4. Traverse functions

4. Traverse functions

4. Traverse functions

The traverse functions, togehter with the traverse information, establish the geometric part of the program. They consist of the address letter G and a 10-digit number. One block may contain 8 traverse functions.

If the traverse functions and the appropriate traverse information are programmed in different blocks, the traverse functions in the program should always precede the traverse information. Traverse functions become effective before the programmed traverse information.

The following table contains the traverse functions made available by the control.

Traverse functions within a group overwrite each other mutually (in addition G92 is overwritten by G53 to G59).

The traverse functions preset at the start of a program are identified by an *.

The traverse functions in the program are displayed when the ‘?’ key is pressed (except for the traverse functions which act block-by-block).

4. Traverse functions (continued)

Traverse functions		Effect
G00 G01* G02 / G03 G123	Positioning at rapid traverse Linear interpolation Circular interpolation, CW/CCW Automatic selection of linear and circular interpolation	modal modal modal modal
G04	Dwell	block-by-block
G05 / G06 / G07	Driving direction of round axes	block-by-block
G08 / G09	Precision stop, on/off	block-by-block
G10 / G11*	Polar coordinate programming, on	modal
G12	Contour path, rapid programming	block-by-block
G13*/ G14	Tangential axis off/on	modal
G15*/ G16	Polar transformation off/on	modal
G17 / G18 / G19	Plane selection XY / XZ / YZ	modal
G28 / G29*	Switching on/off precision stop	modal
G40*/ G41 / G42	Milling cutter path correction offset, clearing / left / right	modal
G43*/ G44	Axis correction off/on	modal
G45 G46*	Turning on/off	modal
GG47/G48/G49	Robot transformation off / Tool / workpiece coordinates	modal
G147	Transformation off	modal
G50/G51/G52	Spline interpolation	modal
G53* G153 G54...G59	Machine zero point Zero point shift off Zero points	modal modal modal
G60*/ G61 / G62	Mirror imaging, off	modal

4. Traverse functions (continued)

Traverse functions		Effect
G63 / G64*	Switching on feed rate 100%	modal
G66	Switching off all offsets	block by block
G90*	Absolute dimension input	modal
G91	Incremental dimension input	modal
G92	Zero point shift	modal
G94	Feed in mm/min	modal
G95	Feed in mm/r	modal
G96	Constant cutting speed	modal
G97	Number of revolution in 1/min	modal

4.1 G00 positioning at rapid traverse rate

Positioning at rapid traverse rate is called up by G00.

All axes can be traversed simultaneously as long as the machine tool is designed for this. Absolute and incremental dimension input are both possible.

G00 acts modally and can be overwritten by G01, G02, G03, G50, G51 and G52.

When traversing at rapid rate the programmed point is homed into via the shortest route.
The axis with the longest programmed traverse distance is traversed at rapid rate and determines the positioning time. The speed of the other axes is chosen by the control system such that they reach their end point simultaneously with the fastest axis.
The feed rate override potentiometer is also operative during positioning at rapid rate.

Any feed rate stored in front of G00 again becomes operative after a rapid traverse through G01, G02, G03, G50, G51 and G52.

Example: positioning at rapid rate

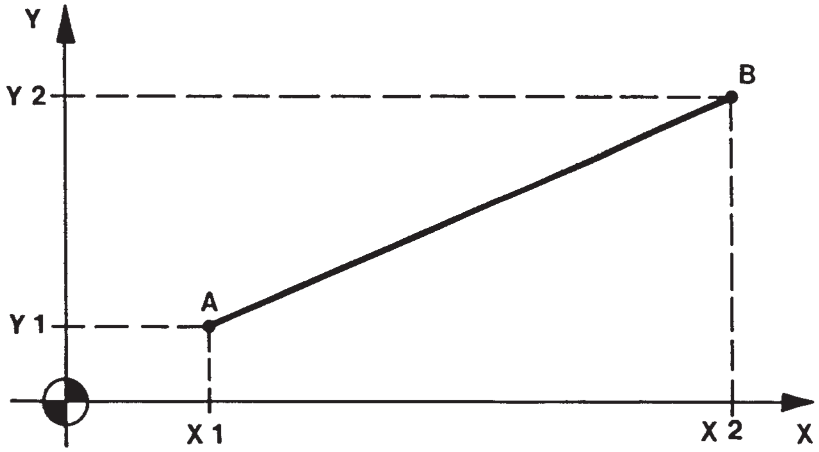


Figure 4-1

4.2 G01 Linear interpolation

The programmed target position is homed into by G01 along a straight line. The programmed feed rate is the contouring rate.

G01 acts modally and can be overwritten by G00, G02 and G03. G01 is automatically preset at start of the program.

Example: linear interpolation

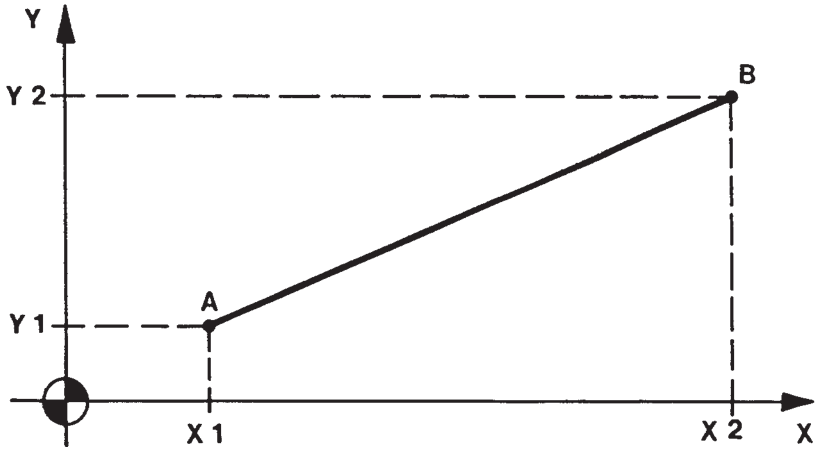


Figure 4-2

4.3 G02 / G03 Circular interpolation

When inputting G02 and G03, the programmed target position is homed into along a circle segment having a centre point determined by the interpolation parameters I, J and K and/or having a radius determined by R.

G02 signifies clockwise circular interpolation and
G03 signifies counter-clockwise circular interpolation

The G02 / G03 functions act modally and overwrite each other mutually and can be cleared by G00, G01, G50, G51 and G52.

Circle centre point

The circle limit point and the interpolation parameters can be input simultaneously using either absolute or incremental dimensions. For complete circle programming the limit point is equal to the starting point.

Interpolation parameter address letter	Circle centre point distance in direction
I	X axis
J	Y axis
K	Z axis

The speed at which the circle is started is proportional to the stored feed rate which can be adjusted between 0 and 120% by the feed rate override potentiometer.

4.3 G02 / G03 Circular interpolation (continued)

The interpolation parameters can be input in such a way that the deviation at A is less than or equal to 10 increments. If A is greater than 10 increments the program is still not shut down. In all cases the control system recalculates the circle centre point where R = (R1 + R2)/2.

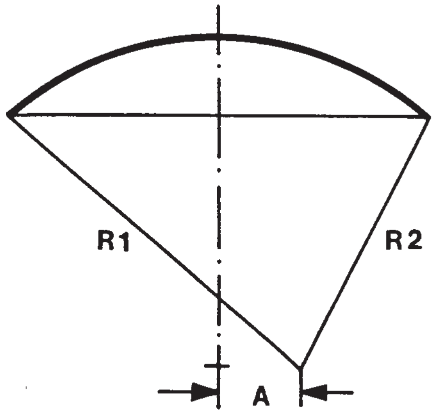


Figure 4-3

Three-dimensional circular interpolation

A programmed circle is three-dimensional driven off, if in a block 3 axes (X, Y, Z) and 3 circle center points (I, J, K) are indicated.

With programmed
G02 is executed a long arc,
G03 is executed a short arc.

The area levels G17, G18 and G19 are here without meaning.

4.3 G02 / G03 Circular interpolation (continued)

The main planes for the circular interpolation and tool correction offsets are selected by G17, G18 and G19 (see also 4.12 Plane selection).

Main plane	Parameters for circle centre point
XY	IJ
ZX	KI
YZ	JK

Example: clockwise circular interpolation

End point and circle centre point are programmed in absolute dimensions (G90).

A = Starting point
B = End point

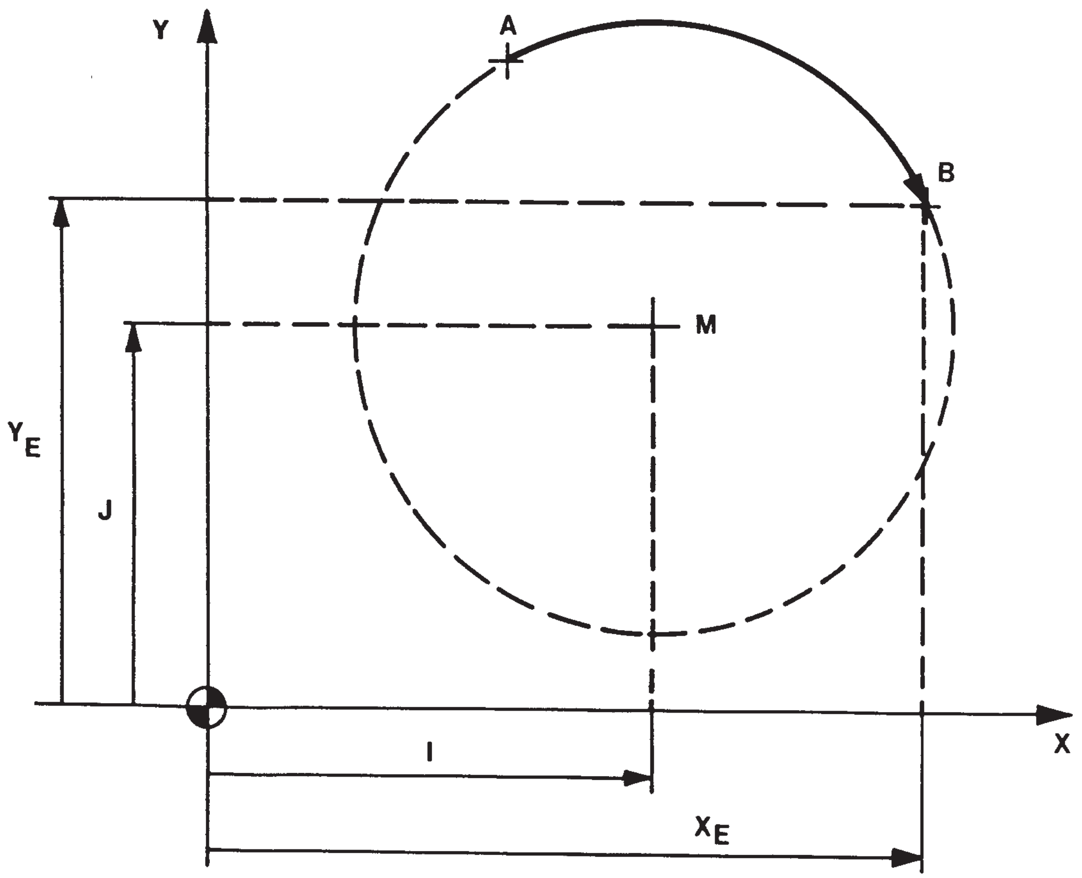


Figure 4-4

4.3 G02 / G03 Circular interpolation (continued)

The circle end point can be input in absolute or incremental dimensions. Full circle programming is not permitted.

The smaller and larger angular paths are described by positive and negative radius information respectively.

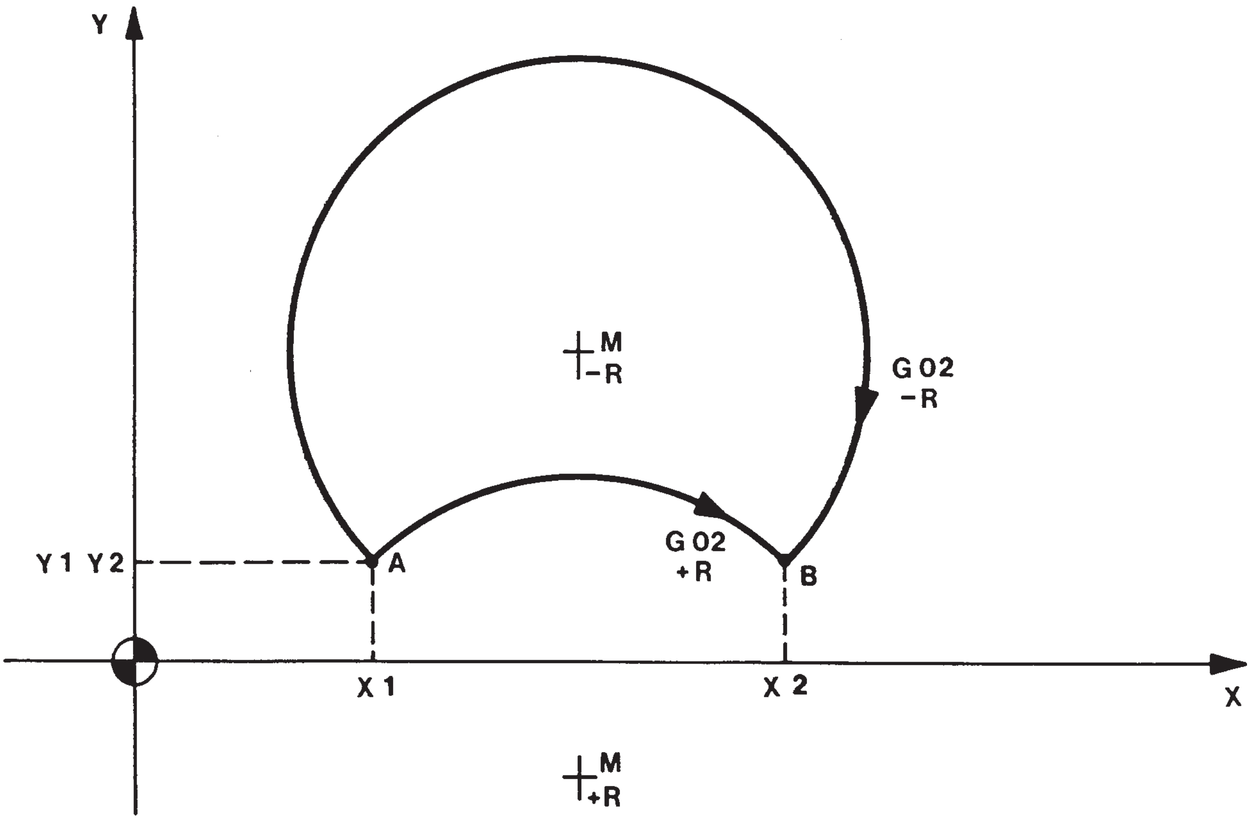


Figure 4-5

Radius R must be greater than or equal to the semi-chord AB/2 in order to give a centre point. If the R being input is less than the semi-chord, message 1407 is initiated and the control system selects an R = AB/2 without the program being shut down.

4.4 Automatic selection of linear and circular interpolation

G123 selects automatically after analysis of the position of three points G01 / G02 / G03:

1st point	position
2nd point	programmed position in the actual block
3rd point	programmed position in the next block

Example G123 -> G01

N10	G01	X20	Y50	position
N20	G123	X70	Y45	position in actual block
N30		X130	Y40	position in next block

Example G123 -> G02

N10	G01	X20	Y50	position
N20	G123	X70	Y71,5	position in actual block
N30		X130	Y63	position in next block

Example G123 -> G03

N10	G01	X20	Y50	position
N20	G123	X70	Y18	position in actual block
N30		X130	Y18	position in next block

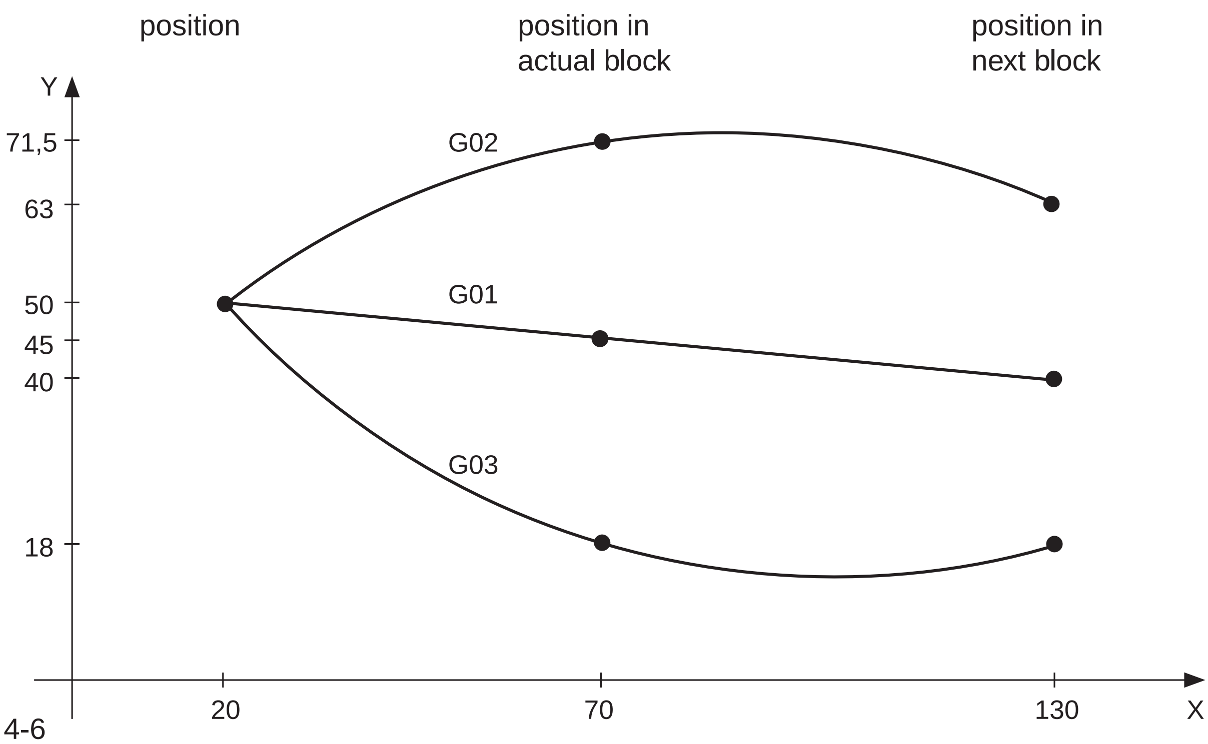


Figure 4-6

4.4 Automatic selection of linear and circular interpolation

Program example

```
P2050
N2                                                  { G123 free outline }
N10   G49 G56 G0 F3000   X:0       Y:0      Z:2
N20                      X:21.1    Y:9.300  Z:-1    { point 1 }
N30   G123 G1            X:19.6    Y:54.3           { point 2    selection G02 }
N40                      X:45.0    Y:61.0           { point 3 }
N50   G123               X:63.0    Y:46.7           { point 4    selection G01 }
N60                      X:82.9    Y:30.5           { point 5 }
N70   G123               X:81.2    Y:18.8           { point 6    selection G03 }
N80                      X:108.8   Y:21.6           { point 7 }
N90   G123               X:101.0   Y:33.0           { point 8    selection G02 }
N100                     X:110.3   Y:64.4           { point 9 }
N110  G123               X:108.2   Y:40.6           { point 10   selection G03 }
N120                     X:131.8   Y:25.5           { point 11 }
N130  G123               X:135.7   Y:13.8           { point 12   selection G01 }
N140                     X:138.447 Y:4.4            { point 13 }
N150                                        Z:2
N160  M30
```

