ROMI®

PROGRAMMING AND OPERATION MANUAL

MACHINE

C420 / C510 / C620 / C830 / C1000

CNC SIEMENS 802D

T22911A

INDÚSTRIAS ROMI S/A

MARKETING DIVISION:
Rua Coriolano, 710 Lapa
05047-900 São Paulo - SP - Brazil
Phone (11) 3873-3388
Telex 1183922
Facsimile (11) 3865-9510

HEADQUARTERS:
Avenida Pérola Byington, 56 Centro
13453-900 Santa Bárbara D'Oeste - SP - Brazil
Phone (19) 3455-9000
Telex 191054
Fac-simile (19) 3455-2499

CONTENTS

PART I - PROGRAMMING

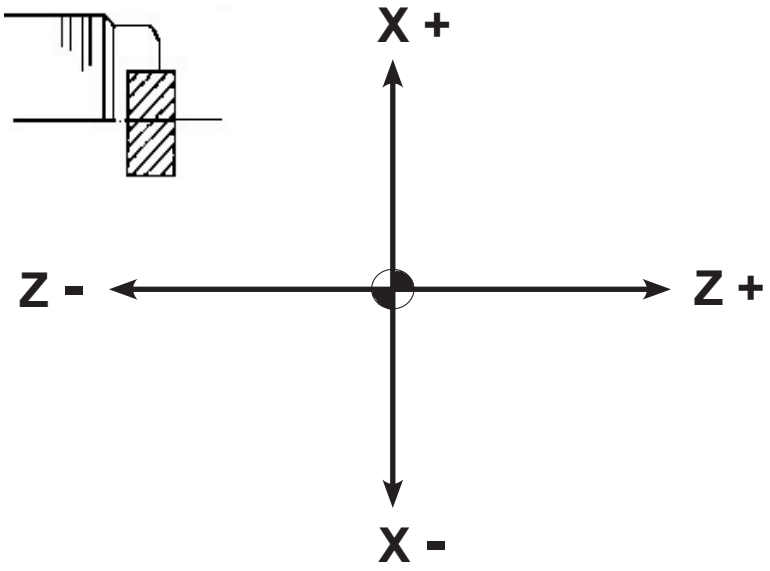
PART II - OPERATION

PART I

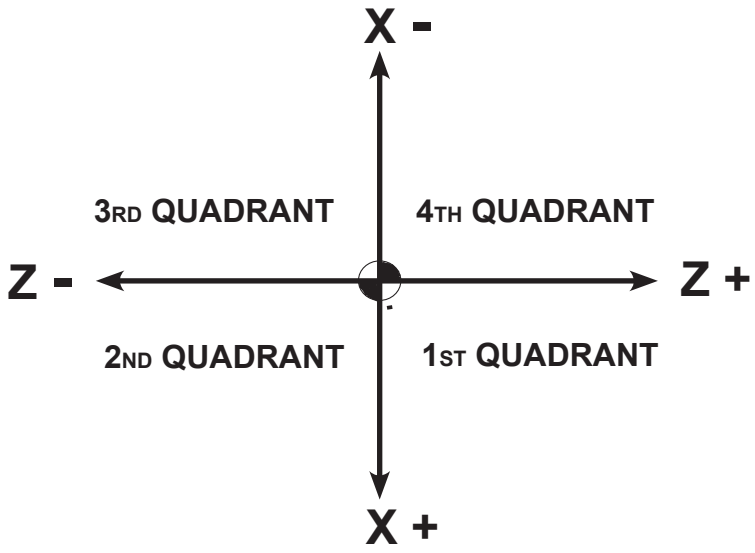
PROGRAMMING

1 - COORDINATE SYSTEM

All the workpiece geometry is transmitted to the CNC based on Cartesian Coordinate System. The universal representation of this system with 2 axes is the following:



NOTE: *On machines equipped with front turret, the universal coordinate system quadrants are adapted according to the illustration below:*



The coordinate system is defined on the plane formed by the intersection of a line parallel to the longitudinal movement (Z), and a line parallel to the cross movement (X).

Each movement of the tool tip is described on this XZ plane, in relation to a pre-established origin (X0,Z0). Remenber that X is always the diameter value.

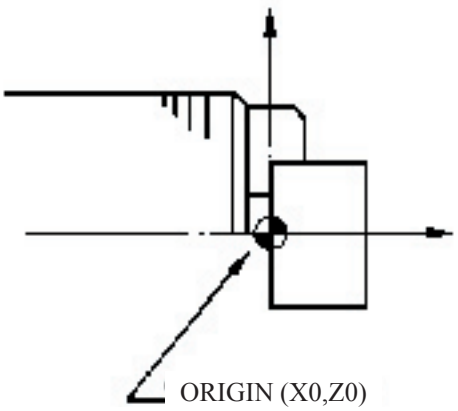
NOTE: *The point that intersects the two lines or axes is defined as "WORK OFFSET" and it is represented by the symbol:*

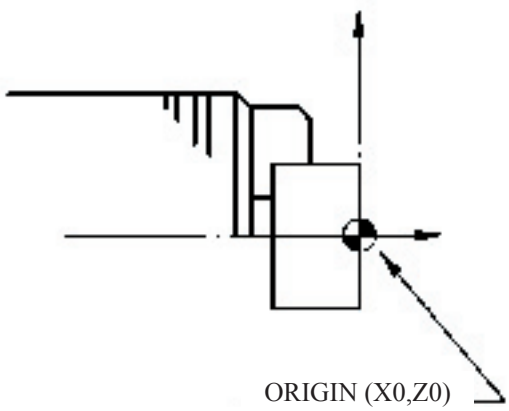
1.1- ABSOLUTE COORDINATE SYSTEM

In this system, the origin is established according to the workpiece to be executed, therefore, it is possible to establish it on any space point to ease programming. This process is called “work offset”.

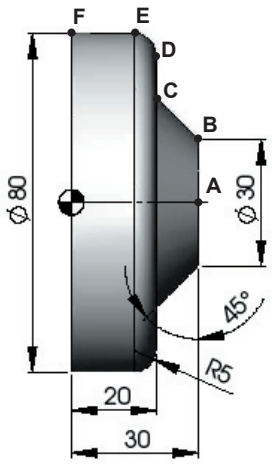
Thus, the system origin was set as points X0, Z0. The point X0 is defined by the spindle center line. The point Z0 is defined by any line perpendicular to the spindle center line.

During programming, the origin (X0, Z0) is usually pre-established at workpiece base (jaws face), or at workpiece face, according to the illustration below:



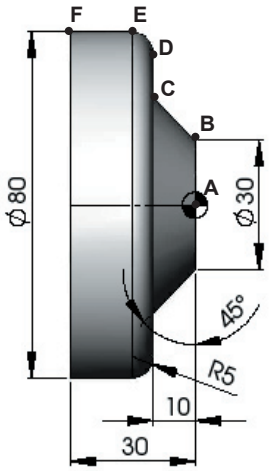


PROGRAMMING EXAMPLE:



ORIGIN AT WORKPIECE BASE:

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



ORIGIN AT WORKPIECE FACE:

ABSOLUTE COORDINATES		
POINT	AXIS	
	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

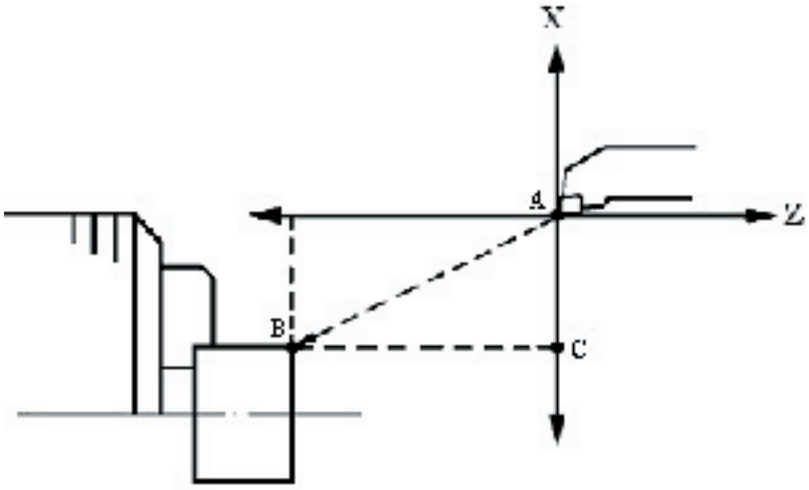
The origin of this system is established for each tool movement.

After any tool shift, a new origin occurs, that is, for any point reached by the tool the coordinates origin will be this reached point.

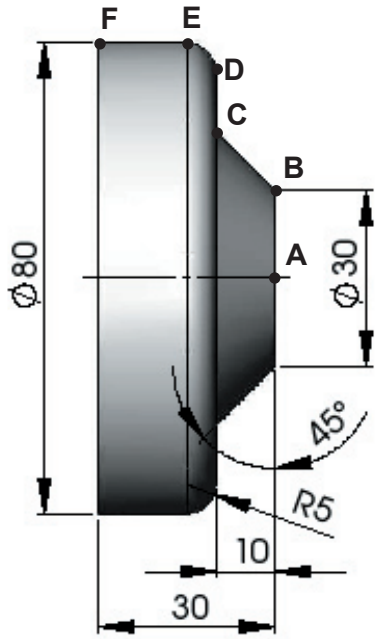
All measurements are performed through the distance to be shifted.

If the tool shifts from point A to B (any two points), the coordinates to be programmed will be the distances between the two points, measured (projected) on X and Z.

Note that point A is the shift origin for point B and B is the origin for the shift up to point C and so on.



PROGRAMMING EXAMPLE:



MOVEMENT		INCREMENTAL COORDINATES	
START	END	AXIS	
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 - WORKPIECE FILES MANAGEMENT

For a more flexible data and programs management, they may be visualized, stored and organized according to different criteria.

The programs and files are stored in different directories; that is, they will be stored in folders according to their functions and characteristics.

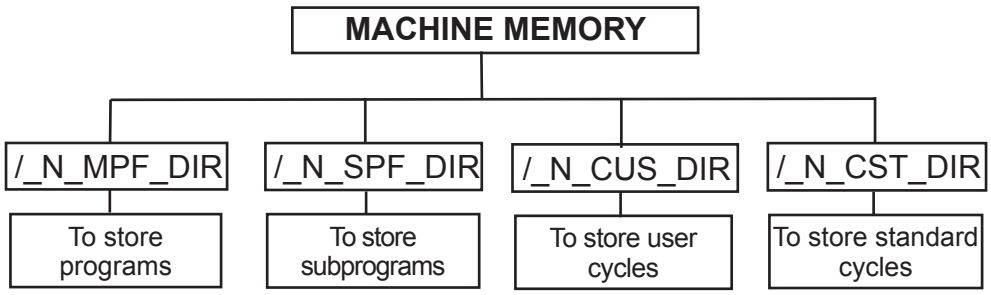
Examples of directories :

- Programs
- Subprograms
- User cycles
- Siemens cycles

Each program corresponds to a file and each file has an extension which will inform the file type used.

- CMA Machine manufacturer cycles
- CST Siemens cycles
- CUS User cycles
- MPF Main programs
- SPF Subprograms
- INI Start up files (tool data)

To store CNC program files (machine), through RS232 (serial port), it is required to address them to the respective directories according to the file type to be stored.



To load a program from a microcomputer to the machine, it is required that the program has a specific header for transmission.

Header syntaxis for programs transmission:

%_N_(program name)_(extension type according to file type)
;$PATH=(corresponding address, refer to graphic above).)

Program header example:

%_N_PART1_MPF
;$PATH=/_N_MPF_DIR
:
:

3 - FUNCTION TYPES

3.1 - POSITIONING FUNCTIONS

FUNCTION X:

Application: Cross axis positioning (absolute)

Format: X +- 5.3 (millimeter)

FUNCTION Z:

Application: Longitudinal axis positioning (absolute)

Format: Z +- 5.3 (millimeter)

3.2 - SPECIAL CODES

3.2.1 - Code: N

Application: Blocks identification.

Function N has the purpose to sequentially number the programming blocks, and its use is optional.

Example:
N10 ...
N20 ...
N30 ...

The required sequence to introduce a command N is the following:

- Press "*PROGRAM MANAGER"* key.
- Press directional keys (◄, ►, ▲, ▼) to place the cursor on the program to be numbered.
- Press "*INPUT"* key.
- Press *[RENUMBER]* softkey.

3.2.2 - Code: Slash (/)

Application: Inhibit blocks execution (BLOCK SKIP).

The Slash Function (/) is used when it is required to inhibit blocks execution in the program without changing programming.

If the character "/" is typed in front of some blocks, these will be ignored by the CNC since the operator has selected the option "SKIP". If this option is not selected, the CNC will normally execute the blocks, even the ones preceded by the character "/" (slash).

To select the option BLOCK SKIP the instructions below must be followed:

– Press "*POSITION*" key.
– Press *"AUTO"* key.
– Press *[PROGRAM CONTROL]* softkey.
– Press *[SKIP]* softkey.

3.2.3 - Code: F

Application: determine the feed rate

The feed rate is an important data for the machining process and it is obtained by taking into consideration the type of material, the tool and the operation to be executed.

In CNC lathes, feed rate is usually defined in mm/rotation (function G95), but it may also be set in mm/min (function G94).

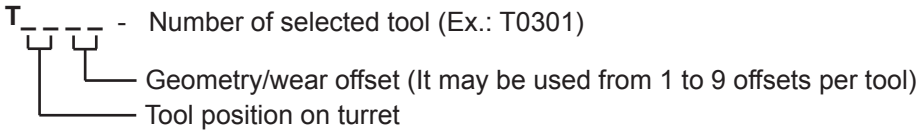
3.2.4 - Código: T

Application: tool selection

T Function is used to select the tool, informing to the machine its preset, insert radius, cutting directions and offsets.

"T" code must be followed by four digits maximum in its programming: the first two digits indicate tool position on turret or tool holder (in case there is no optional for electric turret) and the last two digits indicate the selected tool offset.

The syntaxis for programming is:



Example:

T0101
:
:
T0201
:
:
T0301

4 - PRELIMINARY FUNCTIONS

Application: This group of functions, also called “G Codes”, define what the machine should do, preparing it to execute a certain operation, or to receive a certain information.

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

MODAL: Functions that when programmed stay in the CNC memory, and are valid for all the subsequent blocks, unless modified by another or the same function.

NON-MODAL: Functions that must be programmed when required, and are valid only on the blocks that include them.

LIST OF PRELIMINARY FUNCTIONS			
G Code	Function	Modal	Non-Modal
G00	Positioning (rapid traverse)	X	
G01	Linear interpolation (programmed feed)	X	
G02	Circular interpolation (clockwise)	X	
G03	Circular interpolation (counter clockwise)	X	
G04	Dwell time		X
G20	Input system inch (in)	X	
G21	Input system metric (mm)	X	
G28	Return axes to reference position		X
G33	Thread cutting with interpolation (constant lead)	X	
G40	Cancel tool radius compensation	X	
G41	Activate tool radius compensation (tool at left)	X	
G42	Activate tool radius compensation (tool at right)	X	
G53	Cancel work offset coordinates (activate machine offset)		X
G54	Activate 1st work offset coordinate system	X	
G55	Activate 2nd work offset coordinate system	X	
G56	Activate 3rd work offset coordinate system	X	
G57	Activate 4th work offset coordinate system	X	
G58	Activate 5th work offset coordinate system	X	
G59	Activate 6th work offset coordinate system	X	
G70	Finishing cycle	X	
G71	Longitudinal roughing cycle		X
G72	Cross roughing cycle		X
G73	Roughing cycle parallel to the contour		X
G74	Longitudinal roughing or axial drilling cycle		X
G75	Facing or grooving cycle		X
G76	Automatic threading cycle		X

LIST OF PRELIMINARY FUNCTIONS			
G Code	Function	Modal	Non-Modal
G77	Longitudinal or taper roughing cycle	X	
G78	Semiautomatic threading cycle	X	
G79	Cross or taper roughing cycle	X	
G80	Cancel drilling cycles	X	
G83	Axial drilling cycle	X	
G84	Axial tapping cycle	X	
G90	Absolute Coordinate System	X	
G91	Incremental Coordinate System	X	
G92	Maximum rotation (RPM)	X	
G94	Feed per minute (Millimeters/inches)	X	
G95	Feed per rotation (Millimeters/inches)	X	
G96	Activate constant surface speed (m/min)	X	
G97	Cancel constant surface speed (programming in RPM)	X	

5 - INTERPOLATION FUNCTIONS

5.1 - FUNCTION: G00

Application: Rapid traverse positioning (approach and retraction).

The axes move to the programmed point at the maximum feed speed available.

Syntaxis:
G0 X__ Z__

where:

X = coordinate to be reached (diameter values)

Z = coordinate to be reached

The function G0 is Modal and cancels functions G1, G2, G3

NOTE: *In the Centur 30D the rapid traverse speed is 10 m/min for "X" and "Z" axes.*

5.2 - FUNCTION: G01

Application: Linear interpolation (machining with programmed feed)

This function enables rectilinear movements with any angle, calculated through coordinates and predetermined feed (F) by the operator.

Syntaxis:
G1 X__ Z__ F__

where:

X = coordinate to be reached (diameter values)

Z = coordinate to be reached

F = work feed (mm/rot)

NOTE: *The function G1 is Modal and cancels functions G0, G2, G3.*

5.3 - FUNCTION: G02 AND G03

Application: Circular interpolation (radius).

Both G2 and G3 execute machining operations of predefined arcs through an adequate and simultaneous axes movement.

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

where:

X = arc end point

Z = arc end point

I = arc center coordinate (in relation to the tool tip)

K = arc center coordinate (in relation to the tool tip)

R = radius value

(F) = feed rate

NOTE: *When programming an arc, these rules should be followed:*
• The arc start point is the tool start position.
• Circular interpolation G02 or G03 directions are programmed (clockwise or counter clockwise on the universal programming concept).
• Together with the interpolation direction, the arc end point coordinates are programmed with X and Z .
• Together with the arc direction and the final coordinates, the R function (radius function), or I and K functions (arc center coordinates) are programmed.

5.3.1 - Function: R

Application: Arc defined by radius.

It is possible to program “circular interpolation” up to 180 degrees through R function, always setting radius value with positive signal.

5.3.2 - Function: I and K

Application: Arc defined by polar center.

Functions I and K define the arc center position, where:

I is parallel to X axis and K is parallel to Z axis.

NOTES:

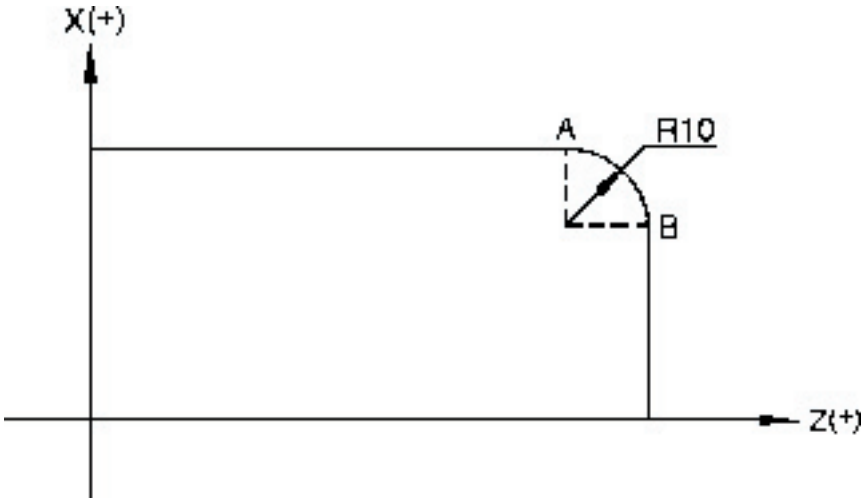
• Functions I and K are programmed taking as reference the distance from tool start point to arc center, giving the corresponding signal to the movement.

• Function “I” must be programmed in radius.

EXAMPLE:

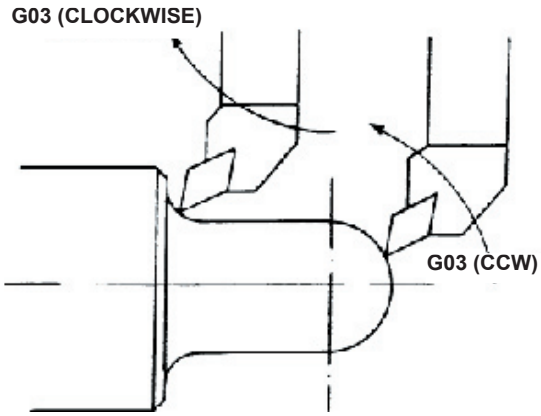
DIRECTION A-B: I-10 K0

DIRECTION B-A: I0 K-10

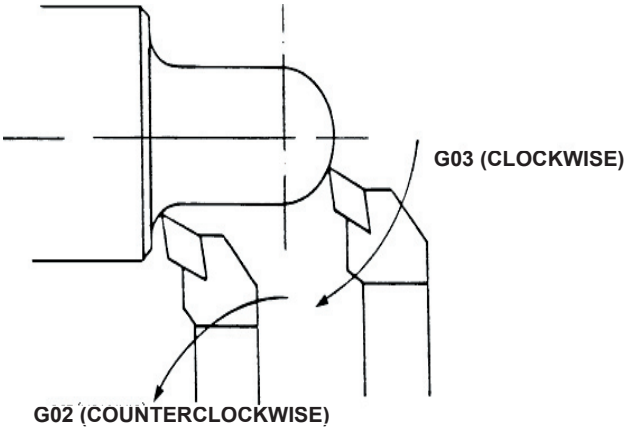


The arc machining direction defines if it is clockwise or counterclockwise, according to the illustrations below:

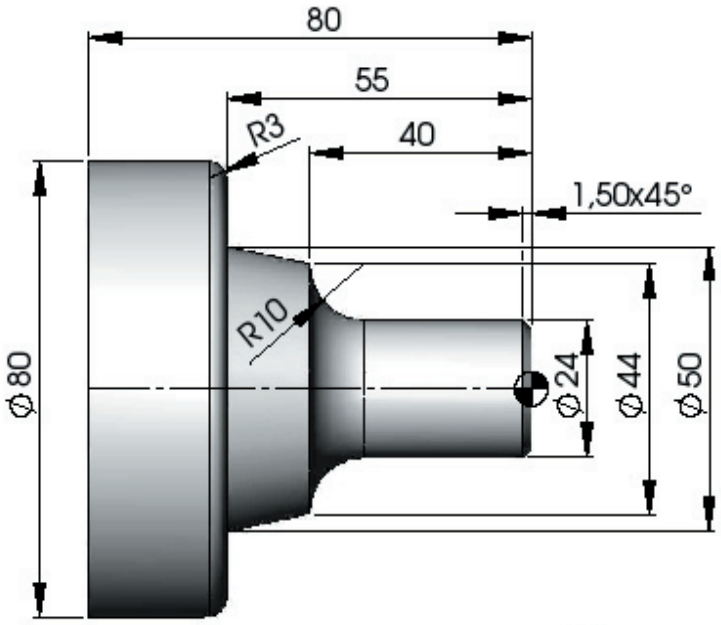
UNIVERSAL PROGRAMMING



ADAPTED PROGRAMMING FOR C420 - SIEMENS 802D



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

NOTE: *Functions G2 and G3 are Modal and cancel functions G0 and G1.*

5.4 - FUNCTION: “,R” / “,C”

Application: Round / chanfer.

Function “,R” and “,C” executes round / chanfer.

Syntaxis:

G01 X__ Z__, R__

G01 X__ Z__

ou

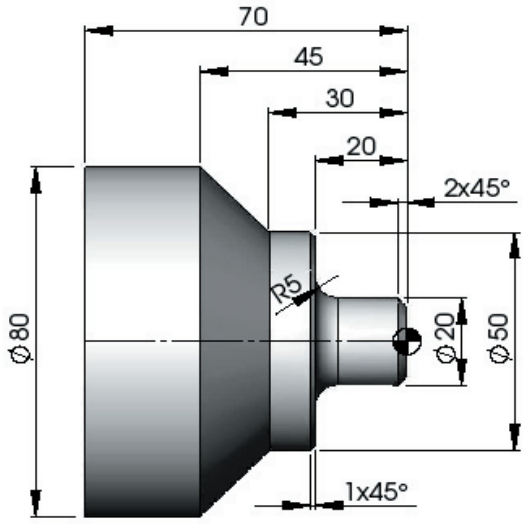
G01 X__ Z__, C__

G01 X__ Z__

where:

,R = round value
,C = chanfer value

EXAMPLE:




```
:
:
:
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 T00
N170 G54 G00 X200 Z200
:
:
```

5.5 - FUNCTION: G33

Application: Thread cutting with constant lead

Function G33 executes the threading cycle on X and Z axes, where each depth is explicitly programmed in a separate block.

