MT101

SINUMERIK HANDBOOK

Programmable Frames **TRANS, ROT, MIRROR & SCALE**

This document shows working with frames and ready examples to utilize on Milling application.

SIEMENS

Higher productivity with SINUMERIK

Whether it’s a basic part or a complex workpiece, the Siemens control system offers you a wealth of functionality with a very high degree of operator friendliness and manufacturing efficiency.

Description of the document:

The effective programming techniques which can be reduced with greater flexibility with frames

- TRANS, ROT, SCALE, MIRROR — Overview
- TRANS, ATRANS — Swift the work offset
- ROT, AROT — Rotation of the coordinate systems
- SCALE, ASCALE — Scale the feature dimensions
- MIRROR, AMIRROR — Mirror the features

Applicability:

- Technology — 3 axis Milling & Milling 3+2 application
- SINUMERIK controllers — 802Dsl, 810D, 840D powerline, 808D, 828D, 840Dsl & SINUMERIK ONE

Index

Programmable zero offset - **TRANS & ATRANS**

Benefit of TRANS / ATRANS:

- **Easily swift the reference point of part in machining**
- **Reduce the programming efforts which need to be milled repeatedly**
- **Reduce the time to create many sub-programs**
- **Handle Cam programs effectively by shifting WCS**

Function:

TRANS/ATRANS can be used in program to swift the reference point of work / zero offset for path and positioning axes in the direction of the specified axis. This allows to work with different zero points.

Syntax

TRANS X... Y... Z...
(Programming absolute coordinate system)

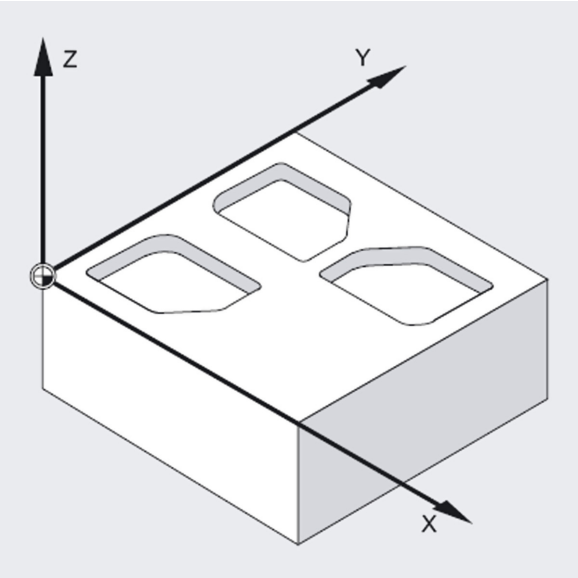
ATRANS X... Y... Z...
(Programming offset additive / incremental coordinate system)

Meaning

TRANS ; zero point offset absolute, referred to the currently
; valid work piece zero point selected with G54 to G599

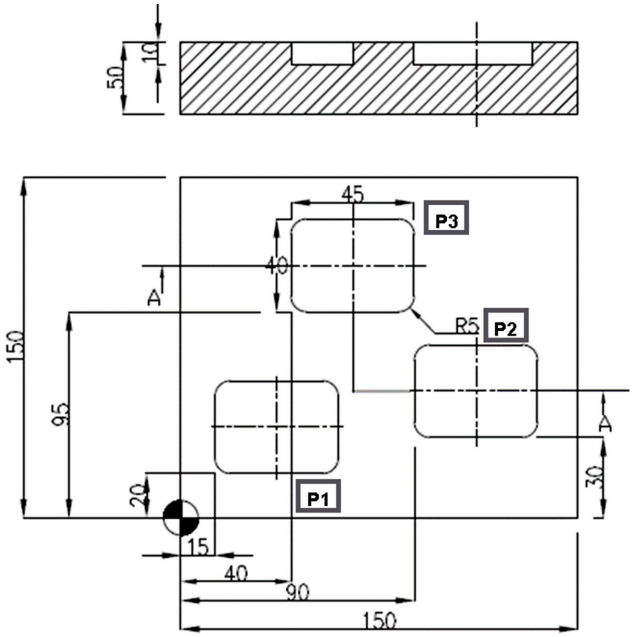
ATRANS ; as TRANS, but with additive / incremental offset

X Y Z ; Shifting value in direction of the stated geometrical axis



Programming example:

Refer the below drawing where three rectangular pockets need to be milled at various positions.



Create a Main program in NC directory.

