following fields appear:

PART TOTAL - Represents a total number of parts to be done

PART REQUIRED - Represents a limit/goal to be reached. When this limit/goal is reached, the command sends an internal signal to its logical PMC.

PART COUNT - Represents the partial number of machined parts

10.2 - TO ZERO PARTS COUNTER

Mode 1

– To press key "**OFS/SET**"
– To press sofkey [**OPRT**]
– To press sofkey [**PTSPRE**]
– To press sofkey [**EXEC**]

The Field "CONT. PEÇAS" will be automatically zeroed.

Mode 2

– To press key "**MDI**"
– To press key "**OFS/SET**"
– To press sofkey [**SETTING**]
– To press key "**PAGE ↓**" until the Field "**CONT. PEÇAS**" appear:
– To type "**0**".
– To press key "**INPUT**"

The Field "**PART COUNT**" will be automatically zeroed.

11 - PROGRAMS RUNNING

11.1 - TO EXECUTE A PROGRAM FROM MACHINE MEMORY:

All program, after tested, shall be available for automatic running. In order to do this, we shall:

- To select program.
- To press key “**AUTO**”
- To press key “**PROGR**”
- To press key “**RESET**”
- To press sofkey [**ALL**]
- To press key “**CYCLE START**”

OBSERVATION: *If you want to execute a program step-by-step, please press the key “SINGL BLOCK”, and, for running of each block, press key “CYCLE START”.*

11.2 - TO RUN A PROGRAM DIRECT FROM MEMORY CARD

To run programs directly from card, we shall set some communication parameters (item 11.2.1), and select and run the program (item 11.2.2).

OBSERVATION: *For more details on PCMCIA card, please consult item 4.3.1.*

11.2.1 - To set communication parameters

- To press key “MDI”
- To press key “OFS/SET”
- To press sofkey [SETTING]
- To place cursor at “I/O DEVICE”
- To type 4 for communication by PCMCIA port.
- To press key “INPUT”

11.2.2 - To run program

- To press key “**AUTO**”
- To press key “**DNC**”
- To press key “**PROG**”
- To press sofkey [►]
- To press sofkey [**DIR**]
- To press sofkey [**OPRT**]
- To press softkey [**DEVICE CHANGE**]
- To press sofkey [**MEMORY CARD**]
- To type the file number (lef column). Example: 5
- To add sofkey [**DNC SET]** (The name of selected program shall appear in front of “**DNC FILE NAME**”.
- To press “**CYCLE START**” (machining shall start)

11.3 - TO ABORT A PROGRAM RUN

– To press the key **“CYCLE STOP”**
– To press the key **“RESET”**

11.4 - TO SELECT OPTIONAL STOP:

– To press the key **“OPT STOP”**

OBSERVATION:
- This function activates an optional stop defined in program by function M01.
- The operator shall select this function before progran run starts.
- To deactivate the function, Just press the key “OPT STOP” again.

11.5 - TO DELETE A PROGRAM BLOCK (“BLOCK DELETE”)

– To press the key **“BLOCK DELET”**

OBSERVAÇÕES:
- - If the option “BLOCK DELETE” is active, the command will ignore any information block preceded by the code “/” (bar).
- If the option “BLOCK DELETE” is not active, all blocks shall run, including those with the function (/).

12 - SPECIAL FUNCTIONS

12.1 - PROGRAM EDITION WITH EXTENDED FUNCTIONS

This function allows to do the following operations, using a program already in memory:

A – Total or partial copy of a program to other.
B – Inclusion of two programs in only One.
C – Change a data to other.

This function allows to copy a program, not deleting it, to prepare a program including a similar part of another program, or to invert some programming order.

12.1.1 - Total copy of a program to other:

– To select the program to be copied – according item 3.2.
– To press key "**EDIT**"
– To press key "**PROG**"
– To press sofkey [**PROGRAM**]
– To press sofkey [**OPTR**]
– To press sofkey [►]
– To press sofkey [**SELECT ALL**]
– To press sofkey [**COPY**]
– To access or to create a new program, which will receive the copy.
– To place cursor on the block where the copied data shall be inserted using the Keys: ↑ and ↓.
– To press sofkey [**PASTE**]
– To press sofkey [**BUFFER EXEC**]

12.1.2 - Partial copy of a program to other

– To select the program to be copied – according item 3.2.
– To press key "**EDIT**"
– To press key "**PROG**"
– To press sofkey [**PROGRAM**]
– To place cursor on the first block to be copied.
– To press sofkey [**OPTR**]
– To press sofkey [►]
– To press sofkey [**SELECT**]
– To place cursor on the last block to be copied using the Keys: ↑ and ↓.
– To acess or to create a new program to receive the copy.
– To press sofkey [**COPY**]
– To place cursor on the block where the copied data shall be inserted using the Keys: ↑ and ↓.
– To press sofkey [**PASTE]**
– To press sofkey [**BUFFER EXEC**]