It is possible to thread inner and outer diameters, and they can be parallel or tapered, single or multiple, progressive, etc.

Function G33 requires:

X = threading final diameter

Z = thread length final position

Q = spindle angle for thread start (thousandth of degrees)

R = "X" axis incremental taper value (radius/negative for outer and positive for inner)

F = thread pitch

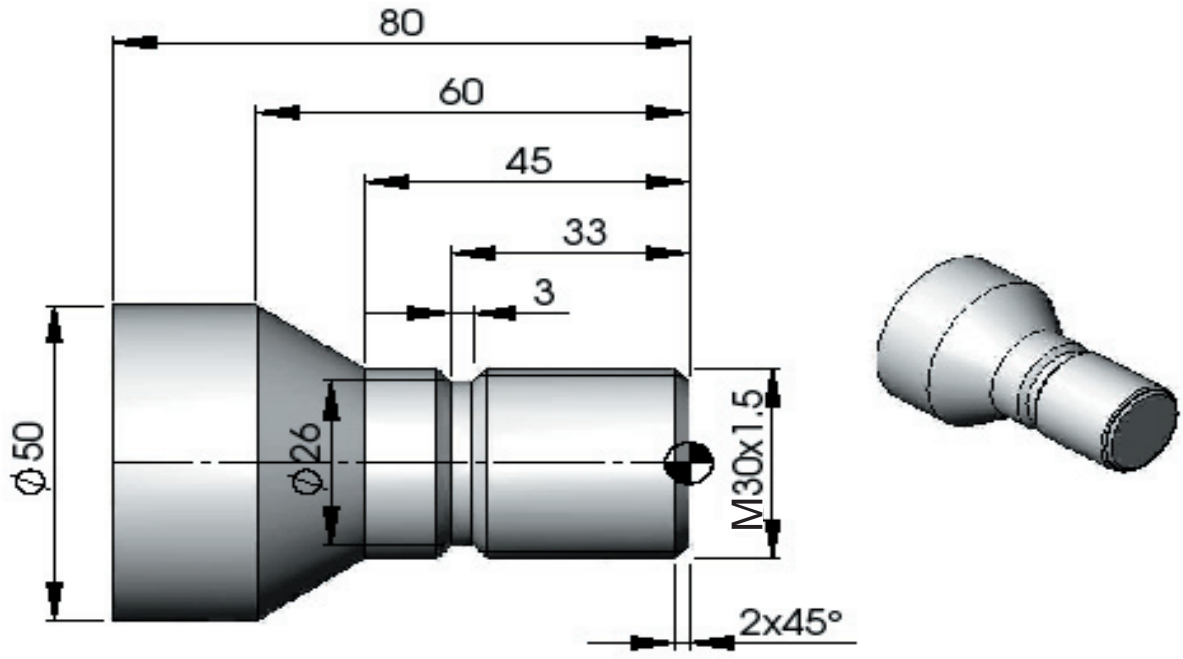
NOTES:

• It is not required to repeat the pitch value (F) in the subsequent blocks of G33.

• During approaching, it is advisable to leave a minimum backlash of two times the thread pitch on "Z" axis

• Function G33 is modal.

Example 1: Metric Thread M30x1.5



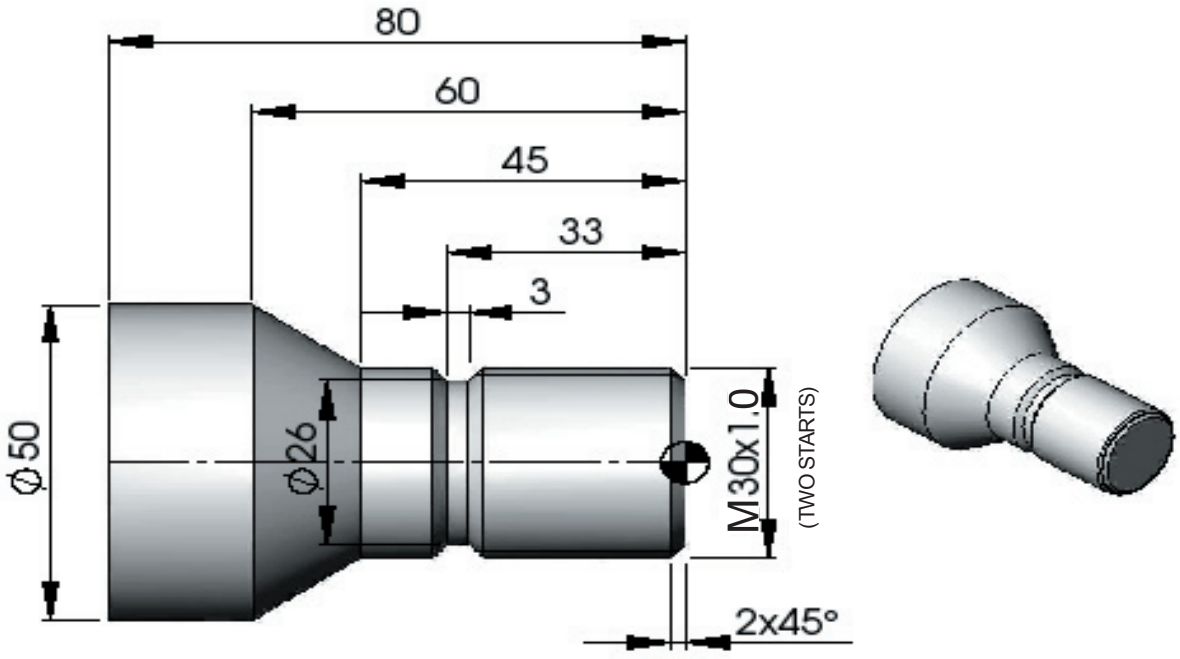

```
%_N_THREAD_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G55 G0 X200 Z200
N50 T0101 (THREAD M30X1.5)
N60 G97 S1000 M3
N70 G0 X35 Z3
N80 X29.35
N90 G33 Z-31.5 F1.5
N100 G0 X35
N110 Z3
N120 X28.95
N130 G33 Z-31.5
N140 G0 X35
N150 Z3
N160 X28.55
N170 G33 Z-31.5
N180 G0 X35
N190 Z3
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 T00
N280 G55 G0 X200 Z200
N290 M30
```

CALCULATION:

1º) Fillet 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

Example 2: Metric Thread M30x1.0 (2 starts)




```
%_N_THREAD_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G54 G0 X180 Z200
N50 T0101 (THREAD M30X1)
N60 G97 S1000 M3
N70 G0 X35 Z3
N80 X29.35
N90 G33 Z-31.5 F2.0 Q0
N100 G0 X35
N110 Z3
N120 X28.95
N130 G33 Z-31.5 Q0
N140 G0 X35
N150 Z3
N160 X28.7
N170 G33 Z-31.5 Q0
N180 G0 X35
N190 Z3
N200 X29.35
N210 G33 Z-31.5 Q180000
N220 G0 X35
N230 Z3
N240 X28.95
N250 G33 Z-31.5 Q180000
N260 G0 X35
N270 Z3
N280 X28.7
N290 G33 Z-31.5 Q180000
N300 G0 X35
N310 T00
N320 G54 G0 X180 Z200
N330 M30
```

CALCULATION:

1º) Fillet height (P):
P = (0.65 x pitch)
P = (0.65 x 1.0)
P = 0.65

2º) Final diameter (X):
X = Start diameter - (P x 2)
X = 30 - (0.65 x 2)
X = 28.7

NOTE: *For multiple-start threads it is required to calculate feed according to this formula:*

Feed rate (**F**):
F = Pitch x Number of starts
F = 1.0 x 2
F = 2.0

6 - DWELL

6.1 - FUNCTION: G04

Application: Dwell time

Between one and other tool shift, it may be programmed a certain tool dwelling. The function G04 executes a dwell which duration is defined by an associated value “P”, “U” or “X” that defines the time in seconds.

Function G04 requires:

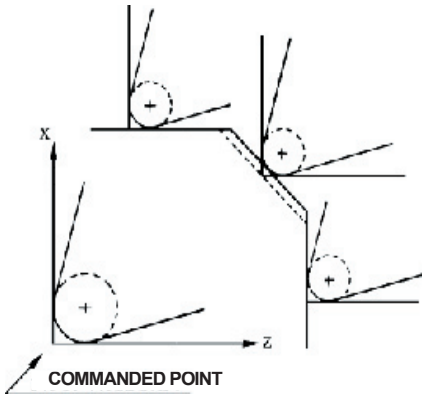
G04 X__ (seconds)
or
G04 U__ (seconds)
or
G04 P__ (thousandths of seconds)

EXAMPLE: (1.5 SECONDS)

G04 X1.5
G04 U1.5
G04 P1500

7 - TOOL RADIUS COMPENSATION

The tool tip radius compensation function is used to correct the measurement differences generated by the tool tip when a shift on “X” and “Z” axes is simultaneously executed, as shown in the illustration below:



7.1 - FUNCTION: G40

Application: Cancel tool radius compensation

Function G40 must be programmed to cancel previously requested functions as G41 and G42.

Function G40 is a Modal code and it is active when the control is ON.

7.2 - FUNCTION: G41

Application: Activate tool radius compensation (at left)

Function G41 selects the tool tip radius compensation value, when the tool is at left of the workpiece to be machined, viewed in relation to cutting feed direction.

Function G41 is Modal, therefore cancels G40

NOTE: *For a successful radius compensation, it is required to access the page “OFFSET PARAM / TOOL LIST.” and to inform tool radius and quadrant (chapter 7.4).*

7.3 - FUNCTION: G42

Application: Activate tool radius compensation (at right)

This function is similar to Function G41, except that compensation direction is at right.

Function G42 is Modal, therefore cancels G40.

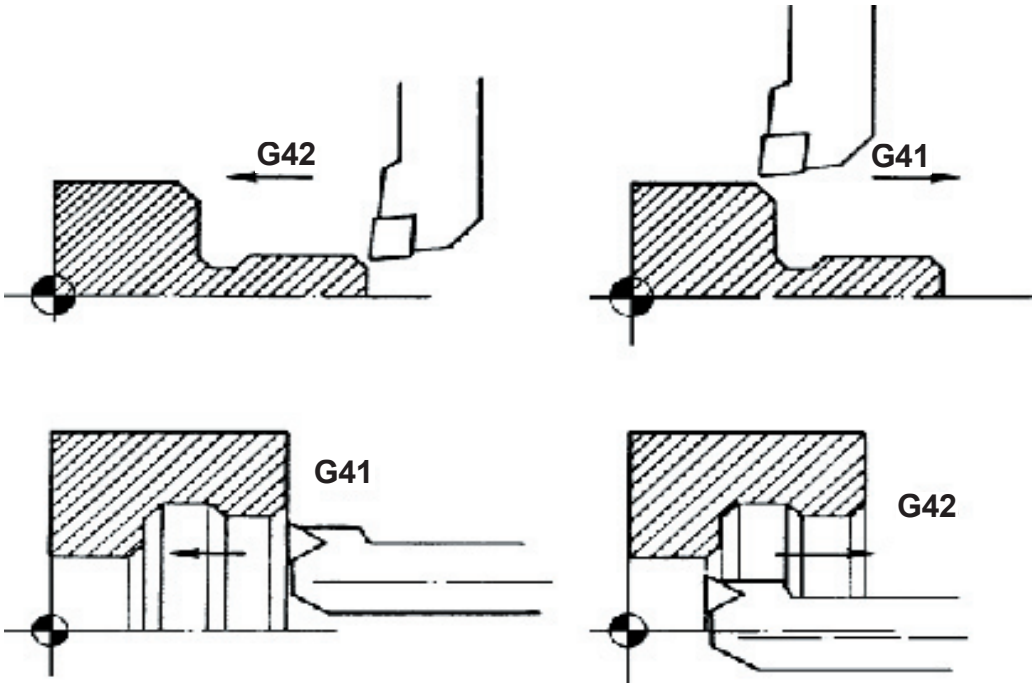
NOTES:

• The first shift after radius compensation must be higher than the insert radius value.

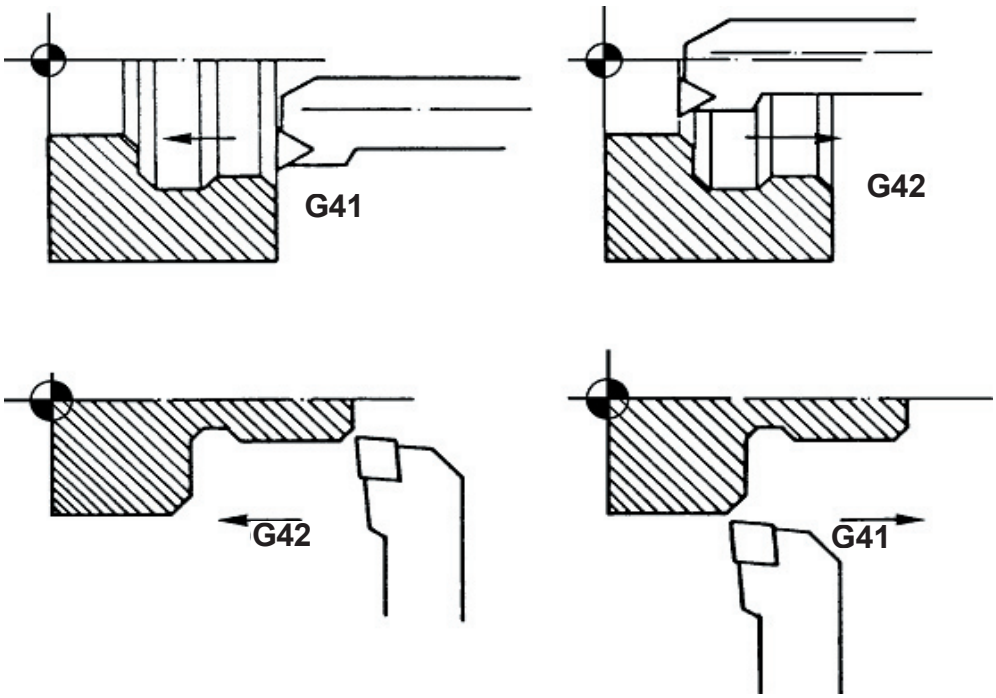
• The tool must not be in contact with the material to be machined when compensation functions are activated in the program.

TOOL RADIUS COMPENSATION:

(UNIVERSAL PROGRAMMING)

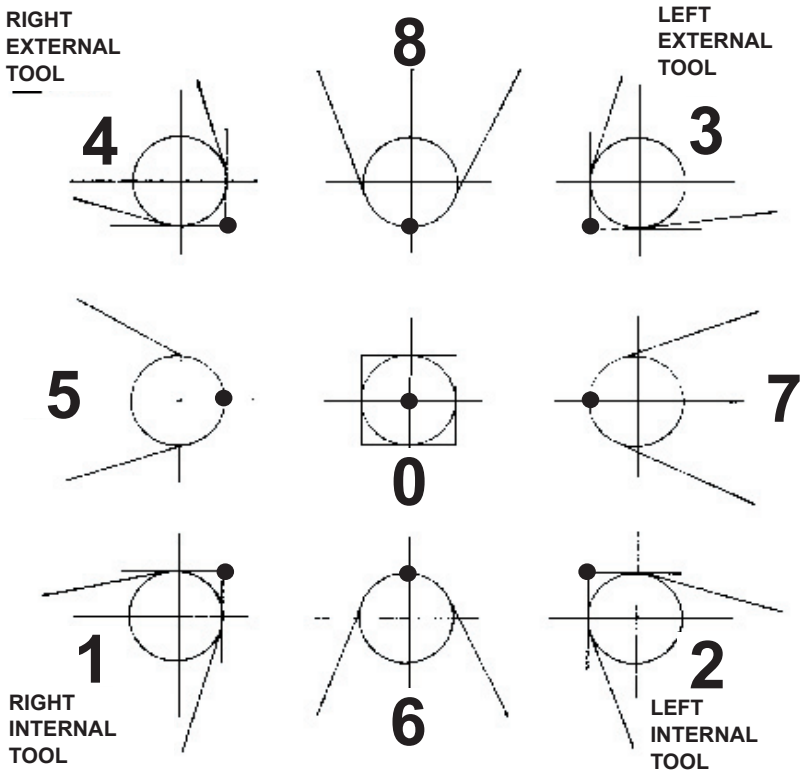


(ADAPTED PROGRAMMING FOR C420 - SIEMENS 802D)

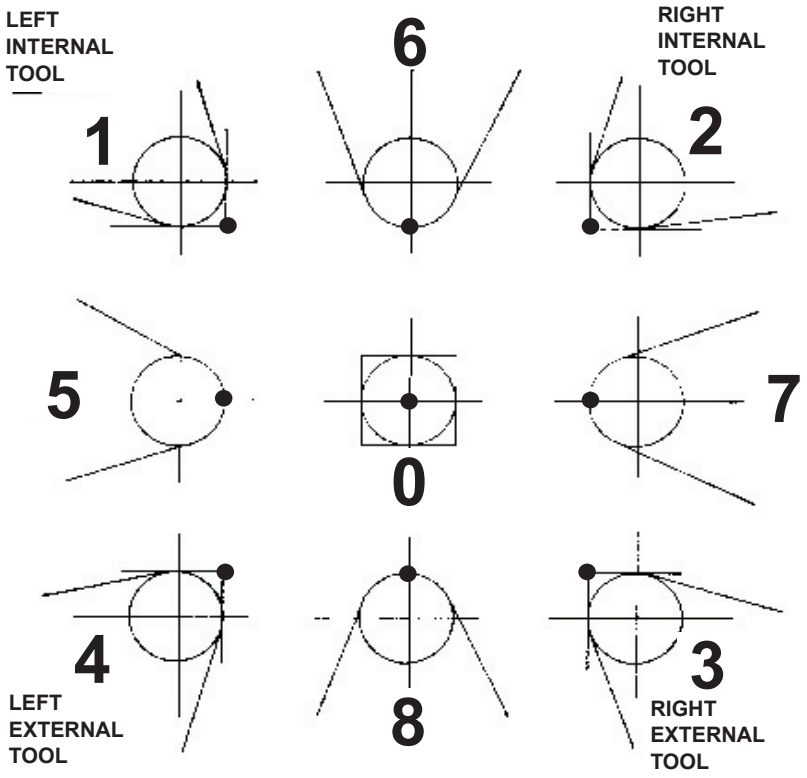


7.4 - TOOL QUADRANTS FOR RADIUS COMPENSATION

(UNIVERSAL PROGRAMMING):

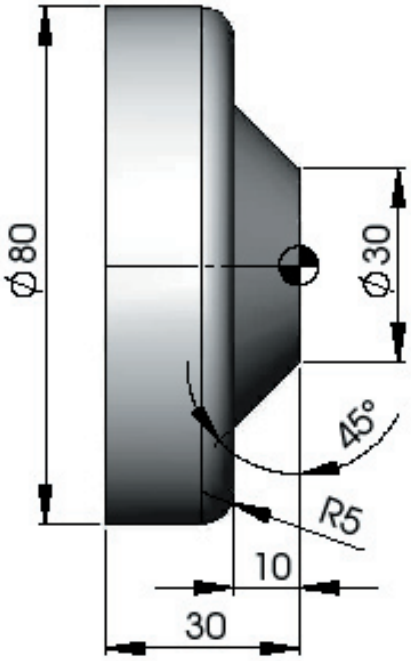


(ADAPTED PROGRAMMING FOR C420 - SIEMENS 802D)



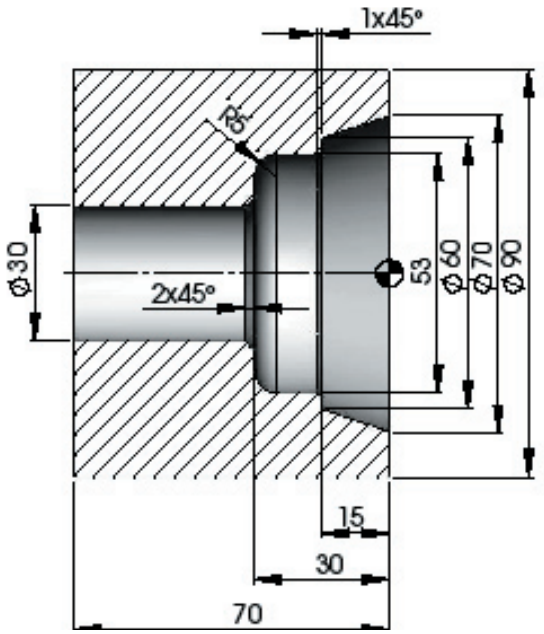
7.5 - EXAMPLES TOOL RADIUS COMPENSATION:

Example1: External




```
:
:
:
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 T00
N190 G54 G00 X200 Z200
:
:
```

Example 2: Intern




```
:
:
N60 G01 X74 Z2
N70 G41
N80 G01 Z0 F.2
N90 X70
N100 X30 Z-15
N110 X53, C1
N120 Z-30, R5
N130 X30, C2
N140 Z-72
N150 X27
N160 G40
N170 G00 Z2
N180 T00
N190 G54 G00 X200 Z200
:
:
```

8 - SINGLE CYCLES

8.1 - FUNCTION: G77

8.1.1 - Parallel turning cycle.

Function G77 may be used as a turning cycle parallel to “Z” axis that cuts with successive passes up to the desired diameter.

Function G77 as a turning cycle requires:

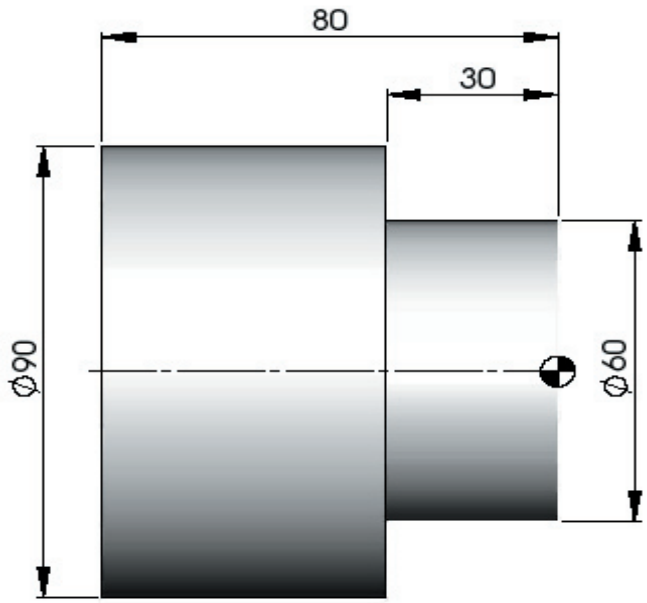
G77 X__ Z__ F__; where:

X = first pass diameter

Z = final position (absolute)

F = work feed

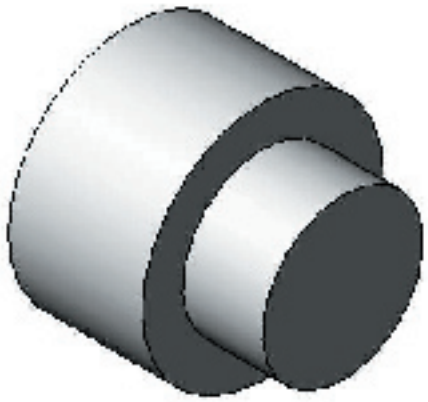
PROGRAMMING EXAMPLE:




```
%_N_ROUGHING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G57 G0 X180 Z300 T00
N50 T0101 (rough.)
N60 G96 S150
N70 G92 S2500 M3
N80 G0 X90 Z2
N90 G77 X84 Z-30 F.3
N100 X78
N110  X72
N120  X66
N130 X60
N140 T00
N150 G57 G0 X180 Z300
N160 M30
```

Cutting depth = 3 mm

Feed = 0.3 mm/rot



8.1.2 - Taper turning cycle.