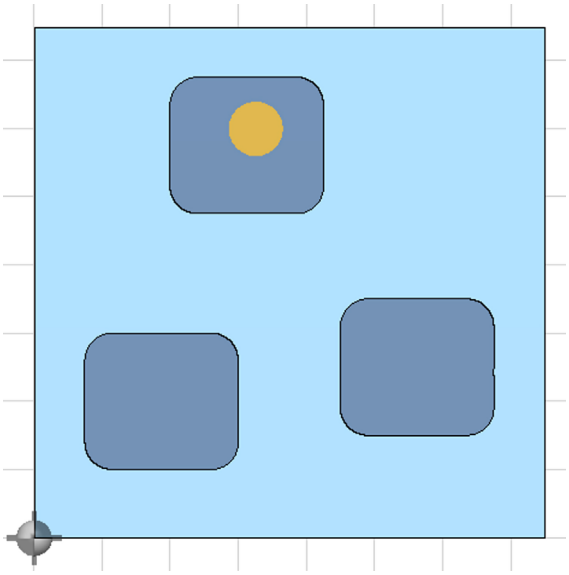
Program		Comments
N10 G90 G40 G17 G71 G94	;	Program header
N20 T=”CUTTER 16”		
N30 M6 D1		
N40 S4500 M3		
N50 G0 G54 X0 Y0 Z10		
N60 Z1		
N70 G1 Z0 F100	;	Approach starting point
N80 TRANS X15 Y20	;	Absolute offset for first pocket P1
N90 LL10 P10	;	Subprogram call for machining
N100 G0 Z10		
N110 TRANS X90 Y30	;	Absolute offset for 2nd pocket P2
N120 G01 Z0		
N130 LL10 P10	;	Subprogram call & Looping 10 times
N140 G0 Z10		
N150 ATRANS X-50 Y65	;	Incremental offset reference to previous TRANS for 3rd pocket P3
N160 G01 Z0		
N170 LL10 P10	;	Subprogram call
N180 TRANS	;	Cancellation of programmable offset
N190 G0 Z100		
N200 M30	;	End of program

Sub program call for looping purpose:

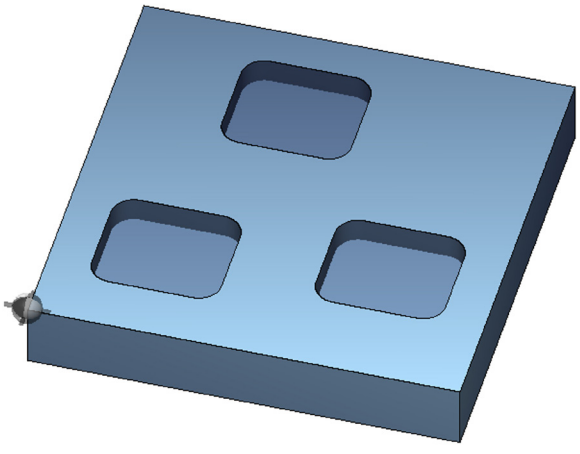
Create a subprogram under subprogram directory in NC and define the name as LL10.SPF

Program		**Comments**
N1 G0 X22.5 Y20	;	Center point of pocket
N2 G1 Z=IC(-1) F100	;	Incremental depth similar to G91
N3 G41 X45 F500		
N4 Y35		
N5 G3 X40 Y40 CR=5	;	CR is define the radius
N6 G1 X5		
N7 G3 X0 Y35 CR=5		
N8 G1 Y0 RND=5	;	RND is define the radius without G2/G3
N9 X45 RND=5	;	Start point of radius is not required when following in path
N10 Y25		
N11 X35		
N12 G0 G40 X25		
N13 M17	;	End of sub program

Simulation:



2D Simulation



3D Simulation

Programmable rotation - **ROT & AROT**

Benefits:

- **Effective programming concepts for 2 ½D machining**
- **Minimize the CAM requirement for 2 ½D machining**
- **Utilize 2D programming for orientation of feature**
- **Minimize the calculation & increase machine availability**
- **No additional resource is not essential like CAM**

Function:

ROT/AROT the workpiece coordinate system can be rotated about any of the three geometrical axes X, Y, Z or by an angle RPL in the selected working plane G17 to G19 (respectively about the vertical infeed axis). This permits the machining of oblique faces or several work piece faces in single setting.