12.1.3 - To transfer (to move) a part of a program to another program

– To select the program to be copied – according item 3.2.
– To press key “**EDIT**”
– To press key “**PROG**”
– To press sofkey [**PROGRAM**]
– To place cursor on the first block to be moved, using the Keys: ↑ and ↓.
– To press sofkey [**OPTR**]
– To press sofkey [►]
– To press sofkey [**SELECT**]
– To place the cursor on the last block to be moved.
– To press sofkey [**CUT**]
– To access or to create a new program, which will receive the copy.
– To place cursor on the block where the copied data shall be inserted using the Keys: ↑ and ↓.
– To press sofkey [**PASTE**]
– To press sofkey [**BUFFER EXEC**]

NOTE: *The part of the program is moved off of the original program.*

12.1.4 - To join two programs

– To select the program which shall receive information– according item 3.2.
– To press key “**EDIT**”
– To press key “**PROG**”
– To press sofkey [**PROGRAM**]
– To place cursor on the position where the other program shall be placed.
– To press sofkey [**OPTR**]
– To press sofkey [►]
– To press sofkey [**PASTE**]
– To type “O” and the number of the program which shall be inserted in this position. Ex. **O0010**
– To press sofkey [**SPEC PRG**]

12.1.5 - To replace data

– To select the program to be changed.
– To press key “**EDIT**”
– To press key “**PROG**”
– To press sofkey [**PROGRAM]**
– To press sofkey [**OPTR**]
– To press sofkey [►] until you find the key [**CHANGE**]
– To press sofkey [**CHANGE**]
– To type new data. Example: **X300**
– To press sofkey [**AFTER**]
– To press sofkey [**EXEC ALL**]

NOTES:
– *When pressing [**EXEC ALL**], all data after cursor are changed.*
– *When pressing [**EXEC SINGLE**], only the first data found is changed.*
– *When pressing softkey [**SKIP**], selected data is not changed.*

12.2 - BACKGROUND EDITION

This function allows to program edition with machine working.

– With the machine running, please proceed the following:
– To press key “**PROG**”
– To press sofkey [**OPRT**]
– To press sofkey [**BG EDIT**]
– To press sofkey [**EDIT EXEC**]
– To press sofkey [**DIR**]
– To type “**O**” and the number of the new program. Example: **O1000**
– To add Keys “**INSERT**”, “**EOB**”, “**INSERT**”
– To type information

OBSERVATION: *When edition is finished, it is necessary:*
– *To add softkey [OPRT]*
– *To add softkey [BG END]*

13 - BAR FEEDED

13.1 - PROGRAMMING OF BAR FEEDER FOR G / GL / GLM LINE

13.1.1 - Conditional deviation – M80

Function M80 was developed with the purpose to control the program flow from an external example, that is, this command shall make the program to jump to a specified block always the bar feeder send a signal “Bar End”.

The program will deviate the normal flow only when it receives this signal of end of bar, otherwise, the program flow shall run normally.

Syntax:

M80 A___

Where:

M80 : calls the function of conditional deviation.

A__: identifies the program block where the flow shall be deviated (N__).

Example:

```
:
M80 A500 ..... End of Bar? – If positive, go to block N500
M20
G4 X0.5
M21
:
N500 G0 Z200 ..... Returns turret for the fall of bar remain.
:
```

OBSERVATION: *Due to the argument “A” is mandatory, the program shall always be written considering the use of identifiers of block “N”.*

13.1.2 - To turn on/off bar feed

To activate and to deactivate bar feed, the following functions shall be programmed:

M20 - opens collet/plate and activates bar feeding

M21 - closes collet/plate and deactivates bar feeding

13.1.3 - Programming examples

For the use of the Bar Feeder, we shall program, beyond the functions described in previous items, some events and movements of the machine, as infeed/ return the parts catchers, bar top facing, stop position for feeding, etc. These movements change according the type of Bar Feeder used.

Below some examples of programming for machines equipped with the feeder **FEDEK**.

NOTES :

- Beyond the main program with bar feeding functions (O1000), we shall elaborate part machining program (O1001), finishing it with function M99, according the example "c) Sub-program for FEDEK".

- Examples below were developed for machines with tool actuated.

