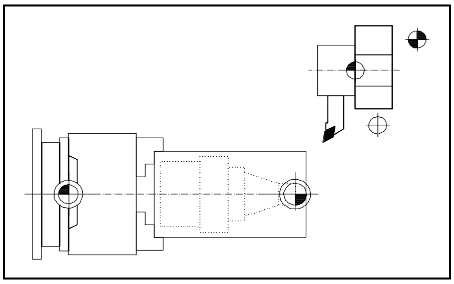
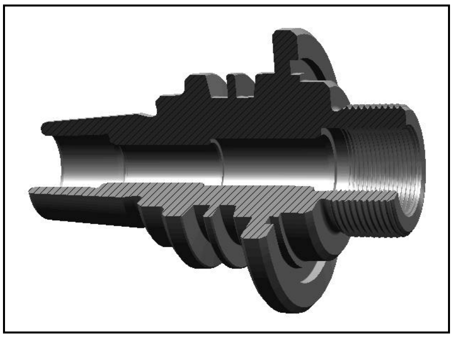
MATHEMATISCH TECHNISCHE SOFTWARE-ENTWICKLUNG GMBH

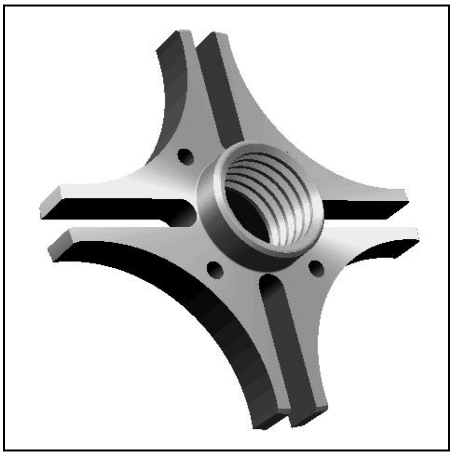


Teachware

CNC Technology



Contents



MATHEMATISCH TECHNISCHE
SOFTWARE-ENTWICKLUNG GMBH

CNC Basics - Excerpt

MTS TeachWare Student's Book - © MTS GmbH 1999

MTS Mathematisch Technische Software-Entwicklung GmbH • Kaiserin-Augusta-Allee 101 • D-10553 Berlin
Phone: +49 / 30 / 349 960 - 0 • Fax: +49 / 30 / 349 960 -25 • World Wide Web: http://www.mts-cnc.com • email: mts@mts-cnc.com

1.3 Characteristics of modern CNC machine tools

Controllable feed and rotation axis

Work part machining on CNC machine tools requires controllable and adjustable infeed axes which are run by the servo motors independent of each other. The hand wheels typical of conventional machine tools are consequently redundant on a modern machine tool.

CNC lathes (see figure 3) have at least 2 controllable or adjustable feed axes marked as X and Z.

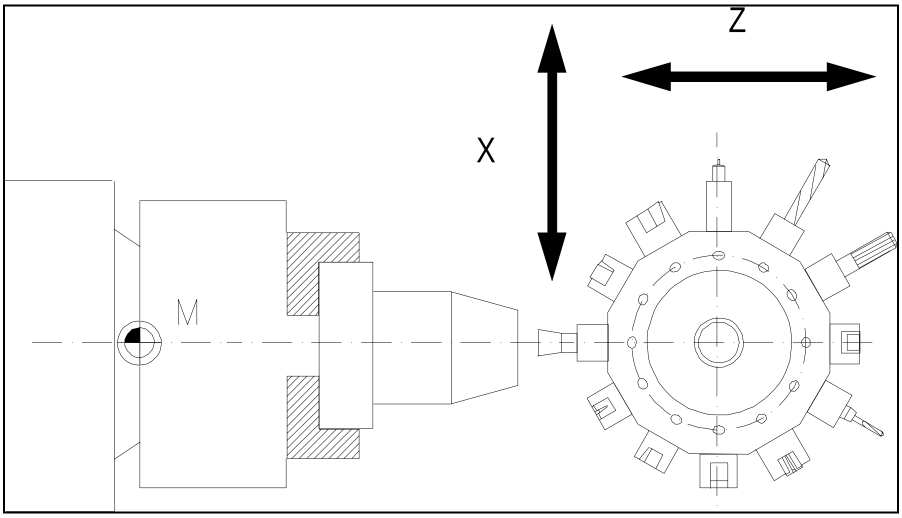


Figure 3
Controllable NC axes on an automatic lathe

CNC- milling machines (see figure 4) on the other hand have at least 3 controllable or adjustable feed axes marked as X, Y, Z.

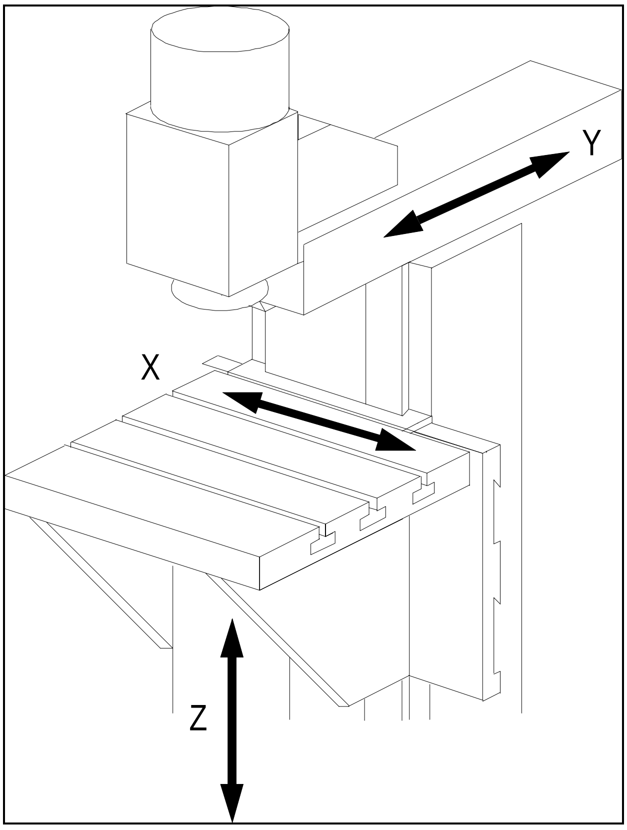


Figure 4
Controllable NC axes on a milling machine

In CNC milling the main function of the work part clamping devices is the correct positioning of the work parts. The work part clamping should allow a work part change which is as quick, easy to approach, correctly and exactly positioned, reproducible as possible. For simple machining controllable, hydraulic chuck jaws are sufficient. For milling on all sides the complete machining should be possible with as few re-clamping as possible. For complicated milling parts milling fixtures, also with integrated automatic rotation, are being manufactured or built out of available modular systems to allow, as far as possible, complete machining without re-clamping. Work part pallets, which are loaded with the next work part by the operator outside the work room and then automatically taken into the right machining position, are increasingly being used.

Tool change facilities

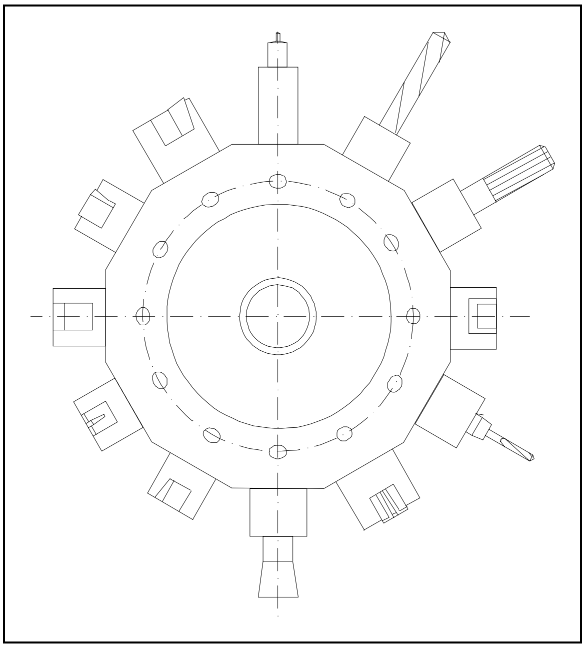


Figure 12
Example of a turret

CNC tool machines are equipped with controllable automatic tool change facilities. Depending on the type and application area these tool change facilities can simultaneously take various quantities of tools and set the tool called by the NC program into working position. The most common types are:

- the tool turret
- the tool magazine.

The tool turret (see figure 12) is mostly used for lathes and the tool magazine for milling machines.

If a new tool is called by the NC program the turret rotates as long as the required tool achieves working position. Presently such a tool change only takes fractions of seconds.

Depending on the type and size, the turrets of the CNC machines have 8 to 16 tool places. In large milling centers up to 3 turrets can be used simultaneously. If more than 48 tools are used tool magazines of different types are used in such machining centers allowing a charge of up to 100 and even more tools. There are longitudinal magazines, ring magazines, plate magazines and chain magazines (see figure 13) as well as cassette magazines.

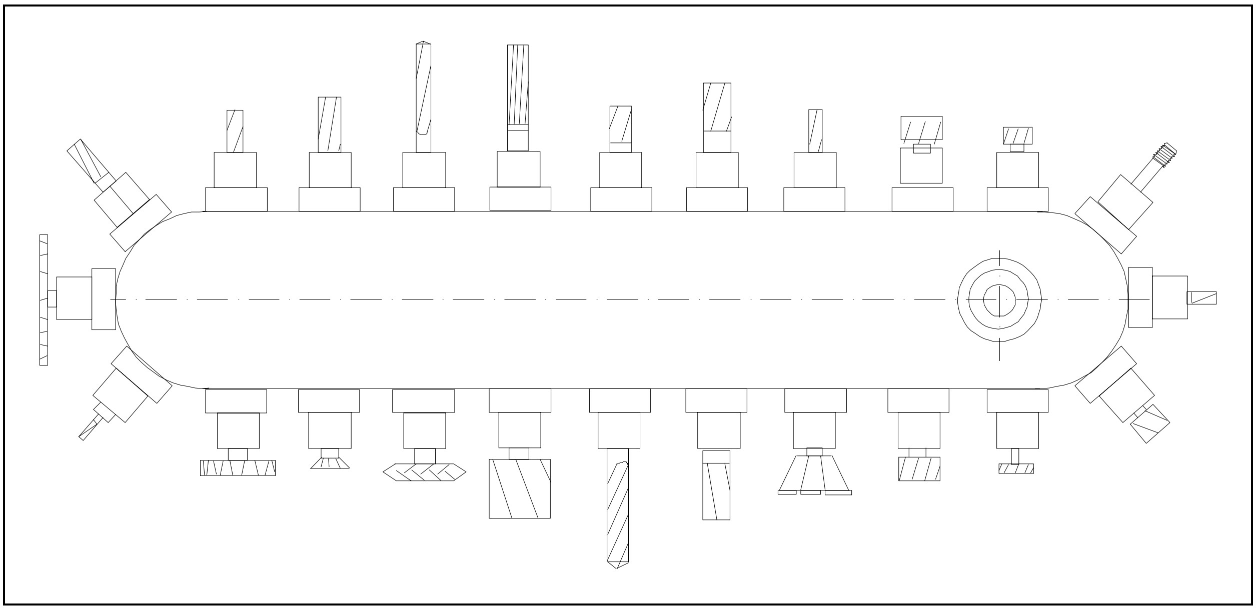
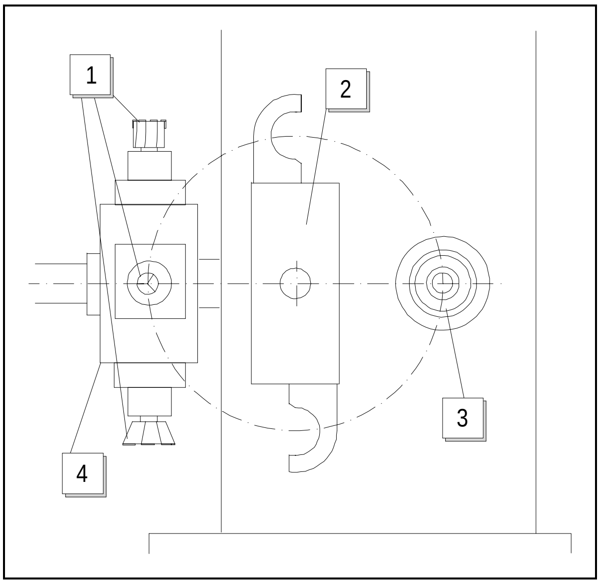


Figure 13
Example of a chain magazine



1 milling tools
2 tool gripper (tool changer)
3 work spindle
4 tool magazine

Figure 14
Automatic tool change facility

In the tool magazine the tool change takes place using a gripping system also called tool changer (see figure 14). The change takes place with a double arm gripping device after a new tool has been called in the NC program as follows:

- Positioning the desired tool in magazine into tool changing position
- Taking the work spindle into changing position
- Revolving the tool gripping device to the old tool in the spindle and to the new tool in the magazine
- Taking the tools into the spindle and magazine and revolving the tool gripping device
- Placing the tools into the spindle sleeve or magazine
- Returning the tool gripping device into home position

The tool change procedure takes between 6 to 15 seconds, whereby the quickest tool changers are able to make the tool change in merely one second.