Syntax

ROT X... Y... Z...	; Replacing instruction for rotation in space
ROT RPL=...	; Replacing instruction for rotation in the plane
AROTX... Y... Z...	; Additive instruction for rotation in space
AROT RPL=...	; Additive instruction for rotation in the plane

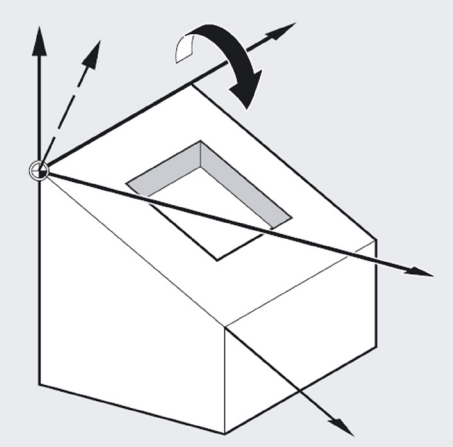
All instructions must be programmed in their own NC-block.

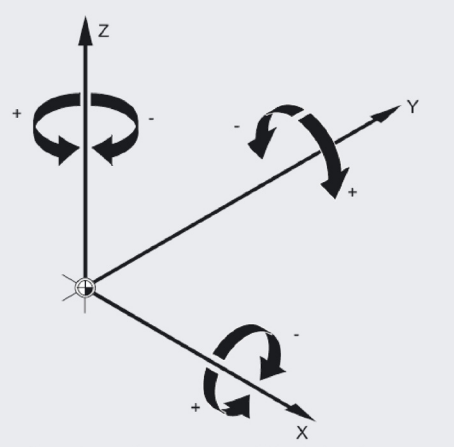
Meaning

ROT	Rotation absolute, as referred to the currently valid work piece zero point selected with G54 to G599.
RPL	Rotation in the plane: Angle by which the co-ordinate system is rotated (plane selected with G17-G19). The sequence in which the rotation is to be carried out can be specified by means of a machine datum. In the standard version the RPY-notation (= Roll, Pitch, Yaw) with Z,Y,

Direction of rotation:

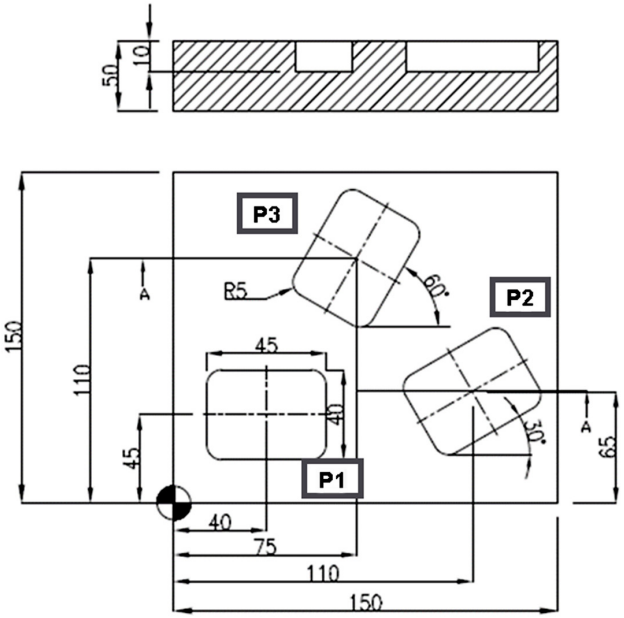
The positive direction of rotation: Clockwise when looking in the positive direction of the coordinate.





Programming example:

Refer the below drawing where three rectangular pockets need to be milled at various orientations.



Programming example for above drawing

```
N10 G90 G40 G17 G71 G94          ;  Program header
N20 T="CUTTER 16"
N30 M6 D1
N40 S4500 M3
N50 G0 G54  X0 Y0
N60 WORKPIECE(,"",,"BOX",112,-1,-25,-80,0,0,150,200)
N70 Z10                          ;  Approach starting point
N80 TRANS X40 Y45                ;  Absolute offset from work offset for P1
N90 G1 Z0 F100
N100 LL10 P10                    ;  Subprogram call & looping for 10 times
N110 G0 Z10
N120 TRANS X110 Y65              ;  Absolute offset from work offset for P2
N130 AROT Z30                    ;  Rotation of the coordinate 30°
N140 G1 Z0 F100
N150 LL10 P10                    ;  Subprogram call
N160 G0 Z10
N170 TRANS X75 Y110              ;  Absolute offset from work offset for P3
N180 AROT RPL=60                 ;  Additive rotation through 60° + TRANS
N190 G1 Z0 F100
N200 LL10 P10                    ;  Subprogram call
N210 TRANS                       ;  Cancel the TRANS
N220 ROT                         ;  Cancel the ROT
N230 M30                         ;  End of program
```