For machines without these optionals, we shall replace blocks N11, N12 and N13, referring to spindle guiding, By the two following blocks:

N11 M19.....GUIDES THE SPINDLE

N12 G64 C20.....PLACES SPINDLE AT 20º

a) Main Program for FEDEK (O1000)

```
O1000 ( MAIN )
G21 G40 G90 G95
G54 G0 Z300 T00
T0808 ( STOP )
G54
M05 ..... Turns spindle off
G0 X0
Z10
G94 G1 Z0.2 F2000 .....Positioning for bar stop (infeed in mm/min)
M80 A1000 .....end of bar? If positive, go to block N1000
M20 .....Opens ollet/plate and activates bar feeding
G4 X0.5.....Time to execute feeding
M21 .....Closes collet/plate and activates bar feeding
G4 X0.5.....Time to close collet/plate
G54 G0 Z300 T00
N10 M98 P1001 (SUB-PROG.) .....Call of machining program
M99 .....Returns to the beginning of the program
N1000 G0 Z200 .....Returns turret for fall of bar remain
N11 M19.....Activates spindle guiding
N12 G28 C0 .....Guides the spindle
N13 G0 C20 .....Positions shaft "C" at 20°
M38 .....Approaches parts catcher
M20 .....Opens collet/plate and activates bar feeding
G4 X0.5.....Tume to remove bar remain
M39 .....Returns parts catcher
G94 G1 Z1 F2000 .....Approach for stop with infeed in mm/min
M21 .....Closes collet/plate and activates bar feeding
G4 X0.5.....Time to close collet/plate
M18 .....Cancells spindle guiding
G54 G0 Z300 T00
T0303 (TO FACE NEW BAR)
G54
G95 .....Infeed in mm/rev.
G96 S150
G92 S3000 M4
G0 X48
Z-30.....Positionging to face bar
G1 X-2 F0.2 .....Facing of new bar
G54 G0 Z300 T00
M99 P10.....Returns to block N10
M30
```

c) Sub-program for FEDEK (O1001)

```
O1001 (BAR FEEDER SUB-RPOGRAM – MACHINING PROGRAM)
G21 G 40 G90 G95
G54 G0 Z300 T00
T0707 (ROUGHING)
G54
G96 S200
G92 S4000 M4
:
:
G54 G0 Z300 T00
T0202 (FINISHING)
G54
G96 S300
G92 S4000 M4
:
:
:
G54 G0 Z300 T00
T0303 (TO CUT-OFF)
G54
G96 S150
G92 S2000 M4
G0 Z40
Z-3
G1 X7 F.15
U1
M38
G97 S1000 M4
G1 X-2 F.06
M39
G0 W2
G54 G0 Z300 T00
M99
```