Security precautions on CNC machine tools

The target of work security is to eliminate accidents and damages to persons, machines and facilities at work site.

Basically the same work security precautions apply to working on CNC machines as to conventional machine tools. They can be classified in three categories:

- Danger elimination
  Defects on machines and on all devices necessary for work need to be registered at once.
  Emergency exits have to be kept free.
  No sharp objects should be carried in clothing.
  Watches and rings are to be taken off.

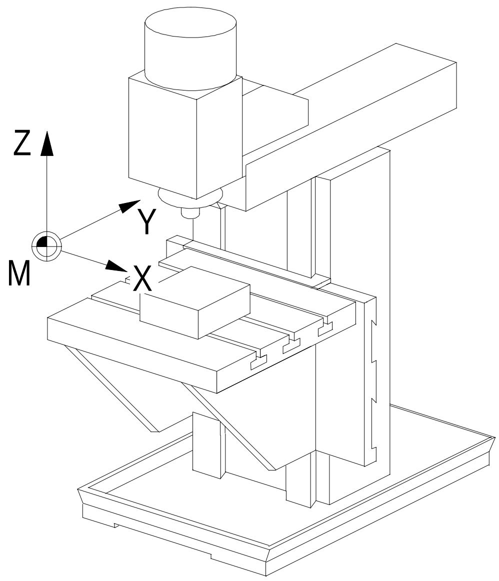
- Screening and marking risky areas:
  The security precautions and corresponding notifications are not allowed to be removed or inactivated.
  Moving and intersecting parts must be screened.

- Eliminating danger exposure
  Protective clothing must be worn to protect from possible sparks and flashes.
  Protective glasses or protective shields must be worn to protect the eyes.
  Damaged electrical cables are not allowed to be used.

Coordinate system definition with reference to machine or work part

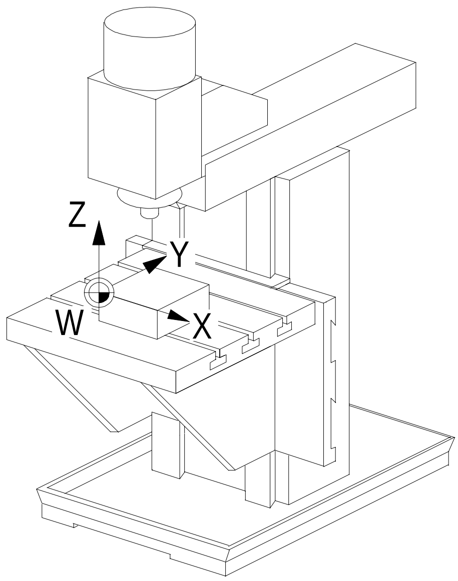
Machine coordinate system
The machine coordinate system of the CNC machine tool is defined by the manufacturer and cannot be changed. The point of origin for this machine coordinate system, also called machine zero point M, cannot be shifted in its location (see figure 21).

Work part coordinate system
The work part coordinate system is defined by the programmer and can be changed. The location of the point of origin for the work part coordinate system, also called work part zero point W, can be specified as desired (see figure 22).



M Machine zero point

Figure 21
Machine coordinate system



W Work part zero point

Figure 22
Work part coordinate system

CNC milling machine
The design of the CNC machine specifies the definition of the respective coordinate system. Correspondingly, the Z axis is specified as the working spindle (tool carrier) in CNC milling machines (see figure 23), whereby the positive Z direction runs from the work part upwards to the tool.

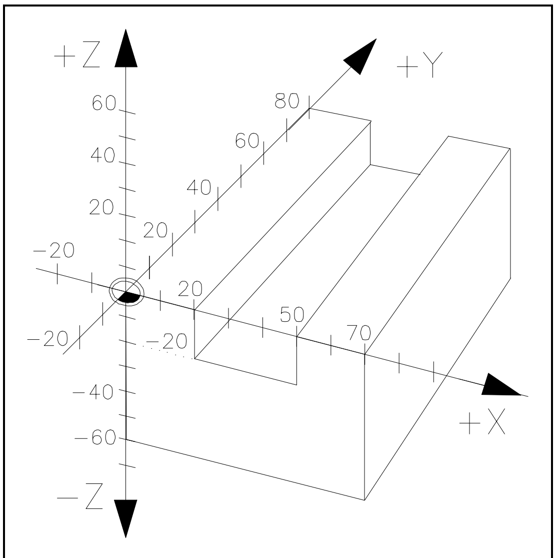


Figure 23
Milling part in three-dimensional Cartesian coordinate system

The X axis and the Y axis are usually parallel to the clamping plane of the work part.

When standing in front of the machine the positive X direction runs to the right and the Y axis away from the viewer.

The zero point of the coordinate system is recommended to be placed on the outer edge of the work part.

For an easier calculation of the points needed for programming it is advisable to use the outer edges of the upper (see figure 24) or the lower area (see figure 25).

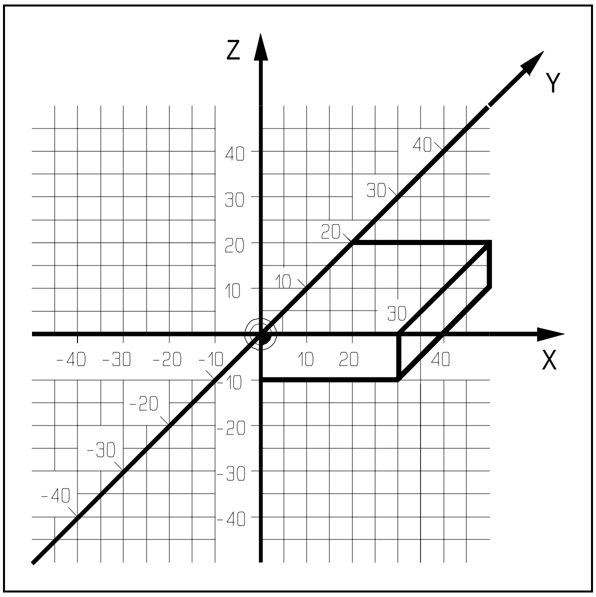


Figure 24
Work part zero point in the upper left outer edge

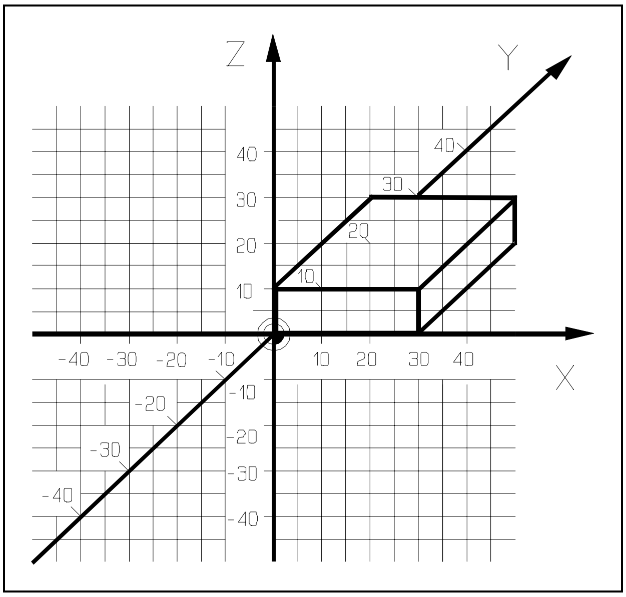


Figure 25
Work part zero point in the lower left outer edge

CNC lathes
In the CNC lathes the working spindle (tool carrier) is specified as Z axis. This means the Z axis is identical to the rotation axis (see figure 26 and 27). The direction of the Z axis is specified so that the tool withdraws from the work part when moving to the positive axis direction.

The X axis is located in a right angle to the Z axis. However, the direction of the X axis always depends on if the tool is located in front of (see figure 26) or behind (see figure 27) the rotation center.

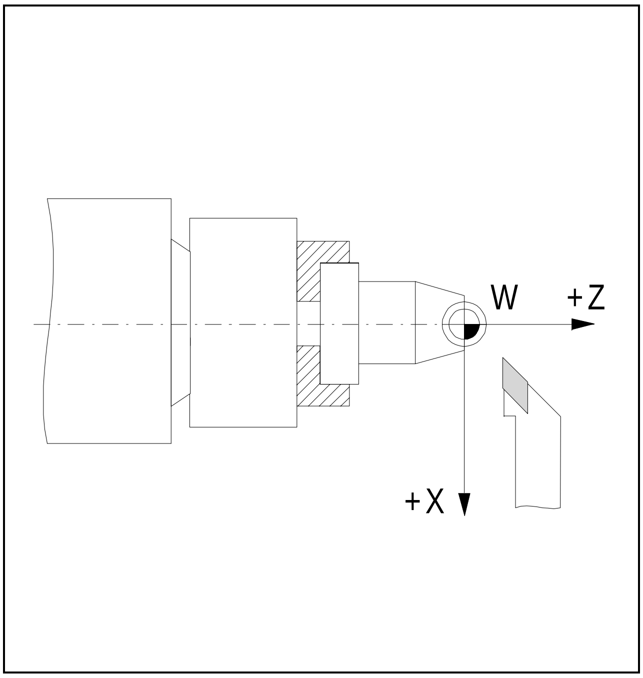


Figure 26
Milling work part in Cartesian coordinate system with 2-axis tool <u>in front of</u> the rotation center

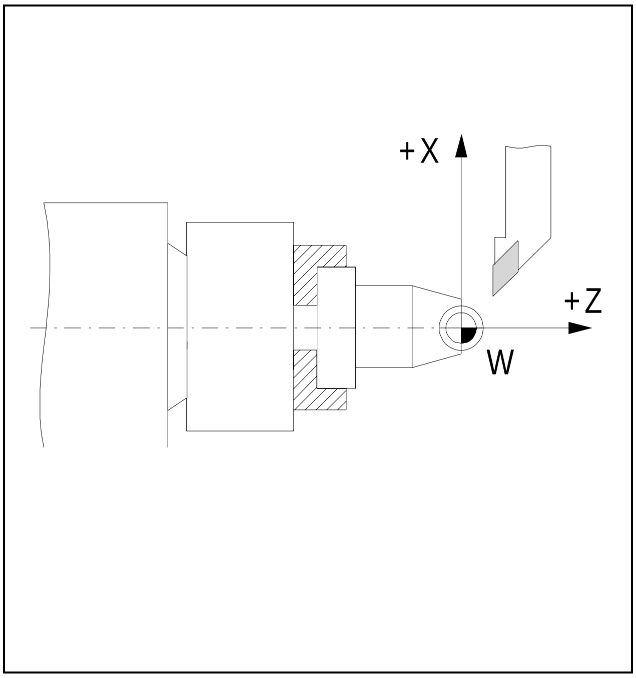
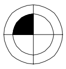
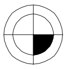


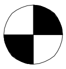
Figure 27
Milling work part in Cartesian coordinate system with 2-axis tool <u>behind</u> the rotation center

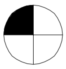
2.3 Zero and reference points on CNC machine tools

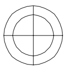
Types of zero and reference points

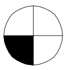
 **M** machine zero point

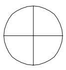
 **W** work part zero point

 **R** reference point

 **E** tool reference point

 **B** tool setup point

 **A** tool shank point

 **N** tool change point

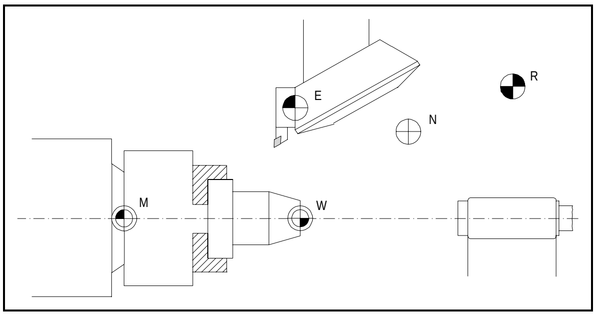


Figure 43
Location of the zero and reference points for turning

Machine zero point M

Each numerically controlled machine tool works with a machine coordinate system. The machine zero point is the origin of the machine-referenced coordinate system. It is specified by the machine manufacturer and its position cannot be changed. In general, the machine zero point M is located in the center of the work spindle nose for CNC lathes and above the left corner edge of the work part carrier for CNC vertical milling machines.