Function G77 as a taper turning cycle requires:

G77 X__ Z__ R__ F__; where:

X = first pass diameter

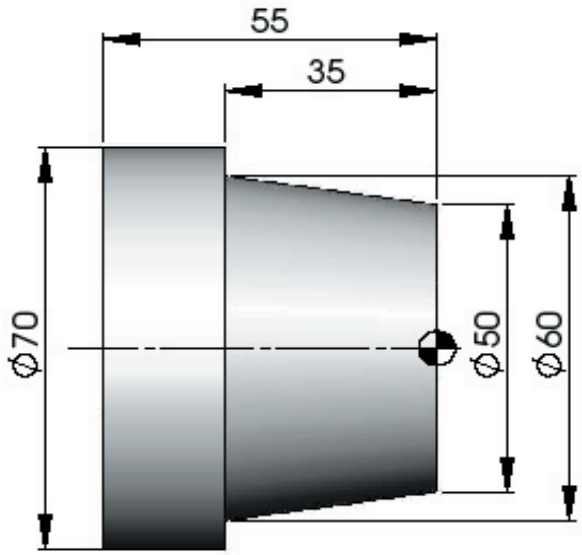
Z = final position (absolute)

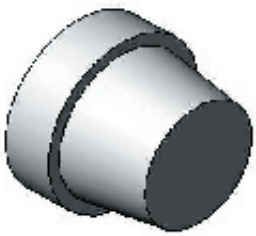
R = “X” axis incremental tapered angle between start and final points (radius)

F = work feed

> ***NOTE:*** *When positioning tool on “X” axis, the “R” value (in the diameter) must be added to define the coordinate to be programmed, in relation to the blank.*

PROGRAMMING EXAMPLE





```
%_N_ROUGHING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G59 G0 X195 Z300
N50 T0201 (rough.)
N60 G96 S250
N70 G92 S3500 M3
N80 G0 X80 Z2
N90 G77 X76 Z-35 R-5 F.2
N100 X72
N110  X68
N120  X64
N130 X60
N140 T00
N150 G59 G0 X195 Z300
N160 M30
```

Cutting depth = 2 mm
Feed = 0.2 mm/rot

8.2 - FUNCTION: G78

Application: Semiautomatic threading cycle

Function G78 requires:

G78 X__ Z__ (R__) F__; where:

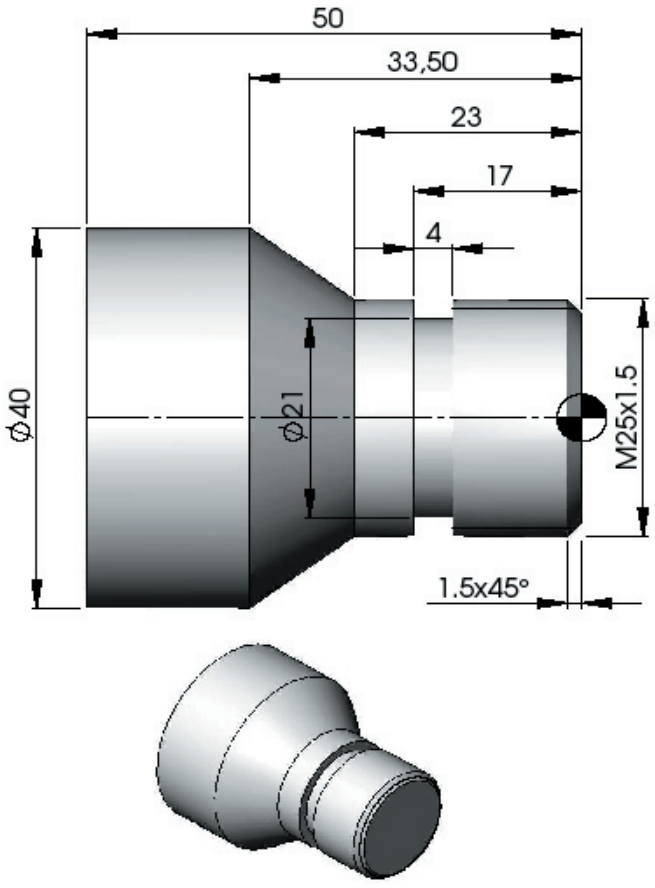
X = threading diameter

Z = threading final position

R = "X" axis incremental tapered angle value (tapered thread)

F = thread pitch

Example 1: Thread M25x1.5




```
%_N_THREAD_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G58 G0 X200 Z270
N50 T0301 (THREAD M25X1.5)
N60 G97 S1500 M3
N70 G0 X30 Z3
N80 G78 X24.2 Z-15 F1.5
N90 X23.6
N100 X23.2
N110 X23.05
N120 T00
N130 G58 G0 X200 Z270
N140 M30
```

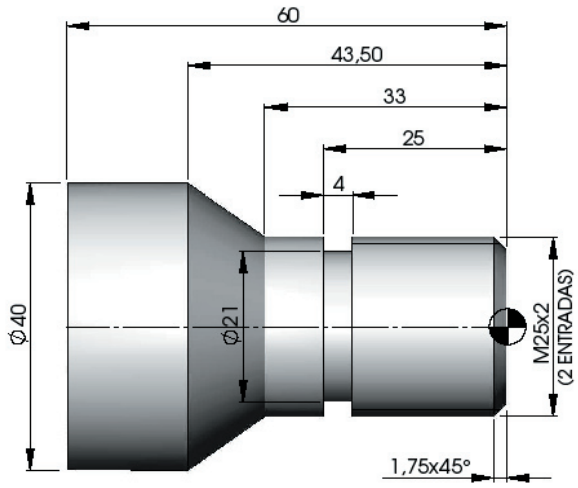
DEPTH IN THE EXAMPLE:
1st pass = 0.8mm
2nd pass = 0.6mm
3rd pass = 0.4mm
4th pass = 0.15mm

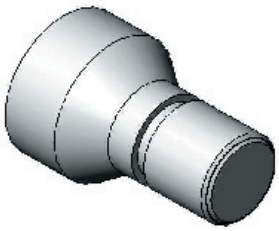
CALCULATION

1º) Fillet 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 = 25 - (0.975 x 2)
X = 23.05

Example 2: Thread: M25x2 (2 starts)





```
%_N_THREAD_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G54 G0 X170 Z270
N50 T0301 (THREAD M25X2)
N60 G97 S1500 M3
N70 G0 X28 Z8 ; (1 ST START)
N80 G78 X24 Z-23 F4
N90 X23.2
N100 X22.6
N110 X22.4
N120 G00 Z10 ; (2ND START)
N130 G78 X24 Z-23 F4
N140 X23.2
N150 X22.6
N160 X22.4
N170 T00
N180 G54 G0 X170 Z270
N190 M30
```

DEPTH IN THE EXAMPLE:
1st PASS = 1.0mm
2nd PASS = 0.8mm
3rd PASS = 0.6mm
4th PASS = 0.2mm

CALCULATION:

1º) Fillet height (P):
P = (0.65 x pitch) P = (0.65 x 2)
P = 1.3

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

NOTE: *For multiple-start thread it is required to calculate feed (F) as follows:*

Programmed feed (**F**) :
F = Pitch x Nº of starts
F = 2 x 2
F = 4

8.3 - FUNCTION: G79

8.3.1 - Parallel facing cycle.

Function G79 describes its cycle parallel to “X” axis, helping roughing operations as facing cycle.

Function G79, as a facing cycle requires:

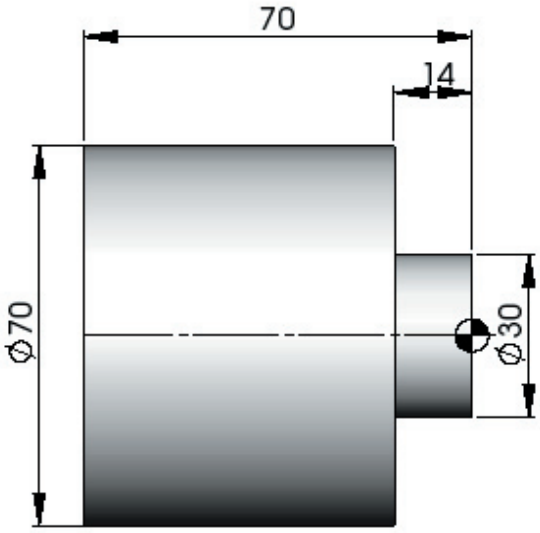
G79 X__ Z__ F__; where:

X = facing final diameter

Z = final position (absolute)

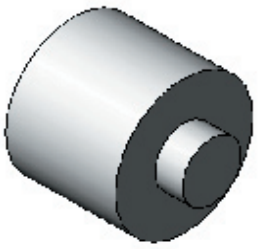
F = work feed

PROGRAMMING EXAMPLE




```
%_N_FACING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G59 G0 X200 Z250
N50 T0201 (rough.)
N60 G96 S250
N70 G92 S3500 M3
N80 G0 X74 Z0
N90 G79 X30 Z-2 F.15
N100 Z-4
N110  Z-6
N120  Z-8
N130  Z-10
N140  Z-12
N150 Z-14
N160 T00
N170 G59 G0 X200 Z250
N180 M30
```

Cutting depth = 2 mm
Feed = 0.15 mm/rot



8.3.2 - Taper facing cycle.

Function G79, as a taper facing cycle requires:

G79 X__ Z__ F__; where:

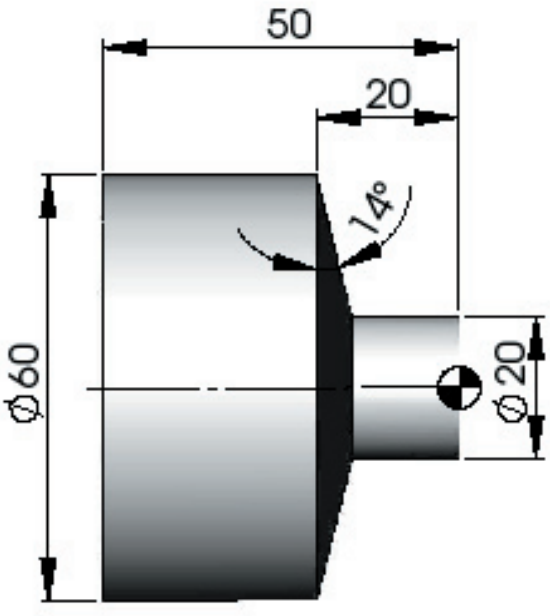
X = facing final diameter

Z = final position (absolute)

R = incremental tapered angle (negative for external and positive for internal)

F = work feed

PROGRAMMING EXAMPLE




```
%_N_FACING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G55 G0 X170 Z300
N50 T0201 (rough.)
N60 G96 S220
N70 G92 S3800 M3
N80 G0 X64 Z5.485
N90 G79 X20 Z2.485 R5.485 F.15
N100 Z-1.485
N110  Z-4.485
N120  Z-7.485
N130  Z-10.485
N140  Z-13.485
N150 Z-15.013
N160 T00
N170 G55 G0 X170 Z300
N180 M30
```

9 - MULTIPLE REPETITIVE CYCLES

9.1 - FUNCTION: G70

Application: Finishing cycle.

This cycle is used after roughing cycles G71, G72 and G73 for workpiece finishing without need to repeat all sequence of the contour to be executed.

The function G70 requires:

G70 P__ Q__ F__; where:

P = number of block that defines the contour start

Q = number of block that defines the contour end

F = work feed used for finishing

Function F specified between the contour start block (P) and contour final block (Q) is valid for code G70 cycles only and it does not have effect during roughing cycles (G71, G72, and G73).

NOTE: *After G70 cycle execution, the tool automatically returns to the point used for positioning.*

The finishing cycle automatically enables tool tip radius compensation. Thus, it is not required to program G41/G42 commands on the workpiece contour.

9.2 - FUNCTION: G71

Application: Longitudinal roughing automatic cycle

The function G71 must be programmed in two subsequent blocks because the values related to cutting depth and finishing allowance on the cross and longitudinal axes are informed by “U” and “W” functions, respectively.

The function G71 in the 1st block requires:

G71 U__ R__; where:

U = cutting depth value during cycle (radius)

R = distance value in the cross axis to return to initial Z (radius)

The function G71 in the 2nd block requires:

G71 P__ Q__ U__ W__ F__ ; where:

P = number of block that defines the contour start

Q = number of block that defines the contour end

U = “X” axis finishing allowance (positive for external and negative for internal / diameter)

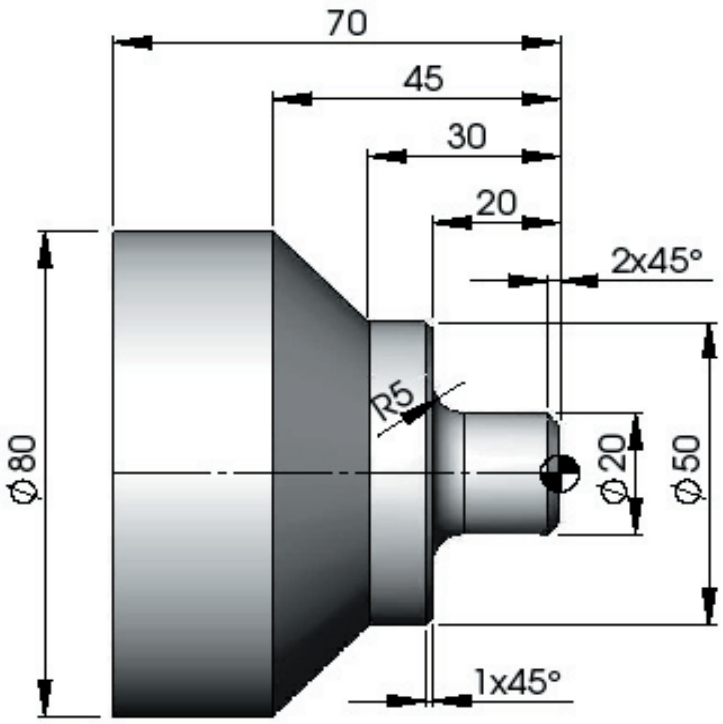
W = “Z” axis finishing allowance (positive for stock removal at right and negative for left machining)

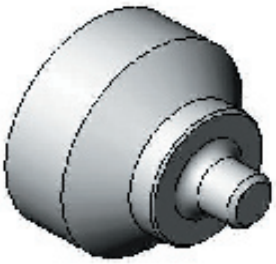
F = work feed

NOTE: *After execution of the cycle, the tool automatically returns to the positioning point.*

The last coordinate programmed in “X” within the profile is the one that the machine understands as being blank, i.e. the machine uses the last programmed diameter for machining increment start.

Example 1: External machining



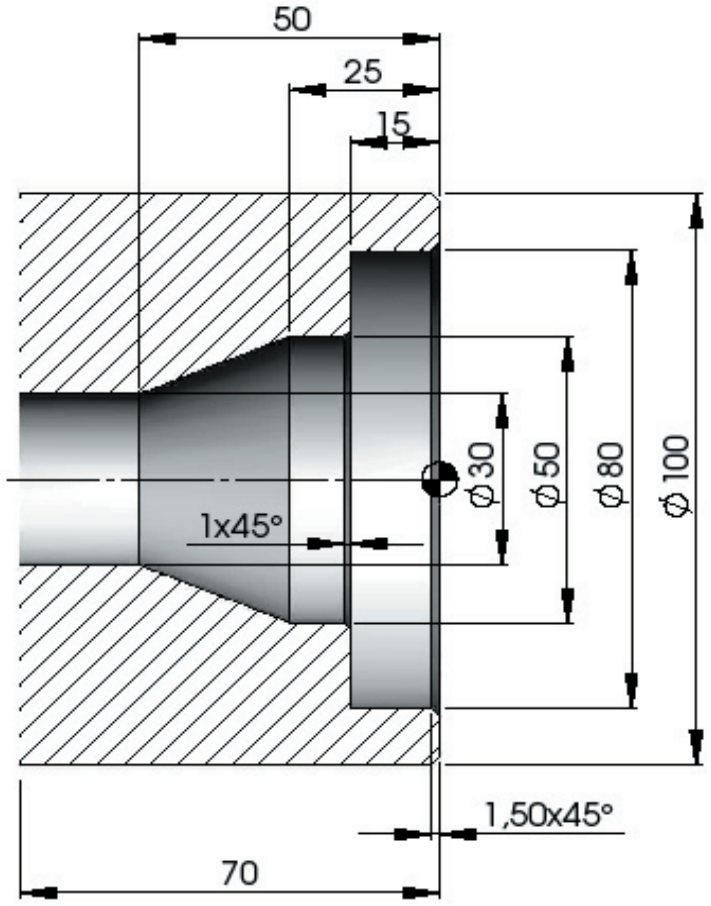


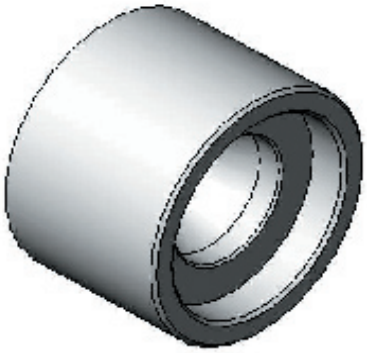
```
%_N_ROUGHING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N10 G21 G40 G90 G95
N30 T00
N40 G58 G0 X200 Z280
N50 T0101 (ROUGH)
N60 G96 S200
N70 G92 S2500 M3
N80 G0 X80 Z2
N90 G71 U2.5 R2
N100 G71 P110 Q190 U1 W.3 F.25
N110 G0 X16
N120 G1 Z0
N130 X20 Z-2
N140 Z-15
N150 G2 X30 Z-20 R5
N160 G1 X48
N170 X50 Z-21
N180 Z-30
N190 X80 Z-45
N200 G70 P110 Q190 F.2
N210 T00
N220 G58 G0 X200 Z280
N230 M30
```

Cutting depth = 2.5 mm
Roughing feed = 0.25 mm/rot
Finishing feed = 0.2 mm/rot

NOTE:
In this example it was considered that roughing and finishing cycles have used the same tool.

Example 2: Internal machining





```
%_N_ROUGHING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G59 G0 X190 Z200
N50 T0101 (ROUGH.)
N60 G96 S200
N70 G92 S2500 M3
N80 G0 X25 Z2
N90 G71 U3 R2
N100 G71 P110 Q180 U-1. W.3 F.3
N110 G0 X83
N120 G1 Z0
N130 X80 Z-1.5
N140 Z-15
N150 X50 ,C1
N160 Z-25
N170 X30 Z-50
N180 Z-71
N190 T00
N200 G59 G0 X190 Z200
N210 T0201 (FINISH))
N220 G54
N230 G96 S250
N240 G92 S3500 M4
N250 G0 X25 Z2
N260 G70 P110 Q180 F.2
N270 T00
N280 G59 G0 X190 Z200
N290 M30
```

Cutting depth = 3 mm
Roughing feed = 0.3 mm/rot
Finishing feed = 0.2 mm/rot

NOTE: *In this example it was considered that roughing and finishing cycles have used different tools.*

9.3 - FUNCTION: G72

Application: Cross roughing automatic cycle

The function G72 must be programmed in two subsequent blocks, because the values relating to cutting depth and finishing allowance on longitudinal axis are given by function "W"

The function G72 in the 1st block requires:

G72 W__ R__; where:

W = cutting depth during cycle

R = distance value in the longitudinal axis to return to initial "X"

The function G72 in the 2nd block requires:

G72 P__ Q__ U__ W__ F__ ; where:

P = number of block that defines the contour start

Q = number of block that defines the contour end

U = "X" axis finishing allowance (positive for external or negative for internal / diameter)

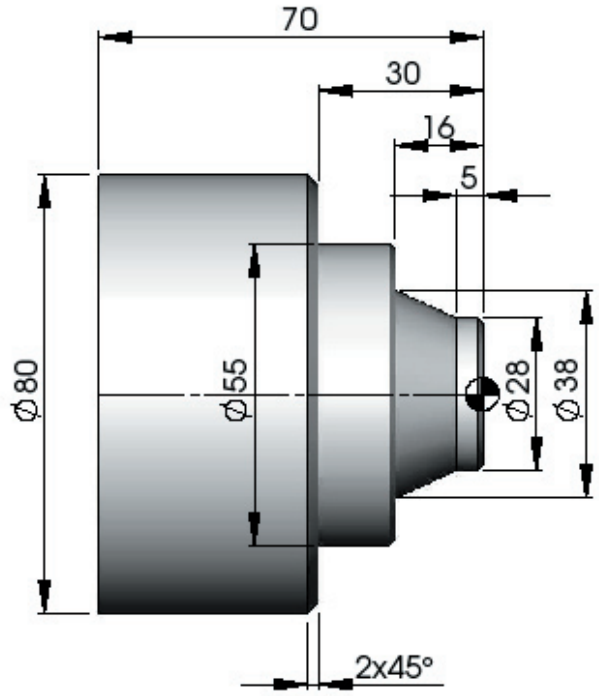
W = "Z" axis finishing allowance (positive for stock removal at contour right or negative for stock removal at contour left)

F = work feed

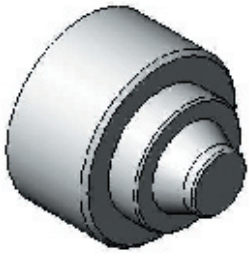
NOTE: *After execution of the cycle, the tool automatically returns to the positioning point.*

IMPORTANT*: THE PROGRAMMING OF WORKPIECE FINISHING CONTOUR MUST BE DEFINED FROM LEFT TO RIGHT.*

Example 1: External machining



Chamfer not indicated corner with 1x45°

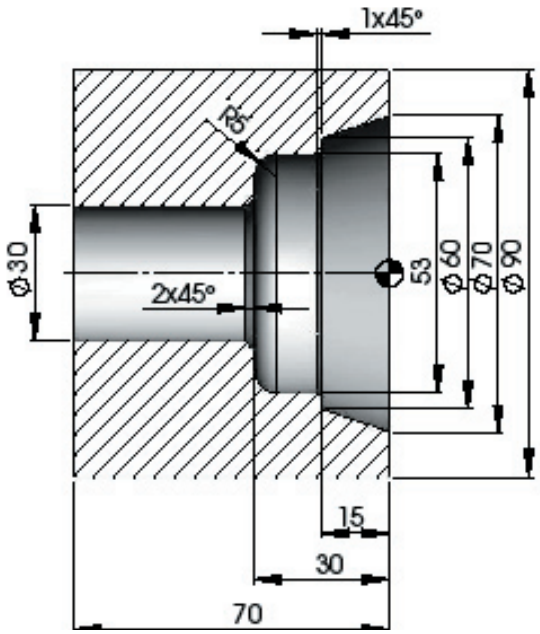


```
%_N_ROUGHING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G54 G0 X200 Z250
N50 T0301 (ROUG.)
N60 G96 S200
N70 G92 S3500 M3
N80 G0 X84 Z1
N90 G72 W2 R1.
N100 G72 P110 Q190 U1 W.3 F.25
N110 G0 Z-32
N120 G1 X80
N130 X76 Z-30
N140 X55
N150 Z-16 ,C1
N160 X38
N170 X28 Z-5
N180 Z-1
N190 X26 Z0
N200 G70 P110 Q190 F.18
N210 T00
N220 G54 G0 X200 Z250
N230 M30
```

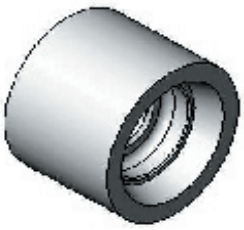
Cutting depth = 2 mm
Roughing feed = 0.25 mm/rot
Finishing feed = 0.18 mm/rot

NOTE: *In this example it was considered that roughing and finishing cycles have used the same tool.*

Example 2: Internal machining



Chamfer corners with 1x45°



```
%_N_ROUGHING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G56 G0 X180 Z200
N50 T0701 (ROUGH.)
N60 G96 S240
N70 G92 S2500 M3
N80 G0 X28 Z1
N90 G72 W2.5 R1.5
N100 G72 P110 Q170 U1 W.3 F.3
N110 G0 Z-32
N120 G1 X30
N130 X34 Z-30
N140 X43 ,R5
N150 Z-15 ,C1
N160 X60
N170 X70 Z0
N180 G70 P100 Q180 F.2
N190 T00
N200 G56 G0 X180 Z200
N210 M30
```

Cutting depth = 2.5 mm
Roughing feed = 0.3 mm/rot
Finishing feed = 0.2 mm/rot

NOTE: *In this example it was considered that roughing and finishing cycles have used the same tool.*

9.4 - FUNCTION: G73

Application: Roughing automatic cycle parallel to final contour.