Explanation of block N130:

N130 AROT Z30

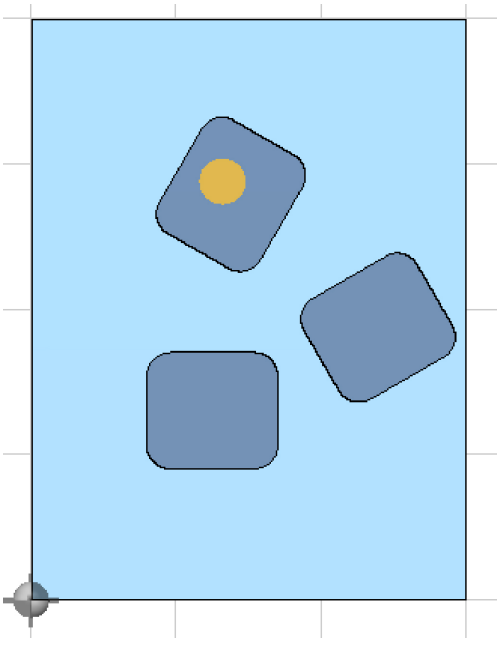
- ROT & AROT - Functionality remains same to rotate the coordinate system.
- When AROT is used with other frames commands then AROT consider the previous transformation like swift of WCS in above case.
- Z30 - Rotate coordinate system by keeping axis constant and rotate the remaining axes XY axes.

Sub program call for looping purpose:

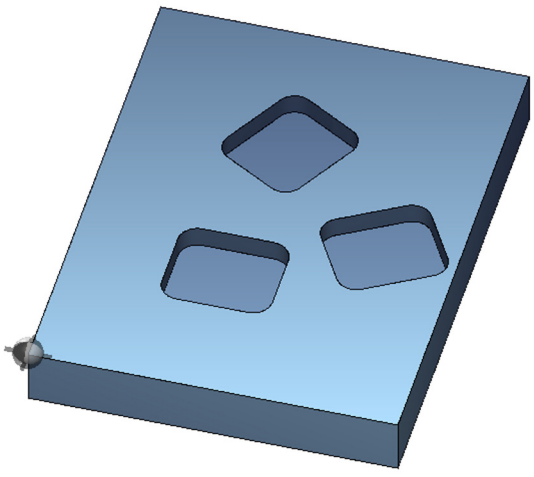
Create a subprogram under subprogram directory in NC and define the name as LL10.SPF

```
N1 G0 X22.5 Y20                ; Centre point of pocket
N2 G1 Z=IC(-1) F100            ; Incremental depth similar to G91
N3 G41 X45 F500
N4 Y35
N5 G3 X40 Y40 CR=5             ; CR is define the radius
N6 G1 X5
N7 G3 X0 Y35 CR=5
N8 G1 Y0 RND=5                 ; RND is define the radius without G2/G3
N9 X45 RND=5                   ; Start point of radius is not required when following in path
N10 Y25
N11 X35
N12 G0 G40 X25
N13 M17                        ; End of sub program
```

Simulation:



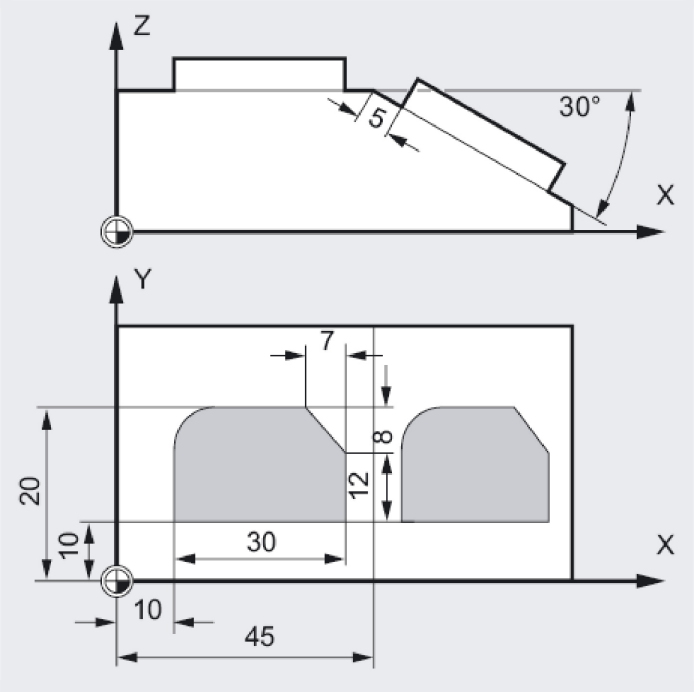
2D Simulation



3D Simulation

Example: Rotation in space

In this example axis-parallel and oblique work piece faces are to be machined in one setting. Pre-requisite: the tool must always be aligned perpendicular to the rotated Z-direction.



Program		Comment
N10 G17 G54	;	Working plane X/Y, work piece zero point
N20 TRANS X10 Y10	;	Absolute offset
N30 L10	;	Sub-program call-up
N40 ATRANS X35	;	Additive offset
N50 AROT Y30	;	Rotation about the Y-axis
N60 ATRANS X5	;	Additive offset
N70 L10	;	Sub-program call-up
N80 G0 X300 Y100	;	Retraction, end-of-program
N90 M30		

Programmable rotation - **MIRROR & AMIRROR**

Benefits:

- **Easily replicate the milling geometries / profiles**
- **Minimize programming efforts**
- **Reduce the time & handle the program effectively**
- **Avoid the calculation & number of programs**

Function:

With MIRROR/AMIRROR it is possible to mirror work piece shapes about coordinate system. MIRROR only change axis sign from positive to negative and vice-versa.

Syntax

MIRROR X0 Y0 Z0 ; (Programming the replacing instruction in a separate NC-block)

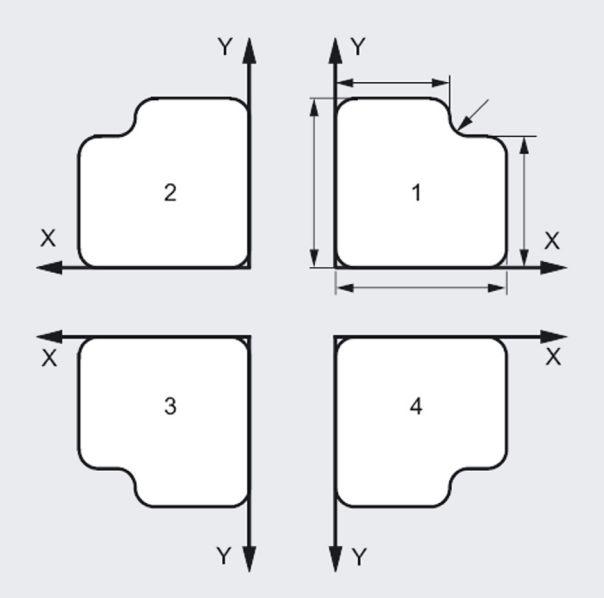
AMIRROR X0 Y0 Z0 ; (Programming the additive instruction in a separate NC-block)

Meaning

MIRROR	;	Mirroring absolute, referred to the currently valid co-ordinate system selected with G54 to G599
AMIRROR	;	Mirroring additive, Takes previous MIRROR as a reference works like incremental MIRROR
X Y Z	;	Geometrical axis, the direction of which is to be changed. The value entered here can be selected at will, e.g. X0 Y0 Z0.

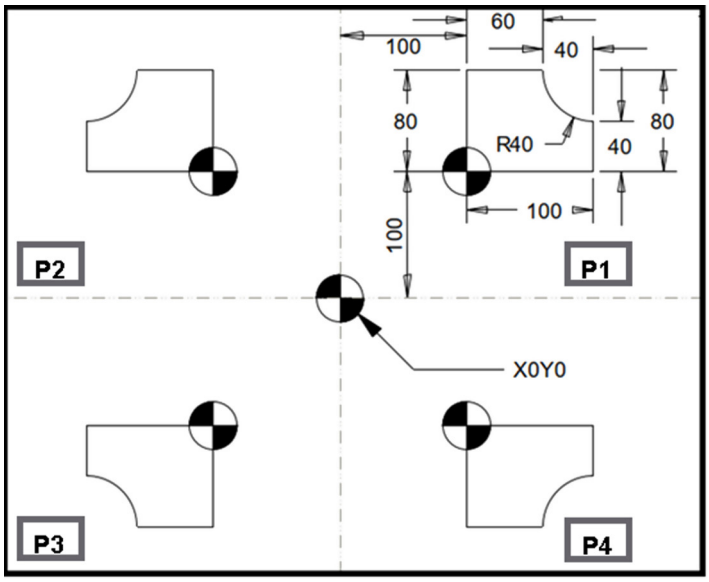
Prerequisites:

- Work offset G54 must be centre of part



Programming example:

Refer the below drawing where three profile spigot need to be milled at various coordinate systems.