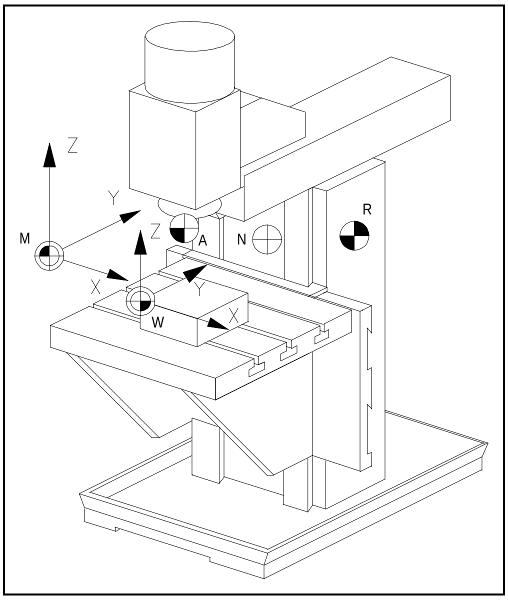
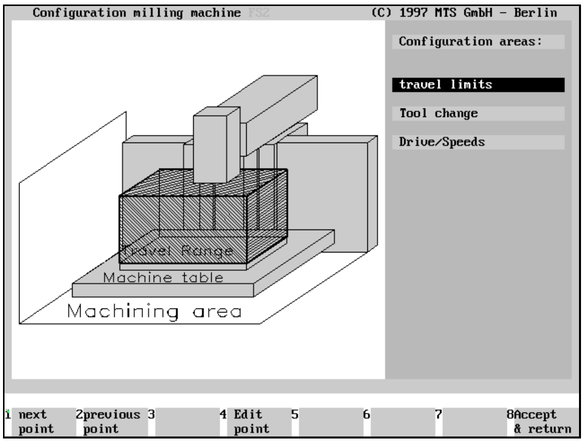
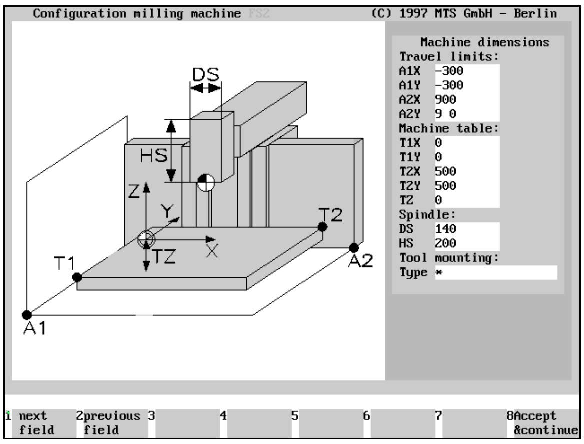


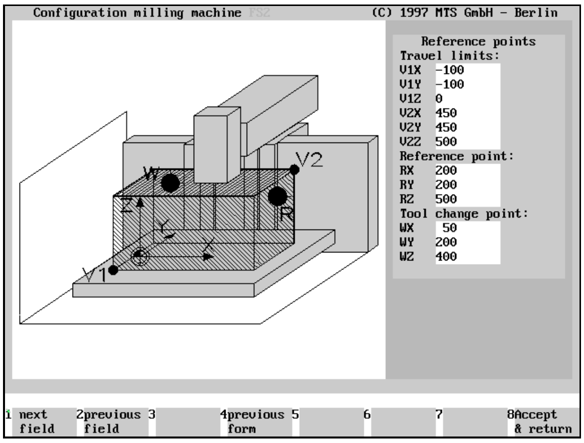
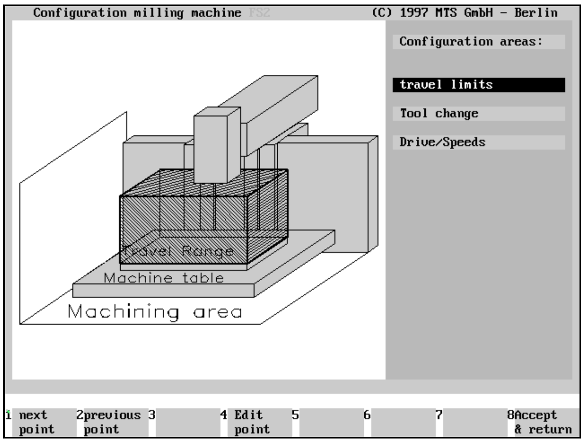
Figure 44
Location of the zero and reference point for milling

Reference point R

A machine tool with an incremental travel path measuring system needs a calibration point which also serves for controlling the tool and work part movements. This calibration point is called the reference point R. Its location is set exactly by a limit switch on each travel axis. The coordinates of the reference point, with reference to the machine zero point, always have the same value. This value has a set adjustment in the CNC control. After switching the machine on the reference point has to be approached from all axes to calibrate the incremental travel path measuring system.

CNC exercise
Generating the machine room of a CNC milling machine

	Description	Entry
1.	Call the configuration in the main menu.	F5 (Configuration)
2.	Select the MTS milling machine.	F1 or select F2
3.	Call the configuration management.	F5 (Config managm)
4.	Generate a new configuration.	F1 (Generate)
5.	Enter a new name, e.g. FS2.	use the keyboard to type the new name „FS2“. F8 (generate)
6.	Select default values, for example, MAKINO FX 650	↑ or select ↓ F8 (Default data)
		
7.	Select the configuration point „machine room“.	F1 or select F2
8.	Change the machine room data.	F4 (Edit point)
		
9.	Enter the machine room data.	F1 or select the individual points F2 Use the keyboard to type in the values. F8 (Accept & Continue)

<table>
<tr><td></td><td colspan="2">
</td></tr>
<tr><td>10.</td><td>Enter the data for the reference points.</td><td>F1 or select the individual points F2
Use the keyboard to type in the values.
F8 (Accept & Continue)</td></tr>
<tr><td></td><td colspan="2">
</td></tr>
<tr><td>11.</td><td>Quit the menu configuration for milling machine.</td><td>F8 (Accept & Return)</td></tr>
<tr><td>12.</td><td>Quit the main menu „configuration“</td><td>F8 (Accept & Terminate)</td></tr>
</table>

2.5 Tool Compensations for CNC Machining

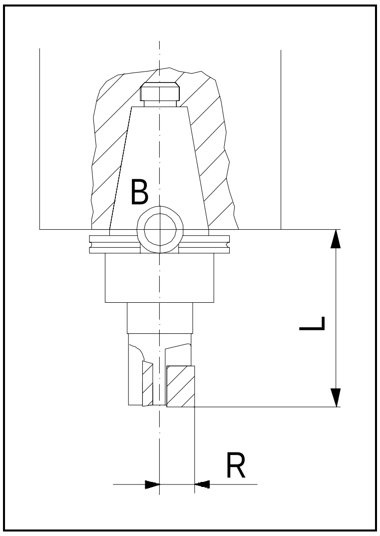
Using tool compensation values

Using the tool compensation values it is easy to program a work part without consideration of the actually applicable tool lengths or tool radii. The available work part drawing data can be directly used for programming. The tool data, lengths as well as radii of the milling machines or indexable inserts are automatically considered by the CNC control.

Tool length compensation for milling and turning

A tool length compensationregarding the reference point enables the adjustment between the set and actual tool length, as in case of tool finishing. This tool length value has to be available for the control. For this it is necessary to measure the length L, i.e. the distance between the tool setup point B and the cutting tip, and to enter it into the control (see chapter on tool measuring page 67 ff.).

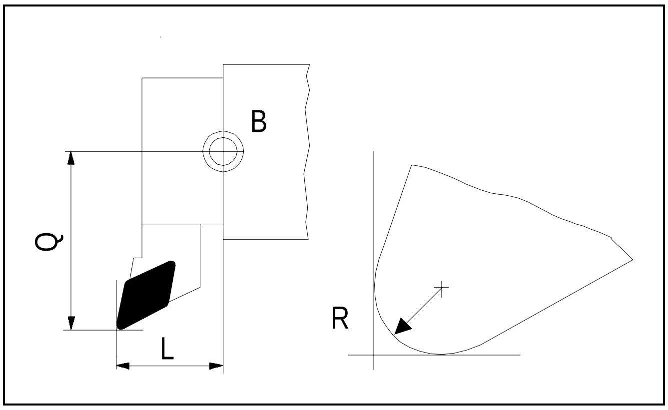
In case of milling tools the length is defined in Z direction (see figure 71).



- B tool setup point
- L length = distance of the cutting tip to the tool setup point in Z
- R radius of the milling tool

Figure 71
Tool compensation values on a cutting tool

In case of lathe tools the length L is defined in Z direction (see figure 72).



- B tool setup point
- L length = distance of the cutting tip to the tool set-in point in Z
- Q overhang = distance of the cutting tip to the tool setup point in X
- R cutting radius

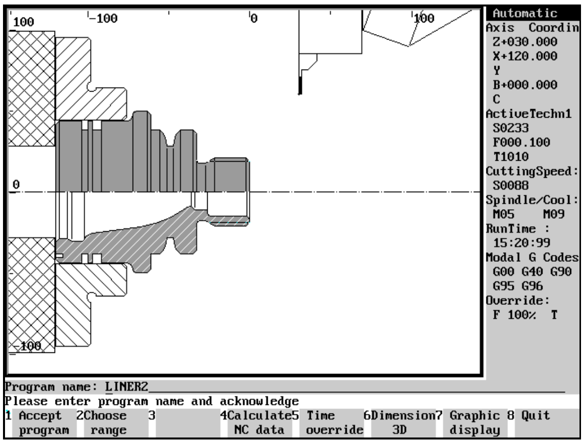
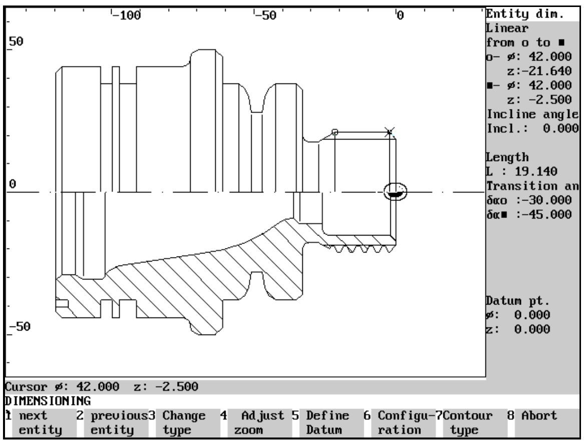
Figure 72
Tool compensation values on a lathe tool

In the CNC control these tool compensation values are stored in the compensation value storage, whereby in most CNC controls it is possible to describe up to 99 tools. These values have to be activated during machining. This is done by calling the data within the NC program, e.g. with the address H or by specific places in the T word.