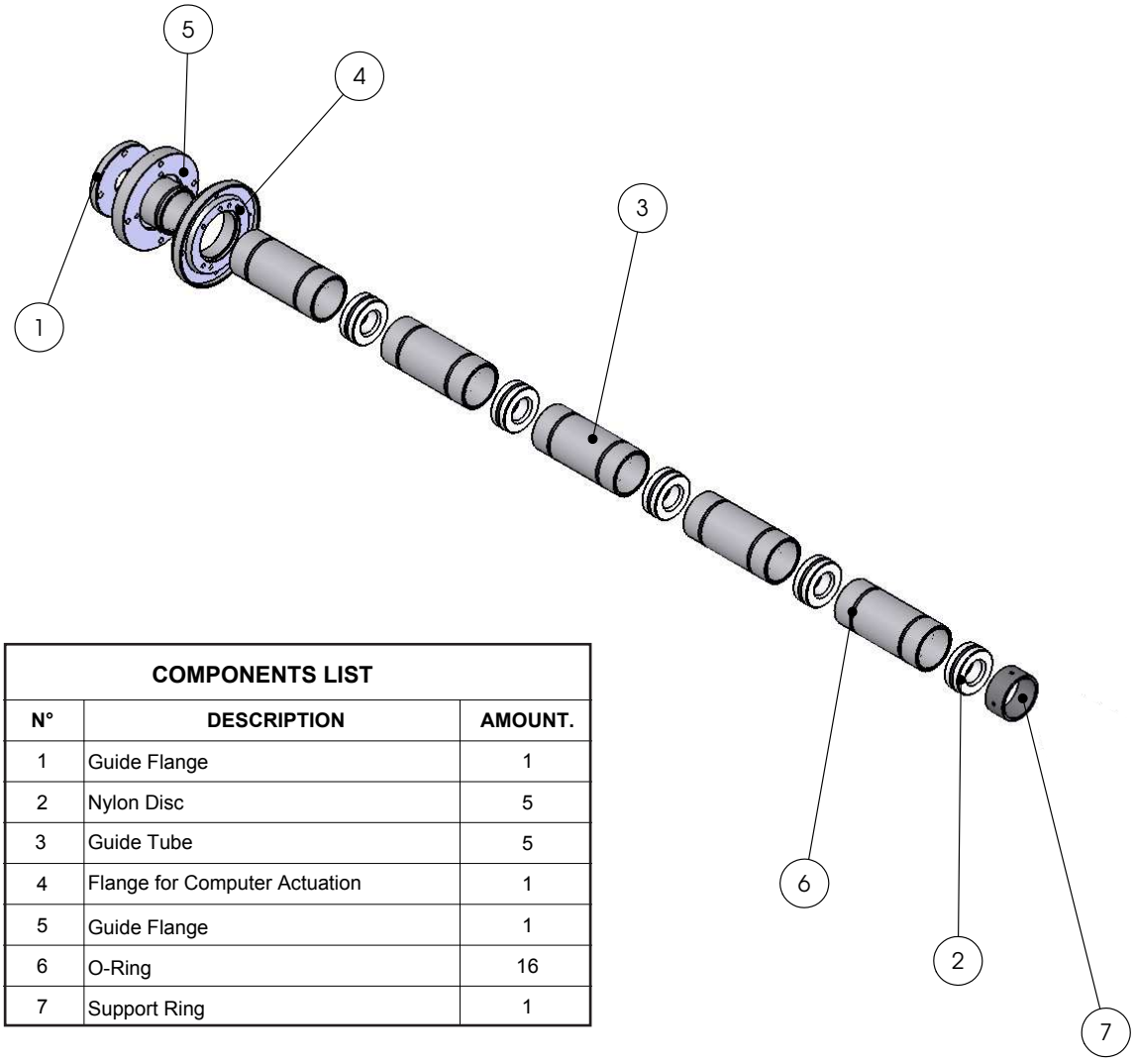
13.2 - MODULAR GUIDE TUBE SYSTEM

The Modula Guide Tube System is the system for reduction of the internal diameter of the cylinder through bars are fed, making the internal diameter of this cylinder to be close to the external diameter of the bar, in order to avoid the entrance of the bar in plate/collet and to reduce bar vibration caused by its shock at the cylinder.

There are many types of guide tubes, in the most different formats and sizes, but, in this item, we will approach the “Modular Guide Tube System for Cylinder”, which is currently sold by Romi.

13.2.1 - Components

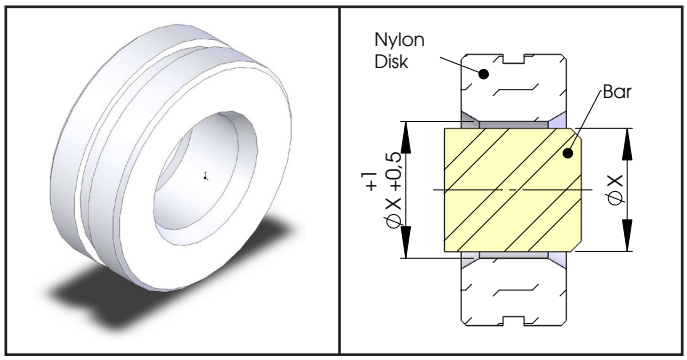
The Modular Guide Tube System for Cylinder is composed by many components, as follows:



COMPONENTS LIST		
N°	DESCRIPTION	AMOUNT.
1	Guide Flange	1
2	Nylon Disc	5
3	Guide Tube	5
4	Flange for Computer Actuation	1
5	Guide Flange	1
6	O-Ring	16
7	Support Ring	1

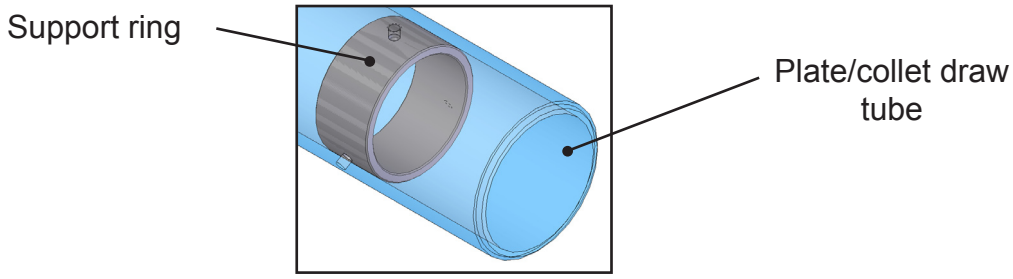
13.2.2 - Preparation of Reduction Tube

The preparation of Modular Guide Tube System is by the internal turningof Guide Flange (1) and Nylon Disks (2), lefting it around 0.5 to 1.0mm clearance in relation to the bar to be fed.



13.2.3 - Insertion / Removal of Support Ring

To insert or to remove the Support Ring from draw cylinder of plate/collet, it is necessary:
- To remove the plate or collet system from machine.
- To loose the three bolts (sets) found close to cylinder end.
- To insert or to remove Support Ring (7).
- To fast three bolts (sets) again.
- To assemble plate or collet system again.