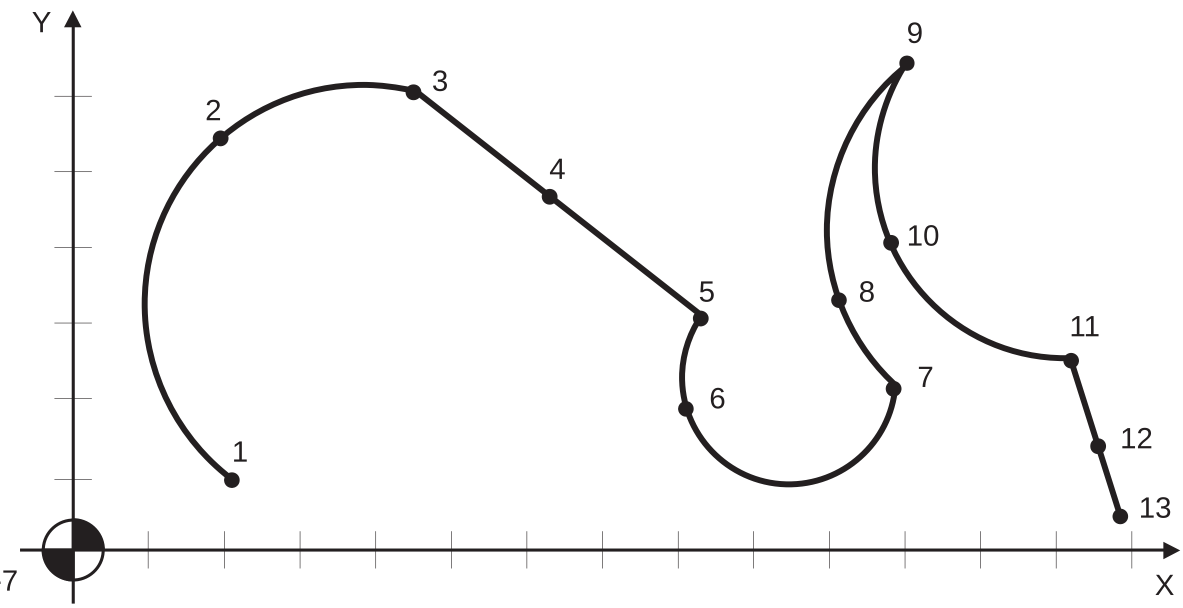


Figure 4-7

4.5 G04 Dwell

A dwell is called up by G04 and can be programmed between 0 and 999.999 s. The dwell must be input in the correct format, i.e. leading and trailing zeros and the decimal point should be input.

Example: G04.001.50 (1.5 s dwell)

The G04 function is only operative in the block in which it was written. G04 is operative after the traverse information and before the M functions subsequently active.

4.6 G05 / G06 / G07 driving direction of round axes

G5 driving direction positively, against clockwise direction.

G6 driving direction negatively, in the clockwise direction.

G7 driving direction neg. / pos., selection shortest path to the target position.

Examples driving direction, if will absolutely proceed!

Function	Start pos	Target pos	Direction.	Drive	Display	
- G5	0	90	pos.	90°	90	
- G5	270	35	pos.	125°	35	
- G5	150	-45	pos.	165°	315	
- G5	350	-10	not applicable	0°	350	initial pos = target pos
- G5/G91	70	30ink.	pos.	30°	100	
- G5/G91	180	-150ink.	neg.	150°	30	

Note: With incremental the technique the functions G5/G6/G7 are ineffective. The driving direction certainly by the sign.

Function	Start pos	Target pos	Direction.	Drive	Display	
- G6	0	90	neg.	270°	90	
- G6	270	35	neg.	235°	35	
- G6	150	-45	neg.	195°	315	
- G6	350	-10	not applicable	0°	350	initial pos = target pos
- G6/G91	70	30ink.	pos.	30°	100	
- G6/G91	180	-150ink.	neg.	150°	30	

Function	Start pos	Target pos	Direction.	Drive	Display	
- G7	0	90	pos.	90°	90	
- G7	270	35	pos.	125°	35	
- G7	150	-45	pos.	165°	315	
- G7	350	-10	not applicable	0°	350	initial pos = target pos
- G7/G91	70	30ink.	pos.	30°	100	
- G7/G91	180	-150ink.	neg.	150°	30	

4.7 G08 / G09 Precision stop block-by-block

G08 precision stop, block-by-block, On
G08 precision stop, block-by-block, Off

G08 initiates a precision stop at the end of a block.

The function overwrites a programmed G29 block by block. The precision stop is automatically set block by block with G00 and G81 to G85.

G09 overwrites a precision stop programmed with G28 block by block

4.8 G10 / G11 Polar coordinate system

G10 Polar coordinate system on
G11 Polar coordinate system off

The function G10 is used to activate the programming of target points in polar coordinates. The function G11 deactivates this function. Both functions act modally.

The coordinates programmed in the block with G10 define the pole, but not the travel of the axes (this is only valid for the two coordinates of the interpolation plane defined with G17...G19). If the coordinates are not programmed, the existing pole is kept.

At the program end or if the program is interrupted, the programmed pole as well as the polar radius (x) and polar angle (c) are cleared. Polar radius (x) and polar angle (c) act modally.

The polar coordinate system plane corresponds to the interpolation plane defined with G17...G19. The polar radius is always interpreted as a positive value!

Polar radius and polar angle can be corrected with G44.

4.8 G10 / G11 Polar coordinates (continued)

Definition of the poles in different interpolation planes

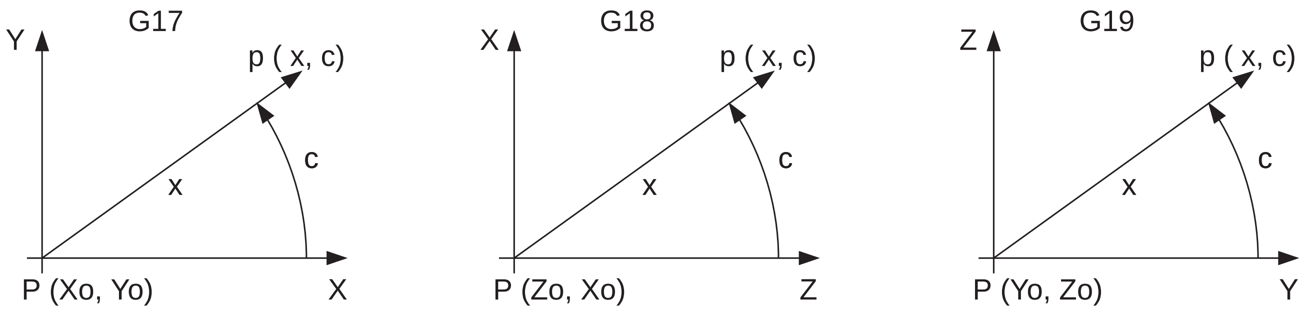


Figure 4-8

Parameters used for polar coordinate system

G10	Activating the polar coordinate input position with radius vector x and polar angle c	
G11	Deactivating the polar coordinates	
G17	XY plane	pole P() is in plane XY, the X axis is the polar axis
G18	ZX plane	pole P() is in plane ZX, the Z axis is the polar axis
G19	YZ plane	pole P() is in plane YZ, the Y axis is the polar axis

q150 Polar coordinate system: identification for radius vector (e.g. ‘x’)
Input of the ASCII-code Default: ‘x’

q151 Polar coordinate system: identification for polar angle (e.g. ‘c’)
Input of the ASCII-code Default: ‘c’

4.8 G10 / G11 Polar coordinates (continued)

Program example: Coordinates in X / Y

```
P656
N05   G55  G49   G00   F3000     T1     M16
N10                    X:0       Y:0    Z:2    { point 1 }
N20              G01                    Z:-2
N30                    X:90      Y:0           { point 2 }
N40                    X:60      Y:25          { point 3 }
N50                    X:90      Y:60          { point 4 }
N60              G03   X:30      Y:85   R:34   { point 5 }
N70              G02   X:45      Y:55   R:18   { point 6 }
N80              G01   X:0       Y:0           { point 1 }
N90                                     Z:2
N100       M30
```

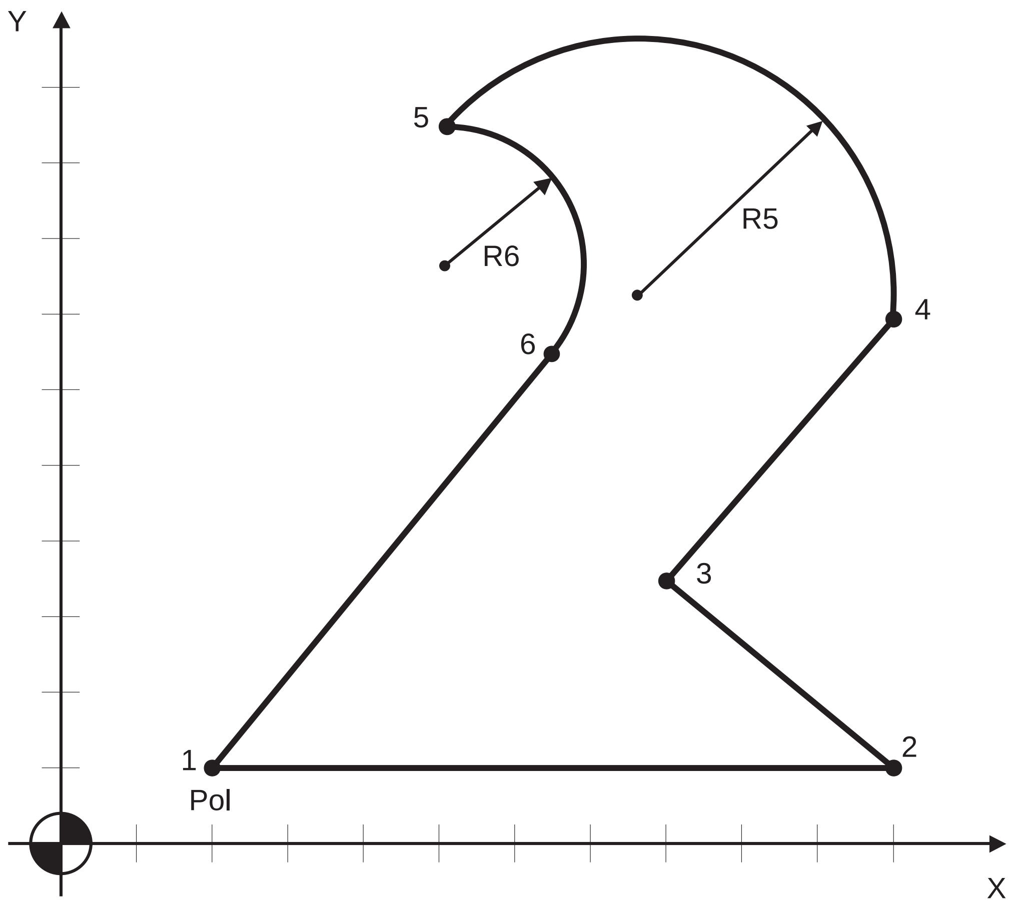


Figure 4-9

4.8 G10 / G11 Polar coordinates (continued)

Program example: Point of pole is the null point X:0 / Y:0

P657									
N05	G55	G49	G00	F3000	T1	M16			
N10					X:0	Y:0	Z:2		{ point 1 }
N20			G01				Z:-2		
N30		G10			X:0	Y:0			{ point 1 pole set }
N40					x:90	c:0			{ point 2 }
N50					x:65	c:22.619			{ point 3 }
N60					x:108.166	c:33.690			{ point 4 }
N70			G03		x:90.138	c:70.559	R:34		{ point 5 }
N80			G02		x:71.063	c:50.710	R:18		{ point 6 }
N90			G01		x:0	c:0			{ point 1 }
N100		G11					Z:2		
N110		M30							

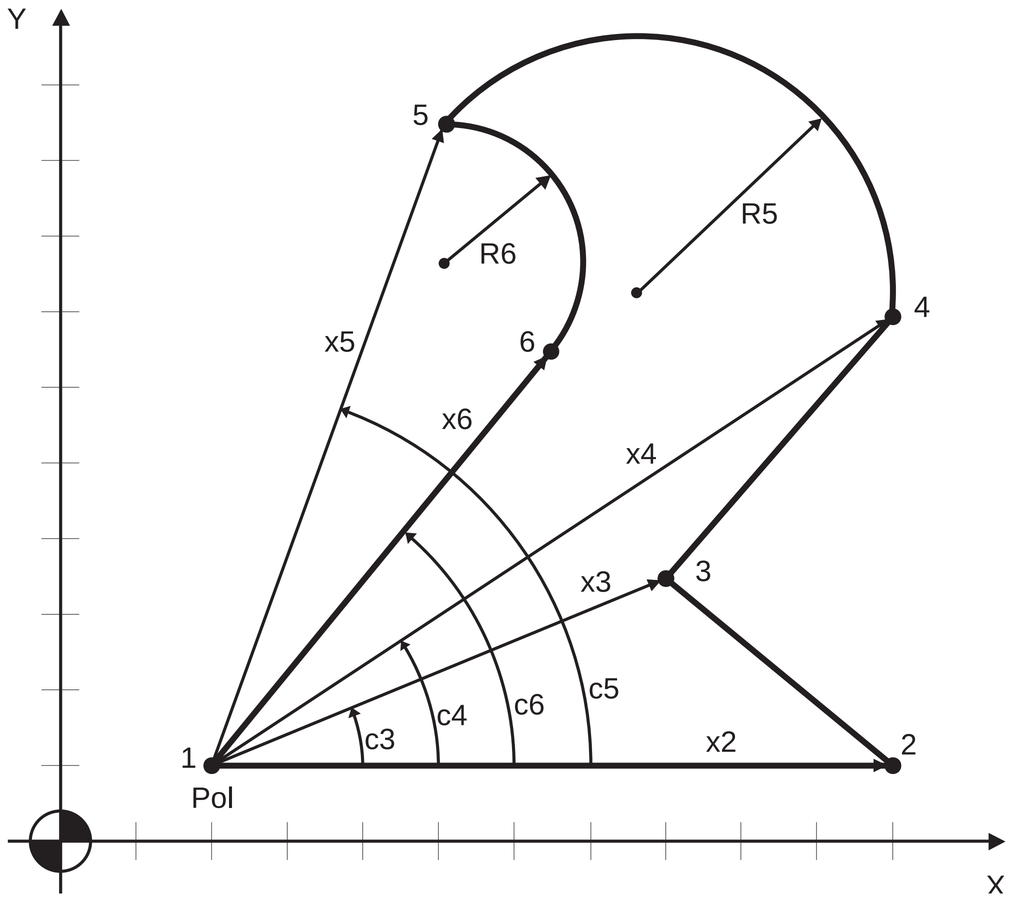


Figure 4-10

4.8 G10 / G11 Polar coordinates (continued)

Program example: Point of pole are thr points 2, 3 und 6

P659								
N05	G55	G49	G00	F3000	T1	M16		
N10				X:0		Y:0	Z:2	{ point 1 }
N20			G01				Z:-2	
N30				X:0		Y:0		{ point 2 }
N40		G10		X:90		Y:0		{ point 2 pole set }
N50				x:39.05		c:140.194		{ point 3 }
N60				X:60		Y:25		{ point 3 pole set }
N70				x:46.09		c:49:398		{ point 4 }
N80		G11	G03	X:30		Y:85	R:34	{ point 5 pole reset }
N90			G02	X:45		Y:55	R:18	{ point 6 }
N100		G10		X:45		Y:55		{ point 6 pole set }
N110			G01	x:71.062		c:230.710		{ point 1 }
N120		G11					Z:2	{ point 1 pole reset }
N130		M30						