The cycle G73 allows a complete workpiece rough machining using only two programming blocks.

The function G73 is specific for cast and forged materials because tool path is always parallel to the defined contour.

The function G73 requires:

G73 U__ W__ R__; where:

U = direction and stock removal on “X” axis per pass (radius).

W = direction and stock removal on “Z” axis per pass.

R = roughing number of pass

Formula for “U” and “W” values calculation:

$$U = \frac{\text{“X” stock removal (radius)} - \text{“X” Finishing allowance (radius)}}{\text{Number of passes (R)}}$$

$$W = \frac{\text{“Z” Stock removal} - \text{“Z” Finishing allowance}}{\text{Number of passes (R)}}$$

G73 P__ Q__ U__ W__ F__; where:

P = number of block that defines the contour start

Q = number of block that defines the contour end

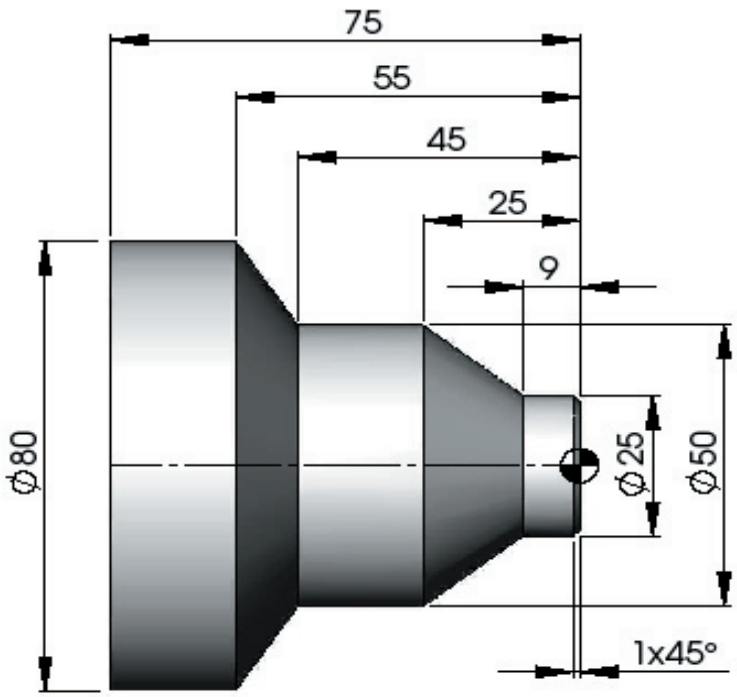
U = “X” axis finishing allowance (positive for external or negative for internal / diameter)

W = “Z” axis finishing allowance (positive for stock removal at contour right or negative for stock removal at contour left)

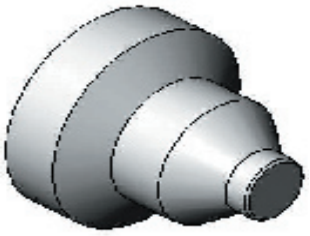
F = work feed

NOTE: *After cycle execution the tool returns automatically to the positioning point.*

Example 1: External machining




```
%_N_ROUGHING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
 N20 G21 G40 G90 G95
 N30 T00
 N40 G55 G0 X200 Z230
 N50 T0201 (ROUGH.)
 N60 G96 S240
 N70 G92 S3000 M3
 N80 G0 X90 Z5
 N90 G73 U2 W1.35 R2
 N100 G73 P100 Q160 U2 W.3 F.2
 N110 G0 X23 Z2
 N120 G1 Z0
 N130 X25 Z-1
 N140 Z-9
 N150 X50 Z-25
 N160 Z-45
 N170 X80 Z-55
 N180 G70 P100 Q170 F.18
 N190 T00
 N200 G55 G0 X200 Z230
 N210 M30
```



In the example it was considered:
2-pass Roughing
"X" stock removal = 10 mm (Ø)
"Z" stock removal = 3 mm
"X" Finishing allowance = 2mm (Ø)
"Z" Finishing allowance = 0.3mm
Roughing feed = 0.2 mm/rot
Finishing feed = 0.18 mm/rot

NOTE: *In this example it was considered that roughing and finishing cycles have used the same tool.*

9.5 - FUNCTION: G74

9.5.1 - Drilling cycle.

The function G74 as a drilling cycle requires:

G74 R__;
G74 Z__ Q__ F__; where:

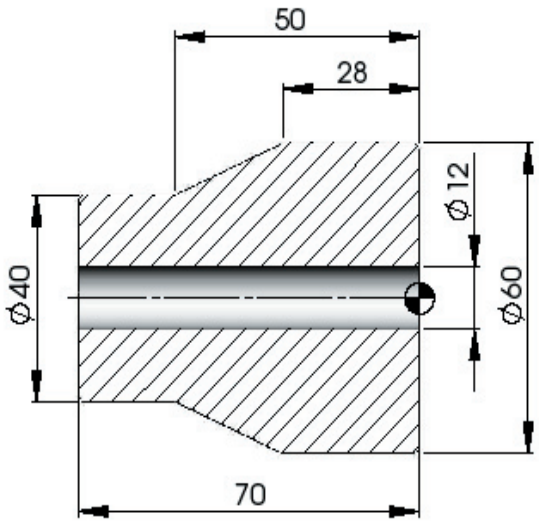
R = incremental return for chip breaking during drilling cycle

Z = final position (absolute)

Q = incremental value for drilling cycle (thousandths of millimeter).

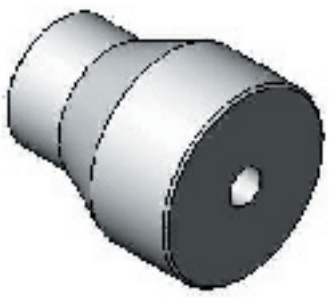
F = work feed

> ***NOTE:*** *After cycle execution the tool returns automatically to the positioning point.*
>
> ***NOTE:*** *When cycle G74 is used as a drilling cycle, functions "X" and "U" can not be informed in the block.*




```
%_N_DRILLING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G59 G0 X170 Z250
N50 T0501 (DRILL D12)
N60 G97 S1200 M3
N70 G0 X0 Z5
N80 G74 R2
N90 G74 Z-74 Q15000 F.12
N100 T00
N110 G59 G0 X170 Z250
N120 M30
```

Drilling increment = 15 mm
Feed = 0.12 mm/rot



9.5.2 - Turning cycle.

The function G74 as a turning cycle requires:

G74 X__ Z__ P__ Q__ R__ F__; where:

X = turning final diameter

Z = final position (absolute)

P = cutting depth (radius/ thousand millimeter)

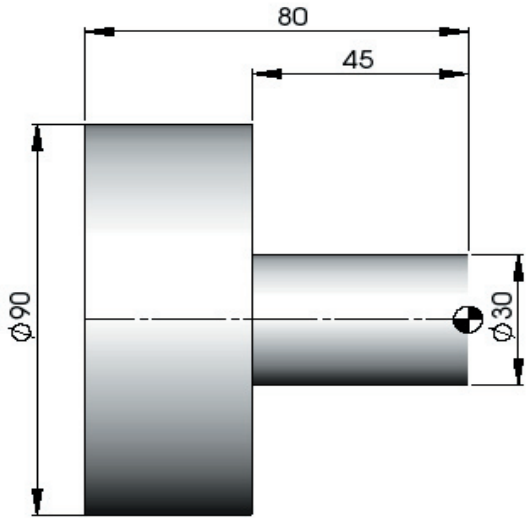
Q = cutting length (incremental/ thousand millimeter)

R = distance value in cross axis (radius)

F = work feed

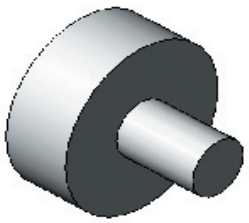
NOTES: *1. To execute this cycle, tool must be positioned on the first pass diameter.*

2. After cycle execution the tool returns automatically to the positioning point.




```
%_N_TURNING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G55 G0 X200 Z250
N50 T0201 (ROUGH.)
N60 G96 S250
N70 G92 S3500 M3
N80 G0 X84 Z2
N90 G74 X30 Z-45 P3000 Q47000 R1 F.2
N100 T00
N110 G55 G0 X200 Z250
N120 M30
```

Cutting depth = 3 mm
Feed = 0.2 mm/rot



9.6 - FUNCTION: G75

9.6.1 - Grooving cycle.

The function G75 as a grooving cycle requires:

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

R = incremental return for chip breaking (radius)

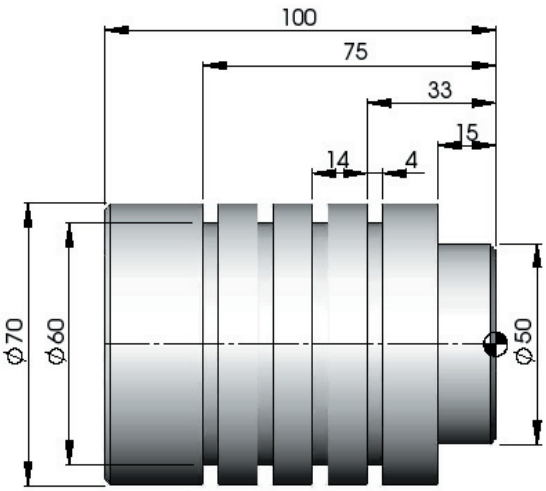
X = groove final diameter

Z = final position (absolute)

P = cutting increment (radius / thousand millimeter)

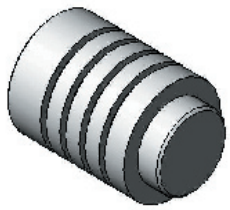
Q = distance between grooves (incremental / thousand millimeter)

F = work feed




```
%_N_GROOVE_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G54 G0 X195 Z300
N50 T0201 (GROOVES)
N60 G96 S130
N70 G92 S2000 M3
N80 G0 X75 Z-33
N90 G75 R2
N100 G75 X60 Z-75 P7500 Q14000 F.2
N110 T00
N120 G54 G0 X195 Z300
N130 M30
```

Feed = 0.2 mm/rot



9.6.2 - Facing cycle.

The function G75 as a facing cycle requires:

G75 X__ Z__ P__ Q__ R__ F__; where:

X = facing final diameter

Z = final position (absolute)

P = “X” axis cutting increment (radius / thousand millimeter)

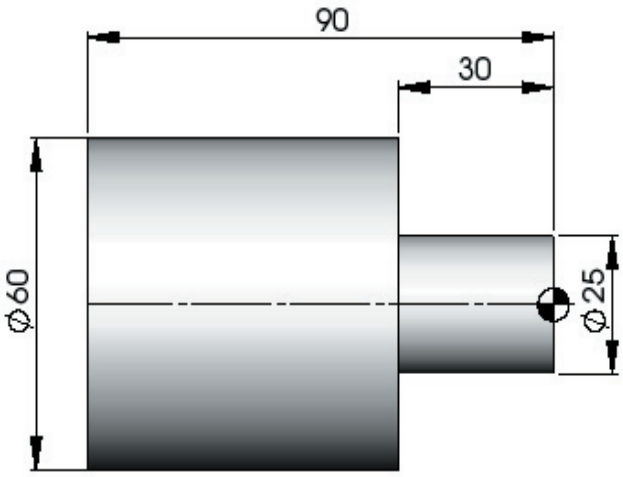
Q = cutting depth per pass on “Z” axis (thousand millimeter)

R = distance value in the longitudinal axis for returning to initial “X” (radius)

F = programmed feed

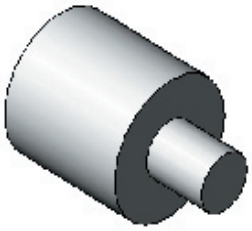
NOTES: *1. To execute this cycle, tool must be positioned on the lenght of 1st roughing pass.*

2. After cycle execution the tool returns automatically to the positioning point.




```
%_N_FACING_MPF
;$PATH=/_N_MPF_DIR
 N10 G291
 N20 G21 G40 G90 G95
 N30 T00
 N40 G57 G0 X200 Z250
 N50 T0701 (rough.)
 N60 G96 S210
 N70 G92 S3500 M3
 N80  G0 X64 Z-2
 N90  G75 X25 Z-30 P19500 Q2000 R1 F.2
 N100 T00
 N110  G57 G0 X200 Z250
 N120 M30
```

Cutting depth = 2 mm
Feed = 0.2 mm/rot



9.7 - FUNCTION: G76

Application: Automatic threading cycle

The function G76 requires:
G76 P (m) (s) (a) Q__ R__; where:

m = number of repetition of the last pass

s = thread angular output = Number of tapered fillets x 10

NOTE: *The thread tool angular output in this cycle is always 45º.*
The minimum value to be programmed to the thread angular output is 10 (that mean 1 tapered fillets), if you program "00" , won' t be executed the thread angular output.

a = tool angle (0º, 29º, 30º, 55º and 60º, etc.)

Q = minimum cutting depth (radius / thousand millimeter)

R = last pass depth (radius)

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

X = threading final diameter

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

Z = threading final length

W = incremental distance from positioned point (initial "Z") to longitudinal axis final coordinate (final "Z").

R = "X" axis incremental tapering value (radius / negative for external and positive for internal)

P = thread fillet height (radius / thousandth millimeter)

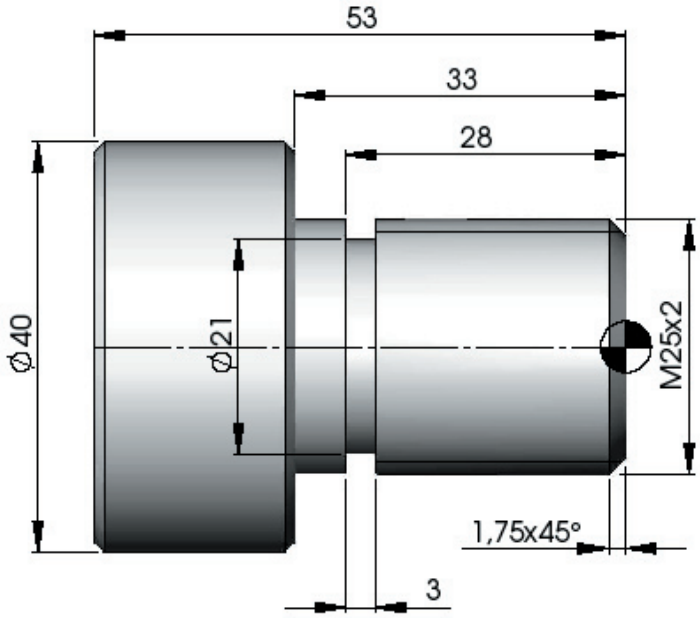
Q = 1st pass depth (radius/ thousandth millimetero)

F = thread pitch

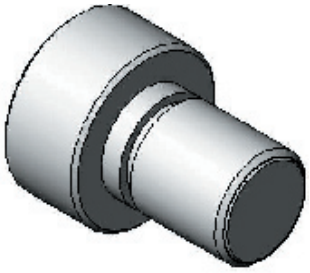
NOTE: *The positioning diameter must be higher than the angular output final diameter.*
Example: To program a M20x2 thread with an angular output equal to 15, the angular output final diameter will be 26, therefore, the positioning must be higher than 26, as shown in the table below:

CORRECT PROGRAM	CALCULATION:
G00 **X27** Z57 G76 P01**15**60 Q100 R.1 G76 X17.4 Z41 P1300 Q392 F2	Angular output final diameter: (angular output x pitch x 2) + Thread diameter (1.5 x 2 x 2) + 20 = 26

Example 1: Thread M25x2




```
%_N_THREAD_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G58 G0 X190 Z200
N50 T0101 ( THREAD M25X2)
N60 G97 S1000 M3
N70 G00 X29 Z4
N80 G76 P010060 Q100 R0.1
N90 G76 X22.4 Z-26.5 P1300 Q392 F2
N100 T00
N110 G58 G0 X190 Z200
N120 M30
```



CALCULATION:

1º) Fillet height (P):
P = (0.65 x pitch)
P = (0.65 x 2)
P = 1.3

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

3º) First pass depth (Q):

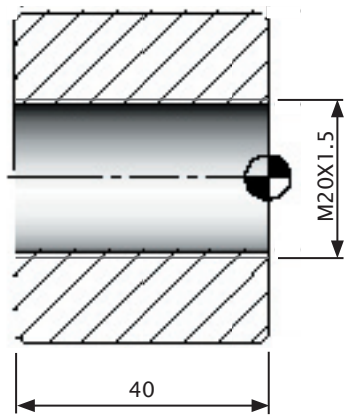
$$Q = \frac{P}{\sqrt{N.\ Passes}}$$

NOTE: In the example, calculation for 11 passes.

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

Q= 0.392

Example 2: Internal Thread M20x1.5




```
%_N_INTERNAL_ THREAD_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G54 G0 X200 Z200
N50 T0201 ( M20X1.5)
N60 G97 S1000 M3
N70 G00 X16 Z4
N80 G76 P010060 Q100 R0.1
N90 G76 X20. Z-43 P975 Q325 F1.5
N100 T00
N110 G54 G0 X200 Z200
N120 M30
```

CALCULATION:

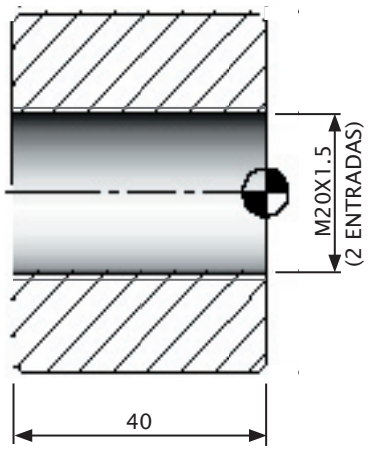
1º) Fillet height (P):
P = (0.65 x pitch)
P = (0.65 x 1.5)
P = 0.975

2º) First pass depth (Q):

$$Q = \frac{P}{\sqrt{N.\ Passes}} \qquad Q = \frac{0.975}{\sqrt{9}} \qquad Q = 0.325$$

***NOTE**: In this example, it was considered a calculation for 9 passes.*

Example 3: Internal Thread M20x1.5 (2 starts)



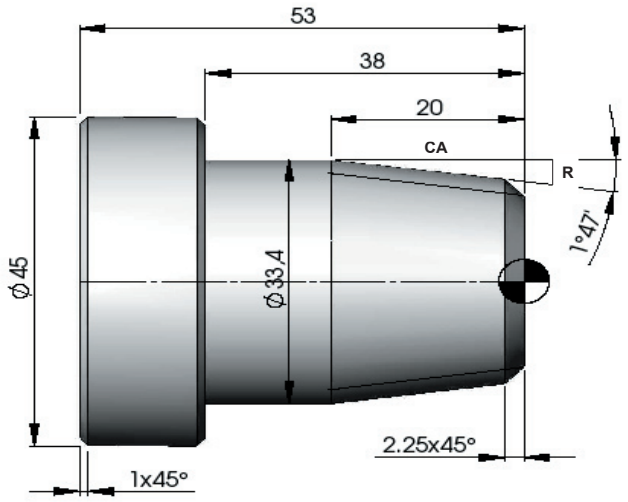

```
%_N_THREAD_INT_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G54 G0 X175 Z200
N50 T0201 (THREAD M20X1.5)
N60 G97 S1000 M3
N70 G00 X16 Z6 (1ST START)
N80 G76 P010060 Q100 R0.1
N90 G76 X20. Z-43 P975 Q325 F3
N100 G00 X16 Z7.5 (2ND START)
N110 G76 P010060 Q100 R0.1
N120 G76 X20. Z-43 P975 Q325 F3
N130 T00
N140 G54 G0 X175 Z200
N150 M30
```

***NOTA:** For multiple-start thread it is required to calculate feed (F) as follows:*

F =Pitch x Nº of starts
F = 1.5 x 2
F = 3

TAPER THREAD:

Example 4: Taper Thread NPT 11.5 fillets/inch
(Inclination: 1 degree 47 min)




```
%_N_THREAD_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G55 G0 X200 Z270
N50 T0201 (THREAD NPT)
N60 G97 S1200 M3
N70 G0 X37 Z5
N80 G76 P010060 Q150 R0.12
N90 G76 X29.574 Z-20 P1913 Q479
R-0.778 F2.209
N100 T00
N110 G55 G0 X200 Z270
N120 M30
```

CALCULATION:

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) Inclination degree conversion:

1° 60’
A° 47’

60 x A = 47 x 1
A =47 / 60
A = 0.783°

Therefore: 1°47’ = 1.783°

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

4º) First pass depth (Q):

$$Q = \frac{P}{\sqrt{\text{Nº PASSES}}}$$

Example: 16 passes.

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

Q = 0.479

6º) “X” axis incremental tapered angle (R):
R = (tan α) x CA
R = tan 1.783° x 25
R = 0.778

10 - DRILLING CYCLES

10.1- FUNCTION : G80

Application : Cancels cycles of G80 series

This function is used to cancel cycles of G80 series, that is, from G83 to G85.

10.2- FUNCTION : G83

Application : Drilling cycle

This cycle enables to execute holes with chip removal. Also a dwell can be programmed at the drilling end point, as shown below:

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

Z = Hole final position (absolute)

Q = Increment value (incremental / thousandth)

P = Dwell time at each increment end (thousandth second)

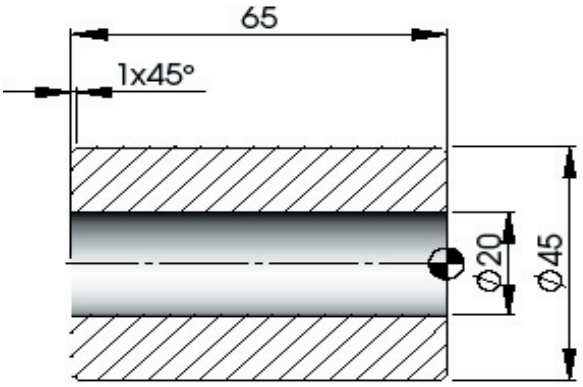
R = Drilling start reference plane (incremental)

F = Feed

NOTES:

- *After cycle execution, the tool returns to initial point.*
- *IF "R" is not programmed, drilling start will be performed from "Z" approach.*
- *To run cycle G83 with "chip unload", perform the procedure below:*
  - Press **"SHIFT"** and **"SYSTEM ALARM"** keys simultaneously
  - Press ***[∧]*** softkey.
  - Press ***[SET PASSWORD]*** softkey.
  - Enter **CUSTOMER** password.
  - Press ***[ACCEPT]*** softkey.
  - Press **"OFFSET PARAM"** key
  - Press ***[USER DATA]*** softkey.
  - Place cursor (▲, ▼) on **ZSFR** parameter **[21]**
  - Enter approach value. Example: **2**
  - Place cursor (▲, ▼) on **ZSFI** parameter **[20]**
  - Enter **"1"** to enable unload function
  - Press **"INPUT"** key
  - Press ***[BACK]*** softkey.