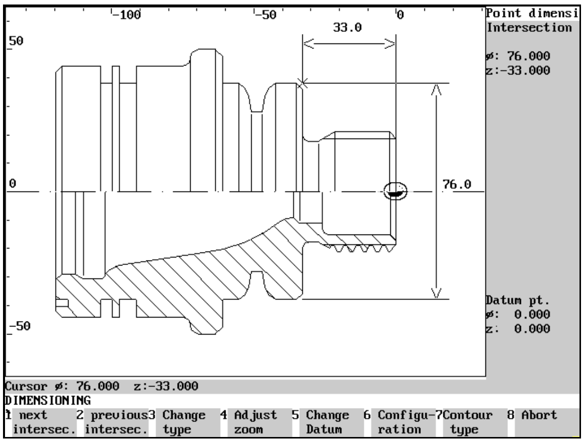
Measuring the work part

A work part can be measured either after machining (automatic run) or during machining after each machining step (in single block run).

Procedure:

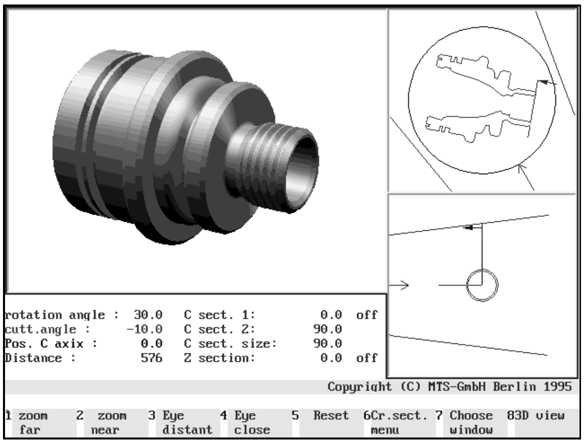
	Description	Entry
1.	Call CNC turning in the main menu.	F1 (Turning)
2.	Select automatic run.	F3 (Automatic mode)
3.	Call a stored NC program, e.g. GEWBU2.	Type „GEWBU2“ using the keyboard and ↵ confirm.
4.	Select simulation mode for automatic run.	F1 (Automatic mode)
		Machining is simulated on the screen
5.	Select measuring menu.	F6 (Dimension 3D)
6.	Select the menu for entity measuring.	F6 (Entity dimension)
7.	Select the entity to be measured.	F1 (next entity) or F2 (previous entity)
		The data for the selected entity are displayed in each case.
8.	Quit the menu for entity measuring.	F8 (Abort)

9.	Select menu for point dimensioning.	F7	(Point dimension)
10.	Select the point to be measured.	F1 F2	(next point) or (previous point)



The data of the selected point are displayed in each case.

11.	Quit the menu for point measuring.	F8	(Abort)
12.	Select the menu for 3D representation.	F1	(3D display)
13.	Generate the 3D representation.	F8	(3D view)



14.	Quit the menu for 3D representation.	ESC ESC	
15.	Quit the measuring menu.	F 8	(Quit)
16.	Quit the simulation mode of automatic run.	F 8	(Quit)

Cutting edge geometry

Each machining process requires its cutting edge geometry. Only this can guarantee ideal production times, long cutting-edge life and high surface quality. The angles of the tool cutting edge play a decisive role here (vgl. Abbildung 103).

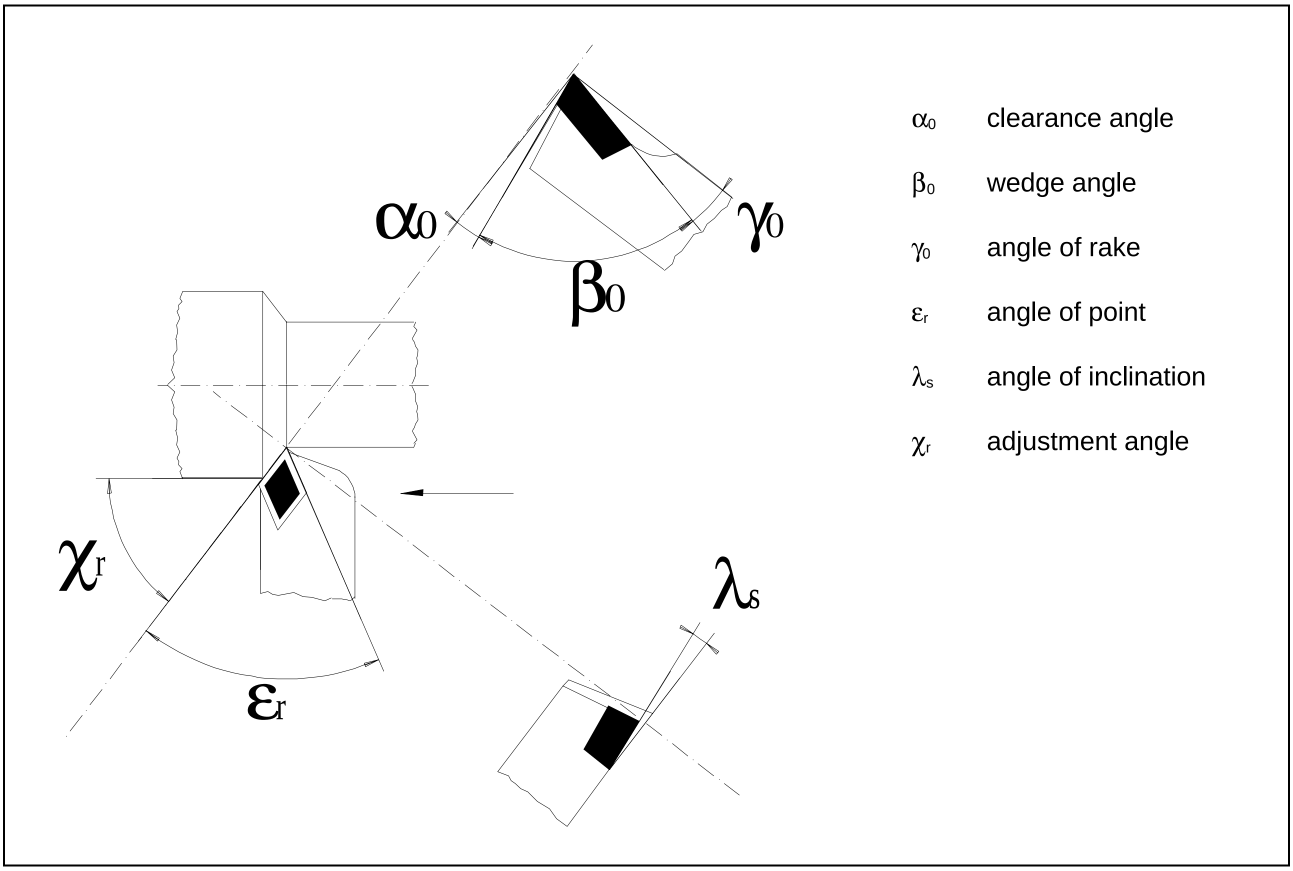


Figure 103
Cutting geometries in turning

Clearance angle α: The clearance angle reduces friction and heating up of the tool edge and the work part.

Wedge angle β: The size of the wedge angle depends on the hardness and toughness of the work part. The smaller the wedge angle the lighter the cutting, however, the larger the edge abrasion and the shorter the cutting edge life.

Angle of rake γ: The angle of rake has an influence on chip building and cutting forces. The larger the angle of rake the smaller the cutting force, however, cutting edge breach and abrasion are increased because of total decarburization. Solid, medium hard materials require an angle of rake of approx. 10°. Hard and brittle materials require a small or even a negative angle of rake.

Adjustment angle χ: In the first place the entering angle has an influence on infeed force, on the forces against the work part clamping and work part as well as on the cutting width and thickness. In case of solid clamping situation an entering angle of 30 to 60° is selected. Only for thin shafts or right angled offsets 90° is selected for the adjustment angle.

Inclination angle λ: For finishing a positive, for roughing a negative inclination angle is frequently selected. When negative angles of rake are used the cutting edge tip is exposed to less stress. When positive inclination angle is used the chip flow is directed away from the work part.

Angle of point ε: The larger the angle of point the better the stability of the tool edge and the better the heat removal.

Cutting value

Turning is a cutting operation with a circular cutting movement and an infeed which can be in any relation to the cutting direction. In most cases the cutting movement is made by the rotation of the work part and the infeed of the tool (see figure 110). The

- cutting speed v_c and the
- infeed speed v_f

overlap and result in a continuous cutting process.

Cutting speed v_c
Cutting speed is the movement between the tool and the work part causing only a single chip removal during one rotation without infeed. The symbol for cutting speed is v_c and is indicated in m/min.

In general the speed indicates the traversed path *s* within a certain period of time *t*. It is calculated as follows:

$$v = \frac{s}{t}$$ in path/time

The traversed path *s* for a work part rotation can be generated in turning using the work part diameter d on the cutting edge tip and the constant π:

$$s = \pi * d$$ in m

The starting point for the calculation of the cutting speed is now a time unit t = 1 min. The result is herewith cutting speed v_c :

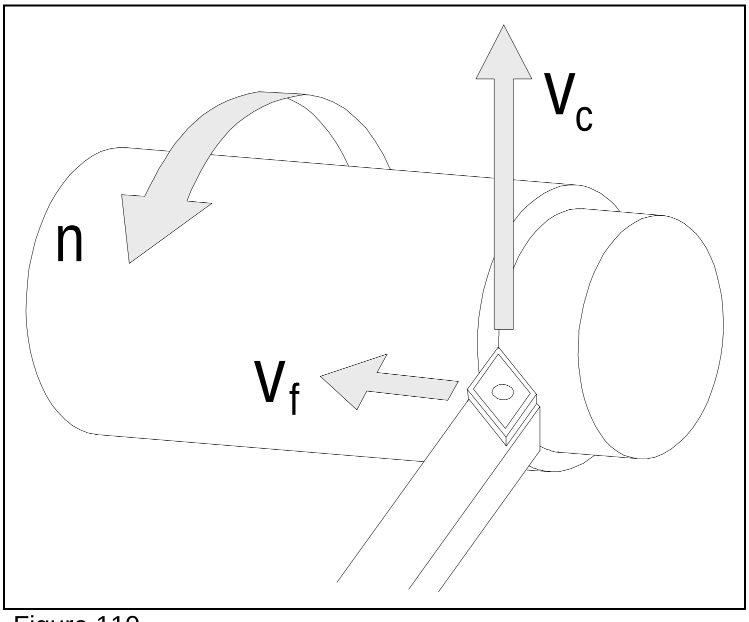
$$v_c = \frac{\pi * d}{t}$$ in m/min

The number or work part rotations in one minute is indicated as a number of rotations n (in rotations per minute):

$$t = \frac{1}{n}$$ in min

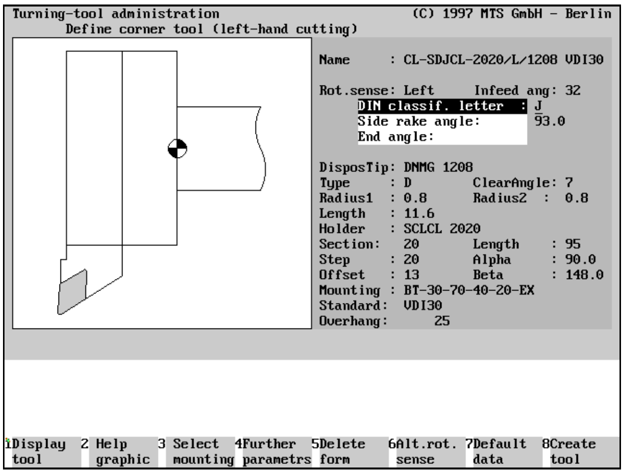
As a result the following formula is achieved for the calculation of the cutting speed v_c:

$$v_c = \pi * d * n$$ in m/min



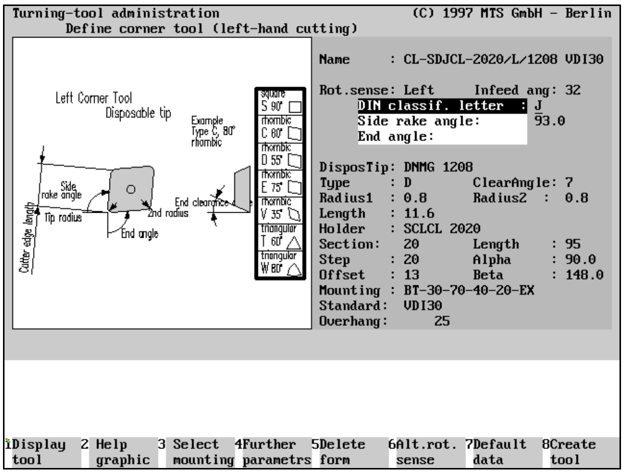
n	number of rotations	in U/min
v_f	infeed speed	in mm/
v_c	cutting speed	in m/min
v_c	$= \pi * d * n$	