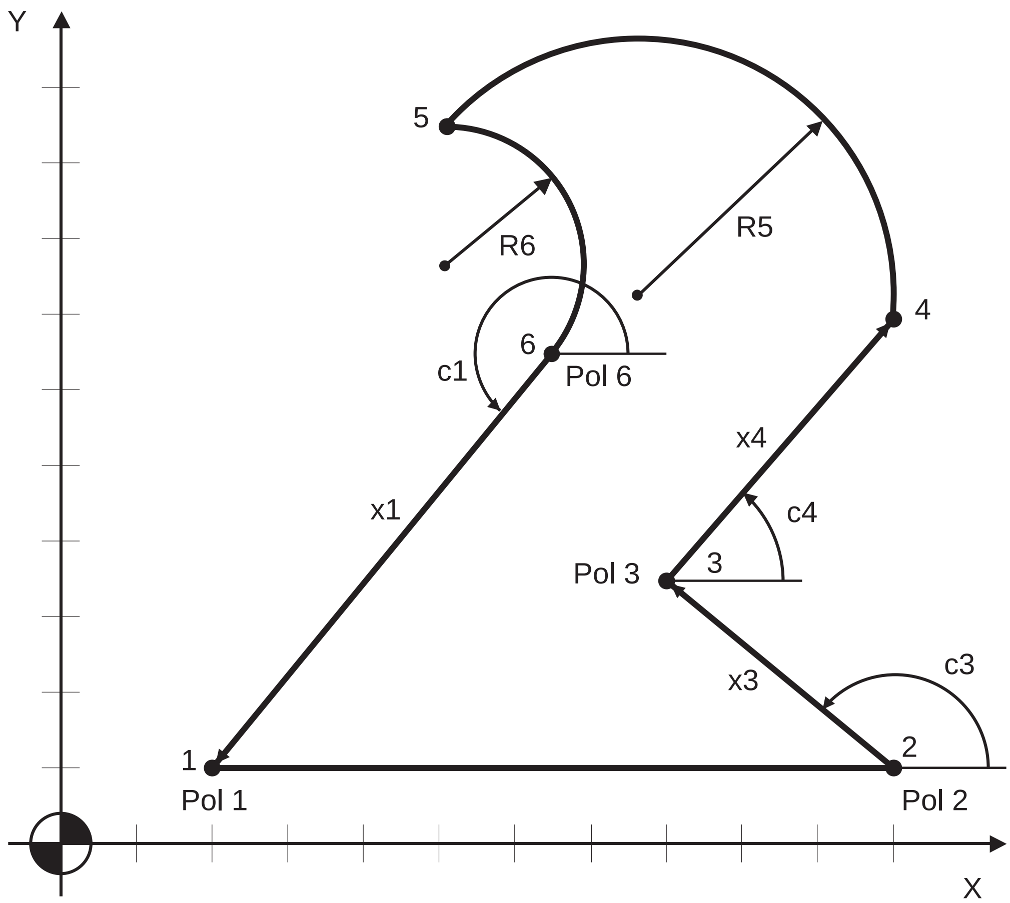


Figure 4-11

4.9 G12 Rapid graphical programming environment

Contour sections can be linked by of straight lines and circles.

This function acts block-by-block.

Parameter	Leg	Radius
	P70	P71

Rapid programming is only carried out with blocks in which the distance to be traversed is unequal to zero (differences in distances are unequal to zero for straight lines, radius is not equal to zero for circles).
Blocks with distances equal to zero are carried out at the start of the chamfer or circle segment.

Linear programming

A chamfer can be inserted at the intersection of two straight lines.
The length of the chamfer is given by P70 and the intersection point SP is programmed in the selected interpolation plane.

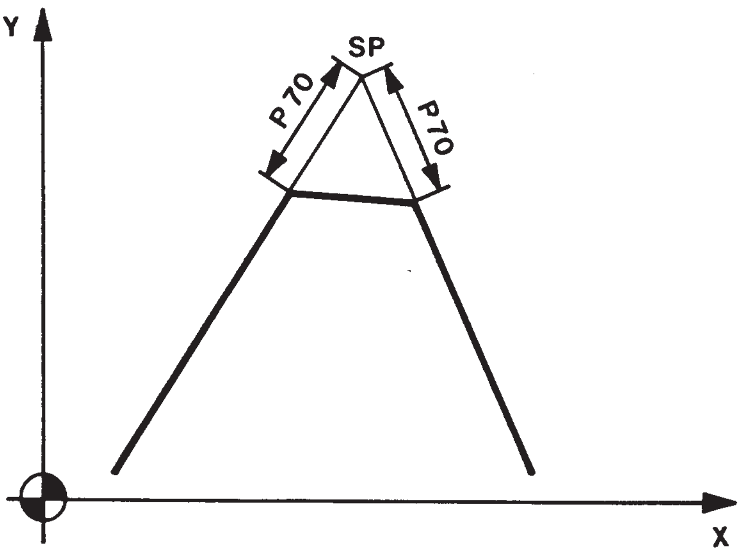


Figure 4-12

4.9 G12 Rapid graphical programming environment (continued)

Radius programming

A circle segment can be inserted at the intersection point of
- two straight lines,
- one straight line and one circle,
- two circles.

The radius of the circle is given by P71 and the intersection point SP is programmed in the selected interpolation plane.

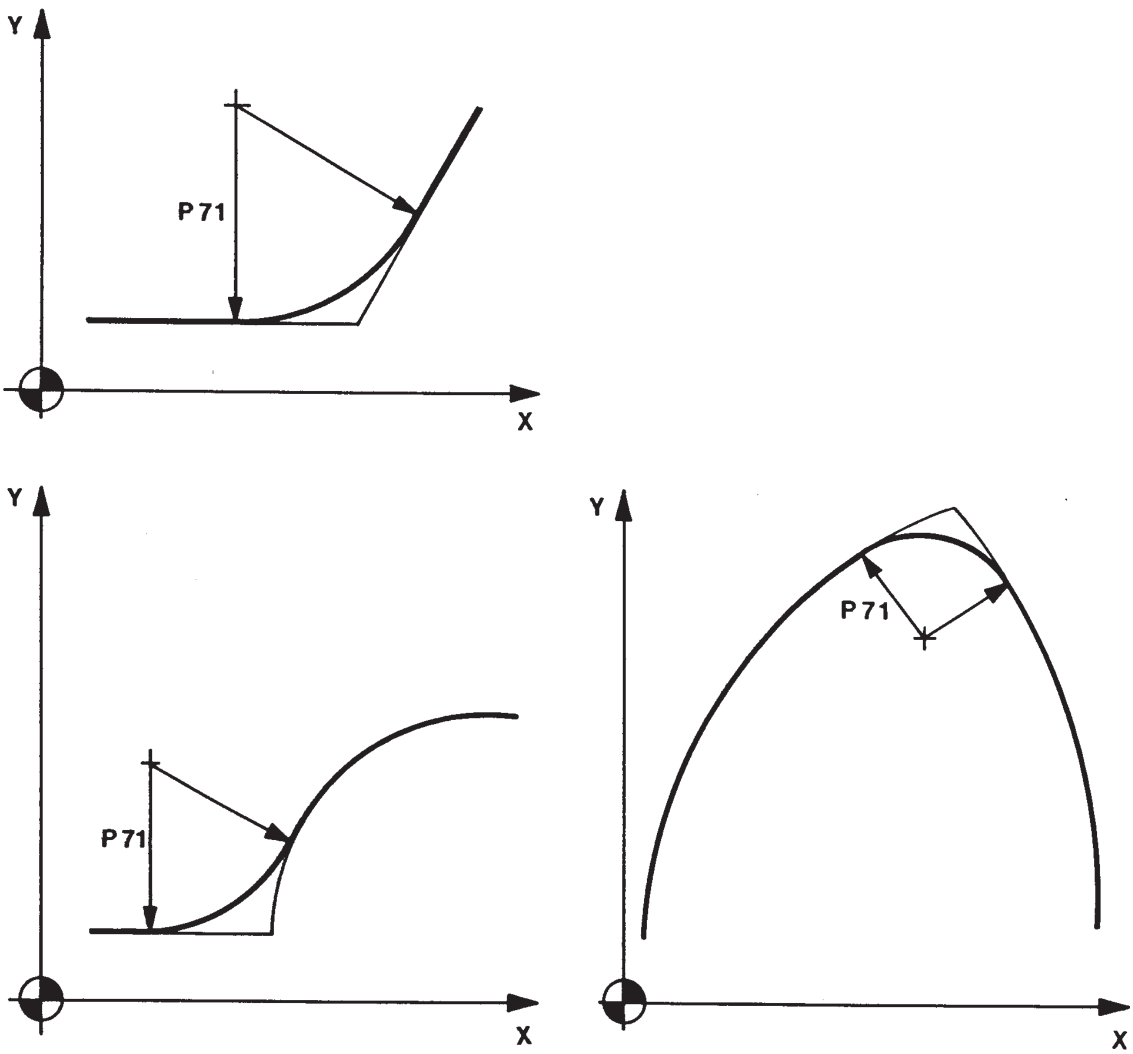


Figure 4-13

4.9 G12 Rapid graphical programming environment (continued)

Program example: Outline path short programming with radius and leg

P391												
N10	G55	G29	G48	G00	F2000	T1	M16	X0	Y0	Z2		
N20		G01								Z:-1		
N30	G12			P71:8				X:40	Y:0		{ point 1	radius }
N40	G12	G02	R65	P71:28				X62.5	Y:91		{ point 2	radius }
N50	G12	G02	R65	P71:15				X95	Y:35		{ point 3	leg }
N60	G12	G01		P71:8				X120	Y:35		{ point 4	radius }
N70	G12	G02		P71:12		I95	J35	X95	Y:10		{ point 5	radius }
N80	G12	G03		P71:10		I70	J10	X45	Y:10		{ point 6	radius }
N90		G01						X34.5	Y:10		{ point 7	radius }
N100				G00						Z:2		
N110	M30											

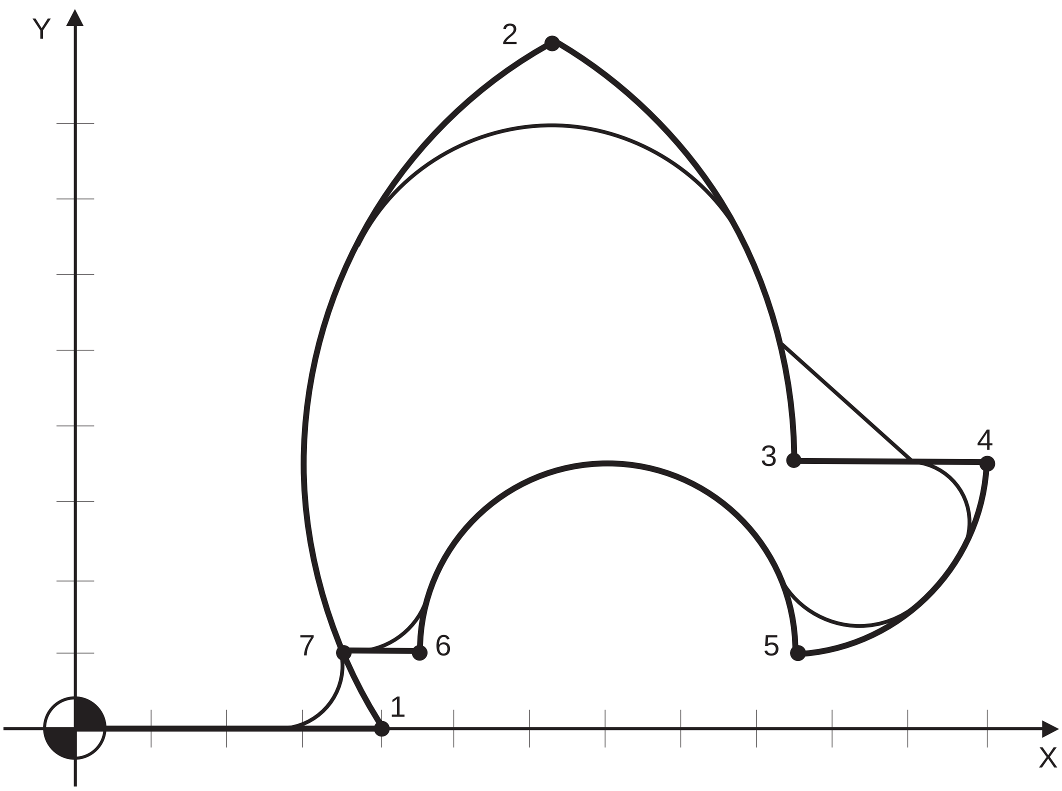


Figure 4-14

4.10 G13 / G14 tangential axis

G13 tangential axis off
G14 tangential axis on

G13 switches the tangential axis off.

G14 switches the tangential axis on.

With this function it is possible to control a round axis so that it is always in a certain position (tangential) to the path of the main axes.
For example for band saws, glass cutting, moist collectors.

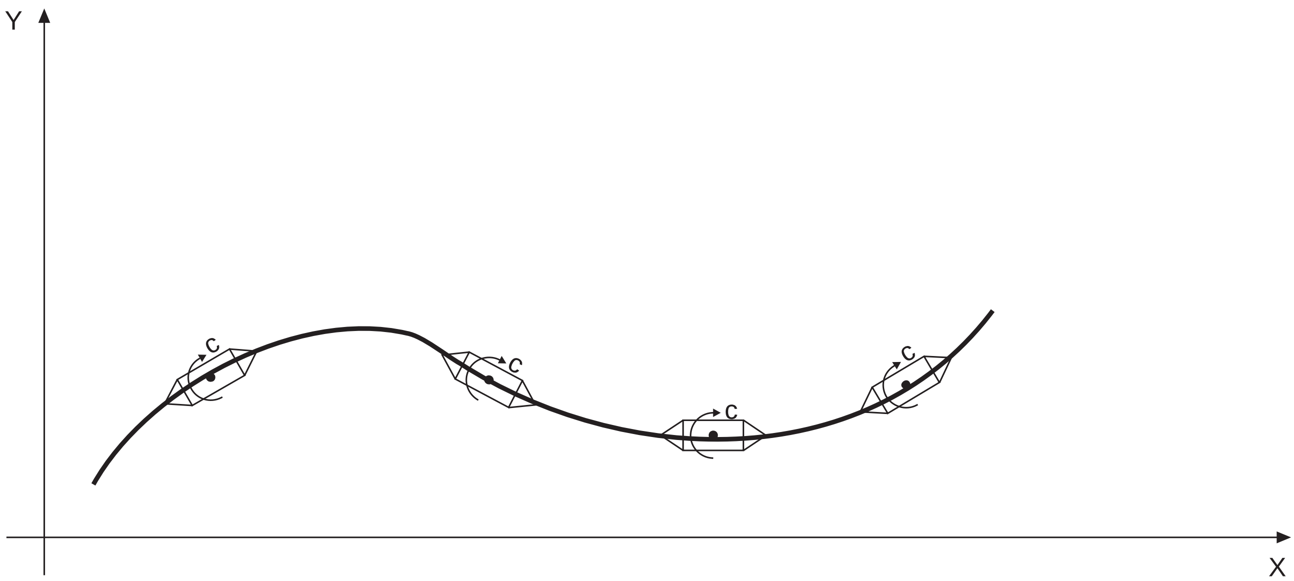


Figure 4-15

4.11 G15 / G16 Polar transformation

G15 Polar transformation off
G16 Polar transformation on

G15 switches the polar transformation off.

G16 switches the polar transformation on.

The polar transformation is applied for example for programming of uneven parts on a grinding machine or lathe with polar coordinate system. Usually the workpiece can be described more easily in the cartesian coordinate system. With the polar transformation G16 the control converts the cartesian coordinates (X/Y/Z) into polar coordinates (radius vector x and polar angle c). This method is very advantageous, because it requires less time for contour featuring.

Definition of the polar coordinate system

With the polar coordinates each point of the coordinates are determined by the two values radius vector ‘x’ and polar angle ‘c’.

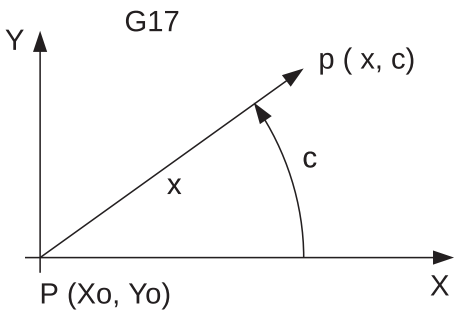


Figure 4-16 Polar coordinates with G 17 (XY - plane)

The radius vector x is the distance of point p(x,c) to the origin P(Xo,Yo).
The polar angle c is between the X-axis and the connecting straight line from the origin P(Xo,Yo) to point p(x,c).
The X - axis is marked as polar axis and the origin P(Xo,Yo) as pole. The polar angle is positive, if it is measured from the pole axis X counter-clockwise direction. The angle is negative in clockwise direction.