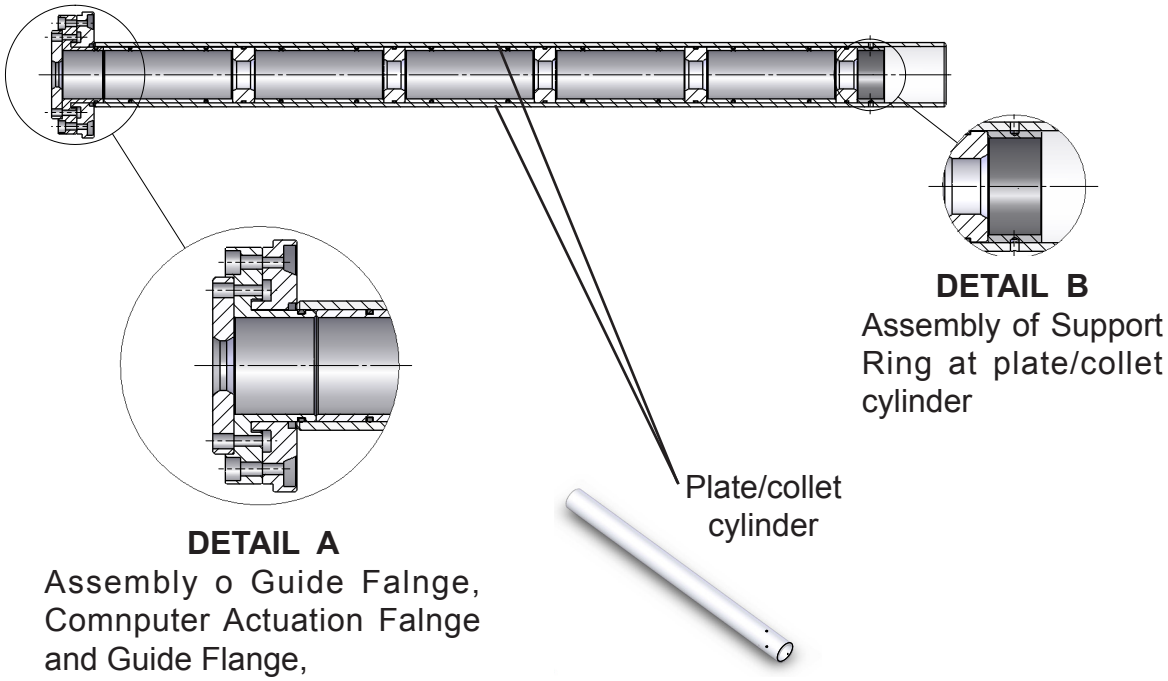


13.2.4 - Assembly of Modular Guide Tube System

The assembly of Modular Guide Tube system within plate/collet draw tube shall be done according the procedure below:

- To remove Bar Feeder from machine.
- To insert Support Ring (7), according previous procedure (14.2.3 – Insertio / Removal of Support Ring).
- To insert O-Rings (6) in Nylon Disks (2) and in Guide Tubes (3)
- To insert Nylon Disks (2) and Guide Tubes, obeying the following order: First a Nylon Disk, up to it touches the Support Ring, next a Guide Tube up to it touches the Nylon Disk, next another Nylon Disk up to it touches the first Guide tube and so on, until the Last Guide Tube touches the Last Nylon Disk.
- To thread the Computer Actuation Flange (4) at the flange of hydraulic cylinder.
- To thread Guide Falnge (5) at Computer Actuation Flange (4).
- To thread Guide Flange (1) at Guide Flange (5)

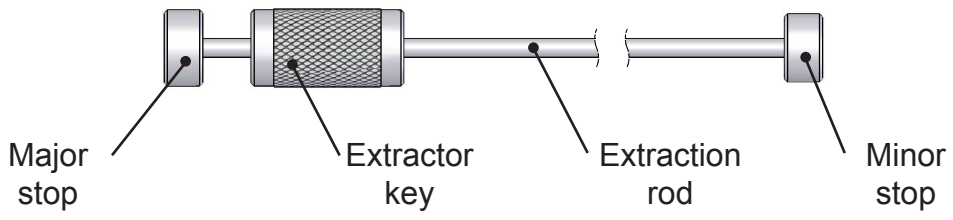
NOTE: *In order to make easier insertion and removal of reduction tube set, we recommend the use of some type of lubricant at the external parts of Guide Tube and Nylon Rings.*



13.2.5 - Removal of Reduction Tubes Set

The removal of Modular Guide Tube System, once it is inside the plate/collet draw tube, shall be done according the procedure below:

- To remove Bar Feeder
- To remove Guide Flange (1)
- To unscrew Guide Flange (5)
- To insert extractor set with no stop in less diameter up to plate / collet face.
- To fix Minor Stop at extractor set
- To hit Major Stop with Extractor Key up to remove the whole set of reduction tubes.