Figure 110
Cutting values in turning

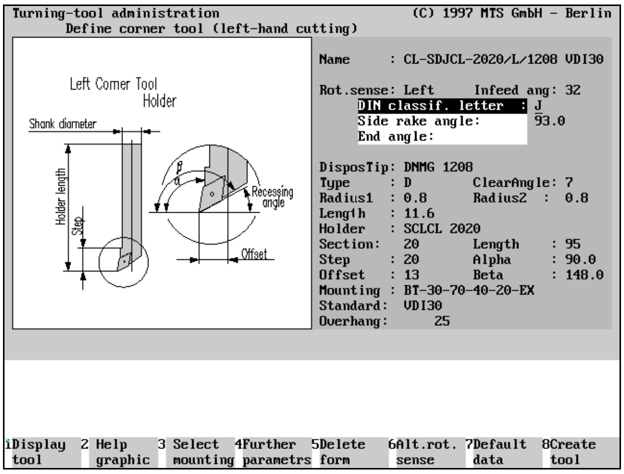


11. If required display further information on turning tool.

1) indexable inserts: — F2 (help graphic)



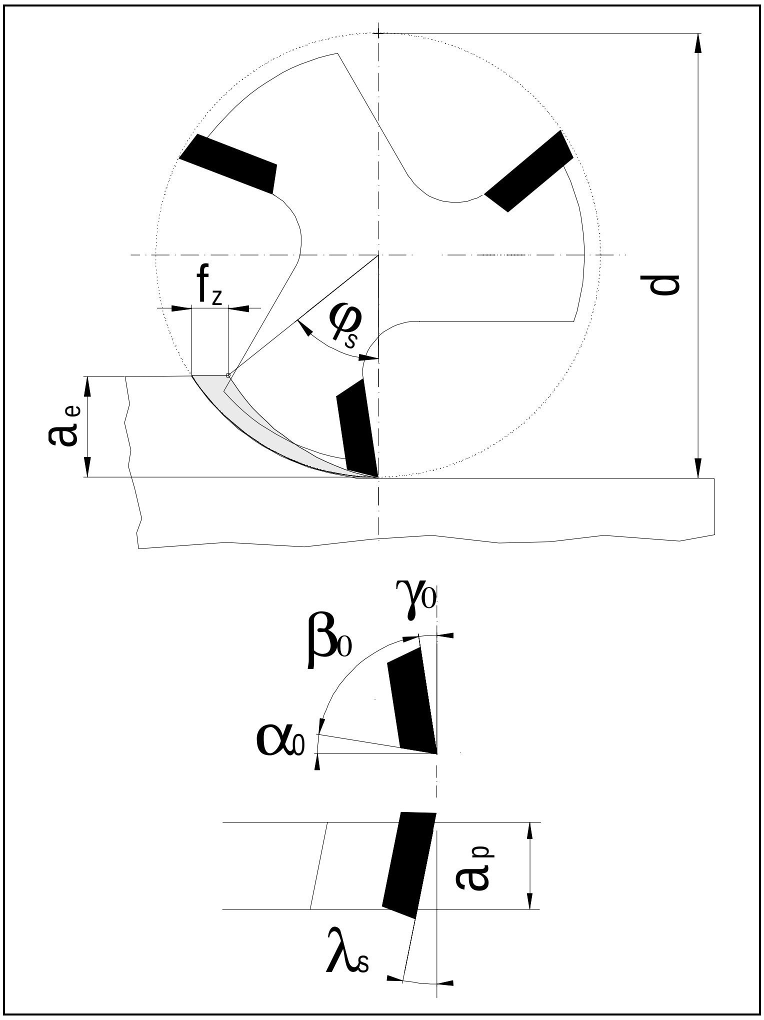
12. 2) tool holder: — F2 (help graphic)



13. 3) tool carrier: — F2 (help graphic)

Cutting geometry

Unlike lathe tools milling tools have several cutting edges (see figure 119). Typical of milling is the discontinuous cut as each cutting edge works only for a time.



d: diameter of the milling tool

z: number of teeth

f_z: feed per tooth

a_e: entering point

φ_S: entering angle

α_0: clearance angle

β_0: wedge angle

γ_0: angle of rake

λ_S: angle of twist of the edges

a_p: cutting width

Figure 119
Cutting geometry milling

Clearance angle α: The clearance angle is to reduce the friction and consequently the heating of the cutting edge and of the work part.

Wedge angle β: The size of the wedge angle depends on the hardness of the work part. The smaller the wedge angle the lighter the cutting, however the greater the cutting abrasion and the shorter the cutting edge life.

Angle of rake γ: The angle of rake influences cutting chip formation and cutting forces. The larger the angle of rake of the chip the smaller the cutting force, however the risk to breach as well as abrasion of the cutting edge are increased due to erosion.

Entering angle φ_S: The entering angle indicates the machining path of the tool with reference to the circumference. It depend on the size of the entering point.

Inclination angle λ: The size of the inclination angle influences the process of chamfering and cutting-out. Since the inclined cutting edges are consecutively engaged the milling tool runs with increased quietness.

3.4 Calculation of technological data for CNC machining

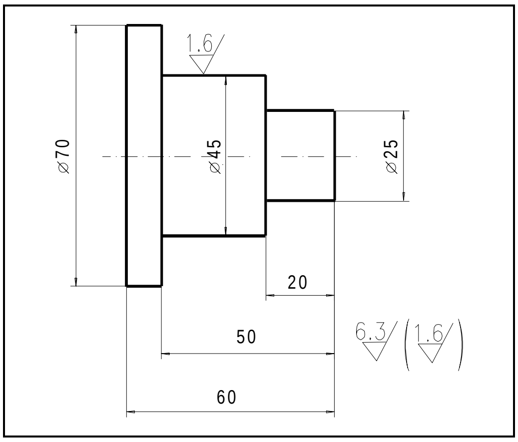
Calculation examples of technological data for CNC turning

1. Example:

On a CNC-lathe the sketched bolt is to be roughed as well as finished in four cuts with cutting depths of 6; 6; 5 and 5 mm and a finishing allowance of 0,5 mm.

The cutting speed for roughing is v_{cv} = 280 m/min and for finishing v_{cf} = 400 m/min.

Calculate the number of rotations for each cut.



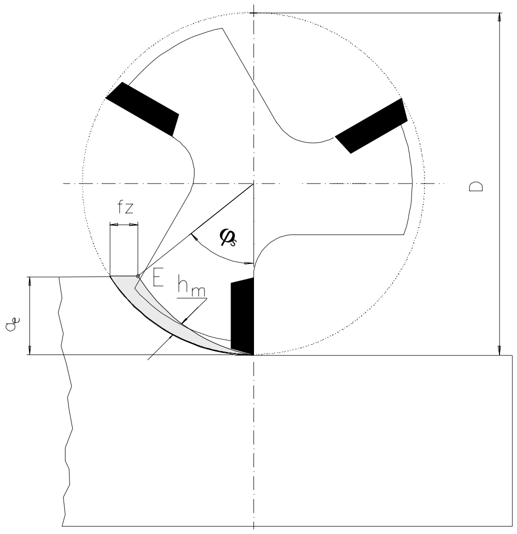
Calculating the number of rotations for roughing (Cut 1-4) and for finishing (Cut 5-6)

datum:	v_{cv} = 280 m/min v_{cf} = 400 m/min
unknown:	n in 1/min
valid :	$n = \frac{v_c}{\pi * d}$

1. Cut ∅ = 58mm v_{cv} = 280 m/min $n_1 = \frac{280m}{\pi * \min * 0{,}058m}$ $n_1 = 1537\ 1/\min$	2. Cut ∅ = 46mm v_{cv} = 280 m/min $n_2 = \frac{280m}{\pi * \min * 0{,}046m}$ $n_2 = 1938\ 1/\min$
3. Cut ∅ = 36mm v_{cv} = 280 m/min $n_3 = \frac{280m}{\pi * \min * 0{,}036m}$ $n_3 = 2476\ 1/\min$	4. Cut ∅ = 26mm v_{cv} = 280 m/min $n_4 = \frac{280m}{\pi * \min * 0{,}026m}$ $n_4 = 3428\ 1/\min$
5. Cut ∅ = 25 mm v_{cf} = 400 m/min $n_5 = \frac{400m}{\pi * \min * 0{,}025m}$ $n_5 = 5393\ 1/\min$	6. Cut ∅ = 45 mm v_{cf} = 400 m/min $n_6 = \frac{400m}{\pi * \min * 0{,}045m}$ $n_6 = 2830\ 1/\min$

Calculating the cutting force and motor power

For calculating the cutting force, the same compensation factors are used for milling as in for turning..



a_p	cutting depth
a_e	entering point
b	chip width
F_{cz}	cutting force per edge (mean)
φ_s	entering angle
h_m	middle chip thickness
f_z	feed per edge
z	number of cutter edges
z_e	number of edges in operation
D	diameter of milling cutter
λ	angle of twist of edges
κ	adjustment angle of edges
k_c	specific cutting force
k_{c1-1}	specific cutting force related to chip diameter $b \cdot h_m$=1 mm
m_c	chip thickness index

These are either taken from a book of specifications or, as in the case of the angle of rake variation factor, calculated with the formula $K_{\gamma_o} = 1 - \frac{\gamma_{o-}\gamma_{ok}}{66.7}$. For milling, the cutting force is:

$F_c = F_{cz} \cdot z_e [N \cdot 1 = N]$. In this formula

$z_e = \frac{z \cdot \varphi_s}{360°}$ and

$F_{cz} = b \cdot h_m \cdot k_c$. Herewith are

$b = \frac{a_e}{\cos\lambda} [mm]$ and

$h_m = f_z \cdot \sin\kappa \cdot \frac{360° \cdot a_e}{d \cdot \pi \cdot \varphi_s} [mm]$.

κ=90°-λ for milling cutters with angle of twist.

Taking into account the compensation factors, the cutting force can be calculated with the formula:

$F_c = z_e \cdot b \cdot h_m \cdot k_c \cdot K_{\gamma_o} \cdot K_v \cdot K_{ver} \left[mm \cdot mm \cdot \frac{N}{mm^2} = N \right]$ and with z_e, b, h_m yields the formula

$$F_c = \frac{z \cdot \varphi_s}{360°} \cdot \frac{a_e}{\cos\lambda} \cdot \frac{360° \cdot a_p}{\pi \cdot \varphi_s \cdot d} \cdot f_z \cdot \sin\kappa \cdot k_c \cdot K_{\gamma_o} \cdot K_v \cdot K_{ver}$$

4.2 NC programming basics

A NC-program comprises a series of commands with which the CNC-machine tool is instructed to manufacture a certain tool.

For each machining process on a CNC-machine tool, the NC-program has a command with relevant information. These commands are alphanumerically coded, i.e. they consist of letters, numbers and characters.

NC programming standards (ISO)

The ISO-Norm 6983 strives for standardizing the NC-programming of machines in the production area. This is however limited to standardizing certain commands as well the general structure of a NC-program. CNC-control manufacturers have considerable liberty for incorporating their own NC-commands in their controls. Subsequently, the general structure of an NC-program according to ISO 6983 is illustrated.

Structure of an NC program

Structure of an NC program:
A complete NC-program consists of the following elements:

% TP0147	NC-program beginning,
N10 G54 X80 Y100...	a series of NC-blocks
...	with the information for machining and
N75 G01 Z-10 F0.3 S1800 T03 M08	
...	
N435 M30	a command for ending the program.

figure 5
Structure of an NC-program

The **program beginning** consists of a character or a command (ex. %) which informs the CNC-control that a NC-program will follow. Additionally, the first line of the NC-program also contains the program name (ex. TP0147). Furthermore, both characteristics are also important for the NC-program manager as well as for calling the NC-programs in the CNC-control.