4.11 G15 / G16 Polar transformation (continued)

Utilized parameter for polar coordinate system

G15	Selecting the polar coordinate transformation
G16	Call up of the polar coordinate transformation, input position with X, Y
G17	XY plane pole P() lies in XY Plane, X axis is polar axis
G18	ZX Plane pole P() lies in ZX Plane, Z axis is polar axis
G19	YZ Plane pole P() lies in YZ Plane, Y axis is polar axis

P11820 Polar coordinate axis (physical axis number)
e.g.
Byte 3, 2, 1: the first 2 axes (x, y) 00 02 01 Hex
Byte 4 : change of direction (Bit 2, 1)

P11821	Radius vector offset (ro) for polar coordinate transformation	[mm]
P11822	Polar angle offset (wo) for polar coordinate transformation	[degrees]
P11823	Pole offset (po) for polar coordinate transformation	[mm]

4.11 G15 / G16 Polar transformation (continued)

Example: Polar transformation G16 for square contour

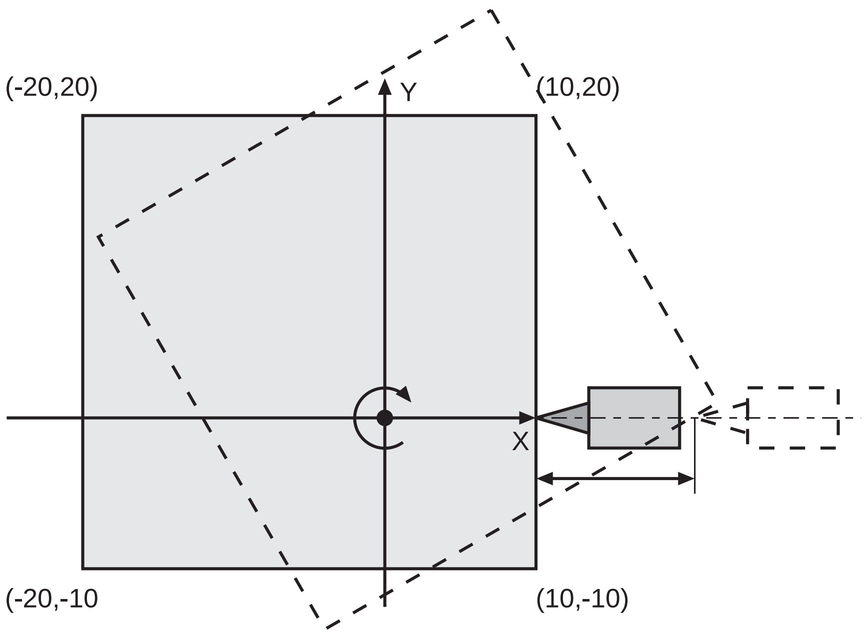


Figure 4-17 Square contour

N10 G17 G1 F1000	;	Definition of coordinate plane
. ..		
N150 G16 X10 Y0	;	Call up polar transformation and X,Y
	;	are transformed accord. equation 2 into polar coorinates
	;	here: Xo = Yo = 0;
N160 X10 Y 20	;	Position X= 10 and Y= 20 are transformed into
	;	polar coordinates. Approach to transformed position
N170 X - 20 Y 20	;	X= -20 and Y= 20 are transformed into polar coordinates.
	;	Approach to transformed position
N180 X - 20 Y-10	;	X= -20 and Y= -10 are transformed into polar coordinates.
	;	Approach to transformed position
N190 X10 Y-10	;	X= 10 and Y= -10 are transformed into polar coordinates.
	;	Approach to transformed position
N200 X10 Y 0 G15	;	X= 10 and Y= 0 are transformed into polar coordinates.
	;	Approach to transformed position
	;	Switch off polar transformation.
N220 . . .		
N230 M30	;	Program end

4.11 G15 / G16 Polar transformation (continued)

Example: Polar transformation G16 for cam contour

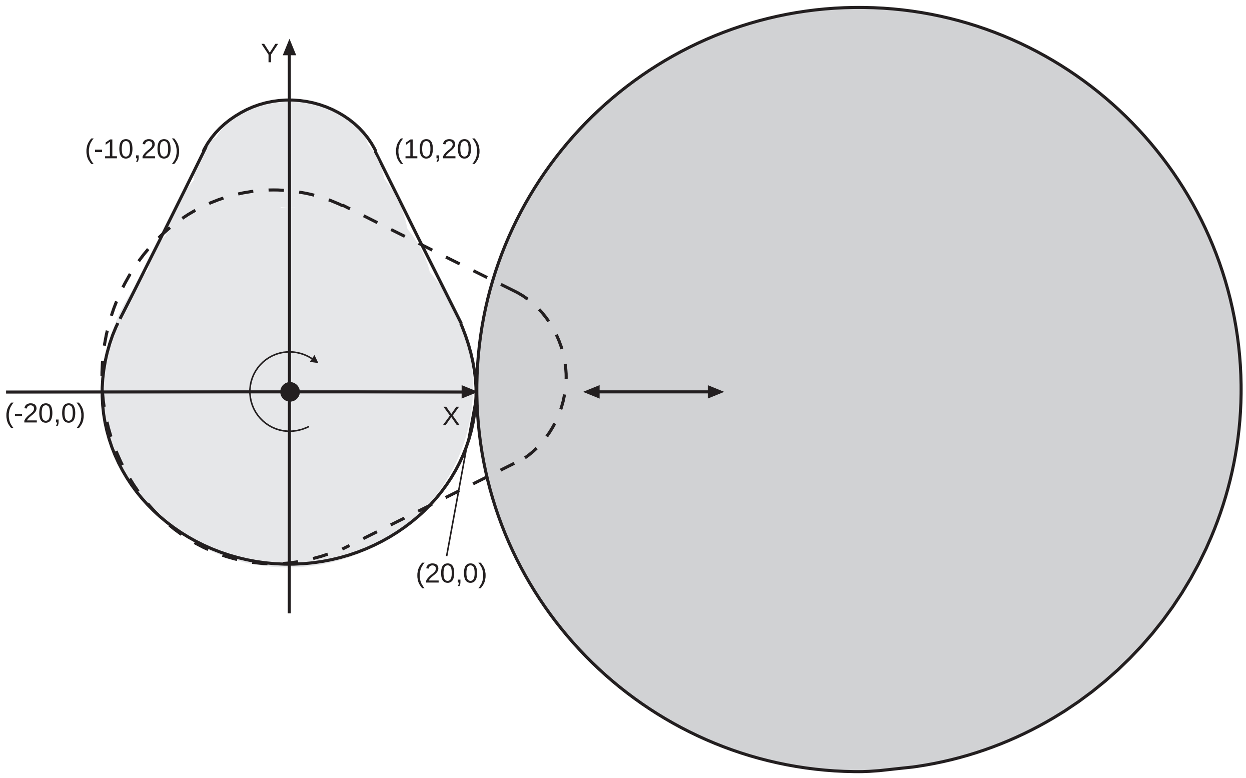


Figure 4-14 Cam contour

```
N10  G17 G1 F1000          ;   Definition coordinate plane
. ..
N150 G16 X20 Y0            ;   Call up polar transformation and X,Y
                           ;   are transformed accord. equation 2 into polar coorinates
                           ;   here: Xo = Yo = 0;
N160  X10 Y 20             ;   Position X= 10 and Y= 20 are transformed into
                           ;   polar coordinates. Approach to transformed position
N170  G2 X -10 Y 20  R11   ;   X= -10 and Y= 20 are transformed into
                           ;   polar coordinates. Approach to G2 until transformed pos.  N180
G1 X -20 Y0                ;   X= -20 and Y= 0 are transformed into polar coordinates
                           ;   Approach to transformed position
N190 G2 X20 Y0 R22         ;   X=20 and Y= 0 are transformed into polar coordinates
                           ;   Approach to G2 until transformed position
N200 G15                   ;   Switch off  polar transformation.
N220 . . .
N230 M30                   ;   Program end
```

4.12 G17 / G18 / G19 Plane selection

The main planes for circle interpolation and tool correction offsets are selected using G17, G18 and G19.

Traverse functions	Main plane	Parameters for circle centre point
G17	XY	IJ
G18	ZX	KI
G19	YZ	JK

These functions act modally and overwrite each other mutually. The plane preset is controlled by the PLC parameter „WZ“ tool axis.
The tool correction offset takes place in the given interpolation plane. The correction offset should be cancelled using G40 before changing the interpolation plane.

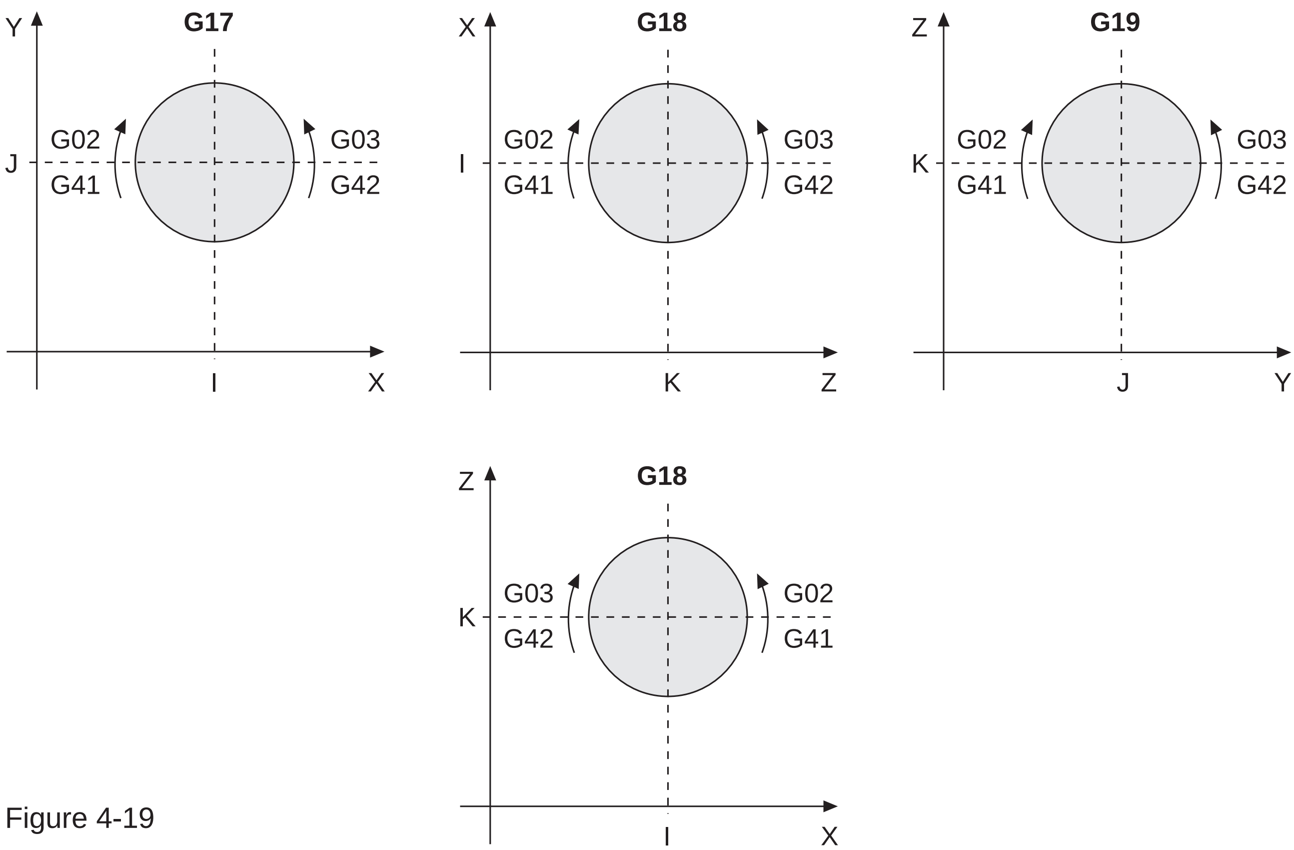


Figure 4-19

4.12 G17 / G18 / G19 Plane selection (continued)

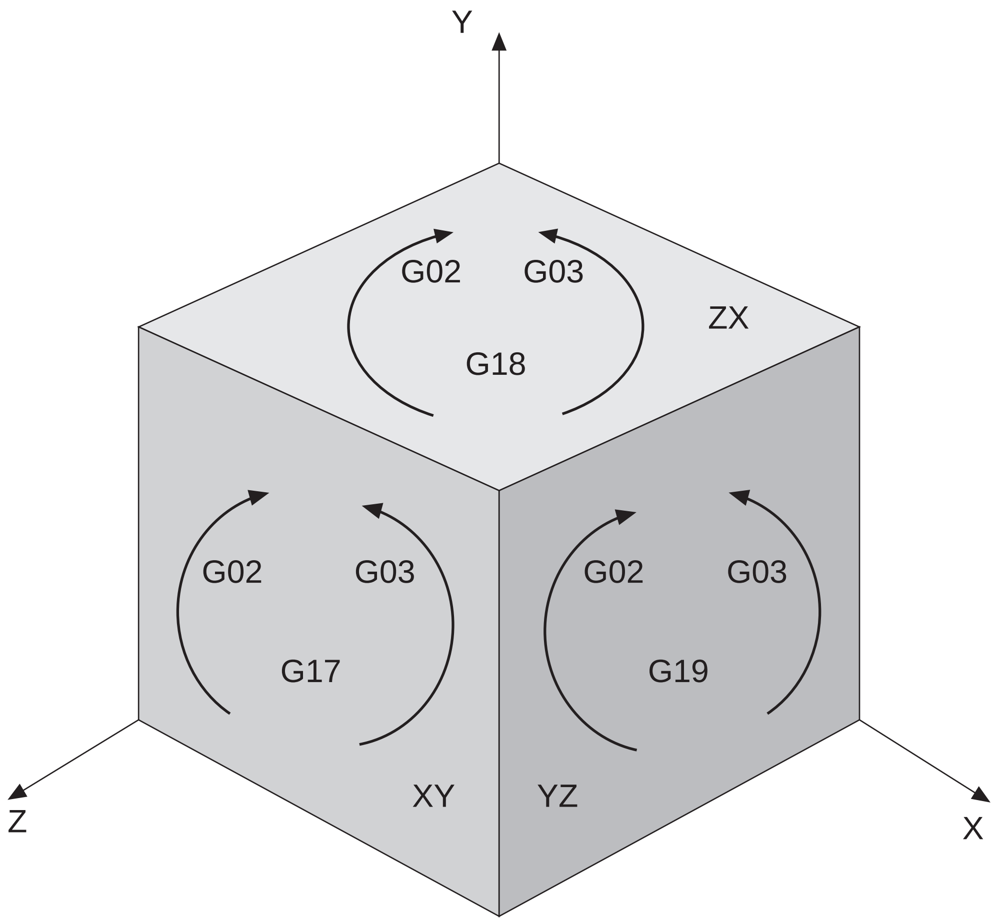


Figure 4-20

4.13 G28 / G29 Precision stop modal

G28 Switching on modal precision stop

G28 initiates a precision stop at the end of a block. The tool traverses to the end point programmed in the block and stops.
Subsequently it traverses to the end point programmed in the next block and stops again.

The function acts modally and can be cleared by G29.

G29 Switching off modal precision stop

G29 switches off a precision stop programmed with G28.

The function acts modally and can be cleared by G28.

G29 is automatically set when the program starts.

4.14 G40 / G41 / G42 Milling cutter path correction offsets

G40 Clearing milling cutter path correction offsets

The milling cutter path correction offset programmed using G41 or G42 is cleared using G40.

G40 acts modally and can be cleared by G41 and G42. G40 is automatically set when the program starts.

G41 / G42 Actuating milling cutter path correction offset

A milling cutter path correction offset can be switched on using G41 and G42.

G41 effects a correction to the left of the programmed path.
G42 effects a correction to the right of the programmed path.

The reference direction is in the feed direction in each case.

The functions act modally, overwrite each other mutually and can be cleared by G40.

The feed rate is proportional to the milling cutter centre point path when the tool correction offset is switched on.

Further details on this subject appear in the section „Tool correction offset“.

4.15 G43 / G44 Axis correction

With G43 / G44 an axis correction can be programmed.

G43 axis correction off
G44 axis correction on

The programmed traverse informations in the block with G44 determine the axis correction, whereby no traverse is driven.
These corrections remain modally.

At program end or program abort these corrections are cleared, or switched off with G43.
A programmed G44 correction is a component, which is added on all further traverse information.

With G44 polar angles and polar radius can also be corrected.

Input: e.g.:

N100	G44 X10 Y20 Z30 axes do not drive !
N120	G01 F1000 X100 Y100 Z100 axes drive to X: 110, Y: 120; Z: 130
N130	G43

4.16 G45 / G46 Turning of coordinates

The „Turn“ function initiates the turning of a coordinate pair about a given angle E.
The traverse function G45 switches turning on, G46 switches the function off. G45 and G46 act modally.

G45 / G46 Geometric definition turning

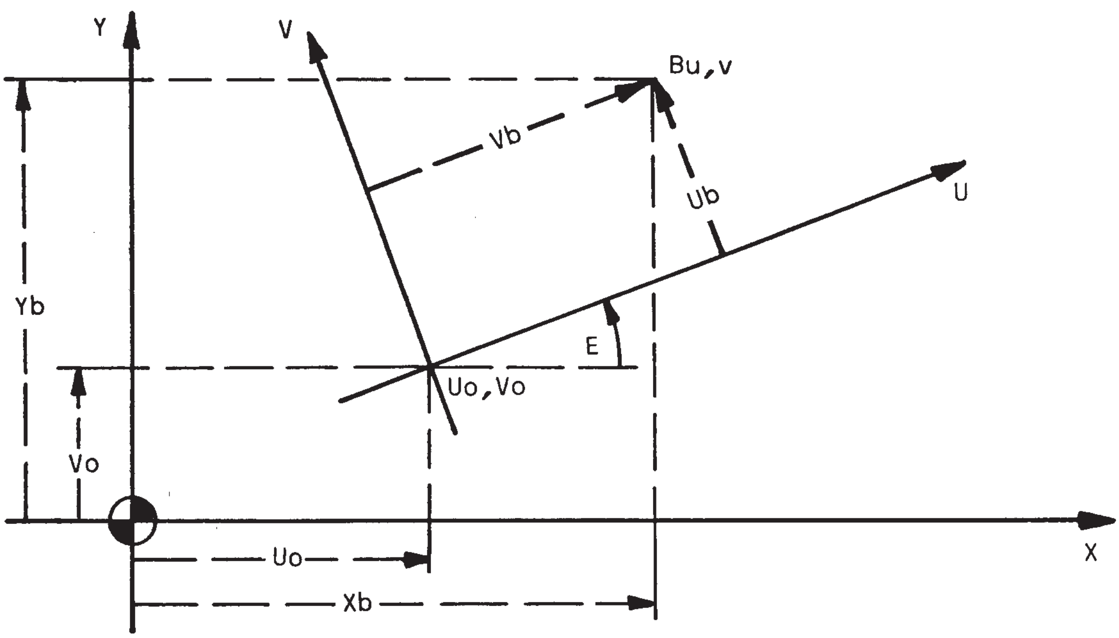


Figure 4-21

X, Y Machine coordinate system
U, V Coordinate axes of the turned coordinate system

E Angle about which the machine system X axis is to be turned in the direction of the machine system Y axis in order to retain the UV coordinate system orientation. Angle E is positive if measured counter-clockwise to the X axis and negative in the clock-wise direction.