Extractor st

13.3 - FEDEK BAR FEEDER

In this item, some basic principles of Feder Bar Feeder will be learned. In order to obtain more detials on this equipment, the manufacturer manual shall be consulted (delivered with equipment).

13.3.1 - To turn feeder on

To turn feeder on, the procedure below shall be done:
- To turn Power on and to energize machine.
- To turn main switch of bar feeder
- To deactivate emergency buttons of the feeder.
- To press the button at remote panel, according shown in figure below

13.3.2 - Operation and use

13.3.2.1 - To work in Manual Mode

For work in manual mode it is necessary:
- To press Button in remote panel, referring to Manual Mode.
- To press movement buttons of the feeder, according description below:

KEY	DESCRIPTION
	It move guide groove forward (pusher mode)
	It move guide groove backward (pré-feed mode)
▶	It feeds pusher
◀	It returns pusher

13.3.2.2 - To Work in Automatic Mode

For the work in automatic mode, it is necessary:

- To place manually the Rod close to bar, according previous procedure.
- To press Button in remote panel, referring to Automatic Mode
- RO press Button in remote panel, to start feeding cycle..
- To run machine program.

13.3.3 - Feeder Parameters

Many movements which occur at bar feeder during the processo f bar feeding or change, as coordinates for bar remain removal, to return pusher after feeding, to signal end of bar, etc. are programmed by parameter pages, here named: Working Parameters.

a) To change Working Parameters

In order to inser values at Working Parameters Page, it is necessary:

-To add the key [**F4/F9**] (“the page **PARAMETER** shall be shown).

- When necessary, you shall insert password (**F0**, **F0**, **F0 SHIFT F5**, **SHIFT F6**, **SHIFT F5**)
- To select parameter, using the keys **PAGE UP** and **PAGE DOWN**. Example: **P1**.
- When the selected parameter presents two different fields to be filled, [**F0/F5**] shall be pressed to change the first field and [F1/F6] shall be pressed to change the second field.
- To type new value. Example: **100**.
- To press the key [**ENTER**] to confirm change.