EXAMPLE :




```
%_N_DRILLING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G57 G0 X180 Z300
N50 T0201 (DRILL)
N60 G97 S1500 M3
N70 G0 X0 Z3
N80 G83 Z-68 Q15000 P1500 R -2 F0.12
N90 G80
N100 T00
N110 G57 G0 X180 Z300
N120 M30
```

10.3 - FUNCTION : G84

Application : Rigid tapping cycle

This cycle enables to cut threads with tap, by using rigid support, that is, without floating support. For this:

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

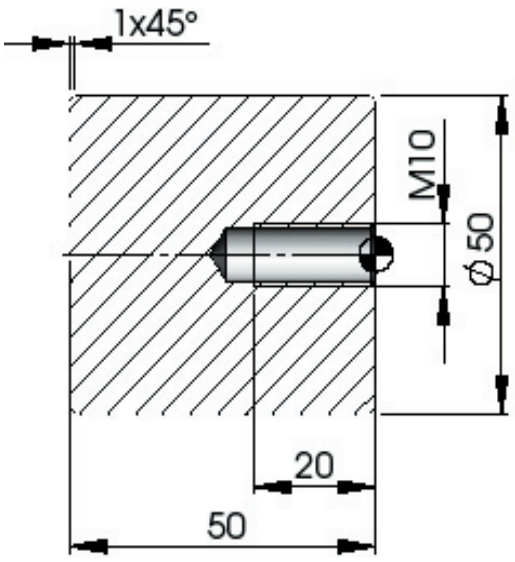
M3 = indicates rotation direction (M3 = right thread and M4 = left thread)

M29 = activates rigid tapping

Z = thread final position

F = thread pitch

EXAMPLE




```
%_N_TAPPING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G56 G0 X190 Z215
N50 T0601 (RIGID TAP)
N60 G97 S500 M3
N70 G0 X0 Z4
N80 M29
N90 G84 Z-20 F1.5
N100 G80
N110 T00
N120 G56 G0 X190 Z215
N130 M30
```

10.4 - FUNCTION : G85

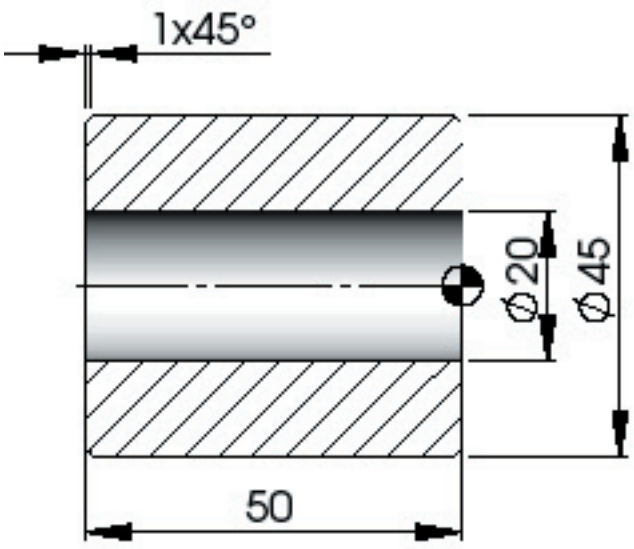
Application : Boring cycle

G85 Z__ F__, where

Z = Final position

F = Feed

EXAMPLE:




```
%_N_BORING_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G59 G0 X150 Z200
N50 T0801 (BORING)
N60 G97 S750 M3
N70 G0 X0 Z2
N80 G85 Z-55 F0.5
N90 G80
N100 T00
N110 G59 G0 X150 Z200
N120 M30
```

NOTE: The outfeed is the double of the programmed value for turning.

11 - OTHER PRELIMINARY FUNCTIONS

11.1 - FUNCTION: G20

Application: Measurement unit reference - Inches

This function prepares the control to compute all data input in inches.

11.2 - FUNCTION: G21

Application: Measurement unit reference - Metric

This function prepares the control to compute all data input in millimeters.

11.3 - FUNCTION: G90

Application: Absolute coordinate system

This code prepares the machine to execute absolute coordinate operations, having a pre-established origin for programming. Function G90 is modal and cancels function G91.

11.4 - FUNCTION: G91

Application: Incremental coordinate system

This code prepares the machine to execute incremental coordinates operations. Therefore all measurements are taken through the distance to be shifted. Thus, coordinate origin of any point is the point before shifting. Function G91 is modal and cancels function G90.

11.5 - FUNCTION: G92

11.5.1 - G92: Establishes rotation limit (RPM)

The function **G92** together with **S____** code (4 digits) is used to limit spindle maximum rotation (RPM). Usually this function is programmed in the block subsequent to function G96, which is used to program cutting speed.

Example: **G92 S2500 M4;** (limits spindle rotation at 2500 RPM)

11.5.2 - G92 : Establishes new origin

The function G92 may also be used to establish a new origin for coordinate system. Therefore, it must be programmed in a block together with one or more machine axes.

Example: **G92 Z0;** (establishes the new origin for coordinate system)

The function G92 is modal and is canceled by function G92.1 .

11.5.3 - G92.1 : Cancels G92 (cancels zero-base)

The function G92.1 is used to cancel command G92. Therefore, it must be programmed in a block together with the axis to be canceled.

Example: **G92.1 Z0;** (cancels coordinate system origin corresponding to “Z” axis).

11.6 - FUNCTION: G94

Application: Establishes feed in mm/minute or inch/minute.

This function prepares the control to compute all feeds in inches/**minutes** (**G20**) or millimeters/minutes (**G21**). The function G94 is modal and cancels the function G95.

11.7 - FUNCTION: G95

Application: Establishes feed in mm/rotation or inch/rotation:

This function prepares the control to compute all feeds in inches/**rotation** (**G20**) or millimiters/rotation (**G21**). The function G95 is modal and cancels the function G94.

11.8 - FUNCTION: G96

Application: Establishes programming in constant cutting speed.

The function G96 selects programming mode in constant cutting speed, where RPM calculation is programmed by function **“S”**.

The maximum RPM reached by constant cutting speed may be limited through function G92 programming.

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

The function G96 is modal and cancels the function G97.

11.9 - FUNCTION: G97

Application: Establishes RPM programming

The function G97 is used to program a spindle fixed rotation (RPM), helped by function S and by using a format (S4).

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

RPM may vary through “Spindle Rotation Selector”, from 50% up to 120% of the programmed rotation. The function G97 is modal and cancels the function G96.

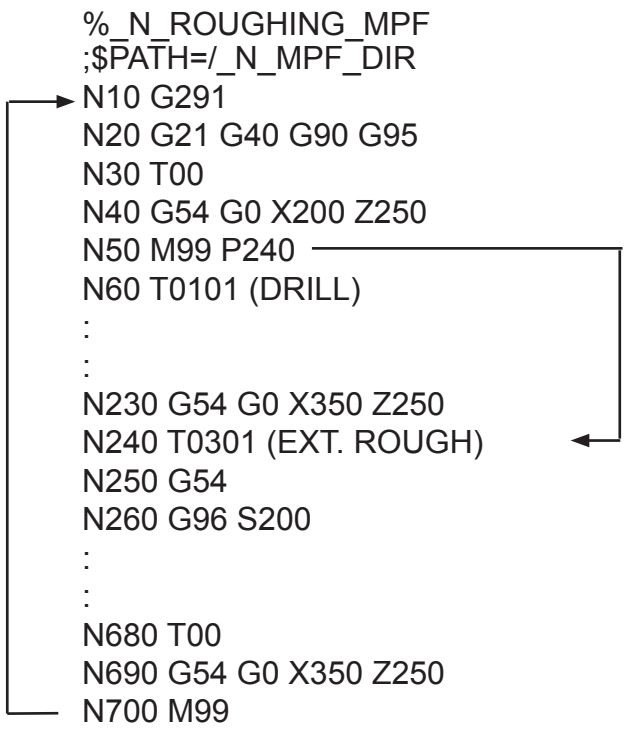
12 - UNCONDITIONAL DEVIATION

Function: M99

Programming function M99 with function “P”, followed by the block number makes the control forward/return to the block indicated by “P”.

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

EXAMPLE:



13 - SUBPROGRAM CALL AND RETURN

Functions: M98/M99

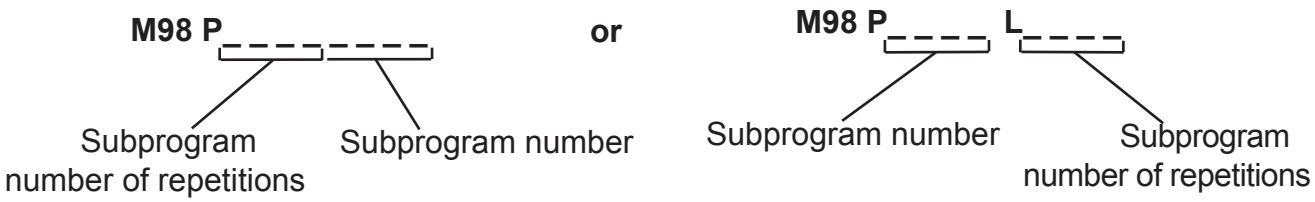
When the machining of a certain operating sequence appears many times in the program, subprogram call resource through M98 function may be used.

The block that provides M98 function must also provide the subprogram number through the function “P”. E.g.: **M98 P1001.**

The subprogram must be ended with a function M99 and its name must have 4 numerical characters.

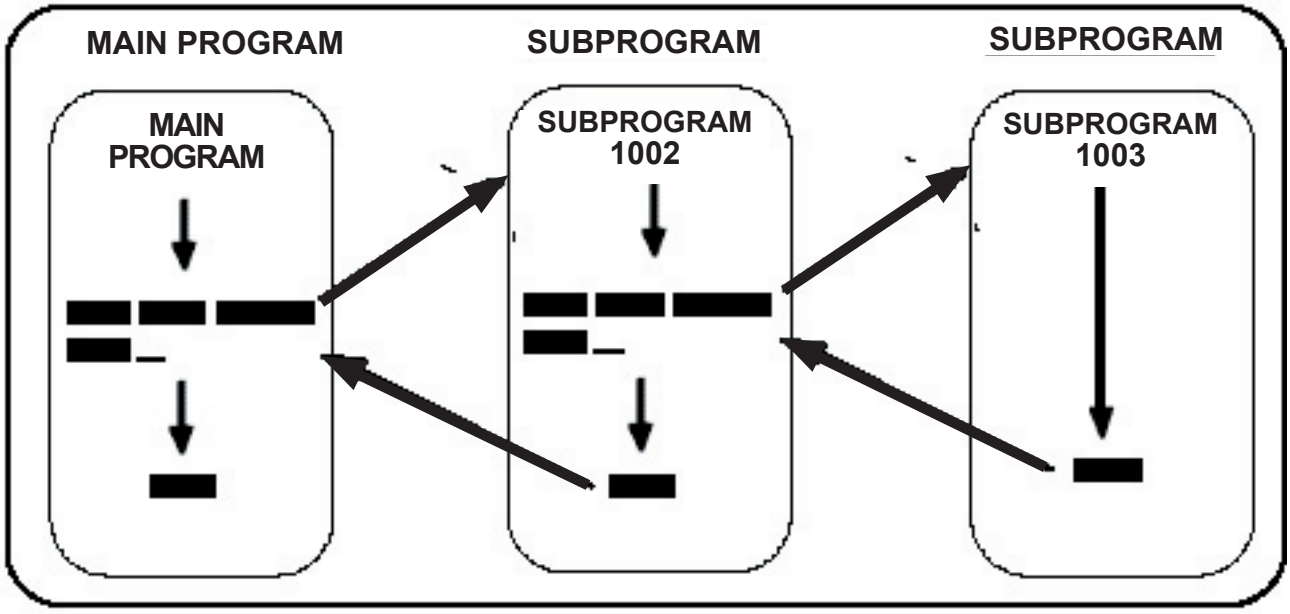
> **NOTE:** The subprogram number is the same found on CNC directory.

The formats for calling a subprogram are the following:



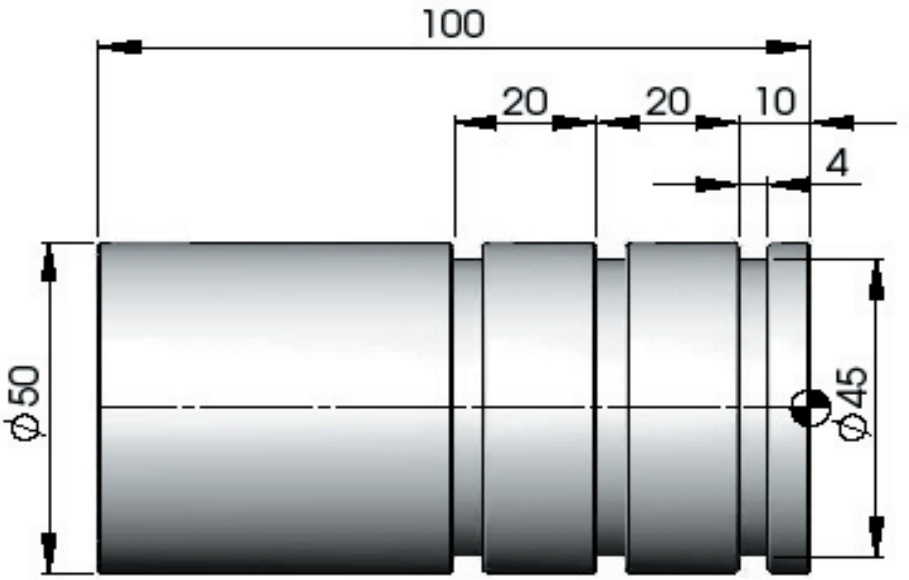
When a subprogram ends its operations, the control returns to the main program.

EXAMPLE:

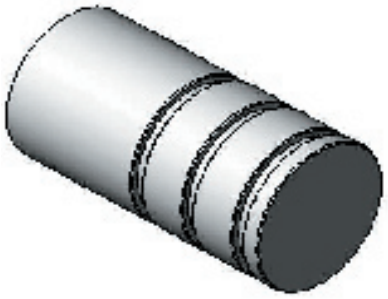


> ***NOTE:*** *If a number of repetitions is omitted, the control will execute the subprogram once.*

EXAMPLE: INCREMENTAL PROGRAMMING AND SUBPROGRAM



Chamfer corners with 0.5x45°



Main program

```
%_N_MAIN_MPF
;$PATH=/_N_MPF_DIR
N10 G291
N20 G21 G40 G90 G95
.
.
.
N90 G0 X55 Z10
N100 M98 P30002
(N100 M98 P2 L3;)
N110 T00
N120 G54 G0 X250 Z130
N130 M30
```

Subprogram 0002

```
%_N_0002_SPF
;$PATH=/_N_SPF_DIR
N10  G0 G91 Z-20;
N20  G1 G90 X45 F.15;
N30  G0 X55;
N40  G91 Z-0.5;
N50  G1 G90 X50;
N60  G91 X-1 Z0.5;
N70  G0 G90 X55;
N80  G91 Z0.5;
N90  G1 G90 X50;
N100  G91 X-1 Z-0.5;
N110  G0 G90 X55;
N120  M99;
```

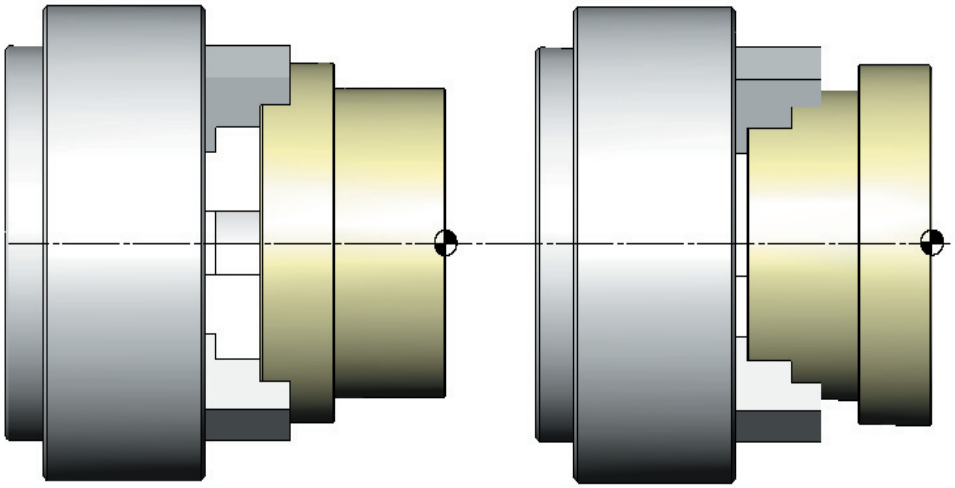
14 – WORK REFERENCE (G54 TO G59)

Work Reference also known as Work Offset corresponds to the point that is the origin for absolute coordinate system, that is, the workpiece point referenced as “X0” and “Z0”.

In some cases more than one work reference are used in a same program to ease programming of certain workpieces. Example: to program the machining of both sides of a workpiece in a same program, it is advisable to use two work offsets so the operator should not to be concerned about some matters as stock removal of both sides of blank, different jaw supports, etc.

NOTE: *On “CENTUR 30D” line machines up to six work offsets can be manually referenced during machine setup: G54, G55, G56, G57, G58 and G59.*

EXAMPLE:



G54 (1ST NECKING) **G55 (2ND NECKING)**

G54 family values must be input on page “OFFSET PARAM” through “WORK OFFSET” softkey .

15 - MISCELLANEOUS OR AUXILIARY FUNCTIONS

Miscellaneous functions are programmed to execute machine operations and resources that are not covered by preliminary functions.

MISCELLANEOUS FUNCTIONS		
Command M	Application	Optional
M00	Program stop	
M01	Program optional stop	
M02	Program end	
M03	Turn spindle clockwise	
M04	Turn spíndle counter clockwise	
M05	Turn spindle OFF	
M07	Turn coolant 2 ON	X
M08	Turn coolant 1 ON	
M09	Turn coolant OFF	
M19	Orient spindle / Turn C axis ON	
M20	Turn bar feeder ON	X
M21	Turn bar feeder OFF	X
M24	Open chuck	X
M25	Clamp chuck	X
M26	Retract tailstock quill	X
M27	Forward tailstock quill	X
M30	Program end	
M34	Select pressure 1 for chuck	X
M35	Select pressure 2 for chuck	X
M36	Open automatic door	X
M37	Close automatic door	X
M38	Forward parts catcher	X
M39	Retract parts catcher	X
M45	Turn protection cleaning ON	X
M46	Turn protection cleaning OFF	X
M47	Turn chip conveyor ON	X
M48	Turn chip conveyor OFF	X
M49	Change bar	X
M52	Open steady rest	X
M53	Closed steady rest	X
M78	Turn mist exhaust ON	x
M79	Turn mist exhaust OFF	X
M81	Select external gripping	
M82	Select internal gripping	
M83	Enable spindle rotation with open chuck	

16 - SEQUENCE FOR HANDWRITTEN PROGAMMING

The programmer should be conscious of all parameters involved in the process and should obtain a proper solution for machining each kind of workpiece. He should also analyze all machine resources that will be required for the workpiece execution.

16.1 - WORKPIECE DRAWING STUDY: FINISHED AND BLANK

The programmer should have ability to compare the drawing (ready workpiece) with the dimension wanted in the machining with a Numercal Control machine.

It is required an analysis on viability of a part execution, taking into consideration required dimensions, stock removal of the previous phase, required tooling, part fixing, etc.

16.2 - PROCESS TO BE USED

It is required to define the machining stages for each part to be machined to determine an adequate holding system for the machining process.

16.3 - CNC TOOLING

Tool selection and its arrangement on the turret is very important. It is required that the tools are installed in such a way to avoid interference between themselves and with the machine. A good program depends a lot on the selection of appropriate tooling and suitable holding.

16.4 - KNOWLEDGE OF MACHINE PHYSICAL PARAMETERS AND OF THE CONTROL PROGRAMMING SYSTEM

The programmer must have this knowledge to carry out operations using all machine and control resources aiming at minimizing operation times and stages and to guarantee the product quality.

16.5 - DEFINITION ACCORDING TO MATERIAL, CUTTING PARAMETERS (FEED RATE, SPEED, ETC.)

Taking into consideration the material to be machined as well as the tool to be used and the operation to be executed, the programmer must establish cutting speeds, feed rates and power required by the machine. Calculations required to obtain these parameters are the following:

17 - CALCULATION

17.1 - CUTTING SPEED (VC)

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

Cutting speed is a value directly proportional to the diameter and to the main spindle rotation given by the formula:

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

where:

VC = Cutting speed (m/min)

$Ø_P$ = Workpiece diameter (mm)

N = Spindle rotation (rpm)

17.2 - ROTATION (N)

When determining the feed rate for a specific tool to execute a machining process, the spindle rotation is given by the following formula:

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

17.3 - CUTTING POWER (NC)

To avoid some inconveniences during machining process such as motor overload and consequent spindle stop during operation, it is required to previously calculate the power to be consumed during the process which may be obtained by the formula:

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

where:

Ks = Specific cutting pressure

Ap = Cutting depth

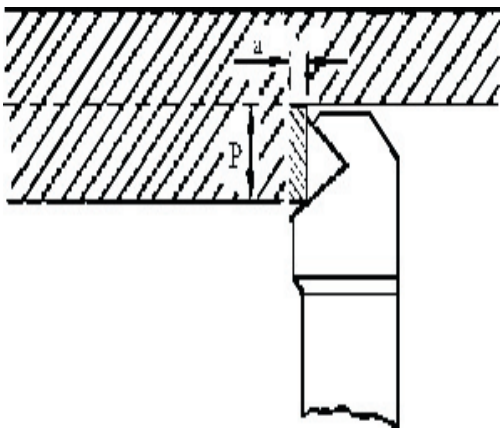
Fn = Feed

Vc = Cutting speed

η = Efficiency:

CENTUR = 0.8

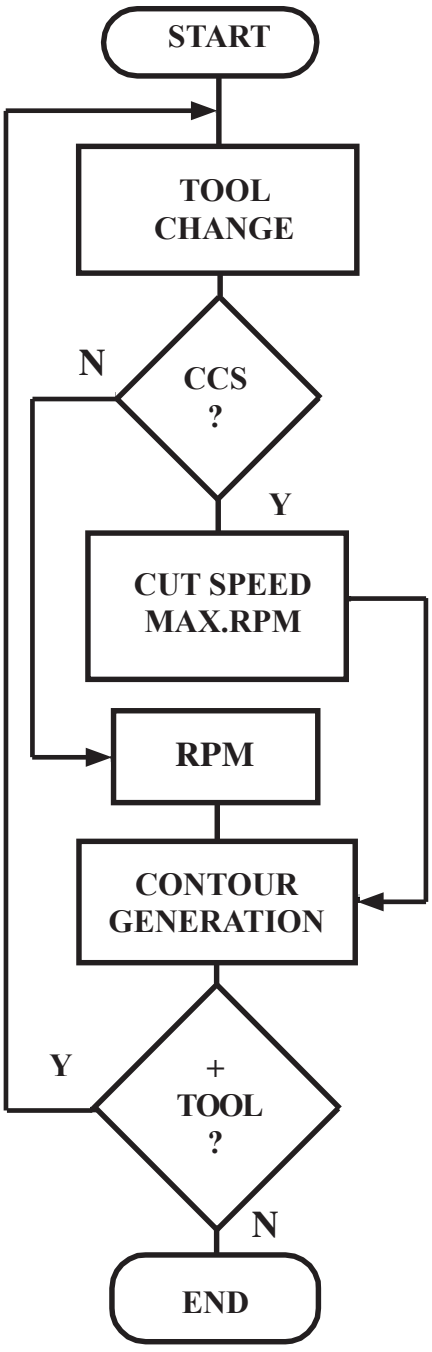
CUTTING AREA FOR 90-DEGREE TOOLS



ORIENTATIVE VALUES FOR SPECIFIC CUTTING PRESSURE (KS)

MATERIAL	TENSION RESISTANCE Kgf/mm² BRINELL HARDNESS		"KS" IN KG/MM2 FEED IN MM/ROT			
	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
MEDIUM CAST STEEL	50 to 70	138 to 192	360	260	190	136
HARD CAST STEEL	OVER 70	OVER 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
TOOLING STEEL	150 to 180	415 to 500	570	410	300	215
HARD MANGANESE STEEL			660	480	360	262
SOFT CAST IRON		UP TO 200	190	136	100	72
MEDIUM CAST IRON		200 to 250	290	208	150	108
HARD CAST IRON		250 to 400	320	230	170	120
HARDENED CAST IRON			240	175	125	92
ALUMINUM		40	130	90	65	48
COPPER			210	152	110	80
COPPER ALLOY			190	136	100	72
BRASS		80 to 120	160	115	85	60
RED BRONZE			140	100	70	62
CAST BRONZE			340	245	180	128

18- PROGRAMMING FLOW CHART SIEMENS 802D



***START**
%_N_*NAME*_MPF - program name and extension
;$PATH=/_N_MPF_DIR - program store directory
G291 - required only for machines with optional MMP
G21 G40 G90 G95 - safety block