Uo, Vo Original UV coordinate system coordinates, referred to the machine XY coordinate system zero point.
Ub, Vb Point B coordinates in UV coordinate system.
Xb, Yb Point B coordinates in XY coordinate system, referred to the effective shift (NP and G92).

Converting the UV system coordinates into those of the XY system
Xb = (U.cosE - V.sinE) + Uo
Yb = (U.sinE + V.cosE) + Vo

4.16 G45 / G46 Turning of coordinates (continued)

Programming the coordinate origins U0, V0 (referred to the machine system zero point) and the turning angle E take place via parameters.

Parameter	Characteristic	Significance
P160	Uo	UV system original coordinate
P161	Vo	UV system original coordinate
P163	E	Turning angle

The parameters act modally.

The parameters P160, P161 and the reference axis for the turning angle are dependant on the plane set.

Plane set	P160	P161	Reference axis for turning angle
G17 (XY)	Xo	Yo	X axis
G18 (ZX)	Zo	Xo	Z axis
G19 (YZ)	Yo	Zo	Y axis

Shifts with additive effect

In addition to the adjustment values P160, P161, P163, shifts can be programmed which act additively with respect to the direction.

Parameter	Characteristic	Significance
P165	U	Shift in U
P166	V	Shift in V
P168	E	Turning angle incrementation

When one of these parameters is called up, it’s value is added to the former value of the corresponding shift. The value obtained in this way acts modally. The original parameters are retained.

Cancellation of the additive shifts is by re-calling up the adjustment values P160, P161 or P163 (or by switching G45 off with G46).
The additive shifts are plane-dependent, as are the adjustment values.

4.16 G45 / G46 Turning of coordinates (continued)

Mirror imaging of the turned coordinate system

The mirror functions G61 and G62 are referred to the UV coordinate system:
G61 mirror images the U axis
G62 mirror images the V axis

The mirror imaged coordinates are expressed in the machine system XY in the following manner:
Xb = - (U.cosE - V.sinE) + Uo
Yb = - (U.sinE + V.cosE) +Vo

The effect on G92

The shifts programmed using G92 are carried out.

The effect on G66 on the turning function

G66 initiates the block-by-block switching off (G46) of turning G45.

Turning the circle centre point coordinates I, J, K

Turning the circle centre point coordinates I, J, K takes place according to the turning of coordinates as described above.

4.16 G45 / G46 Turning of coordinates (continued)

Example 1 Repeated machining with turning

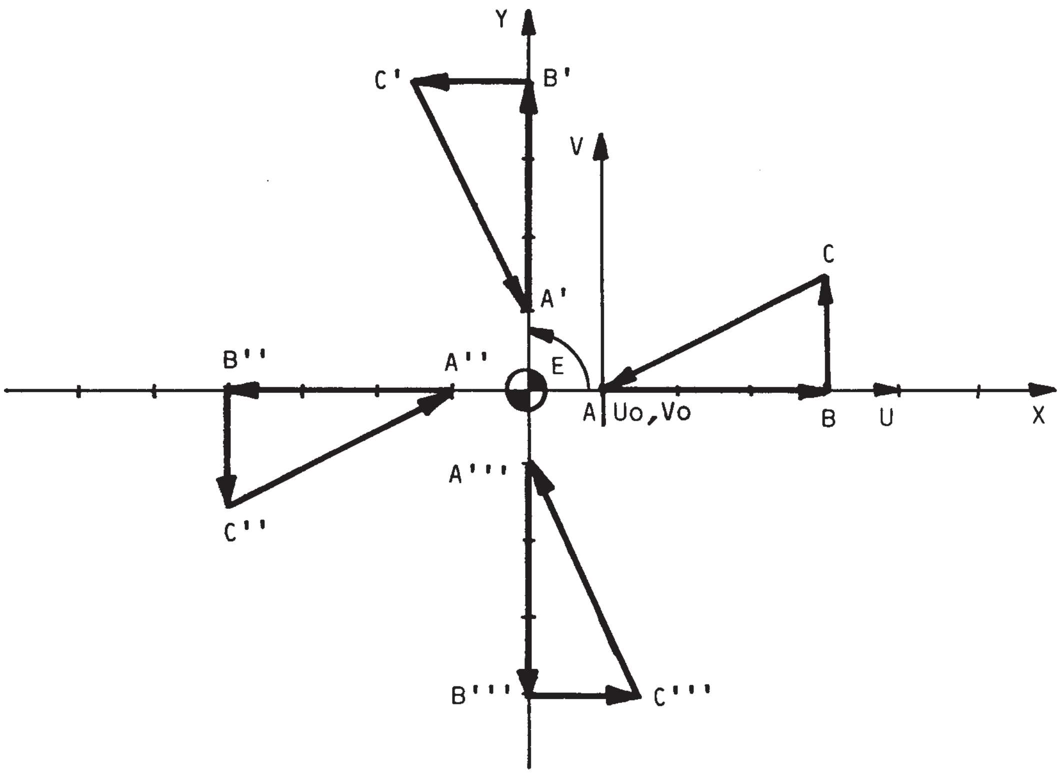


Figure 4-22

P1	N10	G45	G00	G54	Z2	P160:0	P161:0	P163:0	F2000 M24.4
	N20	G00	X10	Y0					
	N30	G01	Z-1,5						
	N40	X40							
	N50	Y15							
	N60	X10	Y0						
	N70	Z2	P168:90	M25					
	N80	M30							

The triangular shape ABC is to be machined four times with appropriate orientation of the +X, +Y, -X, -Y coordinate axes.

The machining sequence on the triangle is:
1st. Traverse A to B
2nd. Traverse B to C
3rd. Traverse C to A

4.16 G45 / G46 Turning of coordinates (continued)

Program run, example 1

Block 10 Switching on the coordinate turning function (G45)

Input set-up data U0=10 (P160) Vo=0 (P161) E=0 (P163)
Uo, Vo coincide with point A
Approach to Z safety dimension, Loop input

Block 20 Positioning to point A = Uo, Vo

Block 30 Tool axis in-feed to machining depth

Block 40 Machining along line AB

Block 50 Machining along line BC

Block 60 Machining along line CA

Block 70 Traversing tool axis away for positioning, incrementing the turning angle to +90 deg., skip back to Block 20

Block 20 Positioning to point „A“

Block 30 In-feed to machining depth
.
.
Block 70

Block 20 Point „A“ Machining, x-orientated
.
.
Block 70

Block 20 Point „A“ Machining, y-orientated
.
.
Block 70

Block 80 End of machining operations

The final machine position is at point „A“.

4.16 G45 / G46 Turning of coordinates (continued)

Example 2 Repeated machining with shift

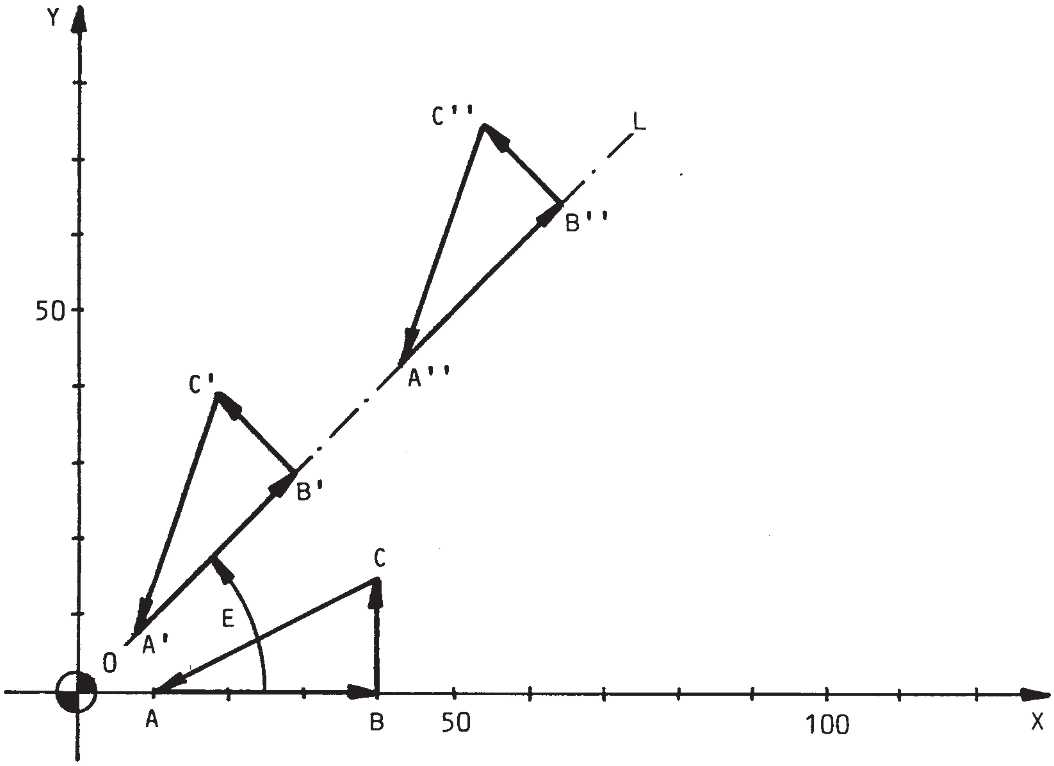


Figure 4-23

```
P2    N10   G45   G0    Z0    P160:0   P161:0   P163:45   F2000   M24.2
      N20   G00   X10   Y0
      N30   G01   Z50
      N40   X40
      N50   Y15
      N60   X10   Y0
      N70   Z70   P165:55   M25
      N80   M30
```

The basic triangular shape ABC is to be machined twice along line OL. Line OL makes an angle E to the axis. The distance between the two triangles A', B', C' and A", B", C" is 55mm (distance A'-A").

The machining sequence on the triangle is:
1st. Traverse A to B
2nd. Traverse B to C
3rd. Traverse C to A

4.16 G45 / G46 Turning of coordinates (continued)

Program run, example 2

Block 10	Switching the coordinate turning function on (G45)
	Set-up data input Uo=10 (P160) Vo=0 (P161) E=45 (P163)
	Approach to Z safety margin, Loop input
Block 20	Positioning to point A'
Block 30	Tool axis in-feed to machining depth
Block 40	Machining along line A' - B'
Block 50	Machining along line B' - C'
Block 60	Machining along line C' - A'
Block 70	tool axis retraction for positioning, shifting the Uo-value P165:55, skip back to Block 20

.
.
.
.

Block 20	Positioning to point A“
Block 30	Tool axis in-feed to machining depth
Block 40	Machining along line A“ - B“
Block 50	Machining along line B“ - C“
Block 60	Machining along line C“ - A“
Block 70	Tool axis retraction
Block 80	End of program

4.16 G45 / G46 Turning of coordinates (continued)

General data for the examples 3, 4 and 5

Program rectangle figure process

```
P452
N10  G0  F3000  Z1.5
N20  G1  G91  X0  Y0  Z-2.5
N30  X30
N40  Y20
N50  X-30
N60  Y-20
N70  Z2.5
N80  G90  M02
#
```

Parameter P8859

the parameter P8859 influences the order of rotation and shift.

P8859:0	with G92, G147, G48 and G49	first turninged, then shifted
P8859:1	with G92, G48 and G49, NP:A>0	first turninged, then shifted
P8859:2	with G92, G147, G48 and G49	first shifted, then turninged
P8859:3	with G92, G48 and G49, NP:A>0	first shifted, then turninged

4.16 G45 / G46 Turning of coordinates (continued)

Example 3 rectangle figure turn with different brackets, absolutely and incremental

```
P451
N10        T1  M16  G0
N20        G55  G49  G60  G0  F3000  X0  Y0  Z2  M28.452      {W 1}
N30  G45   P160:50  P161:10  P163:30  X0  Y0  Z2  M28.452      {W 2 turning}
N40        P165:50  P166:10  P168:30  X0  Y0  Z2  M28.452      {W 3 incremental turning}
N50  G46   G90                        X0  Y0  Z5
N60        M30
#
```

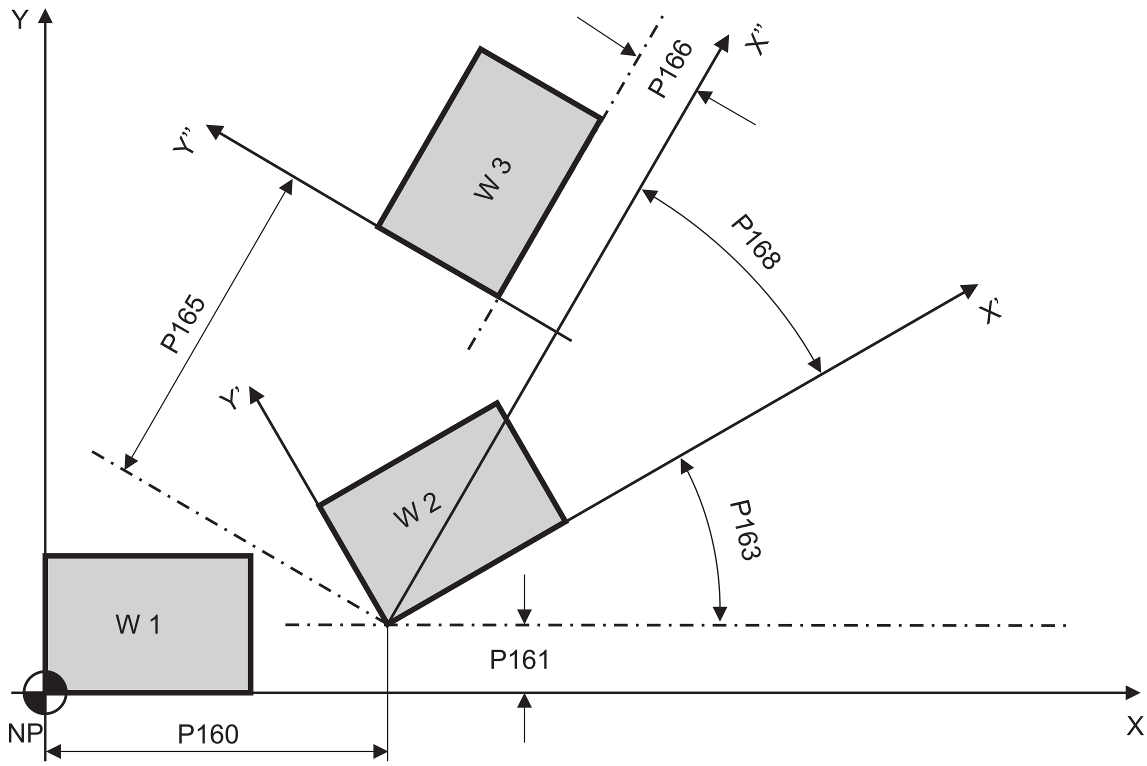


Fig. 4-24

4.16 G45 / G46 Turning of coordinates (continued)

Example 4 rectangle figure shift and turn

P8859:2 first shift by G92 and then turn by G45

```
P451
N10        T1 M16 G0
N20        G55  G147  G0  F3000 X0   Y0  Z2        M28.452    {W 1}
N30  G45   P160:50  P161:10  P163:30  X0   Y0  Z2        M28.452    {W 2 turning}
N40        P165:50  P166:10  P168:30  X0   Y0  Z2        M28.452    {W 3 incremental turning}
N50  G92                              X60 Y0                        {shift}
N60  G45   P160:50  P161:10  P163:30  X10 Y5  Z2 M28.452  {W 2' turning}
N70        P165:50  P166:10  P168:30  X0   Y0  Z2 M28.452  {W 3' incremental turning}
N80  G46   G90                        X0   Y0  Z5
N90        M30
#
```

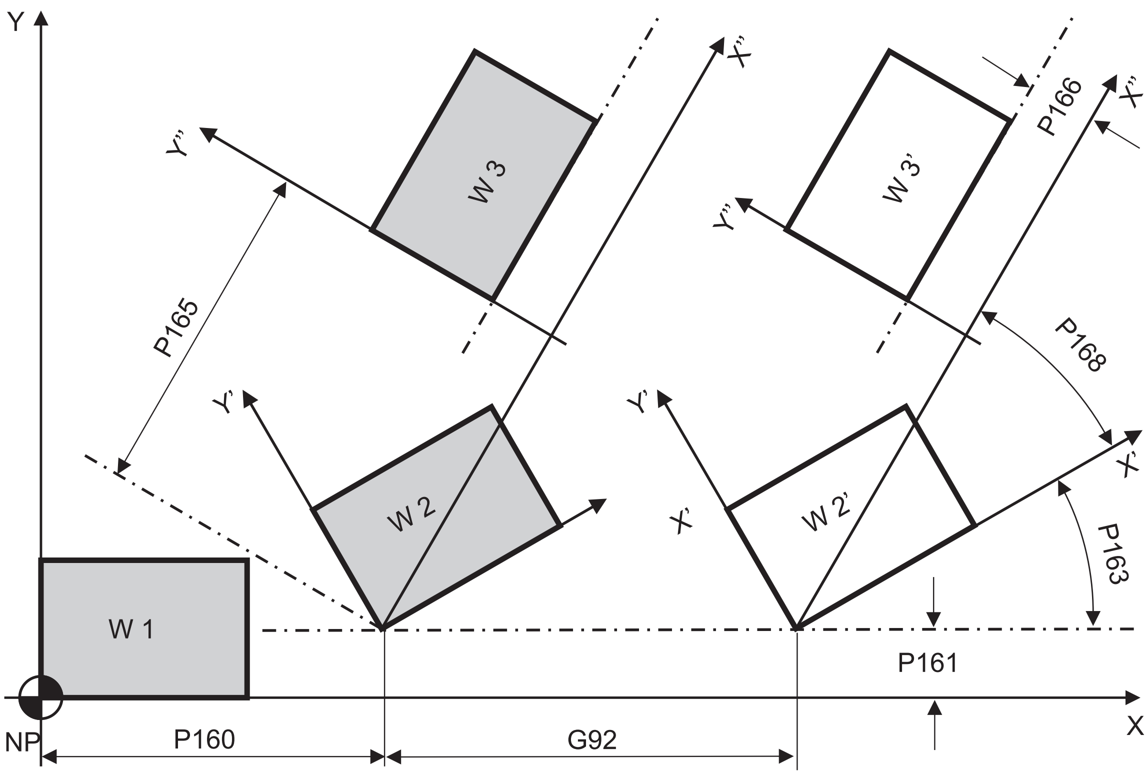


Fig. 4-25

4.16 G45 / G46 Turning of coordinates (continued)

Example 5 rectangle figure turn with different brackets and shift by G92

P8859:0 first turn by G45 and then shift by G92

```
P451
N10          T1 M16 G0
N20          G55  G147  G0  F3000 X0   Y0  Z2         M28.452    {W 1}
N30   G45    P160:50  P161:10  P163:30  X0   Y0  Z2   M28.452    {W 2 turning}
N40          P165:50  P166:10  P168:30  X0   Y0  Z2   M28.452    {W 3 incremental turning}
N58   G92                               X60 Y0                   {shift}
N60   G45    P160:50  P161:10  P163:30  X10 Y5  Z2    M28.452    {W 2" turning}
N70          P165:50  P166:10  P168:30  X0   Y0  Z2   M28.452    {W 3" incremental turning}
N80   G46    G90                        X0   Y0  Z5
N90          M30
#
```