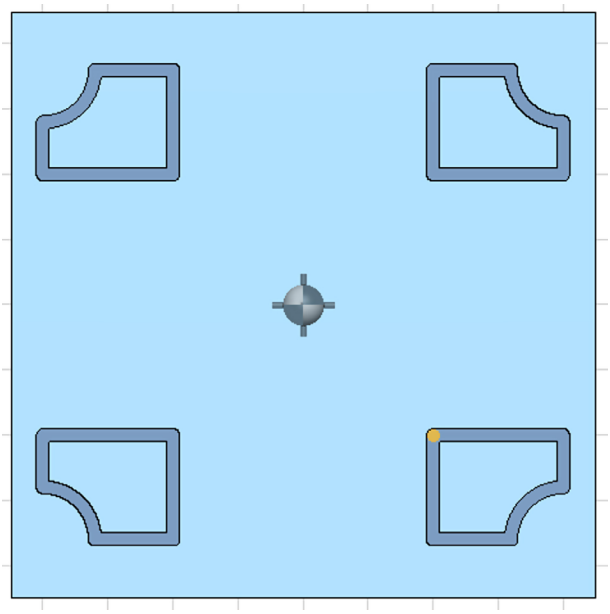
Program		Comment
N10 G90 G40 G17 G71 G94	;	Program header
N20 T="ENDMILL_D10"		
N30 M6 D1		
N40 S4500 M3		
N50 G0 G54 X0 Y0		
N20 LM10	;	Machine first spigot, top right P1
N30 MIRROR X0	;	Mirror X axis (direction is changed in X)
N40 LM10	;	Machine second spigot, top left P2
N50 AMIRROR Y0	;	Mirror Y axis (direction is changed in Y)
N60 LM10	;	Machine third spigot, bottom left P3
N70 MIRROR Y0	;	Mirror Y axis consider from parent P1
	;	(Direction is changed in Y)
N80 LM10	;	Machine fourth spigot, bottom right P4
N90 MIRROR	;	Deactivate mirroring
N100 G0 Z100		
N110 M30	;	End of program

Sub program call for looping purpose:

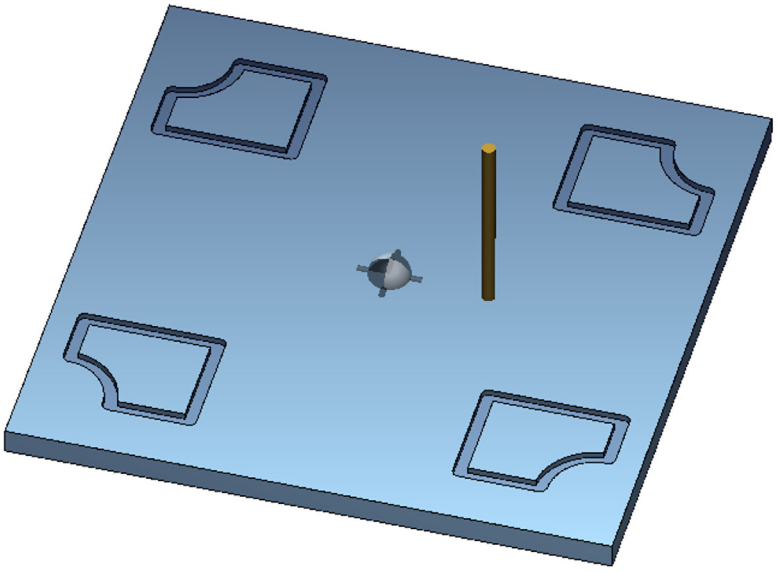
Create a subprogram under subprogram directory in NC and define the name as LM10.SPF

G0 Z1
G1 G40 X100 Y100 F1000
Z-5
Y180
X160
G3 X200 Y140 CR=40
G1 Y100
X100
Z5
G0 G40 Z10
M17

Simulation: Verify the main program using simulation



2D Simulation



3D Simulation

Programmable rotation - **SCALE & ASCALE**

Benefit of TRANS / ATRANS:

- **Simply change the ratio of profile with SCALING**
- **CAM / 2D program can be managed with SCALING**
- **Minimal efforts to manage identical features/profiles**
- **Reduce the time while generating repetitive new programs**

Function:

SCALE/ASCALE can be used to program scale factors for all continuous path, synchronous and positioning axes in the direction of the stated axis. In this manner the size of an element can be changed.