***TOOL CHANGE**
T00 - Cancels tool reference
G5_ G00 X_ _ Z_ _ - Work offset definition (G54-G59) and tool change point
T _ _ _ _ - Desired tool number
G5_ - Work offset definition (G54 - G59)

CONSTANT CUTTING SPEED PROGRAMMING
G96 S_ _ _ - Define CCS value (m/min)
G92 S_ _ _ _ M_ _ - define max. rotation (RPM) and turn ON main spindle (M03 or M04)

***RPM PROGRAMMING**
G97 S_ _ _ _ M_ _ - define fixed rotation (RPM) and turn ON spindle (M03 or M04)

***CONTOUR GENERATION**
(instructions according to the programmer's creativity)

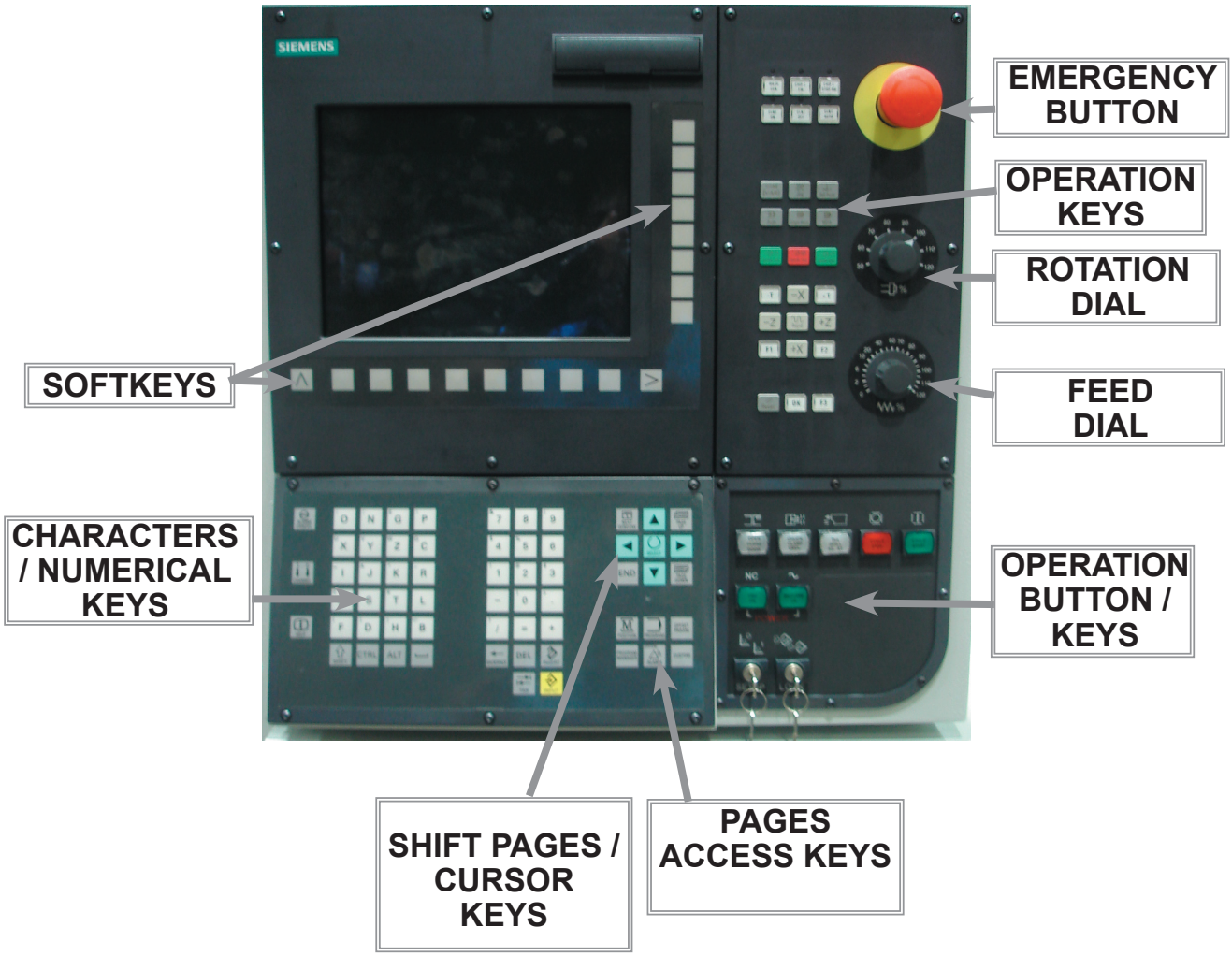
***END OF PROGRAM**
T00 - Cancels tool reference
G5_ G00 X_ _ Z_ _ - Work offset definition (G54-G59) and tool change point
M30 - end of program

PART II

OPERATION

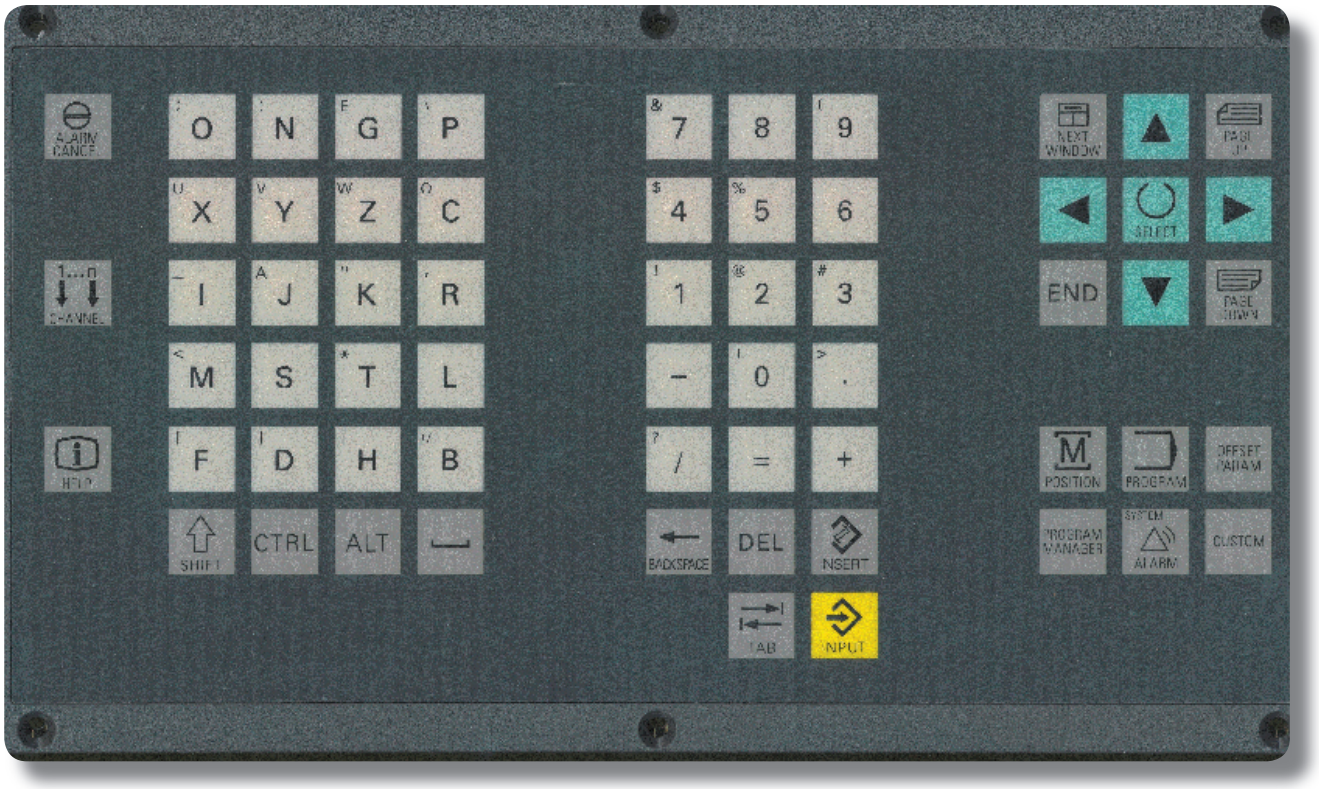
1- CONTROL PANEL

1.1 - CONTROL PANEL - CNC SIEMENS 802D



1.2 - CONTROL PANEL (KEYBOARD DESCRIPTION)

1.2.1 – Characters, numerical, cursor and page access keys

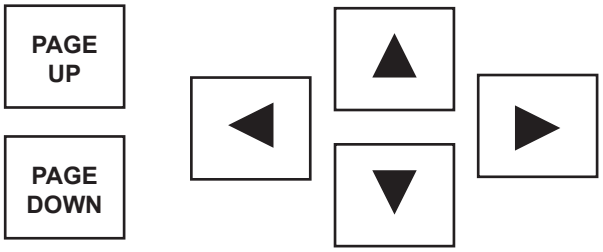


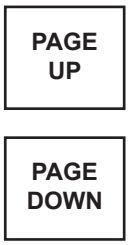
HELP	HELP: Access to Help screen such as: Machine operation or CNC alarm details screen.
PROGRAM MANAGER	PROGRAM MANAGER: Access to program directories.
OFFSET PARAM	OFFSET / PARAM: Access to tool offsets screen and definitions page.
CUSTOM	CUSTOM: Access to Romi customized software
SHIFT	SHIFT: This key enables the second function of alphanumerical and other characters keys.
DEL	DEL: Key used to delete characters in the program.
SYSTEM / ALARM	SYSTEM : Access to parameters, passwords and configuration screen. ALARM: Access to alarms and messages screen.

PROGRAM	PROGRAM: Access to the last opened / edited program.
INPUT	INPUT: This key is used to input an edited value into the program, to open and close directories and to create new directories.
SELECT	SELECT: Selection key. Note: In case an input field offers several parametrization options, it is possible to shift between parameters with this key.
POSITION	POSITION: Access to axes position / operation page.
INSERT	INSERT: Inputs characters and data into the program.
TAB	TAB: Auxiliary function
CTRL	CTRL: Auxiliary function
ALT	ALT: Auxiliary function
⎵	SPACE: Inserts spaces between characters.
BACK SPACE	BACKSPACE: Deletes last typed character or symbol.
END	END: Places cursor at the end of line in program editing.
ALARM CANCEL	ALARM CANCEL: Used to cancel some software alarms.
NEXT WINDOW	NEXT WINDOW: Not used.
CHANNEL	CHANNEL: Not used.

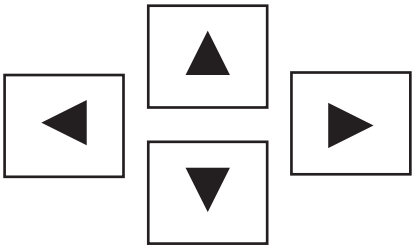
A,B,C...	A, B, C,...: Character keys.
1,2,3,...	1,2,3,...: Numerical keys.

Details of page / screen shifting





PAGE DOWN/PAGE UP: Keys responsible for changing to previous or next page

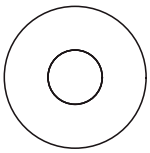


CURSOR MOVEMENT: Keys responsible for cursor movement (right, left, up and down).

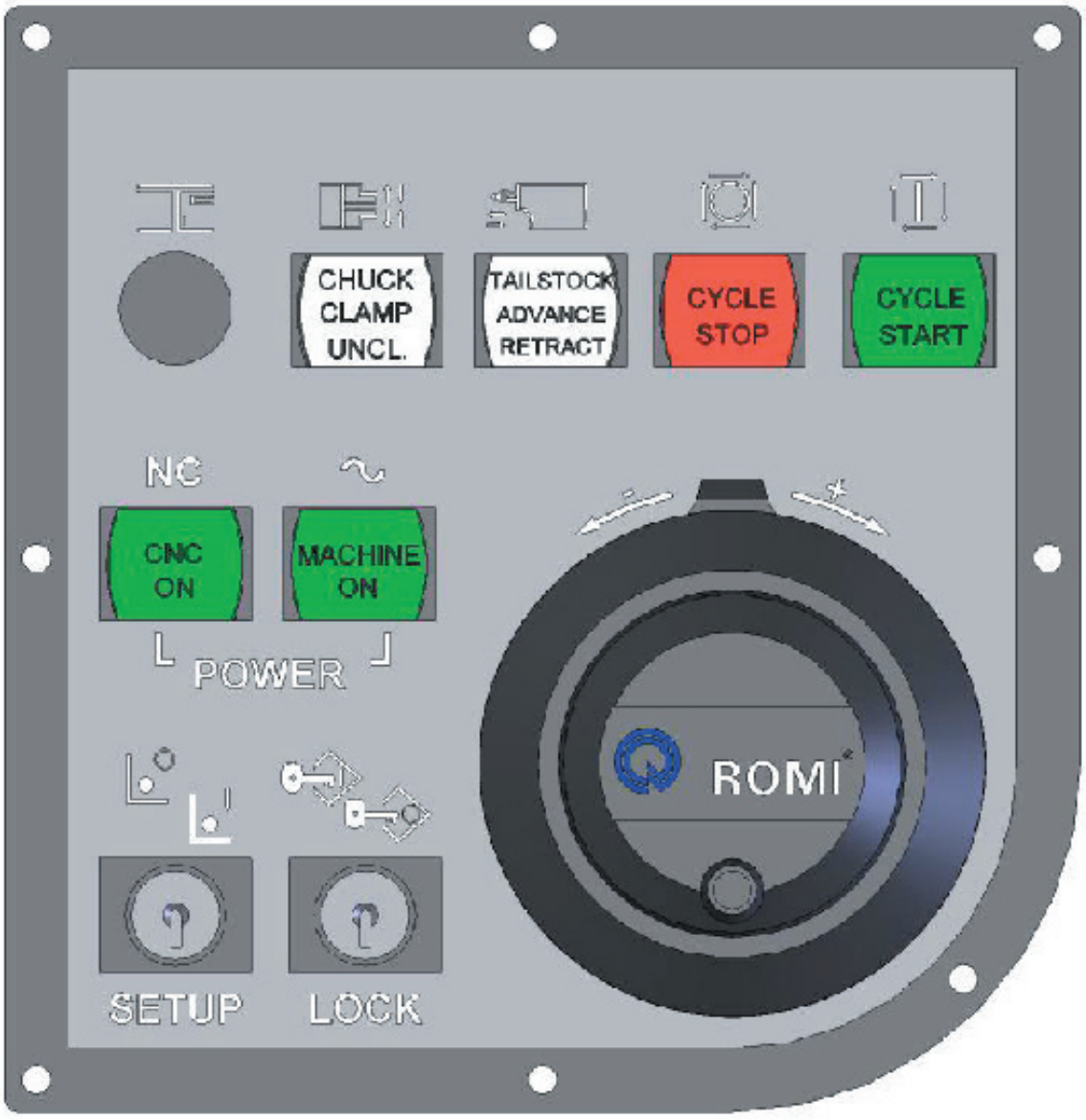
1.2.2 – Operation keys, emergency button, rotation and feed dials.

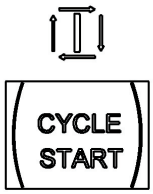
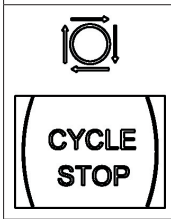
WASH GUN	WASH GUN: ON/OFF.
CHIP C FW	CHIP CONVEYOR FW: Turns ON chip conveyor forward.
CHIP C STOP RW	CHIP CONVEYOR STOP / RW: Turns OFF chip conveyor / Turn OFF chip conveyor in reverse direction.
CLNT ON	CLNT ON: Turns ON coolant system.

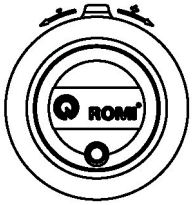
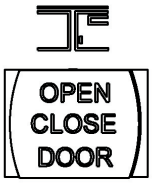
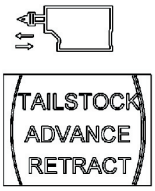
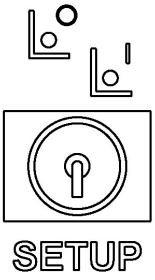
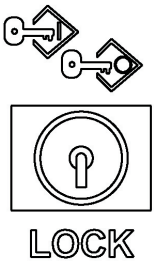
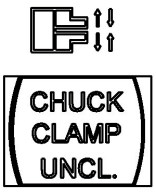
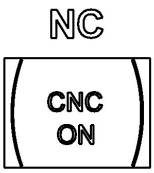
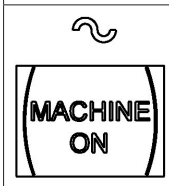
CLNT OFF	CLNT OFF: Turns OFF coolant system.
CLNT AUTO	CLNT AUTO: Enables cooling system to be started from program.
VAR	VAR: Increment selection (1x, 10x, 100x and INC) in handwheel mode.
JOG	JOG: Enables manual operation mode.
REF POINT	REF POINT: Enables machine reference mode.
AUTO	AUTO: Enables automatic mode
SINGLE BLOCK	SINGLE BLOCK: Enables / disables single block operation.
MDA	MDA: Enables data input mode, allowing to insert and execute one or more data blocks manually.
SPINDLE LEFT	SPINDLE LEFT: Turns spindle ON counterclockwise.
SPINDLE STOP	SPINDLE STOP: Turns OFF spindle rotation.
SPINDLE RIGHT	SPINDLE RIGHT: Turns spindle ON clockwise.
RAPID	RAPID: Enables rapid traverse for X and Z axes. It should be simultaneously pressed with the axis directional key (-X, +X, -Z or +Z) .
- X	- X: Moves X axis in negative direction in JOG mode.
+ X	+ X: Moves X axis in positive direction in JOG mode.

- Z	- Z: Moves Z axis in negative direction in JOG mode.
+ Z	+ Z: Moves Z axis in positive direction in JOG mode.
- T	- T: Rotates turret in negative direction. It should be simultaneously actuated with JOG key.
+ T	+ T: Rotates turret in positive direction. It should be simultaneously actuated with JOG key.
RESET	RESET: This key has several functions such as cancel an alarm, interrupt a program in automatic cycle, etc.
OK	OK OPERATOR: Turns OFF sound alarm.
	SPINDLE ROTATION DIAL: Rotating selector that enables to vary spindle speed (50% to 120%)
	FEED DIAL: Rotating selector that enables to vary programmed feed speed (F) or manual feed (1% to 120%).
	EMERGENCY STOP: Interrupts all machine functions including axes movement.
F1	F1: Function 1.
F2	F2: Function 2.
F3	F3: Function 3.

1.2.3 – Operation buttons and switches:



	CYCLE START: Starts the cycle.
	CYCLE STOP: Stops the cycle (X and Z axes).

	ELECTRONIC HANDWHEEL: Rotating switch that defines the axis movement direction. This function only works when function VAR is enabled (increment 1 or 10 or 100).
	OPEN CLOSE DOOR: This button unlocks operator's door when electric lock is installed, or open / close operator's door when automatic operator's door is installed.
	TAILSTOCK ADVANCE RETRACT: Tailstock quill advance / retract button.
	SETUP: Two-position switch that allows operation with open door, if enabled. It is removed only on disabled position.
	LOCK: Two-position switch that blocks program edition when enabled. It is removed only on disabled position.
	CHUCK CLAMP UNCL.: Chuck clamping / unclamping button.
	CNC ON: CNC ON button.
	MACHINE ON: Machine ON button. (This button should only be pressed after CNC ON button is pressed).

1.2.4 – Emergency button

This button should be pressed to stop the machine in case of emergency. If it is pressed, axes will automatically stop and the following messages will be displayed: ”EMERGENCY CIRCUIT FAILURE” and “EMERGENCY BUTTON PRESSED ON OPERATION PANEL.

To restart the machine it is required to solve problems that have caused the emergency, release the emergency button and press “MACHINE ON” button.

1.3 - SERIAL RS-232 AND ELECTRIC PLUGS

Do not disconnect or connect the CNC Tape Reader Cable (signal cable) without turning off the machine power supply. Otherwise the Tape Reader PCB and/or CNC controller PCB may be damaged.

Do not use electric feed from other equipment when connected to the machine. It may damage Tape Reader PCB and/or CNC controller PCB. Always use machine feeding to which it is connected.

PLUG RS232

ELECTRIC PLUG

Extremely caution should be exercised when connecting any device or instrument to the electric plug.

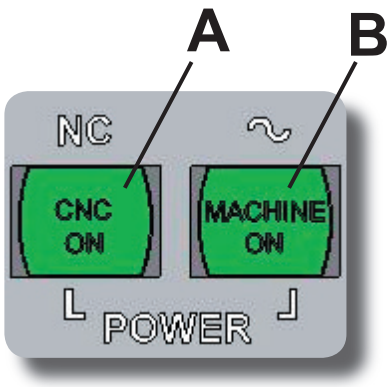
Make sure the instrument connected to the electric plug does not exceed the maximum current delivered by the plug. Make sure the instrument is set to the voltage on the electric plug.

Voltage is always present on plug pins while the main switch is in ON position.
Care should be exercised before connecting any instrument to the plug.
Do not connect any instrument besides those indicated for use.

2- INITIAL OPERATIONS

2.1 - TURN MACHINE ON

– Turn main switch to **"ON"** position.
– Press **"CNC ON"** button located on machine panel (**A**).
– Disable emergency button.
– Open and close machine's door.
– Press **"MACHINE ON"** button (**B**).



2.2 - TURN MACHINE OFF

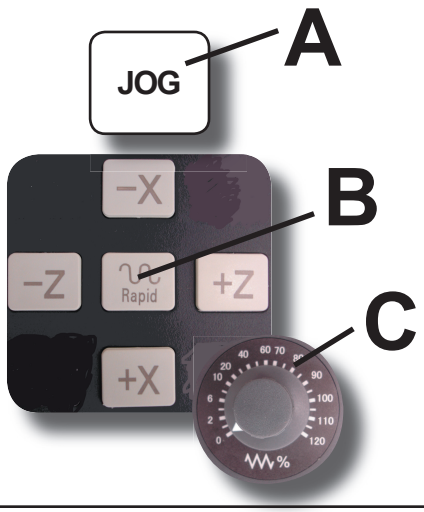
– Press emergency button.
– Turn OFF general switch.

2.3 - MACHINE REFERENCE

– Press **"REF POINT"** key.
– Press **"CYCLE START"** button.

2.4 - MOVING AXES IN CONTINUOUS JOG

– Press **"JOG"** key (**A**).
– Press **"POSITION"** key .
– Press **X+, X-, Z+ or Z-** axes movement key.
– For rapid traverse press the desired axis button and the **"RAPID"** button (**B**) simultaneously.



NOTE: *Axes shift speed may be changed through feed dial (**C**).*

2.5 - MOVING AXES THROUGH ELECTRONIC HANDWHEEL

– Press **“POSITION”** key.
– Press **“JOG”** key.
– Press **“VAR**” key until the ideal speed for movement is shown on machine panel **(“ x 1”, “x 10”, “x 20” or “x 100 “)**.
– Press *[HANDWHEEL]* softkey.
– Select the desired axis (**“X”** or **“Z”**)
– Turn crank to move axes, by observing its turning direction (positive + or negative -).

NOTE: *To move machine axes with optional Manual Machine Plus (MULTIPLIC KIT), turn thehandwheel located on the moving apron plates.*

To change the coordinate (MCS), (REL) (WCS). it’s necessary:

– Press **“POSITION”** key.
– Press **“JOG”** key.
– Press ***[MCS/WCS/REL]*** softkey.
– Select the desired coordinate: **REL** , **WCS** (Work Coordinate system) or **MCS** (Machine Coordinate System).

2.6 - OPERATING WITH OPEN DOOR

– Press **“ JOG ”** key.
– Turn **“ SETUP ”** switch.

2.7 - MANUALLY ROTATING TURRET

– Press **“ JOG ”** key.
– Press **“ JOG ”** and **“ T+ ”** or **“ T- ”** keys simultaneously.

2.8 - OPERATING CONTROL THROUGH M.D.A. (MANUAL DATA INPUT)

– Turn **“SETUP”** switch to work with closed door.
– Close machine door.
– Press **“POSITION”** key.
– Press **“MDA”** key.
– Press **“ RESET “** key.
– Press ***[DELETE MDI PROGR.]*** softkey.