NC-program names can contain alphanumerical or numerical characters. For most CNC-controls 2-6 digit character sequences are used for identification.

An NC-program consists of a chronological sequence of **blocks**. They contain the relevant geometric and technical information that the CNC-control requires for each machining step.

The **program end** is commanded with M30 or M02.

Everything that stands before the character % for commenting the program is ignored by the control. This enables any explanations on the program or tool to be attached preceding the actual program. Comments are also allowed within a program, e.g. for identifying particular blocks. These, however, must be set in brackets.

Structure of a program block

Every NC-block consists of a block number, a number of words as well as a specific control character which informs the CNC-control that the NC-block has ended. This control character is called LF for line feed. It is automatically generated in NC-programming when the enter-key of the CNC-control or the enter-key on the PC-keyboard is pressed.

N75	**G01**	**Z-10.75**	**F0.3**	**S1800**	**T03**	**M08**	LF
Number of the NC-block	Word	Word	Word	Word	Word	Word	invisible block ending character

figure 6
Structure of a program block

Structure of a program word

A word consists of address letters and a number with a plus/minus sign. The definition and sequence are designated in the programming instructions of the CNC-control systems. Depending on the address letter, the number either pertains to a code or a value.

Example	Address	Number	Definition
N75	N	75	For the address N, 75 is the number of the NC-block.
G01	G	01	For the address G, 01 is a code. The NC-command G01 is "Moving the tool along a straight line at infeed speed".
Z-10.75	Z	-10.75	For the address Z, -10.75 is a value. Corresponding to the NC-command G01 of the preceding NC-block example, this means that the tool is to be moved to the position Z=-10.75 in the current tool coordinate system.

figure 7
Structure of a program word

The form of numerical entry depends on the CNC-control: Z-35.5 is equivalent to e.g. the same target coordinates as Z-035.500. For most CNC-controls the positive sign "+" can be excluded in the NC-program.

Generally, three groups of words in an NC-block can be differentiated:

G-Functions	**Coordinates**	**Additional and Switching Functions**
G00	X	F
G01	Y	S
G02	Z	T
G54		M

figure 8
Groups of program words

The sequence of the words in an NC-block is designated as follows:

	Address	Definition
1.	N	block number
2.	G	G-functions
3.	X, Y, Z	coordinates
4.	I, J, K	interpolation parameter
5.	F	feed
6.	S	speed
7.	T	tool position
8.	M	additional functions

figure 9
Sequence of program words

Words that are not needed by a block can be excluded.

Block number N
The block number is the first word in a block and designates it. It can only be conferred once. The block number has no influence on the execution of the individual blocks since they are invoked following the order in which they were entered into the control.

G-function
Together with the words for the coordinates, this word essentially determines the geometric part of the NC-program. It consists of the address letter G and a two-digit code.

Coordinates X, Y, Z
The coordinates X, Y, Z define the target points that are needed for travel.

Interpolation parameters I, J, K
The interpolation parameters I, J, K are e.g. used to define the center of a circle for circular movements. They are usually entered incrementally.

Feed F
The speed at which the tool is to be moved is programmed with the function F. The infeed speed is usually entered in mm/min. For turning, the unit mm/U pertaining to spindle rotation can also be used.

Spindle speed S
The function S is for entering the spindle speed. It can be directly programmed in rotations per minute.

Tool position T
The address T together with a numerical code designates a specific tool. The definition of this address differs according to the control and can have the following functions:

- Saving the tool dimensions in the tool offset table
- Loading the tool from the tool magazine.

Additional functions M
The additional functions, also known as auxiliary functions, primarily contain technical data that is not programmed in the words with address letters F, S, T. These functions are entered with the address letter M and a two-digit code.

4.3 Introduction to manual NC programming

Procedure for manual NC programming

The procedure for manual programming can be divided into four steps:

1. analysis of workshop drawings
2. definition of work plans
3. choice of clamping devices and necessary tools (set-up sheet)
4. generating the NC program (program sheet)

Various documents must be analyzed and plans for production execution must be created. (see fig. 10).

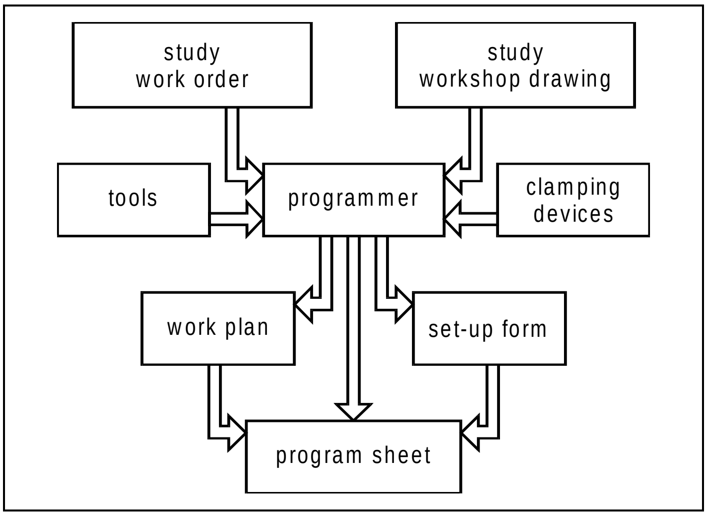


figure 10
Procedure for manual programming

Analysis of workshop drawings

The workshop drawing (see fig. 11) contains the geometric and technical information for the finished part. The dimensions, the surface specifications as well as information on the machining procedure to be used (e.g. cutting, threading, hardening) are taken from the drawing. Information on the work to be executed as well as on the number of work parts and the deadlines is specified in the work order.

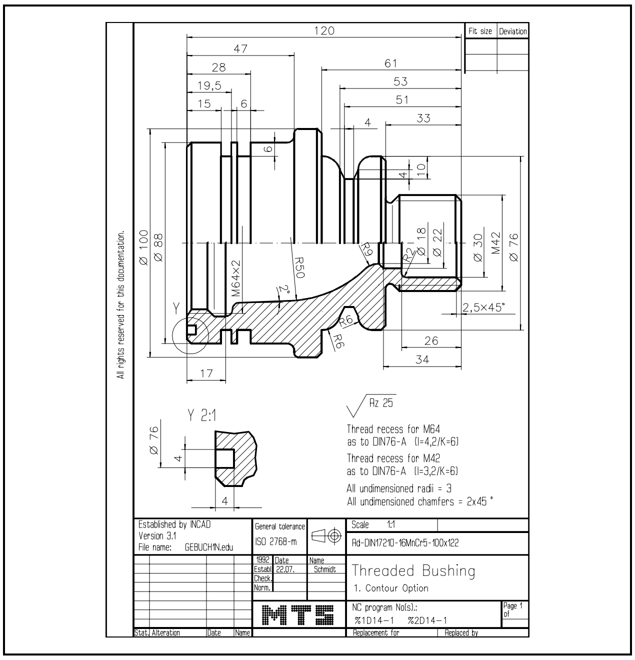


figure 11
Workshop drawing turning

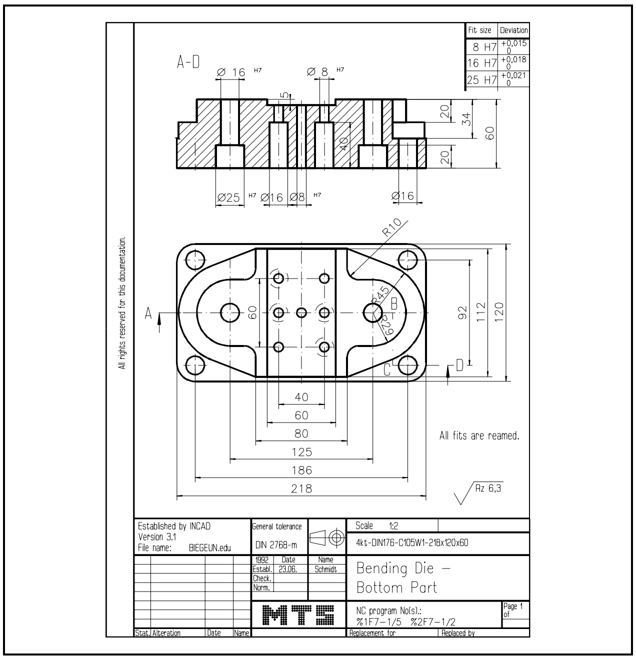


figure 12
Workshop drawing milling

Manual NC programming Turning

CNC exercise

Instructed generation of NC-programs for CNC-turning operations

Task:
An NC-program is to be generated for manufacturing the following part.