Switching-OFF scale factor

For all axes: SCALE (without axis denomination). All frame components of the previously programmed frames will be reset.

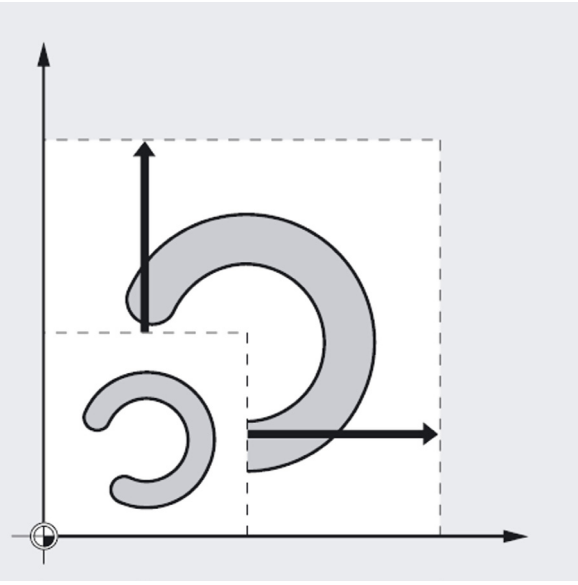
Syntax

SCALE X... Y... Z... ; (Programming the replacing instruction in a separate block)

ASCALE X... Y... Z... ; (Programming the additive instruction in a separate block)

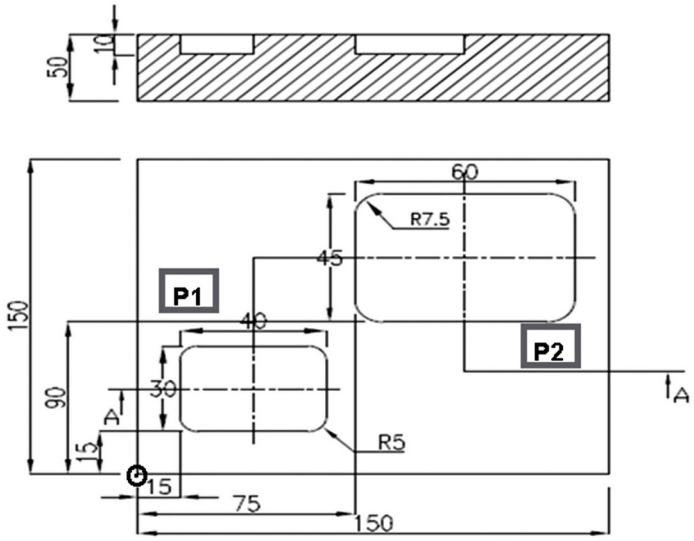
Meaning

SCALE	;	Enlarging/reducing absolute, as referred to the currently valid co-ordinate system selected with G54 to G599
ASCALE	;	Enlarging/reducing additive, as referred to the currently valid, co-ordinate system, either selected or programmed
X Y Z	;	scale factor in direction of the stated geometrical axis
X1	;	X axis run at 1:1 ratio nothing but 100%
Y1.5	;	Y axis run at 1: 1½ means 150%
X0.75 Y1.25	;	X axis run at reducing the scale to 3/4 & Y at 125%



Programming example :

Refer the below drawing where rectangular pocket need to be milled at different scale factor.