– Input the desired instructions:

Example:

N10 T0101 (selects tool 01)

– Press **"INPUT"** key.

N20 G97 S1000 M4 (turns ON spindle counterclockwise at 1000 RPM).

– Press **"INPUT"** key.

– Press **"CYCLE START"** key.

NOTE: *By pressing "RESET" key operation is canceled.*

2.9 - MOVING AXES WITH SPINDLE ON

To move axes with arbor shaft on, proceed as follows:

a) Turn arbor shaft on:

– Turn **"SETUP"** switch to work with closed door.

– Press **"POSITION"** key.

– Press **"MDA"** key.

– Press **"RESET"** key.

– Press ***[DELETE MDI PROGR.]*** softkey.

– Enter G97 S and speed value. Example: G97 S1000 M3

– Press **"INPUT"** key.

– Press **"CYCLE START"** key.

b) Move axes:

– Press **"POSITION"** key.

– Press **"JOG"** key.

– Press **"VAR"** key until the optimal speed for motion (**"x 1"**, **"x 10"**, **"X 20"** or **"x 100"**) is displayed on panel display.

– Press ***[HANDWHEEL]*** softkey.

– Select desired axis (**"X"** or **"Z"**).

– Turn handwheel to move axes, by observing its turning direction (positive + or negative -).

3- EDITING PROGRAMS

3.1 - CREATING A NEW PROGRAM

- Press **"PROGRAM MANAGER"** key.
- Press softkey ***[NC DIRECTORY]***.
- Use cursor keys (▲, ▼) to place the cursor on the desired directory (eg: main program - MPF).
- Press **"INPUT"** key.
- Press ***[NEW]*** softkey**.**
- Press ***[NEW FILE]*** softkey.
- Type in program name (28 characters maximum).
- Press ***[OK]*** softkey.
- Type in the program.

NOTE: *To execute all program edition process it is required to keep the* ***"LOCK"*** *switch in unlocked position.*

3.2 - ACCESSING A PROGRAM IN THE DIRECTORY

- Press **"PROGRAM MANAGER"** key.
- Press ***[NC DIRECTORY]*** softkey.
- Use cursor keys (▲, ▼) to place the cursor on the desired directory (eg: main program - MPF).
- Press **"INPUT"** key.
- Place the cursor on the desired program.
- Press **"INPUT"** key.

3.3 - INSERTING DATA INTO A PROGRAM:

- Select the program (chapter 3.2).
- Press **"PAGE"** key and/or cursor keys (▲,▼) placing the cursor on the position where the desired information will be inserted.
- Type in the program, pressing **"INPUT"** key to change line.

3.4 - SEARCHING FOR DATA WITHIN A PROGRAM

- Select the program (chapter 3.2).
- Press ***[FIND]*** softkey .

– Type in data to be searched. Example: **X100**.
– Press ***[OK]*** softkey.

3.5 - CHANGING DATA IN A PROGRAM

– Select the program (chapter 3.2).
– Use cursor keys (▲, ▼) to place the cursor on the data to be changed.
– Press **"DEL"** to delete all information in front of cursor and/or press **"BACKSPACE**" key to delete all information behind it.
– Type in the correct information.

3.6 - DELETING PROGRAM BLOCKS:

– Select the program (chapter 3.2).
– Place the cursor on the first **"N"** block of the sequence to be deleted.
– Press ***[MARK BLOCK]*** softkey.
– Use cursor keys (▲, ▼) to select the blocks to be deleted.
– Press **[DELETE BLOCK]** softkey**.**

3.7 - DELETING A PROGRAM FROM THE DIRECTORY

– Press **"PROGRAM MANAGER"** key.
– Press ***[NC DIRECTORY]*** softkey.
– Use cursor keys (▲, ▼) to place the cursor on the program to be deleted.
– Press **[DELETE]** softkey**.**
– Pressy **[OK]** softkey.

3.8 - RENAMING A PROGRAM

– Press **"PROGRAM MANAGER"** key.
– Press ***[NC DIRECTORY]*** softkey.
– Use cursor keys (▲, ▼) to place the cursor on the program to be renamed.
– Press ***[MORE...]*** softkey.
– Press **[RENAME]** softkey**.**
– Type in the new program name. Example: **Pecie 2**
– Press **[OK]** softkey**.**

3.9 - COPYING DATA FROM A PROGRAM TO ANOTHER:

– Press **“PROGRAM MANAGER”** key.
– Press ***[NC DIRECTORY]*** softkey.
– Use cursor keys (▲, ▼) to place the cursor on the program to be copied.
– Press **“INPUT”** key.
– Press ***[EDIT]*** softkey.
– Place the cursor on the copy start block.
– Press **[MARK BLOCK]** softkey.
– Use cursor keys (▲, ▼) to select blocks to be copied.
– Press **[COPY BLOCK]** softkey.
– Press **“PROGRAM MANAGER”** key.
– Use cursor keys (▲, ▼) to place the cursor on the program that will receive the copied information.
– Press **“INPUT”** key.
– Press ***[INSERT BLOCK]*** softkey.

4- DATA COMMUNICATION

This chapter comprises the required DATA COMMUNICATION resources to handle, save, load, copy, backup, etc. all the data resident in the machine intended for equipment operation.

This data may be: machine parameters, programs, tool offsets, work coordinate offset, settings, “R” variables and global variables.

All mentioned data remain resident and retained even when the machine is turned off. This is possible through hardware components (battery or capacitor) that store energy and provide machine requirements when it is turned off, a black out occurs or even in case of a failure. Therefore, these components provide energy for a certain period of time which can be hours, days or months.

CNC SIEMENS 802 D-SL CNC is provided with a “capacitor” as an energy supplier and data maintainer with approximate 50-hour capacity. Thus, it is essential that OPERATOR execute some procedures to keep programs stored in the memory when it is turned off during extended period of time, mainly in weekends.

*It is also essential that OPERATOR understands all actions such as **SAVE PROGRAM (COPY/SEND)** and **BACKUP DATA**. For this, read chapters 4.3 and 4.5 carefully.*

The mentioned actions require some elements for its execution, such as SERIAL PORT RS232, COMPACTFLASH, DATA TRANSMISSION APPLICATIONS, OTHER PERIPHERALS, etc., whose instructions are the following.

4.1 - COMMUNICATION THROUGH SERIAL PORT RS-232

Serial communication is performed between the machine and the peripheral (computer, puncher, collector, etc), through serial port.

The microcomputer or peripheral from which communication will be done, should have a cable with a free serial port DB 9 or DB 25. The connector type is irrelevant, since it has perfect connection, without possibitily of a bad contact. The connection cable should have the following configuration:

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

There are a lot of data transmission applications in case of microcomputer transmission. For this reason, this chapter describes the configurations and procedures for machine communication only. For further information on these applications, please refer to the respective manufacturers.

4.1.1 - Configuring communication parameters:

- Press **"SHIFT"** and **"SYSTEM ALARM"** keys simultaneously.
- Press ***[PLC]*** softkey.
- Press ***[STEP 7 CONNECT]*** softkey.
- Press ***[CONNECT OFF]*** softkey, so the option "CONNECT ON" is displayed on the machine panel.
- Press *[/\ ]* softkey.
- Press ***[START-UP FILES]*** softkey.
- Press ***[RS 232]*** softkey.
- Press ***[SETTING]*** softkey.
- Use cursor keys (▲, ▼) to place the cursor on the parameters and press ◯ (SELECT) key to configure parameters.

Suggestion for configuration:

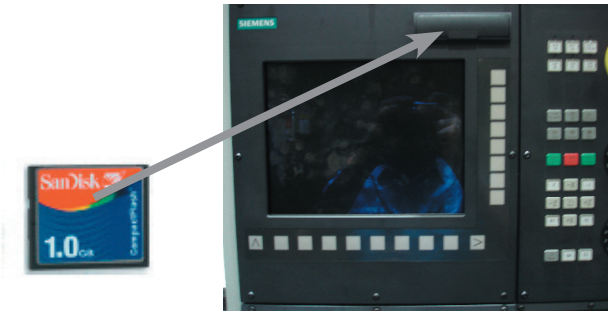
Device	**RTS CTS**
Baud rate	**9600**
Stop bits	**2**
Parity	**None**
Data bits	**7**
Transmission end	**1A**
Overwrite config.	**N**

- Press ***[SAVE]*** softkey.

NOTE: *Computer and CNC should be similarly configured.*

4.2 – COMMUNICATION THROUGH COMPACTFLASH PORT

CompactFlash is an electrical and mechanical structure of a data store system and may be directly connected to the machine without any adaptation system.



The machines interfaced to CNC SIEMENS 802 D-SL are provided with a CompactFlash slot located above the monitor which may be used to transfer different kinds of data, such as programs, machine parameters, tool offsets, etc.

IMPORTANT: *The card to be used must be Compact Flash 5000 – Industrial Grade Sandisk, 1GB memory.*

4.3 - SAVING PROGRAM

4.3.1 - Saving program from external device:

– Prepare the external device (computer, puncher, etc.) to receive data.
– Press **"PROGRAM MANAGER"** key.
– Press ***[NC DIRECTORY]*** softkey.
– Use cursor keys (▲, ▼) to place the cursor on the program (or directory) to be saved.
– Press ***[COPY]*** softkey.
– Press ***[RS 232]*** softkey.
– Press ***[SEND]*** softkey.

4.3.2 - Saving program from memory card

– Prepare the external device (computer, puncher, etc.) to receive data.
– Press **"PROGRAM MANAGER"** key.
– Press ***[NC DIRECTORY]*** softkey.
– Use cursor keys (▲, ▼) to place the cursor on the program (or directory) to be saved.
– Press ***[COPY]*** softkey.
– Press ***[CUSTOMER CF CARD]*** softkey.
– Press ***[PASTE]*** softkey.

4.4 - LOADING PROGRAM

4.4.1 - Loading program from external device:

– Prepare the external device (computer, puncher, etc.) to send data.
– Press **"PROGRAM MANAGER"** key.
– Press ***[RS 232]*** softkey.
– Press ***[RECEIVE]*** softkey.
– Send program from computer to machine.

4.4.2 - Loading program from memory card

– Press **"PROGRAM MANAGER"** key.
– Press ***[CUSTOMER CF CARD]*** softkey.
– Use cursor keys (▲, ▼) to place the cursor on the program to be copied.
– Press ***[COPY]*** softkey.
– Press ***[NC DIRECTORY]*** softkey.
– Press ***[PASTE]*** softkey.

4.5 - BACKUP

BACKUP consists of transfering data from the volatile memory to a machine internal memory area. The transferred data in this process are: workpiece programs, machine data, tool presente and zero point shift.

To save data:

– Presss **"SHIFT"** and **"SYSTEM ALARM"** keys simultaneously.
– Press *[∧]* softkey.
– Press ***[SET PASSWORD]*** softkey.
– Type in password: **CUSTOMER**
– Press ***[ACCEPT]*** softkey.
– Press ***[SAVE DATA]*** softkey.
– Press ***[OK]*** softkey.

NOTE: During saving process no operation can be performed by the machine, and the following message will be displayed:

```
Data being saved.
ATTENTION!!!
Do Not Operate or Switch Off
```

4.6 - RESTORING DATA FROM THE LAST BACKUP

– Press **"SHIFT"** and **"SYSTEM ALARM"** keys simultaneously.
– Press ***[∧]*** softkey.
– Press ***[SET PASSWORD]*** softkey.
– Digitar a senha: **CUSTOMER**
– Press ***[ACCEPT]*** softkey.
– Press ***[STARTUP]*** softkey.
– Press ***[NC]*** softkey.
– Place cursor (▲, ▼) on ***"*POWER-UP WITH SAVED DATA"** option.
– Press ***[OK]*** softkey.

4.6 – FORMATING MEMORY CARD:

To format the memory card follow the procedure below:

– Press **"PROGRAM MANAGER"** key.
– Press ***[CUSTOMER CF CARD]*** softkey.
– Press ***[MARK ALL]*** softkey.
– Press **[DELETE]** softkey.

NOTE: *Since Compactflash memory cards are sensitive instruments, it is recommended to take special care when handling and storing them. Avoid impacts, heat, humidity, and do not disconnect device during data transfer process.*

4.7 - DISPLAYING FILES FROM MEMORY CARD

On the control it is possible to display files from memory card through this procedure:

– Press **"PROGRAM MANAGER"** key.
– Press **[CUSTOMER CF CARD]** softkey.

4.8 - DELETING FILE FROM MEMORY CARD

– Press **"PROGRAM MANAGER"** key.
– Press ***[CUSTOMER CF CARD]*** softkey.
– Use cursor keys (▲, ▼) to place the cursor on the program to be deleted.
– Press ◯ **"SELECT"** key once for each file.
– Press ***[DELETE]*** softkey.

5- PROGRAM TEST

5.1 - TESTING PROGRAMS WITHOUT MOVING THE AXES

To run all machine tests, it is required to operate the machine in some specific modes. Example: Rapid Test (DRY RUN), or Program Test (PRT).

5.1.1 - Executing program test in work feed:

- Press **"PROGRAM MANAGER"** key.
- Use cursor keys (▲, ▼) to place the cursor on the desired program.
- Press **"INPUT"** key.
- Press ***[EXECUTE]*** softkey.
- Press **"POSITION"** key.
- Press **"AUTO"** key.
- Press ***[PROGRAM CONTROL]*** softkey.
- Press ***[PROGRAM TEST]*** softkey, until PRT option is selected on machine panel.
- Press ***[DRY RUN FEEDRATE]*** softkey, until DRY option is disabled on machine panel.
- Press ***[BACK]*** softkey.
- Press **"CYCLE START"** key.

5.1.2 - Executing rapid program test:

- Press **"PROGRAM MANAGER"** key.
- Use cursor keys (▲, ▼) to place the cursor on the desired program.
- Press **"INPUT"** key.
- Press ***[EXECUTE]*** softkey.
- Press **"POSITION"** key.
- Press **"AUTO"** key.
- Press ***[PROGRAM CONTROL]*** softkey.
- Press ***[PROGRAM TEST]*** softkey, until PRT option is selected on machine panel.
- Press ***[DRY RUN FEEDRATE]*** softkey, until DRY option is disabled on machine panel.
- Press ***[BACK]*** softkey.
- Press **"CYCLE START"** key.

NOTE: *After executing all tests it is required to disable "DRY" and "PRT" functions so that the program is normally executed.*

5.2 - EXECUTING PROGRAM TEST IN DRY RUN MODE (RAPID FEED - DRY):

- Press **"PROGRAM MANAGER"** key.
- Use cursor keys (▲, ▼) to place cursor on the desired program.
- Press **"INPUT"** key.
- Press ***[EXECUTE]*** softkey.
- Press **"POSITION"** key.
- Press **"AUTO"** key.
- Press ***[PROGRAM CONTROL]*** softkey.
- Press ***[PROGRAM TEST]*** softkey, until PRT option is disabled on machine panel.
- Press ***[DRY RUN FEEDRATE]*** softkey, until DRY option is selected on machine panel.
- Press ***[BACK]*** softkey.
- Press **"CYCLE START"** key.

NOTE: *After executing all tests it is required to disable "DRY" and "PRT" functions so that the program is normally executed.*

5.3 - GRAPHICAL TEST

This test aims to check if the workpiece contour is correct. It is possible to see the whole path that tool will perform during the machining process. To execute this test, follow the procedures below:

- Press **"PROGRAM MANAGER"** key.
- Place cursor (▲, ▼) on the desired program.
- Press **"INPUT"** key.
- Press ***[SIMULATION]*** softkey.
- Press **"CYCLE START"** key.

When the machine performs a graphical simulation, zoom is automatically set. To change this setting, it is required to:

- Use cursor keys (▲, ▼) to place the cursor (+) on the contour to be enlarged (zoom in) or reduced (zoom out).
- Press ***[ZOOM +]*** softkey to enlarge, or ***[ZOOM -]*** softkey to reduce the contour.

NOTE: *The path showed in graphical simulation in radius compensation cases (during the functions G41, G42 and G70, G71, G72, G73) represent the tool's center.*

6- CREATING / DELETING TOOLS

To perform all tool arrangement and preset (process well kown as “SETUP”) it is required that tools are already created on “OFFSET PARAM” page.

The chapters below describe how to create and delete a tool and how to create an offset.

6.1 - PROCEDURE TO CREATE TOOL:

- Press **“OFFSET PARAM.”** key.
- Press ***[TOOL LIST]*** softkey.
- Press ***[NEW TOOL]*** softkey.
- Press ***[TURNING TOOL]*** softkey.
- Type in tool number. Exemple: *01*
- Press ***[OK]*** softkey.

6.2 - PROCEDURE TO DELETE TOOL:

- Press **“OFFSET PARAM.”** key.
- Press ***[TOOL LIST]*** softkey.
- Place the cursor (▲, ▼) on the tool to be deleted.
- Press ***[DELETE TOOL]*** softkey.
- Press ***[OK]*** softkey.

NOTE: *When deleting a tool all its offsets will be automatically deleted.*

6.3 - PROCEDURE TO CREATE A NEW OFFSET :

- Press **“OFFSET PARAM.”** key.
- Press ***[TOOL LIST]*** softkey.
- Place the cursor (▲, ▼) on the tool which a new offset is added in.
- Press ***[CORRETOR EDGES]*** softkey.
- Press ***[NEW CUT. EDGE]*** softkey.

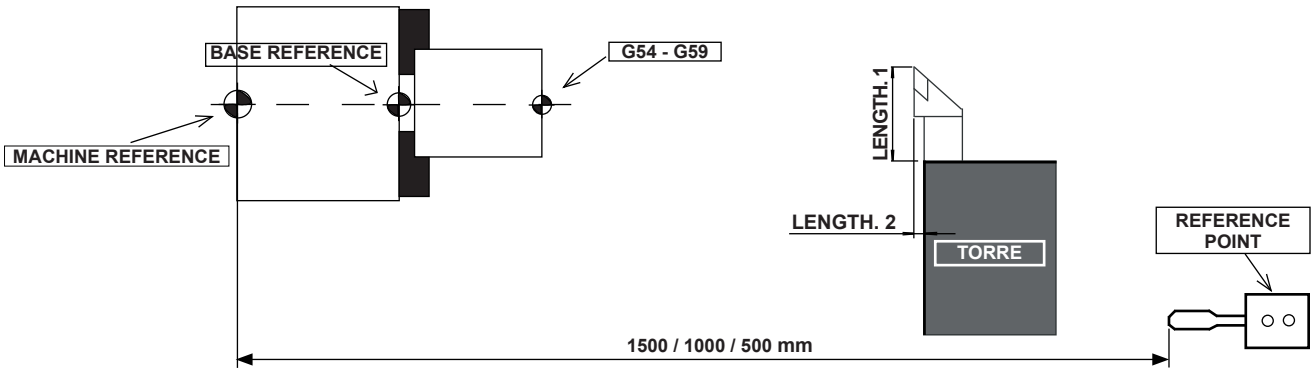
NOTE: *To shift tool offsets display, press softkey* **[D>>]** *and softkey* **[<<D].**

6.4 - PROCEDURE TO RESET CORRECTOR VALUES TO ZERO:

– Press **"OFFSET PARAM."** key
– Press ***[TOOL LIFE]*** softkey.
– Place cursor (▲, ▼) on the corrector of tool to be reset to zero.
– Press ***[CORRETOR EDGES]*** softkey.
– Press ***[INICIAL EDGE]*** softkey.

7- TOOL PRESET

Tool preset is a process that informs to machine the tool dimensions. For this it is required any reference point (usually tool face) in order to compare the distances between tool tips and this reference point on X (length 1) and Z axes (length 1).



The length1's value is met when the tool to be preseted is touched on a known diameter, for example the chuck's diameter, work-piece's diameter, etc..

Execute the chapter 7.1 of this manual to register this value in the machine.

The length2's value is met to comparing the zero-base's measures when the turret is touched in any reference point and the tool is touched in this same point. Due this, is necessary to execute a preparation before to perform the process from the Z axis's preset.

To accomplish this preparation must be used a face like reference, for example, chuck's face, work-piece's face, jaw's face, etc...

The preparation's process is described in the chapter 7.2.

NOTE: *If the function G92 Z0 is activated, then is necessary to execute the function G92.1Z0.*

The preset's process is described in the chapter 7.3.

IMPORTANT: *The turret reference's process (zero base) must be redone every time that the floating reference point (MECHANIC STOP) is shifted.*

7.1 - " X " AXIS PRESET (LENGTH 1)

1) Displace turret up to a safe distance for tool changing:

- Press **" POSITION "** key.
- Press **" JOG "** key.
- Turn **"SETUP"** switch to operate with the door open
- Press **" VAR "** key until the ideal speed for movement (**" X 1"**, **"X 10", "X20"** or **"X 100"**) is displayed.
- Press ***[HANDWHEEL]*** softkey**.**
- Select the desired axis (**" X "** or **" Z "**).
- Displace turret up to a safe distance for tool changing, by rotating handwheel and paying attention to its rotation direction (positive **+** or negative **-**).

NOTE: *To move machine axes with optional Manual Machine Plus (MULTIPLIC KIT), turn the handwheel located on the moving apron plates*

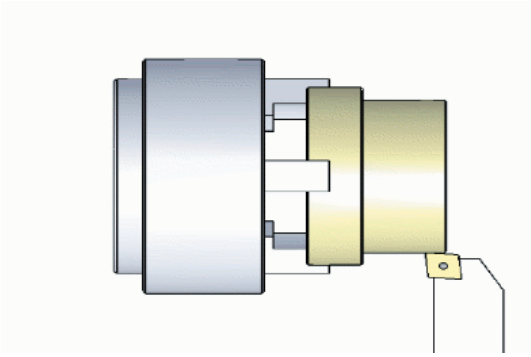
2) Select tool to be preset through MDA:

- Press **"POSITION"** key.
- Turn **"SETUP"** switch to operate with the door closed.
- Press **"MDA"** key.
- Press **"RESET"** key.
- Press ***[DELETE MDI PROGR.]*** softkey.
- Type in command **G291**
- Press **"INPUT"** key.
- Type in the command to select tool to be preset:
  Example: **T0101** (selects tool 01)
- Press **"INPUT"** key.
- Type in command **G290**
- Press **"INPUT"** key.
- Press "CYCLE START" key (Until that the word "ISO" disappear of the upper left corner's of the machine's screen).

Program example MDA:

```
G291
T0101
G290
```

3) Touch the tool tip on measured diameter.



– Press **“ POSITION ”** key.
– Press **“ JOG ”** key.
– Turn **“SETUP”** switch to operate with the door open
– APress **“ VAR ”** key until the ideal speed for movement (**“ X 1”**, **“X 10”, “X20”** or **“X 100”**) is displayed.
– Press ***[HANDWHEEL]*** softkey**.**
– Select the desired axis (**“ X ”** or **“ Z ”**).
– Touch tool on the diameter used as reference, by rotating handwheel and paying attention to its rotation direction (positive **+** or negative **-**).
– Girar a manivela para encostar a ferramenta no diâmetro usado como referência.

NOTE: *To move machine axes with optional Manual Machine Plus (MULTIPLIC KIT), turn the handwheel located on the moving apron plates.*

4) Preset tool:

– Press **“ POSITION ”** key.
– Turn **“LOCK”** switch to input preset data.
– Press ***[MEASURE TOOL]*** softkey**.**
– Press ***[MEASURE MANUAL]*** softkey**.**
– Use cursor keys (▲, ▼) to place the cursor on field “ Ø ”.
– Type in workpiece diameter value. Example: **50**.
– Press **“INPUT”** key.
– Press ***[SAVE POSITION]*** softkey**.**
– Press ***[SET LENGTH 1]*** softkey**.**
– Press ***[BACK]*** softkey *twice*.
– Press **“RESET”** key.

Repeat procedures 1, 2, 3 and 4 for all tools to be used.

7.2 - PREPARATION TO " Z " AXIS PRESET

1) Displace turret up to a safe distance for tool changing:

- Press **" POSITION "** key.
- Press **" JOG "** key.
- Press **" VAR "** key until the ideal speed for movement **(" x 1", "x 10", "X20" or "x 100")** is displayed.
- Press ***[HANDWHEEL]*** softkey**.**
- Select the desired axis (**" X "** or **" Z "**).
- Displace turret up to a safe distance for tool changing, by rotating handwheel and paying attention to its rotation direction (positive **+** or negative **-**).