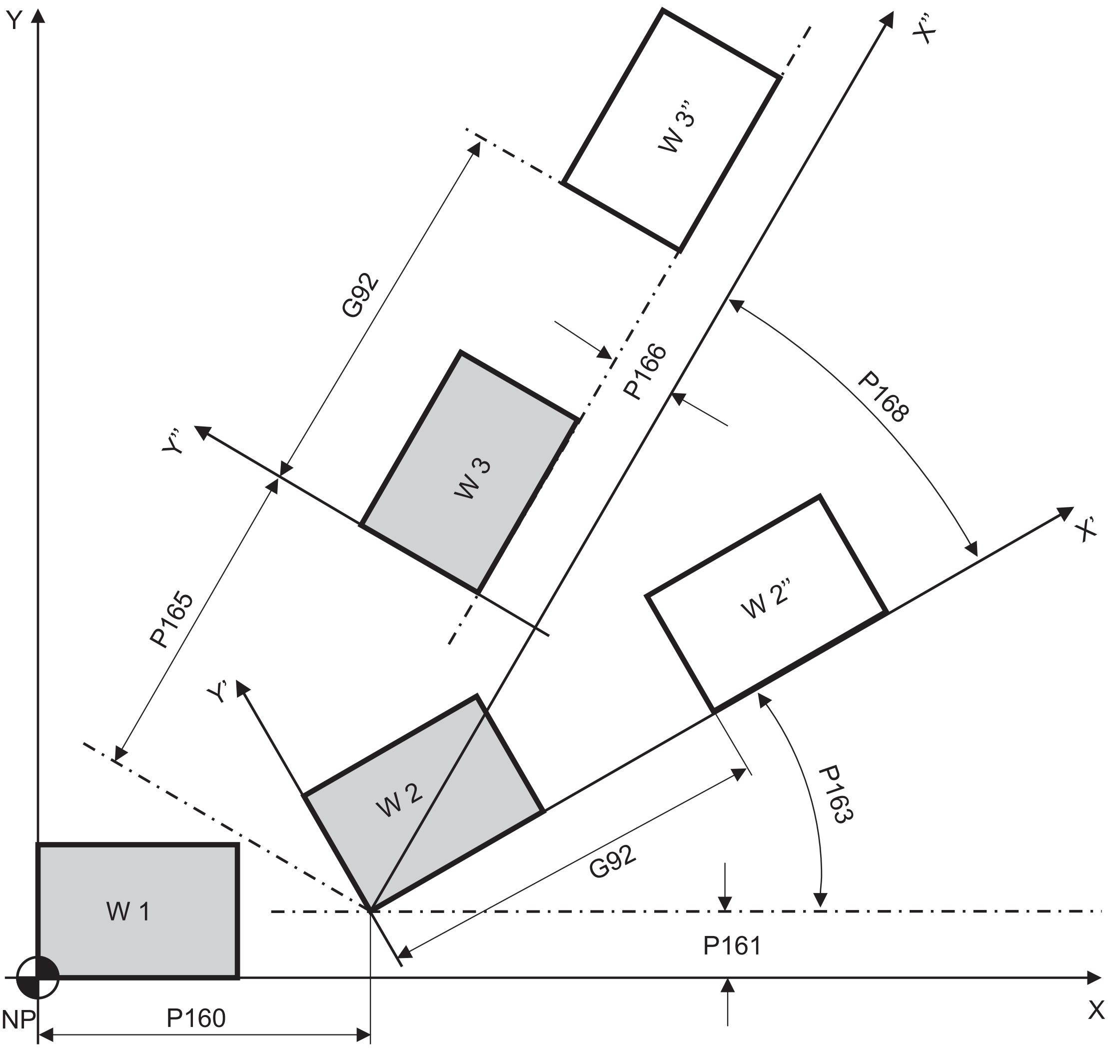


Fig. 4-26

4.17 G47 / G48 / G49 / 147 Robot transformation

It is advantageous for different applications, if the user can select the movements in different mode of coordinates when installing the machine (e.g. machine tool or handling system). With the BWO-ROBOT-system three modes of coordinates can be chosen.

Machine coordinates G47

The general and most frequent case is, that the movement is refered to a machine coordinate system, Thereby the coordinate axes and an origin of coordinates are determined. The cartesian coordinates are used.

Tool coordinates G48

If the movement is refered to the tool or a grip, infeed movements can be executed very simple for machining - or assembling tasks. In this case, the coordinates are oriented at the tool peak.

Workpiece coordinates G49

Programming is facilitated for many machining functions, if the movement is refered to a coordinate system oriented at the workpiece.

Switching between the above named modes of coordinates (G47 / G48 / G49) is an essential help for the user at programming with Teach-in, because e.g. for establishing a NC-program the contours of a workpiece only have to be touched and stored (teached). If equal workpieces are processed in different positions, the same NC program can be used through a single transformation of the coordinates.

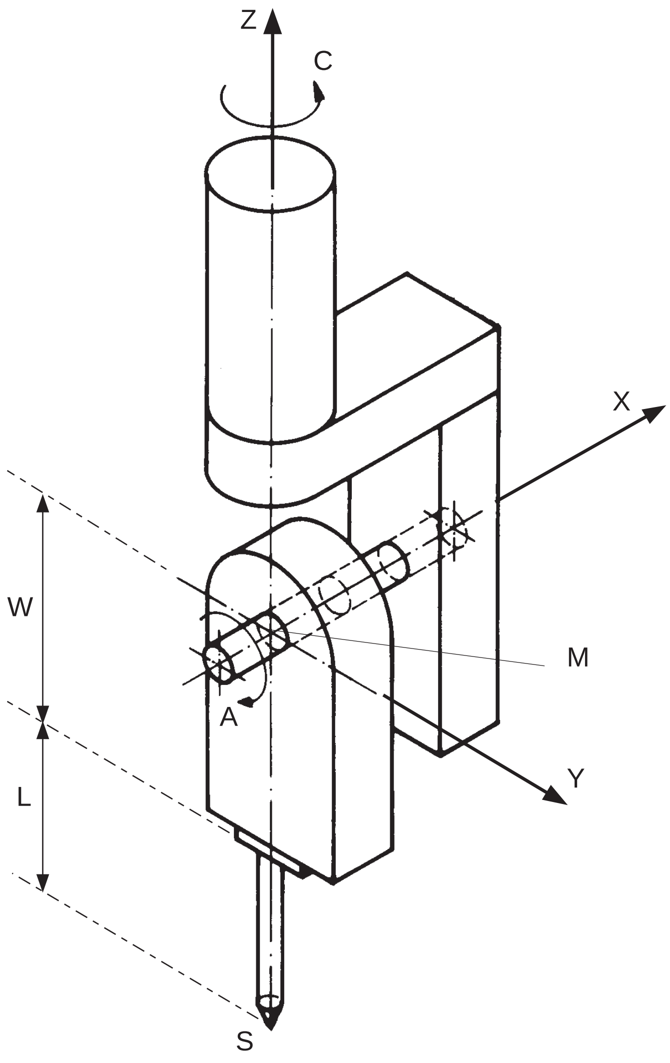
Transformation off by G147

G147 switches the transformation off.

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

The coordinate system demands the following axis configuration:

The A - axis turns around an axis parallel to the X - axis.
The C - axis turns around an axis parallel to the Z - axis.
The spindle axis stands vertically on the A - axis.
The intersection of all three rotation axes is the center point M.
The positive rotating direction of C is seen from positive Z-direction in counter-clockwise direction.
The positive rotating direction of A is seen from positive X - direction in counter-clockwise direction.
The reference point offsets are to be determined, so that in position A = C = 0 the tool system is axis parrallel to the basic coordinate system.



A. C	Angle
M	Center point
W	Tool loader dimension
L	Tool length
S	Tool peak

Figure 4-27 Axis configuration

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

Machine coordinate system (G47)

At the machine coordinate system the coordinates refer to the machine origin.
Thereby the machine axes are defined in cartesian coordinates. The axis movements orientate to the axis coordinates.

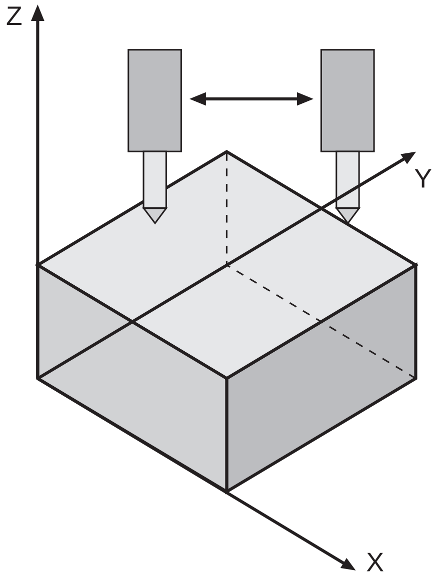


Figure 4-28 Axis movements in the machine coordinate system

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

Tool cordinate system (G48)

At the tool coordinate system the coordinates refer to the tool. The cartesian coordinate system is only used with a firm unit of length (e.g. mm).

If the tool system is arbitrarily shifted and rotated through the NC program, the system must fulfill the following conditions.

The coordinate source is the point, around which the tool is turned, if its orientation is changed. This point is named tool peak.

At rotating tools the Z-axis is the rotation-axis and indicates from the tool peak into the direction of the chuck.

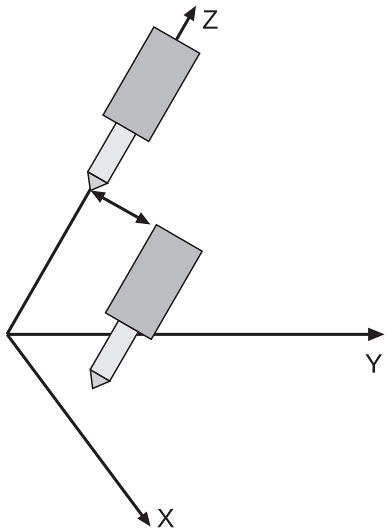


Figure 4-29 Axes movements in too coordinate system

The speed of the tool peak is interpreted as actual working feed.

When activating the tool dimensions the control shifts the tool coordinate system in the Z-direction for an amount corresponding to the tool length.

If the X -, Y - or Z- traverse direction is selected, the X -, Y - or Z - axis traverse according to the position of the rotating angle of the C-axis and the position of the tumbler angle of the A-axis, so that the tool stands always vertically on the X- Y- plane.

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

Workpiece coordinate system (G49)

At the workpiece coordinate system the coordinates refer to the workpiece.
Only the cartesian coordinate system with a firm unit of length (e.g. mm) is used. The advantage of the workpiece system is, that it can be shifted and turned arbitrarily with the NC program.

For each point a coordinate vector can be allocated in reference to the workpiece system, for example:

- If for the linear axis the X -, Y - or Z- traverse direction is selected, the X -, Y - or Z - movement is parallel to the axes coordinates (independently of the position of the rotating angle of the C-axis and the tumbler angle of the A-axis).

- If only axis C is traversed, X and Y are relocated so that the tool peak is always at the same point. The X - and Y - axis describe a circle around this point. The orbit proceeds vertically under the center point.

- If axis A is traversed, additionally the Z - axis is moved in plus - or minus direction according to the tumbler angle position of A.

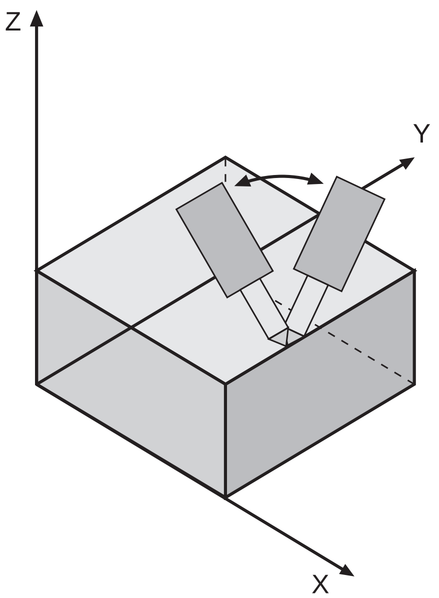


Figure 4-30 Axes movements in workpiece coordinate system

Appearing coordinate transformation when switching over between G47, G48, G49 determines the position and orientation of the tool relative to the workpiece reversable definitely.

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

Offset functions

Sometimes it is necessary, that at the construction of the machine the point of balance does not correspond to the center point M. In this case, a function for compensating this deviation is necessary.

The system offers three offset functions. By using this function, the mechanical offset is corrected automatically.

Offset in X - direction

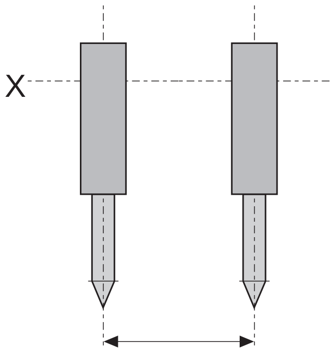


Figure 4-31 Offset in X - direction

Paramter P11802 is offset value [mm].

Offset in Y - direction

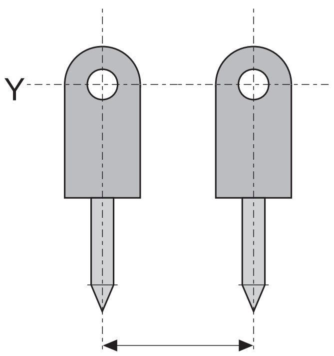


Figure 4-32 Offset in Y - direction

Paramter P11803 is offset value [mm].

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

Offset in Z - direction

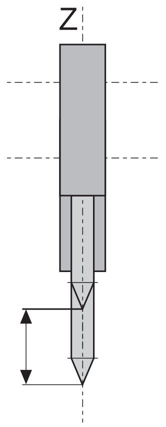


Figure 4-33 Offset in Z - direction

Paramter P11804 is offset value [mm].

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

Tool carrier with oblique angled axes

The configuration of the tool carrier with oblique angled axes (in zero position) is shown in figure 4.24. The A - axis turns around an axis, which does not stand vertically on the rotation axis Z. That means, that the tilt is not parallel to the X - axis. The paramter P11806 is the tilt angle.

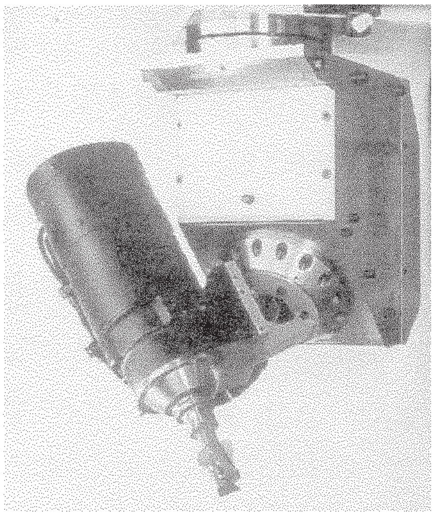


Figure 4-34 Tool carrier with oblique angled axes

Value areas and handling

The parameter values used are limited in the following areas.

For the parameters P11802, P11804 and P11806 positive or negative values are possible.
The amount of P11806 must be smaller than 180 degrees.
If P11806 is = 0, the presetting is 90 degrees, namely $\xi = 90$[degree]
If $\xi = 90$[degree], $\alpha - \alpha 12 = 0$, $\beta - \beta 12 = 0$ and $\theta = 0$

The amount (β) is dependent on the areas of ξ.

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

P8759	Coordinate mode in Manual	
	0 Transformation off	
	15 Polar transformation off	
	16 Polar transformation on	
	47 Robot transformaton off	
	48 Tool coordinate system	
	49 Workpiece coordinate system	
P11800	Robot linear axes (physical axis number)	
	e.g. the first 3 axes (X, Y, Z)	
	Byte 3,2,1 03 02 01 Hex	
	Byte 4 reverse (Bit 3, 2, 1)	
P11801	Robot rotation axes (physical axis number)	
	e.g. the 4th and 5th axis (A, C)	
	Byte 3, 2, 1 00 05 04 Hex	
	Byte 4 reverse (Bit 3, 2, 1)	
P11802	Offset X, rotation axis - tool axis	[mm]
P11803	Offset Y, rotation axis - tilt	[mm]
P11804	Offset Z, length of tool carrier	[mm]
P11805	Angle between tilt - tool axis	[degree]
P11806	Angle between rotation axis - tilt	[degree]
P11807	Angle where the tool shows downwards	[degree]

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

Zero points

The Robot system is given through a block of coordinates, which indicates its position in reference to the machine system. The zero point system G54 to G59 is analogously valid in the Robot system. The zero point indicates the position in reference to the machine system, i.e. the position of the tool is in reference to the workpiece. It can be filed in a zero point memory and is activated by calling up G54 to G59.