Program		**Comment**
N10 G90 G40 G17 G71 G94	;	Program header
N20 T=”CUTTER 16”		
N30 M6 D1		
N40 S4500 M3		
N50 G0 G54 X0 Y0		
N60 WORKPIECE(,””,,”BOX”,112,0,-25,-80,0,0,150,150)		
N70 Z2	;	Approach starting point
N80 TRANS X15 Y15	;	Absolute offset
N90 LS10	;	Subprogram call
N100 TRANS X75 Y90	;	Absolute offset
N110 ASCALE X1.5 Y1.5	;	Increasing pocket size
N120 LS10	;	Subprogram call
N130 TRANS	;	Deactive TRANS
N140 SCALE	;	Deactive SCALE
N150 M30	;	End of program

Explanation

N80 TRANS X15 Y15

G54 is bottom left corner and shifting the reference of Pocket P1 to its left bottom corner
Followed by Sub program to run the program

N100 TRANS X75 Y90

Similarly like N80 for P2

N110 ASCALE X1.5 Y1.5

Increase pocket the size in factors
ASCALE is used to consider TRANS

Sub program call for looping purpose:

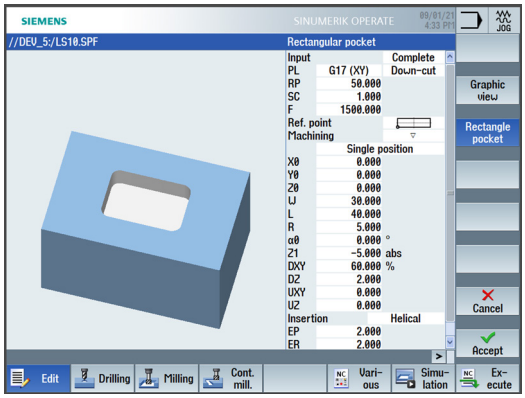
Create a subprogram under subprogram directory in NC and define the name as LS10.SPF

Parameter of rectangular pocket cycle:

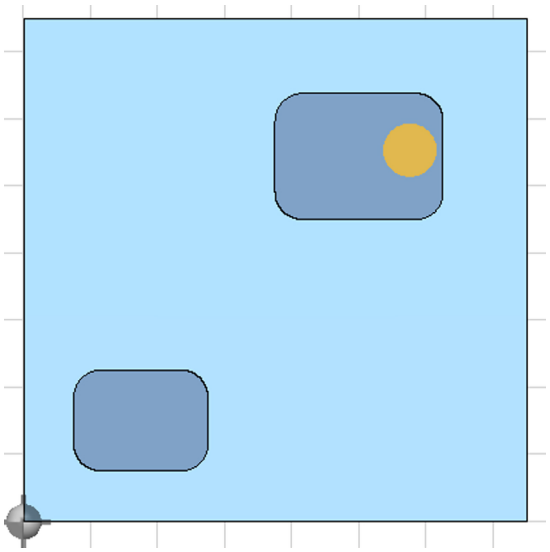
; Added rectangular pocket cycle

POCKET3(50,0,1,-5,40,30,5,0,0,0,2,0,0,1500,0.1,0,21,60,8,3,15,2,2,0,1,2,12100,11,110)

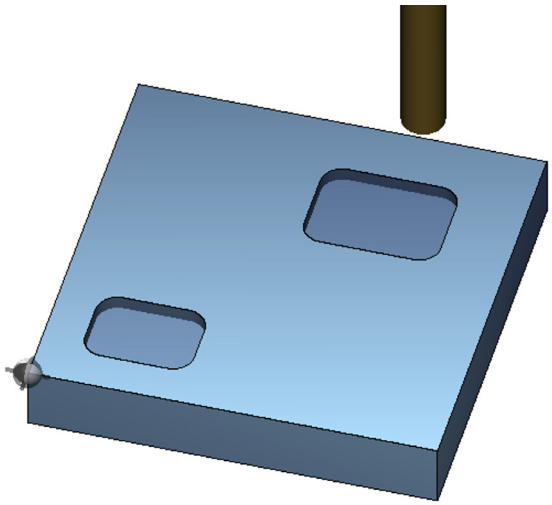
M17



Simulation: Verify the main program using simulation



2D Simulation



3D Simulation

To know more about SINUMERIK

Enrol for Application & Commissioning trainings

: www.siemens.co.in/dex
☎: 080-40975628
✉: tac.training.in@siemens.com

▶ Watch SINUMERIK Video Tutorials Online

▶ SINUTRAIN—Free Download

Published by
Siemens Limited
Digital Industries
Motion Control

XX-XXX-XXX

For online application support

Prakash Somannavar
☎: +91-8980023525
✉: prakash.somannavar@siemens.com

Syed Imtiyaz
☎: +91-9945293532
✉: syed.imtiyaz@siemens.com

For Service Support

:18002090987
✉: ics.india@siemens.com
: www.siemens.co.in/industry-services

S18199