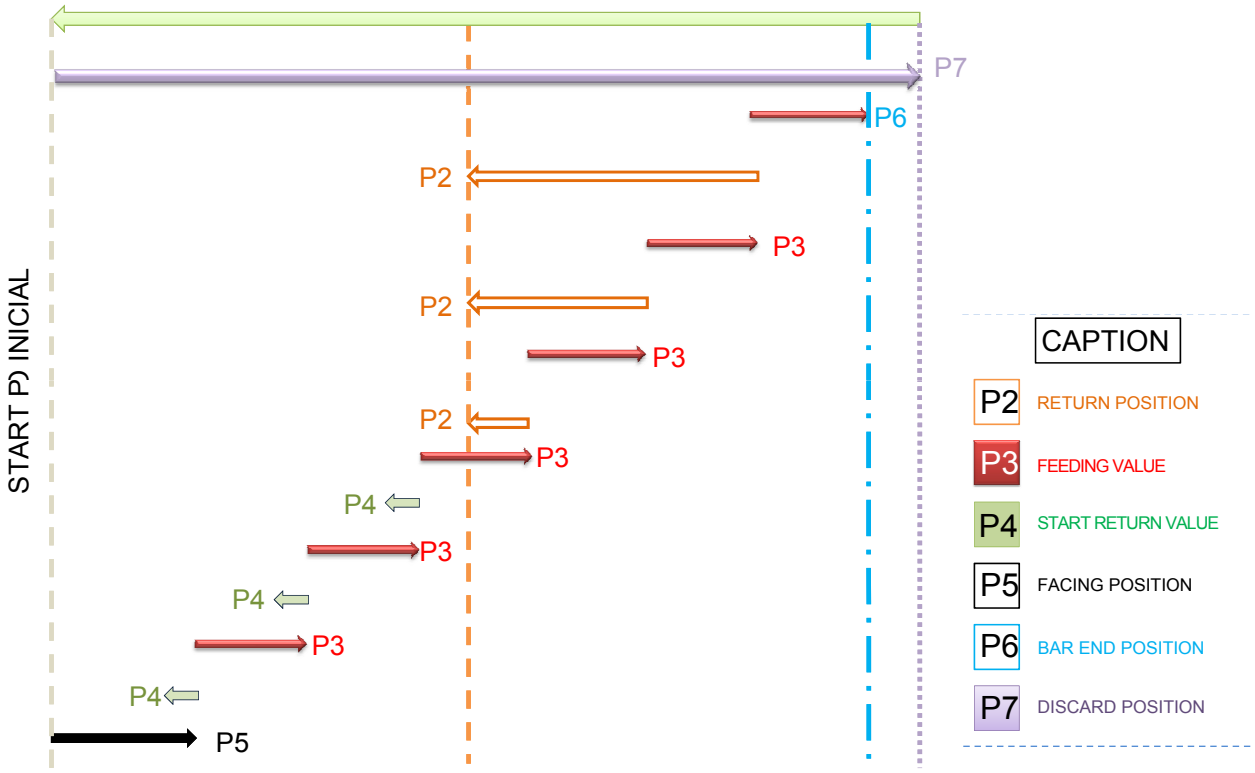
b) List of parameters Page Working

IMPORTANT: *Always when the size of the part to be machined is changed, only parameters P3 and P5 shall be changed. More parameters usually shall have fixed values, that is, they need not to be changed, not depending on the part length or bar used.*

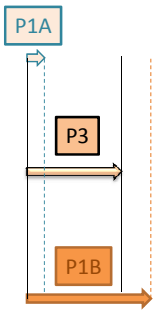
Following a list with description and example of insertion of parameter values for Working Page.

PARAM.	DESCRIPTION		EXAMPLE VALUE
P1	P1A	MINIMUM FEEDING POSITION: Minimum increasing amount at each new part. If the value is overpassed, a alarm will be generated.	USUALLY 0.
	P1B	MAXIMUM FEEDING POSITION: Maximum increasing amount at each new part. If the value is overpassed, a alarm will be generated.	USUALLY MORE THAN P3 VALUE.
P2	RETURN POSITION AFTER BAR IS PUSHED: Absolute position (fixed) of return after bar is pushed when rod is very advanced. This parameter works together parameter P4. After each increasing, the Rod can return uo to P2 or P4. Always the most left.		500
P3	FEEDING VALUE: Part length + Built-in cut-off tool holder width + Collet return. This value also works together Parameter P1.		TO CHANGE ACCORDING PART
P4	START RETURN VALUE: Return value (incremental) after the first movements of Rod feed.		50
P5	POSITION OF FIRST FACING: Start feeding position of the bar for first facing.		TO CHANGE ACCORDING PART
P6	END OF BAR POSITION.		1500
P7	REMAINING MATERIAL DISCARD POSITION: Position to fall end of bar (“stump”). This value usually is a few more than P6.		1590
P8	TOTAL STROKE OF THE FFEDER; Rod position at the end of feeding stroke.		1594
P9	P9A	DWELL TIME AFTER BAR TOUCHES STOP. Time in second.	2
	P9B	DWELL TIME AFTER PLATE IS CLOSED. Time in second.	2
P10	WORKING MODE: 0 = Standard Mode, feeding and pushing the bar. 1 = Working mode, only feeding (in cases with a single feeding, where we work with part puller, large parts or sub-spindle).		0

Explanation diagram for working parameters, P2 – P7:



Explanation diagram for working parameters, P1A, P1B:



14 - PARAMETERS CHANGE

IMPORTANT: *Changing parameters of a machine influences its proper operation. So, the change of any item shall be carefuly done.*

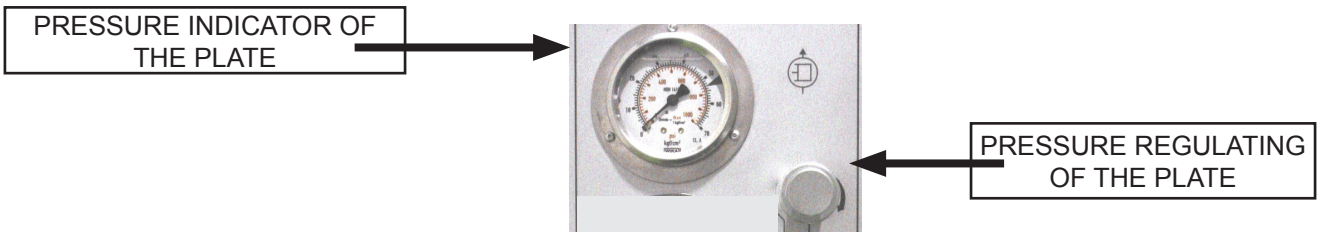
- To press key "**MDI**"
- To press key "**OFS/SET**"
- To press softkey [**SETTING**]
- To place cursor at "**ESCRITA PARAM.**"
- To type: **1**
- To press key "**INPUT**" (in case of alarm, please press –"**CAN" + "RESET**")
- To press key "**SYSTEM**"
- To typo the number of parameter. Example: **6510** (graphic parameter)
- To press softkey [**NO. SRH**]
- When necessary, please place cursor at the "**bit**" to be changed.
- To insert change.
- To press key "**INPUT**"

OBSERVATIONS:
- After the change, return to "ESCRITA PARAM.", please type"0" and press "INPUT".
- When necessary, turn machine off after the change.