The functions G54 to G59 overwrite mutually. Besides they overwrite a shift effective through G92.

Zero point shift

The Robot system can be shifted and turned again in the NC program. The position of the shifted system relative to the old one is given through a block of coordinates. Calling up results through the zero point shift G92. This position of G92 is the reference to the actual zero point coordinates. In manual mode this zero point shift is not disposable.

The position of the tool in the machine system is given through 3 blocks of coordinates

- Zero point coordinates

- Zero point shift

- Coordinates of the position

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

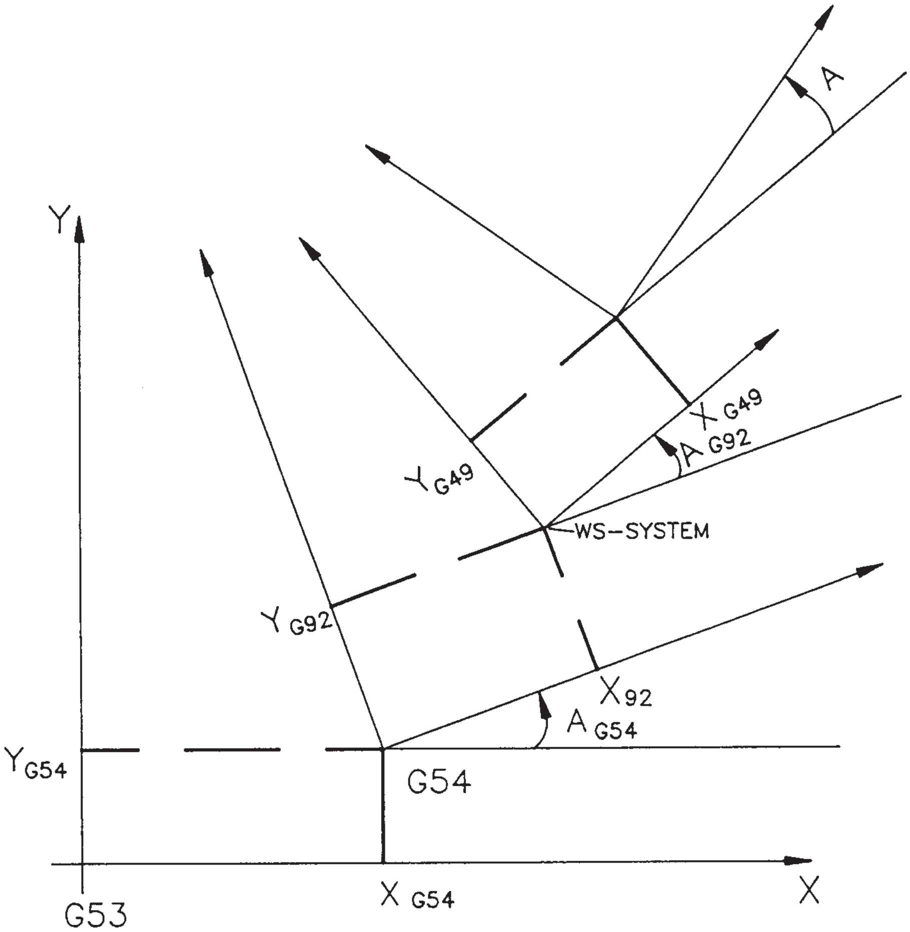


Figure 4-35 Zero point shift

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

Denomination of the 3 axes, e.g. tilt B, linear axes X and Z.

With 3-axes-tool machines, G49 is controlling the axes so that when moving the tilt B, the axes X and Z are always so that the tool peak is always at the same position.

Configuration and definition for the 3-axes-coordinate system

The 3-axes-coordinate system demands the following axis layout:

- X and Z axis are the basic coordinates.

- The B axis is rotating around an axis that is positioned vertically to the plane XZ.

- The positive rotating direction of B is seen from positive X direction counter-clockwise.

- The reference measures are termined so that the tool system is axis-parallell to the basic coordinates in position B=0.

- The offset between the tool peak and the rotating point is determined by the parameters P11802 (offset radius) and P11803 (offset angle).

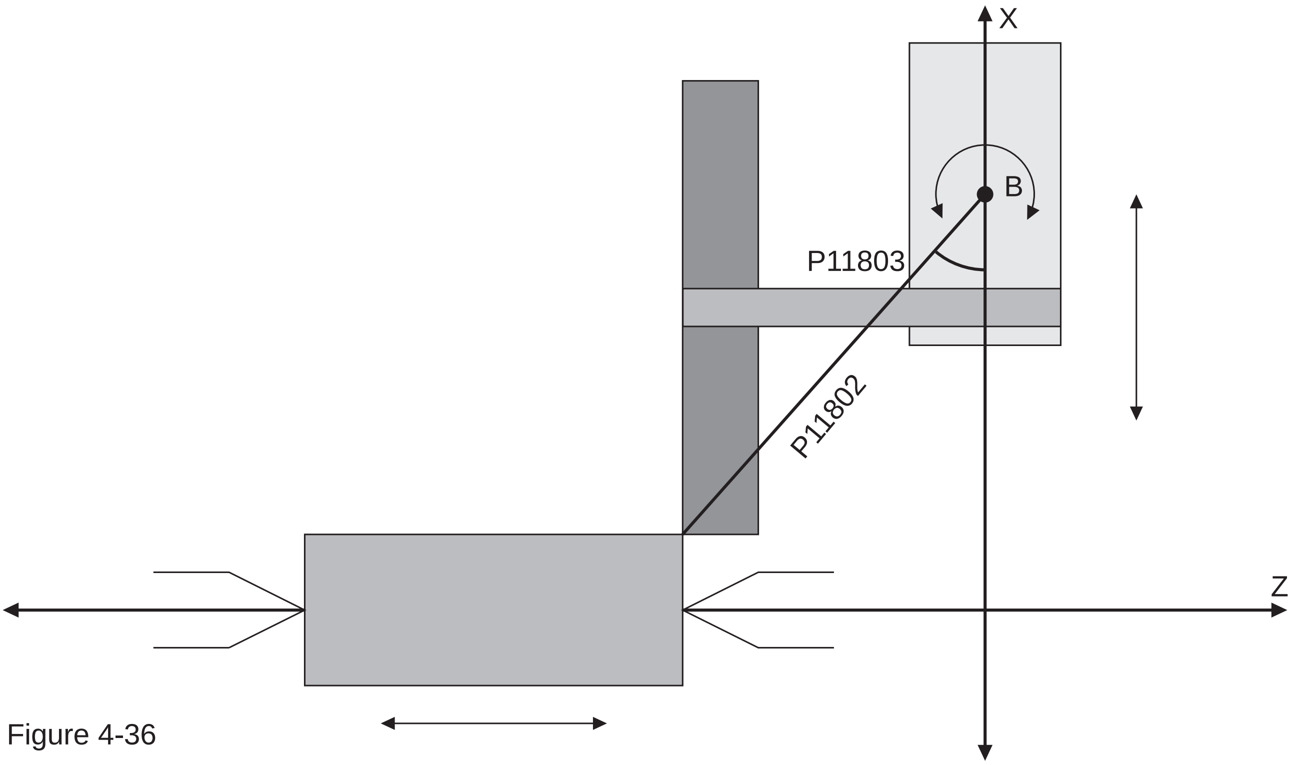


Figure 4-36

4.17 G47 / G48 / G49 / 147 Robot transformation (continued)

Used parameters

P11800 Linear axes(physical axis number)

e.g. the first 2 axes (X,Z)

Byte 3, 2, 1	00 02 01 Hex
Byte 4	reverse (Bit 2, 1)

P11801 Rotation axes (physical axis number)

e.g. the 3rd axis (B)

Byte 3, 2, 1	00 00 03 Hex
Byte 4	reverse (Bit 1)

P11802 Offset radius, tilt - tool peak [mm]

P11803 Offset angle, tilt - tool peak [degree]

4.18 G50 / G51 / G52 Spline interpolation

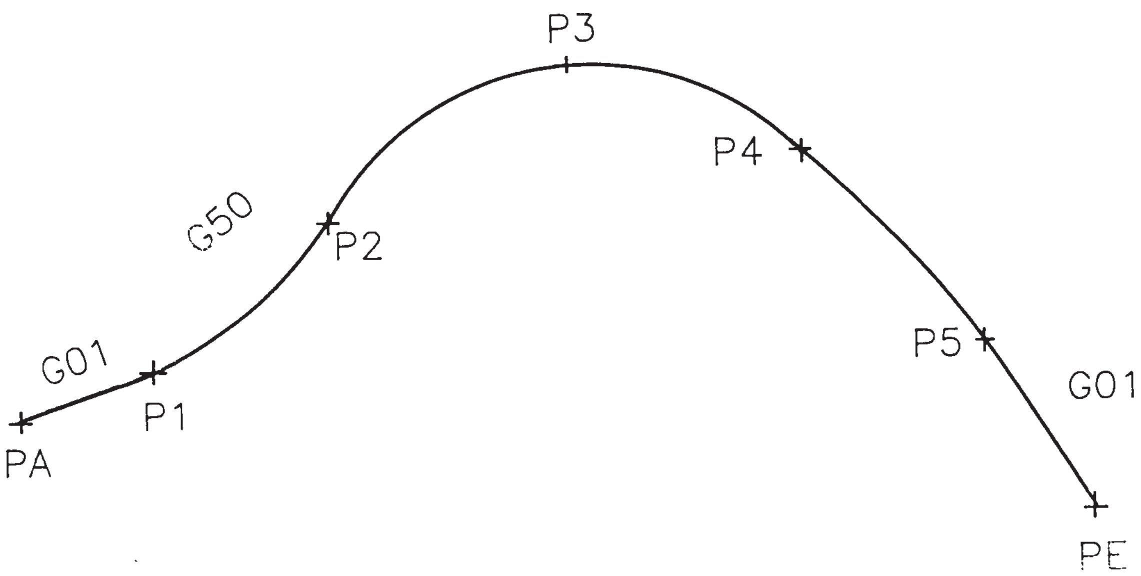
G50 **Spline interpolation**
G51 **Spline interpolation with feed adaptation at the contour**
G52 **Polynom interpolation**

G50 / G51 Spline interpolation

A sequence of points can be joined tangentially with a smooth curve trace with the spline interpolation. The points can be entered both as NC program as well as in the manual operation mode through „Teaching“ with command=target.

The spline curve is started through a traverse, which is run with G00 or G01. The start can also be with G02 or G03, if the starting position is in a main plane (G17, G18, G19).

The spline interpolation is activated in the subsequent block through G50 / G51. There results a tangential transition, whereby the starting tangent of the spline curve is determined through the starting segment.



PA	Starting point
PE	End point
P1...P5	Spline points

Figure 4-37

4.18 G50 / G51 / G52 Spline interpolation (continued)

In the corresponding way the spline curve is finished through programming the @@Ausleitungs-segment with G00, G01, G02 or G03.
The traverse conditions G00, G01, G02, G03 and G50 overwrite themselves mutually.

If the starting segment or the finishing segment are not programmed, the NC program stands still, because the spline interpolation can not be started or finished duly. In this case, the key ‘Manaul’ has to be pressed and the error has to be cleared.

G52 Polynom interpolation

The path a is described with the formula $\vec{P} = \vec{V_3}\, t^3 + \vec{V_2}\, t^2 + \vec{V_1}\, t^1 + \vec{V_0}$,

whereby the variable t can have a value between 0 and 1.

The vectors V_3, V_2, V_1 are programmed in components in the following form:

X : [V_{3X}, V_{2X}, V_{1X}] Y : [V_{3Y}, V_{2Y}, V_{1Y}] Z : [V_{3Z}, V_{2Z}, V_{1Z}]

V_0 is not programmed, since it is the location of the machine at the beginning of the block.

Starting point $\vec{P_S} = \vec{V_0}$

Final point $\vec{P_E} = \vec{V_3} + \vec{V_2} + \vec{V_1} + \vec{V_0}$

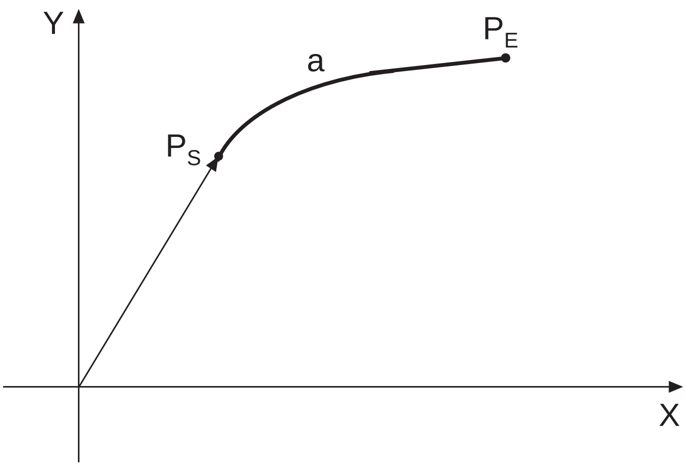
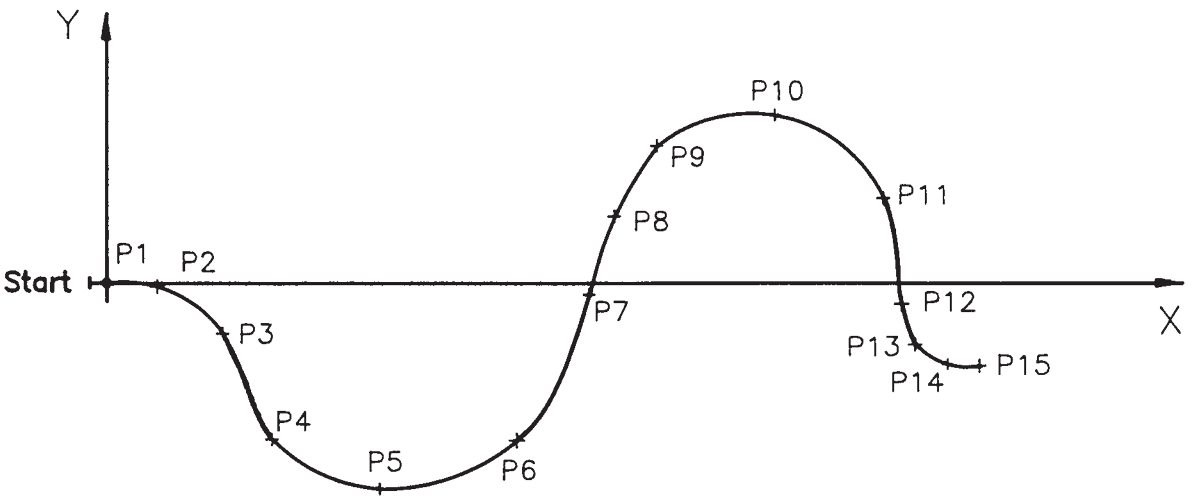


Figure 4-38

4.18 G50 / G51 / G52 Programming examples

Spline contour

N5	G54	G49	G00	F2000	X - 8	Y0	Z5		Starting point
N10	G01				X0	Y0	Z0	point 1	
N20	G50				X6,8	Y-0,5		point 2	Spline on
N30					X15,5	Y-6,7		point 3	
N40					X22	Y-20,6		point 4	
N50					X36,3	Y-27,1		point 5	
N60					X54,3	Y-20,7		point 6	
N70					X63,8	Y-1,5		point 7	
N80					X67,124	Y8,749		point 8	
N90					X72,7	Y18		point 9	
N100					X88,1	Y22		point 10	
N110					X102,5	Y11,1		point 11	
N120					X104,9	Y-2,8		point 12	
N130					X106,7	Y-8,1		point 13	
N140					X111	Y-10,7		point 14	
N150	G01				X115,2	Y-10,9		point 15	Spline off
N160	M30						Z5		



Point 1	Zero point
Point 2	Switching on spline interpolation
Points 3 - 14	Spline points
Point 15	Switching off of spline interpolation

Figure 4-39

4.19 G53 Machine zero point

With the input of G53 with G54 to G59 as well as with G92 programmed zero point shifts become ineffective. The program refers then to the machine zero point.

The function is effective modal and can be overwritten by G54 to G59.
With the program start G53 is automatically adjusted.

G153 switches the zero shift off G53 as well as G54 to G59.

4.20 G 54 to G59 Zero points

With G54 to G59 points of zero can be called within the program.

The functions are effective modal and overwrite themselves opposite-acting. Additionally one is reset by G92 programmed shift.

The points of zero are called in the program with G54 to G59 in the null point memory. The size of the zero shift is determined by the stored values in the null point table.