NOTE: *To move machine axes with optional Manual Machine Plus (MULTIPLIC KIT), turn the handwheel located on the moving apron plates*

2) Select an empty position on turret through MDA:

- Press **" POSITION "** key.
- Turn **"SETUP"** switch to operate with the door closed
- Press **"MDA"** key.
- Press **"RESET"** key.
- Press ***[DELETE MDI PROGR.]*** softkey.
- Type in command **G291**
- Press **"INPUT"** key.
- Type in command to select the empty position:
  Example: **T0301** (selects position 03).
- Press **"INPUT"** key.
- Type in command: **T00**
- Press **"INPUT"** key.
- Type in command: **G290**
- Press **"INPUT"** key.
- Press "CYCLE START" key (Until that the word "ISO" disappear of the upper left corner's of the machine's screen).

Program example MDA:

```
G291
T0301
T00
G290
```

3) Touch the turret face that will be used as reference. Example: chuck face, workpiece face, jaw or ground shim.

- Press **“ POSITION ”** key.
- Press **“ JOG ”** key.
- Turn **“SETUP”** switch to operate with the door open
- Press **“ VAR ”** key until the ideal speed for movement (**“ X 1”**, **“X 10”, “X20”** or **“X 100”**) is displayed.
- Press ***[HANDWHEEL]*** softkey**.**
- Select the desired axis (**“ X ”** or **“ Z ”**).
- Touch the turret face used as reference, by rotating handwheel and paying attention to its rotation direction (positive **+** or negative **-**).

NOTE: *To move machine axes with optional Manual Machine Plus (MULTIPLIC KIT), turn the handwheel located on the moving apron plates*

4) Referencing turret:

- Press **“ POSITION ”** key.
- Turn **“LOCK”** switch to input reference data.
- Press ***[MEASURE WORKPIECE]*** softkey.
- Press ***[OK]*** (If necessary) softkey.
- Use cursor keys (▲, ▼) to place the cursor on : ***“SAVE IN”.***
- Press ◡ “SELECT” to select **“BASE”**
- Press ***[SET WORK OFFSET]*** softkey**.**
- Press ***[BACK]*** softkey.
- Press **“RESET”** key.

7.3 - " Z " AXIS PRESET (LENGTH 2)

1) Displace turret up to a safe distance for tool changing:

- Press **" POSITION "** key.
- Press **" JOG "** key.
- Turn **"SETUP"** switch to operate with the door open
- Press **" VAR "** key until the ideal speed for movement (**" X 1"**, **"X 10", "X20"** or **"X 100"**) is displayed.
- Press ***[HANDWHEEL]*** softkey**.**
- Select the desired axis (**" X "** or **" Z "**).
- Displace turret up to a safe distance for tool changing, by rotating handwheel and paying attention to its rotation direction (positive **+** or negative **-**).

NOTE: *To move machine axes with optional Manual Machine Plus (MULTIPLIC KIT), turn the handwheel located on the moving apron plates*

2) Select tool to be referenced through MDA:

- Press **" POSITION "** key.
- Turn **"SETUP"** switch to operate with the door closed
- Press **"MDA"** key.
- Press **"RESET"** key.
- Press ***[DELETE MDI PROGR.]*** softkey.
- Type in command **G291**
- Press **"INPUT"** key.
- Type in command to select tool to be preset:

  Example: **T0401** (selects tool 04)
- Press **"INPUT"** key.
- Type in command: **G290**
- Press **"INPUT"** key.
- Press "CYCLE START" key (Until that the word "ISO" disappear of the upper left corner's of the machine's screen).

Program example MDA:

```
G291
T0401
G290
```

3) Touch tool tip on the face used as reference:

- Press **“ POSITION ”** key.
- Press **“ JOG ”** key.
- Turn **“SETUP”** switch to operate with the door open
- APress **“ VAR ”** key until the ideal speed for movement (**“ X 1”**, **“X 10”, “X20”** or **“X 100”**) is displayed.
- Press ***[HANDWHEEL]*** softkey**.**
- Select the desired axis (**“ X ”** or **“ Z ”**).
- Touch tool on the face used as reference, by rotating handwheel and paying attention to its rotation direction (positive **+** or negative **-**).

NOTE: *To move machine axes with optional Manual Machine Plus (MULTIPLIC KIT), turn the handwheel located on the moving apron plates*

4) Tool preset:

- Press **“ POSITION ”** key.
- Turn **“LOCK”** switch to input preset data.
- Press ***[MEASURE TOOL]*** softkey**.**
- Press ***[MEASURE MANUAL]*** softkey**.**
- Press ***[LENGTH 2]*** softkey**.**
- Use cursor keys (◄, ►, ▲, ▼), to place the cursor on **“DISTANCE”** field.
- Type in **“ 0 ”**.
- Press **“INPUT”** key.
- Use cursor keys (◄, ►, ▲, ▼), to place the cursor on the field in front of the field **“Z0”.**
- Use key ◯ “SELECT” to select option “**BASE** ”.
- Press ***[SAVE POSITION]*** softkey**.**
- Press ***[SET LENGTH 2]*** softkey**.**
- Press ***[BACK]*** softkey *twice*.
- Press **“RESET”** key.

Repeat procedures 1, 2, 3 and 4 for all tools to be used.

7.4 - TOOL RADIUS AND QUADRANT (CUTTING FACE)

Before presetting tools on " X " and " Z ", it is required to inform RADIUS and QUADRANT values on offset for each one. For this, follow the instructions:

- Turn **"LOCK"** switch to input preset data.
- Press **" OFFSET PARAM. "** key.
- Press ***[TOOL LIST]*** softkey.
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on the column **"RADIUS"** (located on geometry table) of tool whose radius will be informed.
- Type in tool radius value. Example: **0.8**
- Press **"INPUT"** key.
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on the field 

of tool whose cutting face will be informed.
- Type in cutting face value (according to chapter 7.4 - Programming - of this manual). Example: **3**
- Press **"INPUT"** key.

7.5 - TOOL WEAR OFFSET

All tools suffer progressive wear due to their friction with the removed material. Therefore, when the tool is used for calibration, it is required to correct its wear to maintain product quality level related to the dimensional characteristics.

The correction value in the "x" axis is informed in diameter, and to make a correction in this axis, you must put the cursor in the field "lenght 1". To make a correction in the "z" axis, you must put the cursor in the field "lenght 2".

- Turn **"LOCK"** switch to input preset data.
- Press **" OFFSET PARAM "** key.
- Press ***[TOOL LIST]*** softkey.
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on the tool to be offset, in the column **"WEAR".**
- Press " **=** " key (equal signal).
- Type in the value to be offset paying attention to signal to be used.
  Example: **- 0.02.**
- Press **"INPUT"** key.
- Press ***[ACCEPT]*** softkey.

NOTE: *In Manual Machine Plus (KIT MULTIPLIC) it's necessary preess* ***[Tool wear]*** *.*

8- WORK OFFSET DEFINITION

8.1 - WORK COORDINATE SYSTEM (G54 TO G59)

To define work offset using the “WORK COORDINATE SYSTEM” (G54 to G59), follow the procedure below:

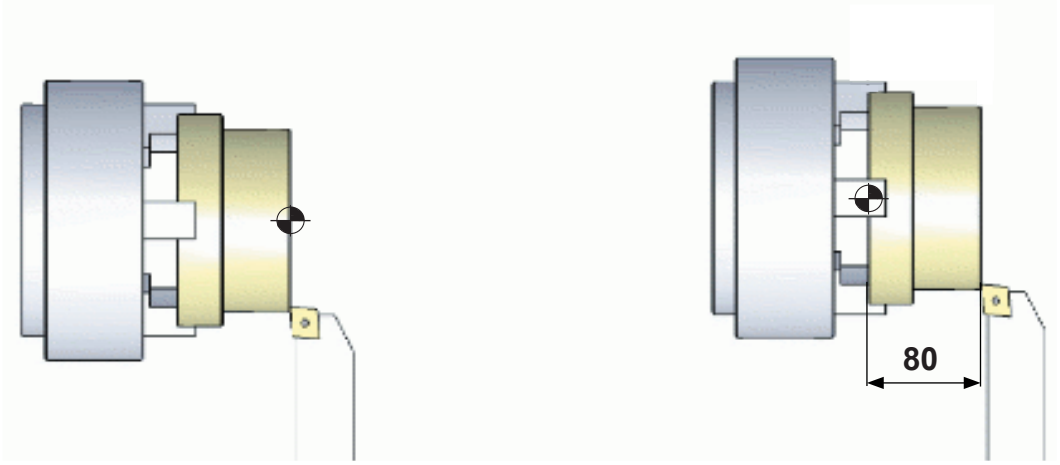
- Press **“ POSITION ”** key.
- Press **“ MDA ”** key.
- Press **“ RESET ”** key.
- Press ***[DELETE MDI PROGR.]*** softkey***.***
- Type in **G291**
- Press **“INPUT”** key.
- Type in **“ T ”** number of the tool already preset and to be used in this process. Example: **T0401**
- Press **“INPUT”** key.
- Type in command **G290.**
- Press **“INPUT”** key.
- Press **“CYCLE START”** key. (Until that the word “ISO” disappear of the upper left corner’s of the machine’s screen).
- Press **“ JOG ”** key.
- Turn **“SETUP”** switch to operate with the door open
- Press **“ VAR ”** key, selecting the desired speed (**“x1”, “x10”, “x20” or “x100”**).
- Press ***[HANDWHEEL]*** softkey***.***
- Select the desired axis (**X** or **Z**) and move axes until tool tip touches workpiece face.

NOTE: *To move machine axes with optional Manual Machine Plus (MULTIPLIC KIT), turn the handwheel located on the moving apron plates*

- Press ***[BACK]*** softkey***.***
- Press ***[MEASURE WORKPIECE]*** softkey***.***
- Press ***[Z]*** softkey***.***
- Use ◡ “SELECT” key to select the desired work offset (G54 - G59)
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on **“DISTANCE”** field.
- Type in **= 0** for work offset on face or value = workpiece length for work offset on jaw face, according to the figure below.

– Press ***[SET WORK OFFSET]*** softkey.

CNC will calculate and automatically define the workpiece zero value.



Work offset on Face = 0

Work offset on Base = 80

8.2- WORK COORDINATE SYSTEM OFFSET (G54 TO G59)

– Press**“ OFFSET PARAM ”** key.
– Press ***[WORK OFFSET]*** softkey.
– Place the cursor on the desired field (**G54 to G59**) (always in “Z”).
– Press “ **=** ” key (equal signal).
– Type in the value to be offset, paying attention to signal to be used.
Example: **- 0.02.**
– Press **“INPUT”** key.
– Press ***[ACCEPT]*** softkey.

9- JAWS MACHINING

This is an important process when it is desired to grip concentric workpiece due to a possible transformation process on it, or to obtain and adequate gripping.

This process uses a set of soft jaws (not hardened) which is prepared so that a small stock removal will contour it according to the diameter to be gripped.

9.1 – MANUAL MACHINING

1) Move tower away to a safe distance fo tool change

- Press **"POSITION"** key.
- Press **"JOG"** key.
- Adjust **"SETUP"** key to work with door open.
- Press **"VAR"** key until the optimal speed for motion (**"x 1"**, **"x 10"**, **"x 20"** or **"x 100"**) is displayed on panel display.
- Press ***[HANDWHEEL]*** softkey.
- Select desired axis (**"X"** or **"Z"**).
- Turn crank until moving the tower away to a safe distance for indexing.

NOTE: *To move machine axes with optional Manual Machine Plus (MULTIPLIC KIT), turn the handwheel located on the moving apron plates*

2) Select the internal tool to be in charge of jaw machining and turning the arbor shaft on:

- Press **"POSITION"** key.
- Adjust **"SETUP"** key to work with door closed.
- Press **"MDA"** key.
- Press **"RESET"** key.
- Press ***[DELETE MDI PROGR.]*** softkey.
- Enter **M83** (to work with plate open, if required).
- Press **"INPUT"** key.
- Enter command to select internal tool. Example: **T0401** (select tool 04)
- Press **"INPUT"** key.
- Enter command to turn arbor shaft on. Example: **N20 G97 S1000 M3** (turn arbor shaft in clockwise direction at 1000 RPM).
- Press **"INPUT"** key.
- Press **"CYCLE START"** key.

3) Manually machine jaws:

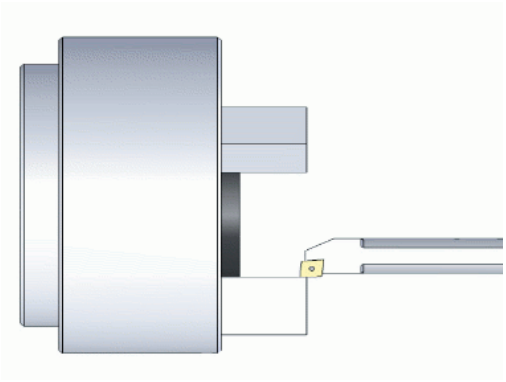
- Press **"JOG"** key.
- Press **"VAR"** key, selecting the desired speed (**"x1", "x10", "x20" or "x100"**).
- Press **"POSITION"** key.
- Press ***[HANDWHEEL]*** softkey.
- Select the desired axis (**X** or **Z**)
- Rotate handwheel, machining jaws up to the desired dimensions.

NOTE: *To move machine axes with optional Manual Machine Plus (MULTIPLIC KIT), turn the handwheel located on the moving apron plates*

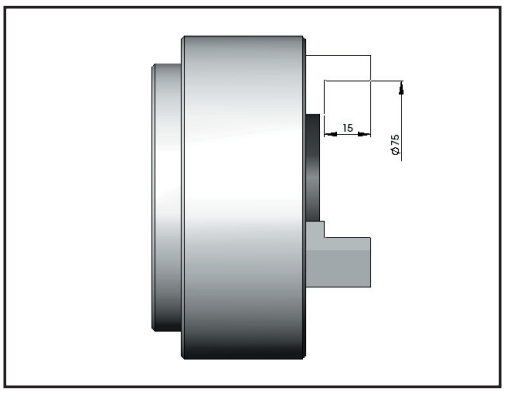
9.2 – MACHINING THROUGH PROGRAM

To machine jaws via program, according to the drawing below, reset tool used for machining to zero, fasten a shim between jaws to eliminate eventual gaps and run the program. In the program, the excess metal to be removed must be enough to establish a backstop for the part to be fastened. In the example below, the preliminary diameter is 60.5 mm, to be machined with 75.5 mm x 15 mm. Thus a 7.5-mm wall will be achieved as backstop or limit.

Before machining



After machining



Program example of jaws machining

```
N10 G291
N20 G21 G40 G90 G95
N30 T00
N40 G00 G54 X350 Z250
N50 T0501; (ROUGH. INT.)
N60 G54
N70 G96 S120
N80 G92 S1000 M03
N90 G00 X59 Z2
N100 G74 X74.5 Z-14.9 P2000 Q17000 R1 F.2
N110 G00 X77 Z2
N120 G01 Z0 F.16
N130 X75 Z-1
N140 Z-13
N150 X76.0 Z-15.
N160 X59
N170 G00 Z2
N180 G00 X350 Z250 T00
N190 M30;
```

10- PROGRAM EXECUTION

NOTE: *Before executing a program, it is required that "DRY" and "PRT" functions are disabled.*

10.1 - EXECUTING A PROGRAM FROM MACHINE MEMORY:

Each program after tested will be available for automatic execution. For this, it is required:

- Press **"PROGRAM MANAGER"** key.
- Use cursor keys (▲, ▼) to place the cursor on the desired program.
- Press **"INPUT"** key.
- Press ***[EXECUTE]*** softkey.
- Press **"AUTO"** key.
- Press **"RESET"** key.
- Press **"CYCLE START"** key.

NOTE: *If it is required to execute program step-by-step, press* **"SINGLE BLOCK"** *key, and for each block execution,* press **"CYCLE START"** key.

10.2 - EXECUTING A PROGRAM DIRECTLY FROM MEMORY CARD

C420 line enables program execution directly from **CompactFlash** card.

- Insert CompactFlash card in the slot.
- Press **"PROGRAM MANAGER"** key.
- Press ***[CUSTOMER CF CARD]*** softkey.
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on the program to be executed.
- Press ***[MORE...]*** softkey.
- Press ***[EXT. EXECUTION]*** softkey.
- Press **"CYCLE START"** key (machining will start)

NOTE: *For further information on COMPACTFLASH card, refer to chapter 4.2*

10.3 - START EXECUTION DURING A PROGRAM

Each program after tested will be available for automatic execution. For this, it is required:

- Press **"PROGRAM MANAGER"** key.
- Use cursor keys (▲, ▼) to place the cursor on the desired program.
- Press **"INPUT"** key.
- Press ***[EXECUTE]*** softkey.
- Press **"AUTO"** key.
- Press **"RESET"** key.
- Press ***[BLOCK SEARCH]*** softkey.
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on block that will start execution. (usually on the tool. Example: **T0101.)**
- Press ***[TO CONTOUR]*** softkey.
- Press **"CYCLE START"** key.

10.4 - ABORTING PROGRAM EXECUTION

- Press **"AUTO"** key.
- Press **"CYCLE STOP"** key.
- Press **"RESET"** key.

10.5 - MOVING AXES THROUGH JOG DURING AUTOMATIC EXECUTION

- Press **"CYCLE STOP"** key.
- Press **"JOG"** key.
- Press **"X+", "X-", "Z+"** or **"Z-"** axis movement key to move turret.
- Press **"SPINDLE STOP"** key.
- Press **"OPEN CLOSE DOOR"** button (If required, open the door to perform any operation. Example: change tool insert).

10.6 - RETURNING FROM JOG TO AUTOMATIC EXECUTION.

- Close the door.
- Press **"AUTO"** key.
- Press **"CYCLE START"** key.

10.7 - OPTIONAL STOP:

This function only interrupts program execution if "M01" function on machine operation panel is actuated. Therefore, M01 function becomes equivalent to M00 function. However, if this options is not actuated, the control will ignore function M01 and will continue program execution normally.

When the stop occurs through this code, "CYCLE START" button must be pressed to continue program execution.

To actuate **"M01"** function on machine panel, it is required:

- Press **"POSITION"** key.
- Press **"AUTO"** key.
- Press ***[PROGRAM CONTROL]*** softkey.
- Press ***[CONDIT. STOP.]*** softkey.
- Check if "**M01"** option is actuated on machine panel.

11- PARTS COUNTER

11.1 - PARTS COUNTER PAGE

To visualize parts counter on CENTUR30D it is required to access the proper page, following these steps:

- Press **"OFFSET PARAM"** key.
- Press ***[SETTING DATA]*** softkey
- Press ***[COUNTER TIME]*** softkey

The parts counter in this page are:"*PARTS IN TOTAL*" **,** "*PARTS REQUIRED*" and " *PARTS COUNT*"**.**

Each counter possesses a corresponding variable that stores the counted values. These variables are the following:

$AC_TOTAL_PARTS	(variable corresponding to TOTAL PARTS)
$AC_REQUIRED_PARTS	(variable corresponding to REQUIRED PARTS)
$AC_ACTUAL_PARTS	(variable corresponding to COUNTED PARTS)

To insert a limit value for parts execution, it is required to:

- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on "*PARTS REQUIRED*" field.
- Type in parts value to be executed.Example **10.**
- Press **"INPUT"** key.

11.2 - PARTS COUNTER PROGRAMMING:

To activate total parts counter, the following program command should be actuated:

```
%_N_COUNTER_MPF
;$PATH=/_N_MPF_DIR
G291
G21 G40 G90 G95
T00
G54 G00 X200 Z250
T0101
G96 S200
G92 S3000 M3
:
G290
$AC_TOTAL_PARTS=($AC_TOTAL_PARTS+1)
G291
M30
```

The program example below generates an alarm when the required parts limit is reached by counter parts (actual parts).

```
%_N_COUNTER_MPF
;$PATH=/_N_MPF_DIR
G291
G21 G40 G90 G95
T00
G54 G00 X200 Z250
T0101
G96 S200
G92 S3000 M3
:
G290
IF $AC_REQUIRED_PARTS<=($AC_ACTUAL_PARTS) GOTOF ALARME
$AC_ACTUAL_PARTS=($AC_ACTUAL_PARTS+1)
$AC_TOTAL_PARTS=($AC_TOTAL_PARTS+1)
G291
M30
G290
ALARM: MSG("PRESET COUNTER")
M00
G291
M30
```

NOTE: *To preset counter, it is required:*

- Press **"OFFSET PARAM"** key.
- Press ***[SETTING DATA]*** softkey.
- Press ***[COUNTER TIME]*** softkey.
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on "*PARTS COUNT".*
- Digitar **0**.
- Press **"INPUT"** key.

12- TOOL USEFUL LIFE MONITORING

Tool useful life monitoring process establishes an execution limit (by time or by parts amount) for each tool and compels the machine to generate an alarm when this limit is reached. To limit the number of these executions, it is required to know how many parts (or how long) the tool is able to machine, before becoming unable for use (very high worn out).

12.1 - USEFUL LIFE MONITORING BY PARTS AMOUNT

To operate with tool life monitoring by parts amount, it is required:

a) Actuate monitoring on tool life page:

- Press **"OFFSET PARAM"** key.
- Press ***[TOOL LIFE]*** softkey**.**
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on the line of desired tool number.
- Use cursor keys (◄, ►) to place the cursor on **"SET.PT."** column on **"QUANTITY"** field.
- Type in the parts limit that tool may machine

  Example: **10.**
- Press **"INPUT"** key.
- Use cursor keys (◄, ►,) to place the cursor on "ACTIV" column and on the line of desired tool number.
- Press ◯ "**SELECT**" key (until "ACTIV" option is selected).
- Press **"RESET MONITOR"** softkey.
- Select options:
  - ☑ Selected tool
  - ☑ Selected cutting edge
  - ☑ Selected monitoring
- Press softkey ***[OK].***

b) Program the following command on program to be executed:

G290
SETPIECE(1)
G291

NOTE: *When machine executes the command SETPIECE(1) a unit is reduced from the residual value. When this value is extinguished, an alarm occurs.*
Then the residual values should be reloaded.

c) Reload residual value:

- Press **“ OFFSET PARAM ”** key.
- Press ***[TOOL LIFE]*** softkey.
- Place cursor (◄, ►, ▲, ▼) on the line with the number of the desired tool.
- Activate ◯ **“ SELECT ”** key (until **“ACTIV”** option is selected).
- Press **[RESET MONITOR]** softkey.
- Press ***[OK]*** softkey.

12.2 - LIFETIME MONITORING IN MINUTES

To work with tool lifetime monitoring in minutes, proceed as follows:

- Press **“ OFFSET PARAM ”** key.
- Press ***[TOOL LIFE]*** softkey.
- Place cursor (◄, ►, ▲, ▼) on the line with the number of the desired tool.
- Place cursor (◄, ►) on **“SET.PT.”** column, located in **“TOOL LIFE [MIN]”** panel.
- Enter time (in minutes) to keep the tool machining parts in machining motion. For example: **10**.
- Press **“ INPUT ”** key.
- Place cursor (◄, ►) on **“ACTIV”** column of the desired tool.
- Press ◯ “SELECT” key (until **“ACTIV”** option is selected).
- Press **[RESET MONITOR]** softkey.
- Select options:
  - ☑ Selected tool
  - ☑ Selected cutting edge
  - ☑ Selected monitoring
- Press ***[OK]*** softkey.

To enable time count by the tool, it is necessary to code the command for tool change in the program. Example: **T0501**

> ***NOTE:*** *The monitoring time count in minutes occurs only when the machine performs a machining motion (G01).*
>
> *When the residual time is elapsed while the tool is machining, the machine triggers an alarm, but only stops execution upon the next tool call (“T” command).*

To reload residual time, proceed as follows:

– Press **“ OFFSET PARAM ”** key.
– Press ***[TOOL LIFE]*** softkey.
– Place cursor (◄, ►, ▲, ▼) on the line with the number of the desired tool.
– Activate ◡ **“ SELECT ”** key (until **“ACTIV”** option is selected).
– Press **[REINIC. MONITOR]** softkey.
– Press ***[OK]*** softkey.