15 - ADJUSTS OF MACHINE ACCESSORIES

15.1 - ADJUST OF PLATE PRESSURE

The pressure of hydraulic fixing systems (plate or collet device) can be viewed and changed by the displays and pressure meters, according the figure below:

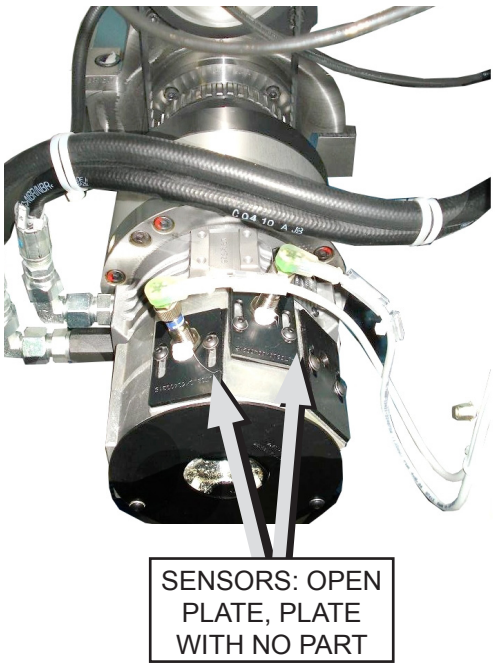


15.2 - ADJUSTS OF PLATE SENSORS:

The plate can work by fastening parts by external diameter or by internal diameter. The hydraulic cylinder have two sensors which detect the part situation at the plate.

By this mechanism, CNC can check the plate position in 3 conditions:

- Plate with no part
- Open plate
- Plate closed, but not at the end of the stroke.



15.2.1. - Sensors adjust to fix part fastening by external

In order to perform the adjust of plate sensors to fix the part by external, we shall obey the procedure below:

– To run command **M41** (to fix by external) via MDI.

– **To fix the part** turning the Button "**CHUCK**".

– To place plate sensors, in a way no one be activated, that is, the Orange LED of each sensor shall be off.

– **To open the plate**, turning Button “CHUCK”.
To place open sensor plate in a way it be active, that is, orange **LED** shall be **on**.

– **To close the plate** (no part) turning Button “CHUCK”.
To place no plate sensor in a way it be active, that is, Orange **LED** shall be **on**.

So, sensors shall be with the following setting:

ADJUST OF PLATE SENSORS (FIXING OF PART BY EXTERNAL):	
CONDITION:	SENSOR TO BE ACTUATED (MESSAGE ON SCREEN):
OPEN PLATE	OPEN PLATE SENSOR
CLOSED PLATE (NO PART)	PLATE WITH NO PART SENSOR
PART FIXED (CLOSING PLATE)	----

OBSERVATION: *: If the sensors are not correctly adjusted, the machine can generate the following alarms:*

“PLATE FAIL WHEN FIXING PART – This alarm occurs when the open plate sensor is actuated when fixing the part.

“PLATE FAIL WHEN LOOSING PART - This alarm occurs when the open plate sensor is not actuated when opening the plate.

15.2.2. - Adjust of Sensors to fix part fastening by internal

In order to perform the adjust of plate sensors to fix the part by internal, we shall obey the procedure below:

– To run command **M40** (to fix by internal) via MDI.

– **To fix the part** turning the Button “**CHUCK**”.

– To place plate sensors, in a way no one be activated, that is, the Orange **LED** of each sensor shall be **off**.

– **To open the plate**, (with no part), turning Button “**CHUCK**”.
To place the no plate sensor in a way it be active, that is, orange **LED** shall be **on**.

So, sensors shall be with the following setting:

ADJUST OF PLATE SENSORS (FIXING OF PART BY INTERNAL):	
CONDITION:	SENSOR TO BE ACTUATED (MESSAGE ON SCREEN):
OPEN PLATE	NO PLATE SENSOR
CLOSED PLATE	OPEN PLATE SENSOR
PART FIXED (OPENING PLATE)	----

OBSERVATION: *If the sensors are not correctly adjusted, the machine can generate the following alarms:*

"PLATE FAIL WHEN FIXING PART – This alarm occurs when the open plate sensor is actuated when fixing the part.

"PLATE FAIL WHEN LOOSING PART - This alarm occurs when the open plate sensor is not actuated when closing the plate.