Example: programming zero points

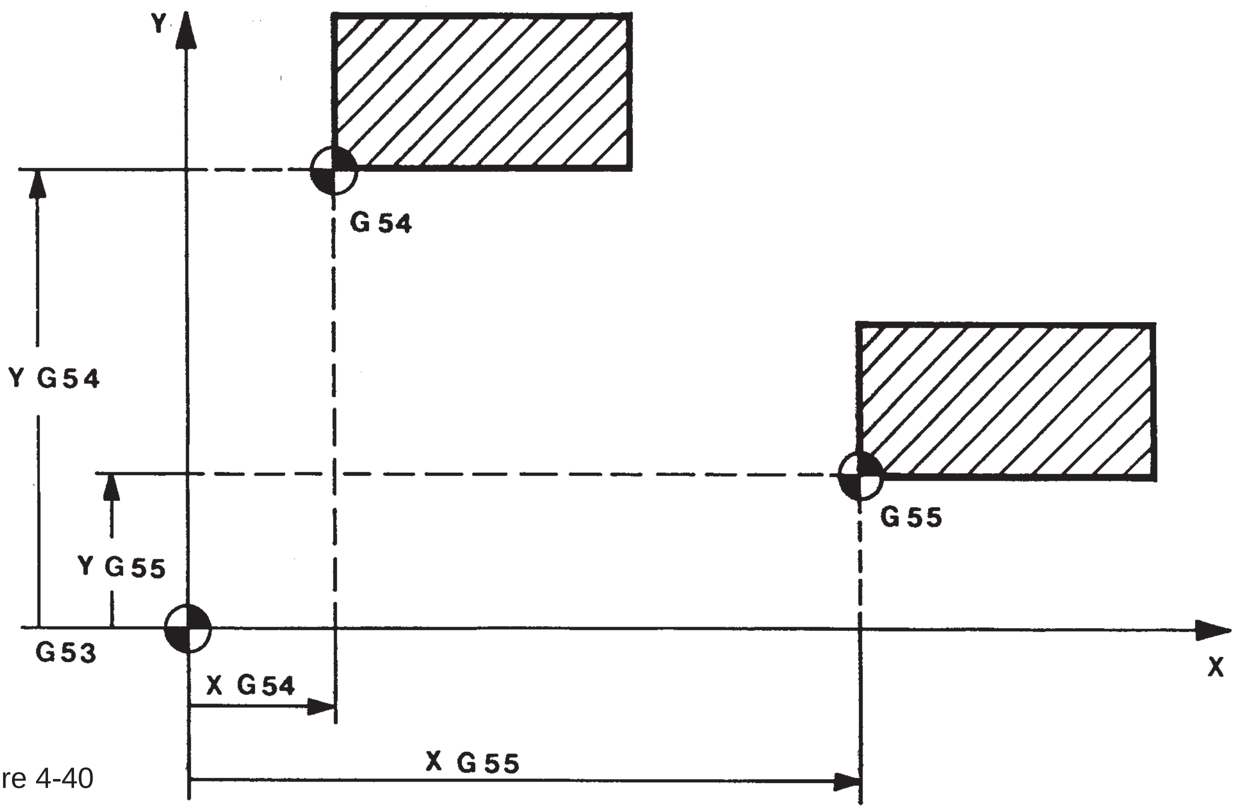


Figure 4-40

4.21 G60 / G61 / G62 Mirror imaging of coordinates

The mirror imaging function inverts the sign of programmed coordinates.
The G61 function inverts the sign of the 1st main axis program values.
The G62 function inverts the sign of the 2nd main axis program values.

Set plane	G61	G62
G17 (XY)	X becomes -X	Y becomes -Y
G18 (ZX)	Z becomes -Z	X becomes -X
G19 (YZ)	Y becomes -Y	Z becomes -Z

G60 cancels the mirror imaging functions G61 and G62 out. G60, G61 and G62 act modally.

Geometric definition

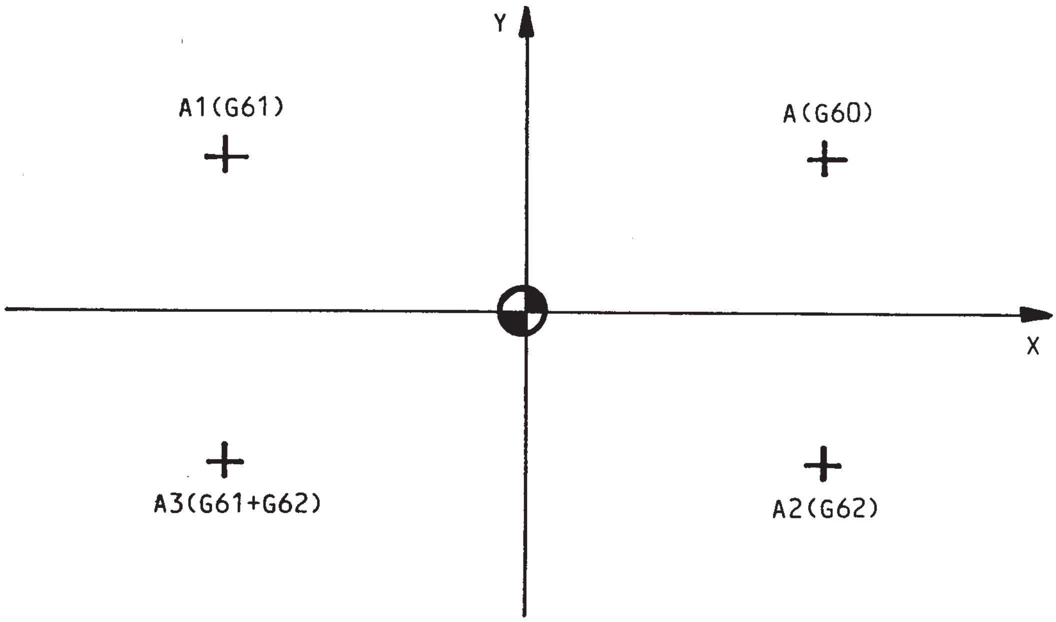


Figure 4-33

The programmed coordinates for point A are Xa and Ya.

When G61 is switched on point A becomes point A1 with -Xa and Ya.
When G62 is switched on point A becomes point A2 with Xa and -Ya.
When G61 and G62 are switched on point A becomes point A3 with -Xa and -Ya.

4.21 G60 / G61 / G62 Mirror imaging of coordinates (continued)

Circle centre point coordinates I, J, K

The circle centre point coordinates are mirror imaged according to the plane and functions set.

Zero points G53 to G59

Zero points G53 to G59 are not mirror-imaged.

Shift G92

The values of shift G92 are mirror imaged according to the appropriate plane and functions set.

Circular interpolation G02 and G03

When circular interpolation is switched on and G61 and G62 are active, G02 becomes G03 and G03 becomes G02. The direction of rotation remains unchanged when G61 and G62 are switched on.

Tool correction offset G41 and G42

When the tool correction offset is switched on and G61 and G62 are active, G41 becomes G42 and G42 becomes G41. The selected correction offset remains unchanged when G61 and G62 are switched on.

Effect of G66

G66 effects the block by block switching off of the mirror-imaging functions G61 and G62.

Incremental dimension programming G91

Using G91, the target point in the system that has not been mirror-imaged is firstly determined and is then mirror-imaged according to the appropriate plane and mirror-imaging functions set.

4.22 G63 / G64 „Feed rate 100%“

G63 Override 100% switch on

With G63 the Override value is set firmly to 100%, i.e. the override is not active any longer. NC program processing runs with the programmed feed.

The function is modal effectively and can by G64 again be switched off.

G64 Override 100% switch off

G64 switches G63 out

The function is modal effectively and can by G63 be overwritten.

With the program start G64 is preset.

4.23 G66 Switching off all correction offsets

When programming G66, the machine zero point is taken as the reference point. All dimension inputs then refer to the machine zero point.

The G66 function is effective only in the block in which it was written.

In this particular block the zero points set by G54 to G59 and shift G92 are inoperative, as are any tool length and tool radius correction offsets.
They remain stored, however, and are effective in the next block again.

4.24 G90 Absolute dimension input

When G90 is being programmed absolute dimension input is chosen, i.e. all dimensions are referred to the program zero point (programmable using G54 to G59 and G92).

G90 operates modally and can be overwritten by G91. G90 is automatically set when the control system is switched on.

Example: Absolute dimension input

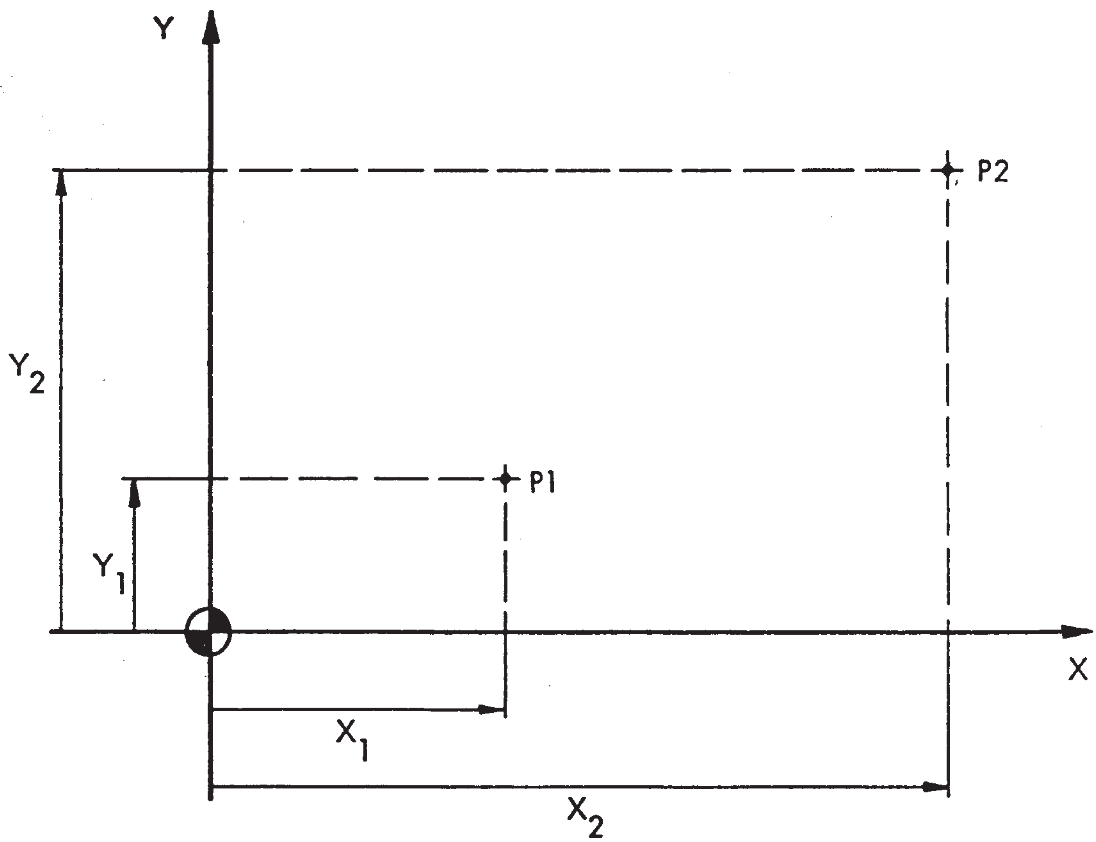


Figure 4-34

4.25 G91 Incremental dimension input

With the programming of G91 the chain measure input is selected. It always refers to the location of the axes started last.

G91 is modal effectively and can by G90 be overwritten.

Example: Incremental dimension input

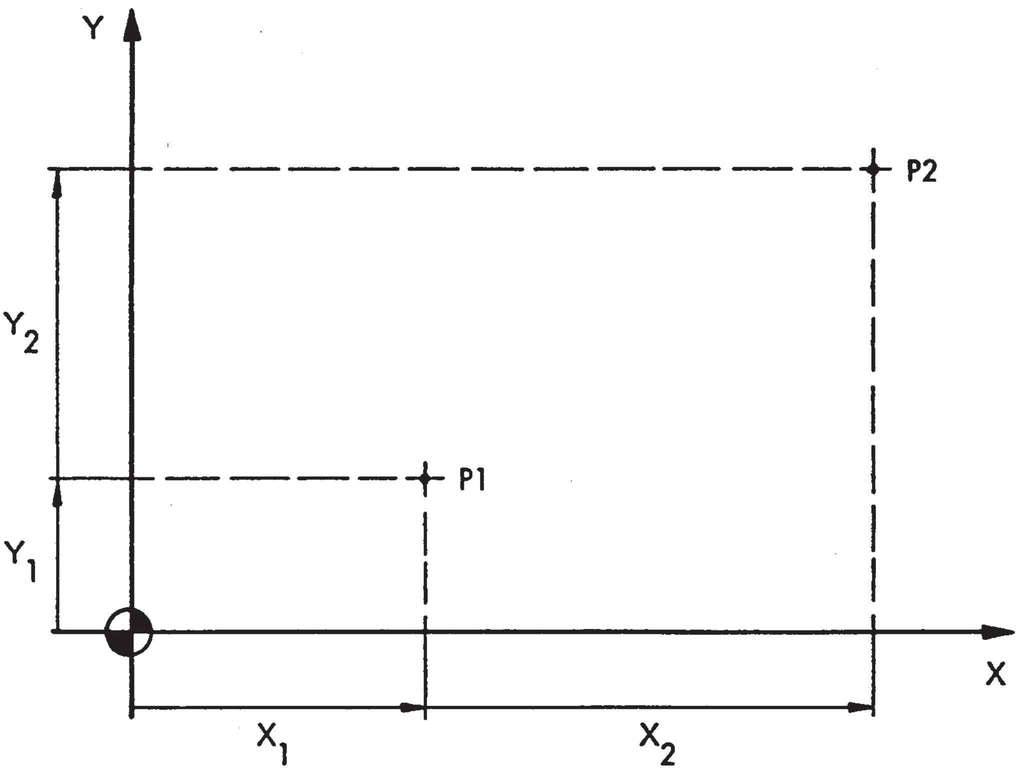


Figure 4-35

4.26 G92 Zero point shift

Zero point shifs programmed by G92 are a fixed part of the NC program and therefore independent of the clamping of the workpiece. In contrast, zero points are programmed outside the NC program by G54 to G59 and activated in the NC program.

Traverse information in the block with G92 determines the zero point shift, but no traverse is activated.

Input: G92 X... Y... Z...

The zero point is shifted by the programmed amounts.

The zero point shift programmed by G92 operates modally and is always referred absolutely to the machine zero point (G53) or to a set zero point (G54 to G59).
G92 can be cleared by G53 to G59 or overwritten by a new G92.

Example: Zero point shift

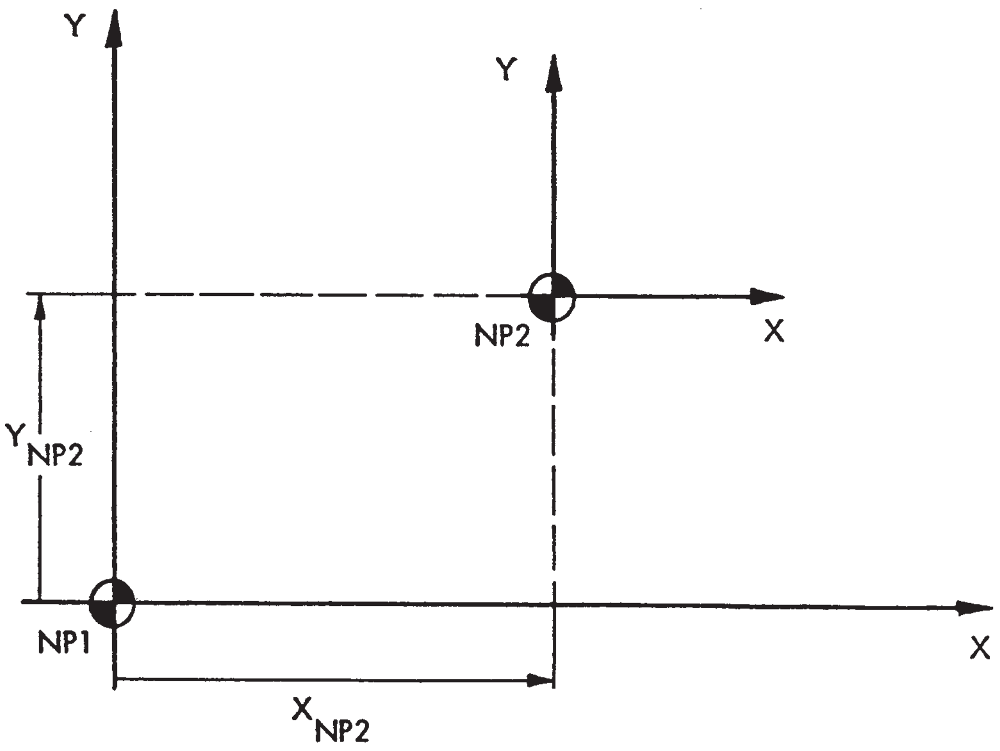


Figure 4-36

4.27 G94 / G95 Feed modification

The functions determine the feed modifications:

G94 Feed in mm/min

G95 Feed in mm/r (revolution)

4.28 G96 / G97 Number of revolution modification

The functions define the speed modifications:

G96 **constant cutting velocity in mm/min**

G97 **speed in U/min**

In order to activate G96 / G97, the following parameters must be set:

P11640 spindle affects axis (physical axis)
(example round axis C corresponds the 6. Axis Input P11640:6)

P11641 datum axis for G96 (physical axis)
(example procedure axis Y corresponds the 2. Axis Input P11641:2)
If P11641 = 0 applies, then the reference position in P11642.

P11642 reference position for G96 [mm]
(e.g.: disk diameters)

P11643 reference factor for G96 [mm/min]
0 or — mm/min
1000 m/min

P11644 max. speed for G96

P11645 max. speed for G97

P11646 reference factor for G97 [U/min]
0 or — U/min
1 degree/min

P11647 speed definition G96 / G97
Presetting 0 = G97
96 = G96

4.28 G96 / G97 Number of revolution modification (continuation)

In the display selection menu the inputs under „ spindle parameters „ can be input.
Spindle axis: 6
Datum axis: 2

Example NC Progr.:
N10 G0 Y:50
N20 G96 G1 F50 S100 Y:0
The speed of the round axis increases the more,
the more near processing toward the position Y:0 comes.

N10 G0 Y:0
N20 G96 G1 F50 S100 Y:50
The speed of the round axis decreases itself the more,
the more near processing toward the position Y:50 comes.

4.29 G99 End feed, traverse dependant feed adaptation

The starting feed is the value programmed under ‘F’, the end feed is written in P608.
This parameter has to be programmed at the latest in the block in which G99 is to be found.