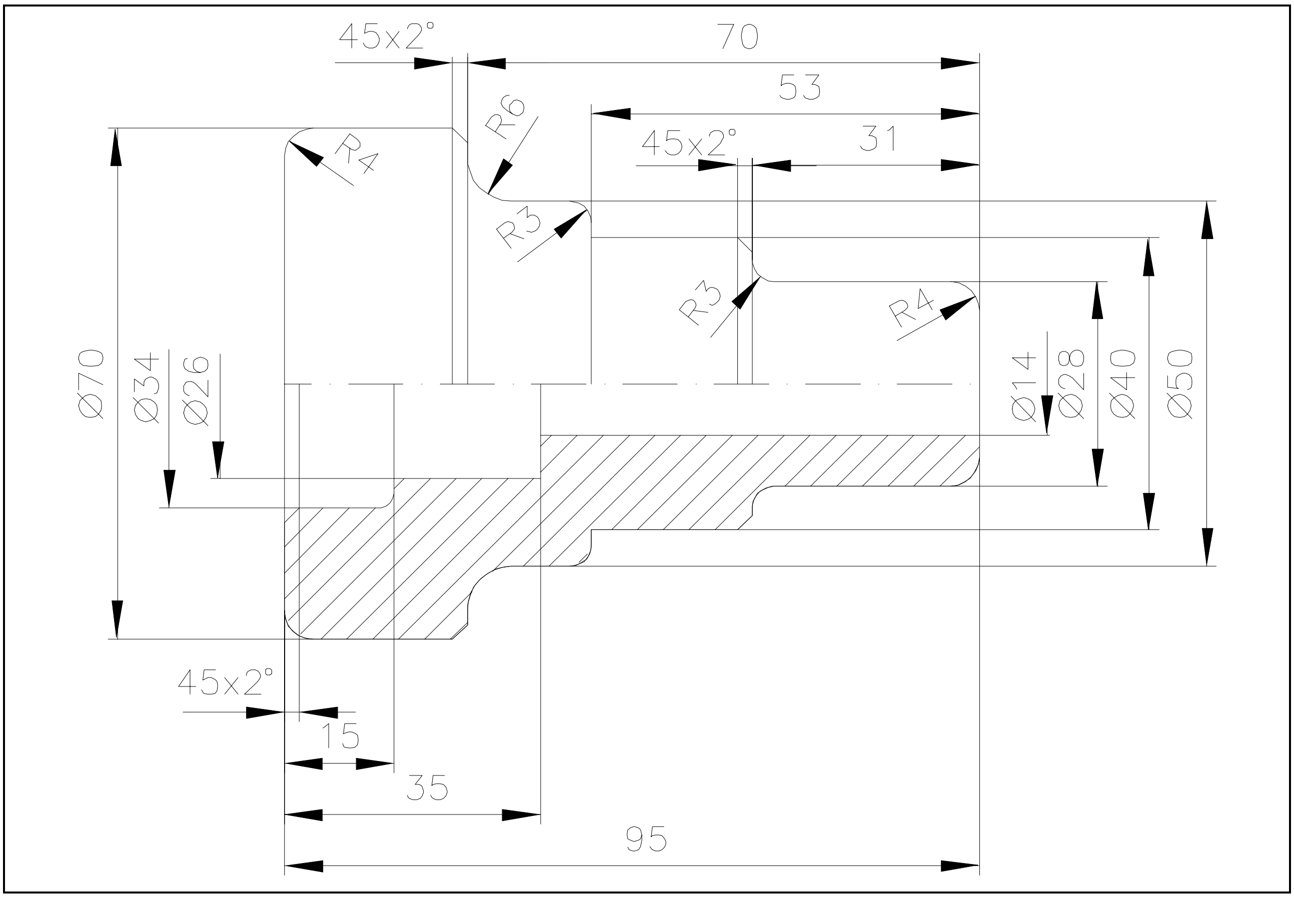


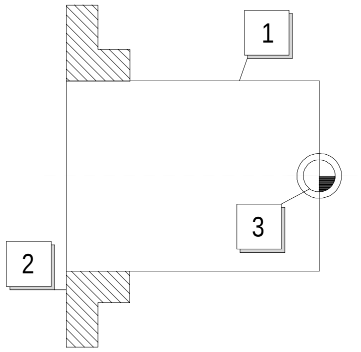
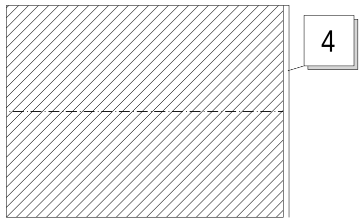
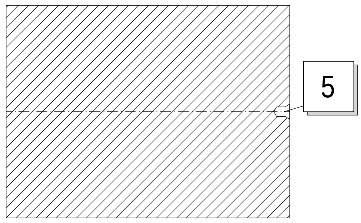
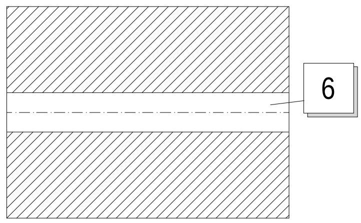
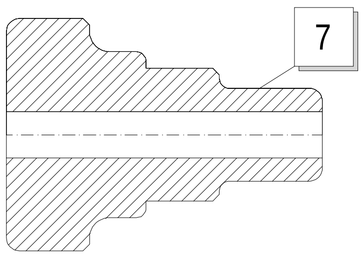
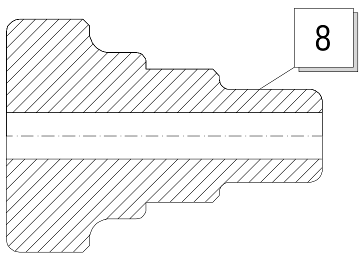
figure 17

Follow the subsequent steps for generating the NC-program:

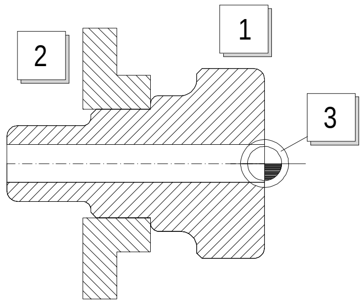
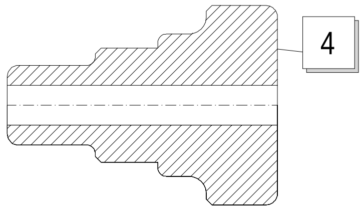
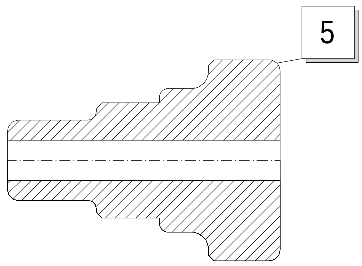
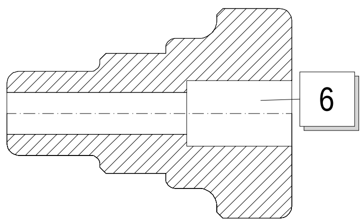
1. definition of the work plan
2. choice of clamping devices and necessary tools
3. generating the NC program
4. simulating the NC program

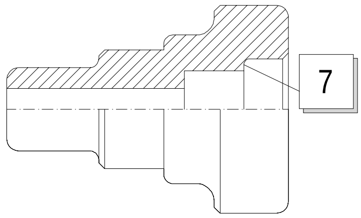
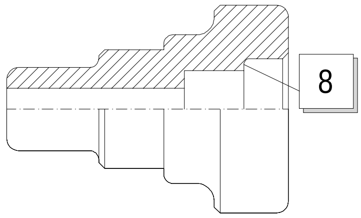
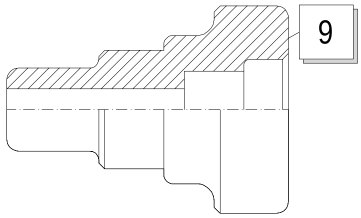
Definition of the work plan

Work plan for machining the first side:

	Machining Sequence	Tool	Turret Position	Cutting Values	Outline
1 2 3	check blank dimensions clamp work part 1.side define work part zero point				
4	Face Turning	Left Corner Tool CL-SCLCL-2020/R/1208	T04	G96 F0.15 S140	
5	Centering	Center Drill CD-03.15/050/R/HSS	T09	G97 F0.16 S1800	
6	Drilling	Twist Drill Ø 14mm DR-18.00/130/R/HSS	T07	G97 F0.22 S1000	
7	Outside contour roughing	Left Corner Tool CL-SCLCL-2020/R/1208	T04	G96 F0.1 S140	
8	Outside contour finishing	Left Corner Tool CL-SVJCL-2020/R/1604	T02	G96 F0.1 S280	

Work plan for machining the second side:

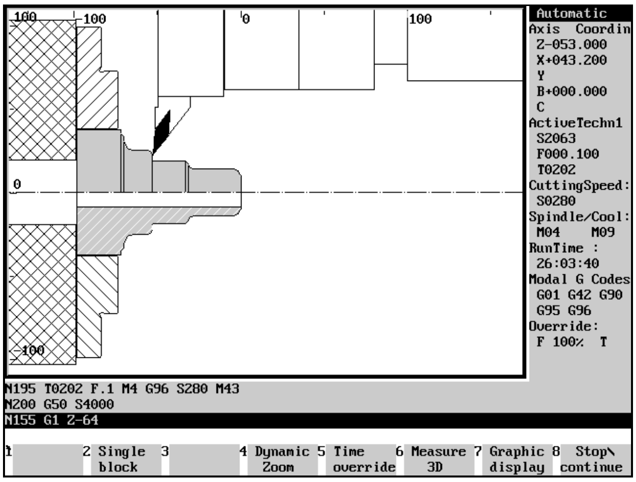
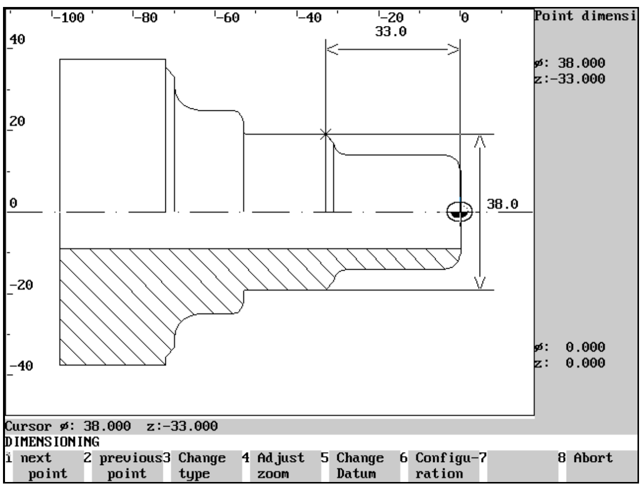
	Machining Sequence	Tool	Turret Position	Cutting Values	Outline
1 2 3	check work part clamp work part 2.side define work part zero point				
4	Face Turning with offset 0.2mm	Left Corner Tool CL-SCLCL-2020/R/1208	T04	G96 F0.28 S140	
5	Outside contour roughing	Left Corner Tool CL-SCLCL-2020/R/1208	T04	G96 F0.28 S140	
6	Predrilling	Reversible Tip Drill Ø 22mm DI-22.00/051/R/HMT	T12	G97 F0.2 S850	

	Machining Sequence	Tool	Turret Position	Cutting Values	Outline
7	Inside contour roughing with offset	Inside Turning Tool Post BI-SDQCL-1616/R1104	T05	G96 F0.2 S120	
8	Inside contour finishing	Inside Turning Tool Post BI-SVQJCL-2020/R/1604	T10	G96 F0.1 S220	
9	Outside contour finishing	Left Corner Tool CL-SVJCL-2020/R/1604	T02	G96 F0.1 S280	

Quality control by measuring work results

A work part can be measured after machining (automatic mode) or during machining after every operation (single block) and can be compared with the values in the drawing.

Procedure:

	Description	Entry
1.	Call CNC turning in the main menu.	F1 (turning)
2.	Select menu automatic mode.	F2 (automatic mode)
3.	Call a present NC program, par example GEWBU2.	Using the keyboard type in„GEWBU2“ and ↵ confirm.
4.	Select the simulation type „automatic mode“.	F1 (Automatic mode)
		On the screen the simulation of the machining starts.
5.	Select menu measurement.	F6 (Dimension 3D)
6.	Select menu point dimension.	F6 (Point dimension)
7.	Select the point for measurement.	F1 (next point) or F2 (previous point)
		For the selected point the data are shown on the screen
8.	Quit the menu measurement.	F8 (Abort) F8 (Quit)

Manual NC programming Milling

CNC Exercise

Instructed generation of NC-programs for CNC-milling

Task:
An NC-program is to be generated for manufacturing the following part:

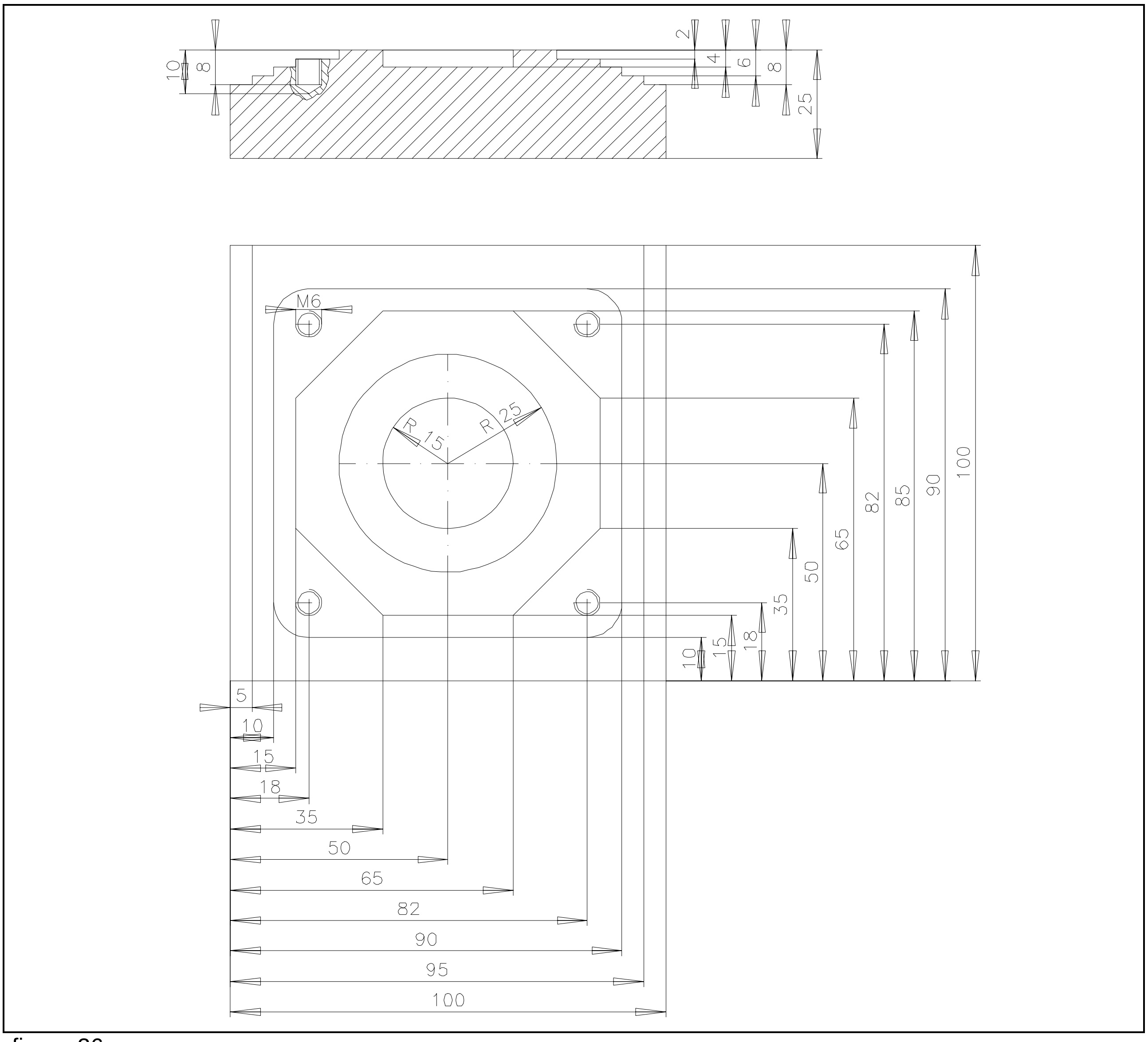


figure 26

Follow the subsequent steps for generating the NC-program:

1. definition of the work plan
2. choice of clamping devices and necessary tools
3. generating the NC program
4. simulating the NC program

Control test „Introduction into NC programming“

1.	List the steps for manual programming.
2.	What is the difference between a work plan and a programming sheet?
3.	Explain the meaning of "switching information".
4.	Name and explain five commands for a CNC-machine.
5.	Explain the structure of an NC-program.
6.	Explain the structure of a program block.
7.	Explain the structure of a program word.
8.	Explain the address letters F, S, T, M, X, Y, Z.
9.	Explain the following program words for a) absolute programming (G90) b) incremental programming (G91)! X 53, Z 184.005
10.	What do the address letters I, J, K express?
11.	Define the following functions with the corresponding program words (G-command or M-command) clockwise circular interpolation activate coolant activate spindle in clockwise rotation
12.	For which cases are constant cutting speeds required? Explain why.
13.	With which G-function is constant cutting speed programmed?
14.	Read and explain the following program block. Illustrate the sequence of motions. G01 G95 X100 Z-5 F0.25 S600 T0101
15.	Read and explain the following program block. Illustrate the sequence of motions. G02 G96 X30 Z-30 I30 K-15 F0.2 S180
16.	Read and explain the following program section!

N5	G90	G96	T0101	S100	M3	M8
N10	G0	X133	Z2			
N20	G1	Z-395	F0.3			
N30	G0	X135	Z2			
N40		X123				
N50	G1	Z-269.8				
N60	G2	X133	Z-274.8	I133	K-269.8	O70
N70	G0	Z2				

MATHEMATISCH TECHNISCHE
SOFTWARE-ENTWICKLUNG GMBH

CNC-Turning - Excerpt

MTS TeachWare Student's Book

MTS Mathematisch Technische Software-Entwicklung GmbH • Kaiserin-Augusta-Allee 101 • D-10553 Berlin
Phone: +49 / 30 / 349 960 0 • Fax: +49 / 30 / 347 960 25 • World Wide Web: http://www.mts-cnc.com • email: mts@mts-cnc.com

1.1.1 CNC turning machine

The CNC Turning Simulator simulates a 2-axis turning machine. In the CNC simulation all positioning and feed movements appear to be made by the tool carrier, so the chuck and the work part have a fixed position and the tool moves in both coordinates.

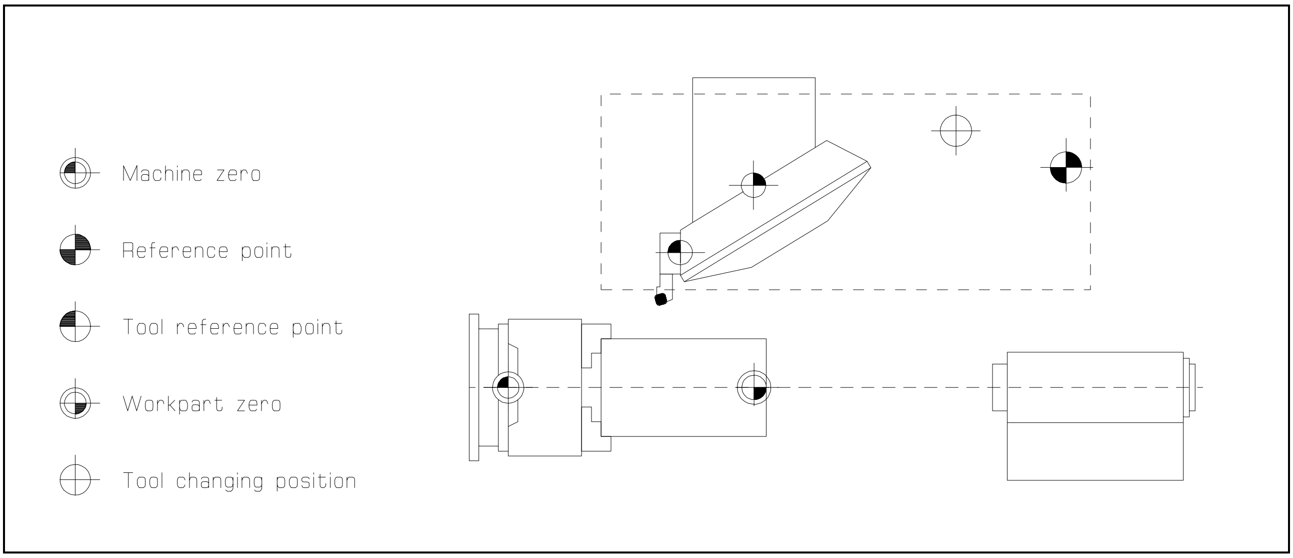


Figure 3
Schematic of the machine configuration

The work part can be clamped by using:·

- lathe chuck with step jaws,
- collet chuck,
- collet,
- face driver·or
- lathe centres.

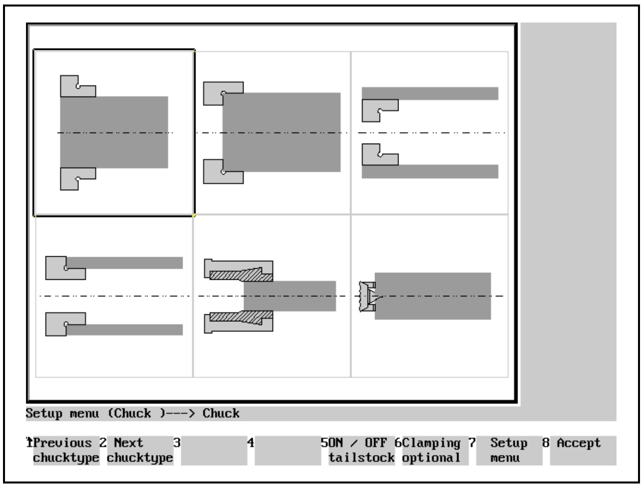


Figure 4
CNC Turning,workpart and clamping definition;"Clamping Fixture Selection" menu.

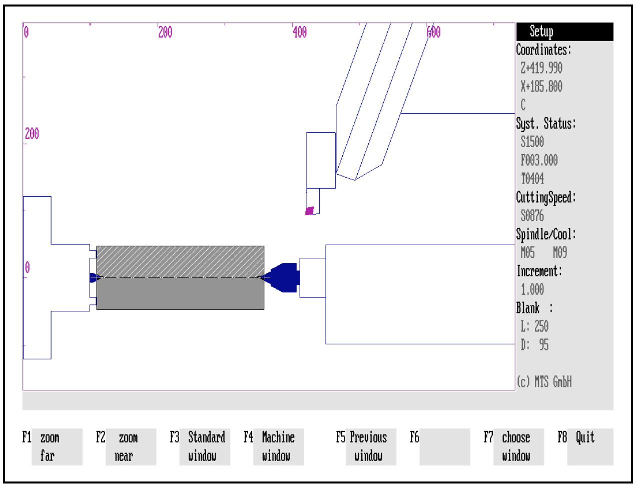
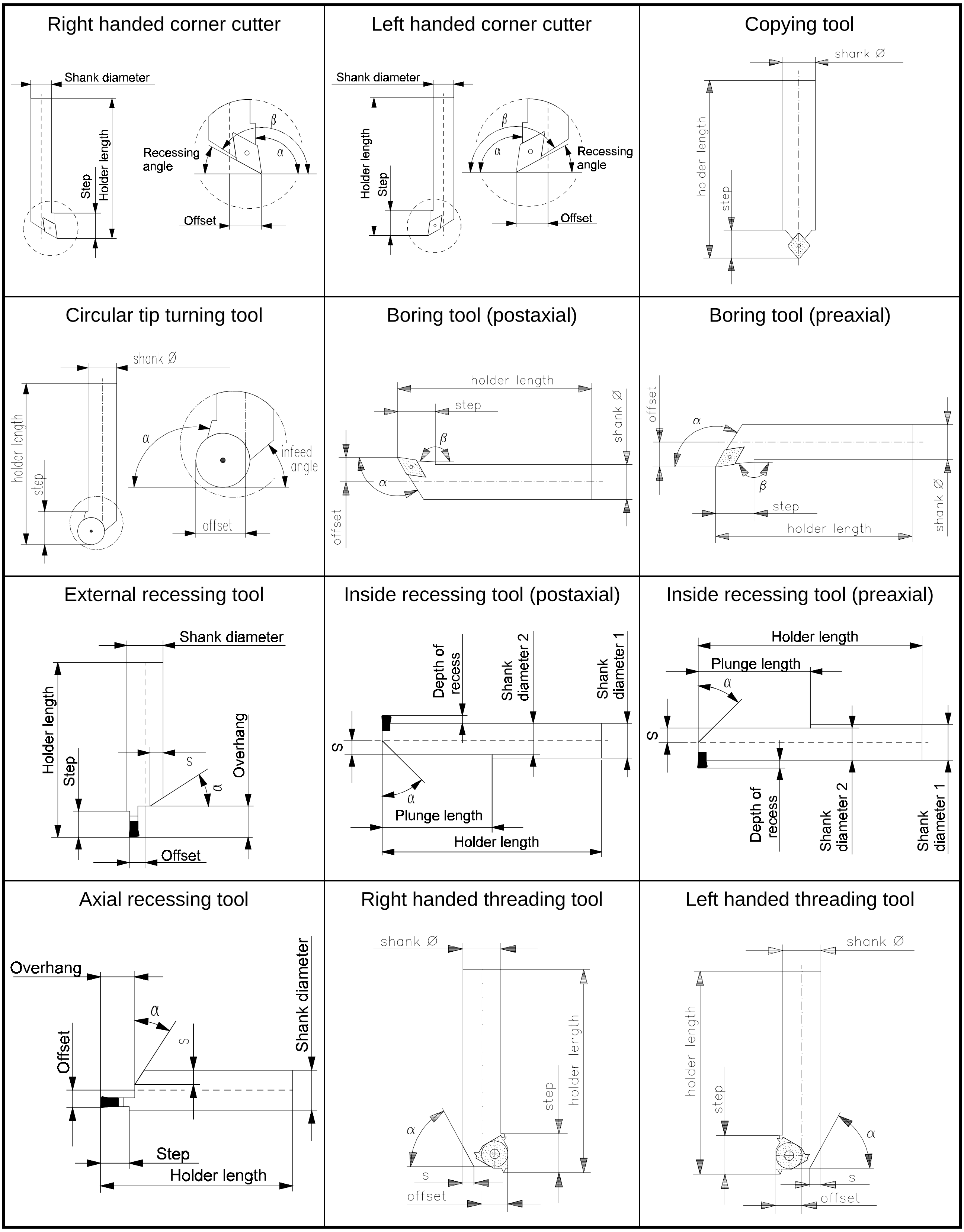


Figure 5
CNC Turning, clamping between centers.

The magazine holds may up to 99 tool positions (pockets) in which the tools are inserted from the tool manager. In the actual configuration we use 12 tools.

The following tool types are available in the Tool Manager:



Available tools in the CNC-Simulator

1.3.4 Data management

The internal data management functions provide a convenient means for documenting and backing up all work results. These functions include:

- NC Program Manager;
- Tool Manager;
- Clamping Fixture Manager;
- Saving created work parts;
- Saving current editing progress;
- Generating various set-up sheets and
- Managing configuration files.

Example: The CNC Simulator has its own tool management function. The program provides almost all ISO tool types and tools as standard options, and allows all common tools to be defined. Naturally, the tool management includes options for editing the available tool files, i.e. modification of existing tools and deletion of those no longer required.

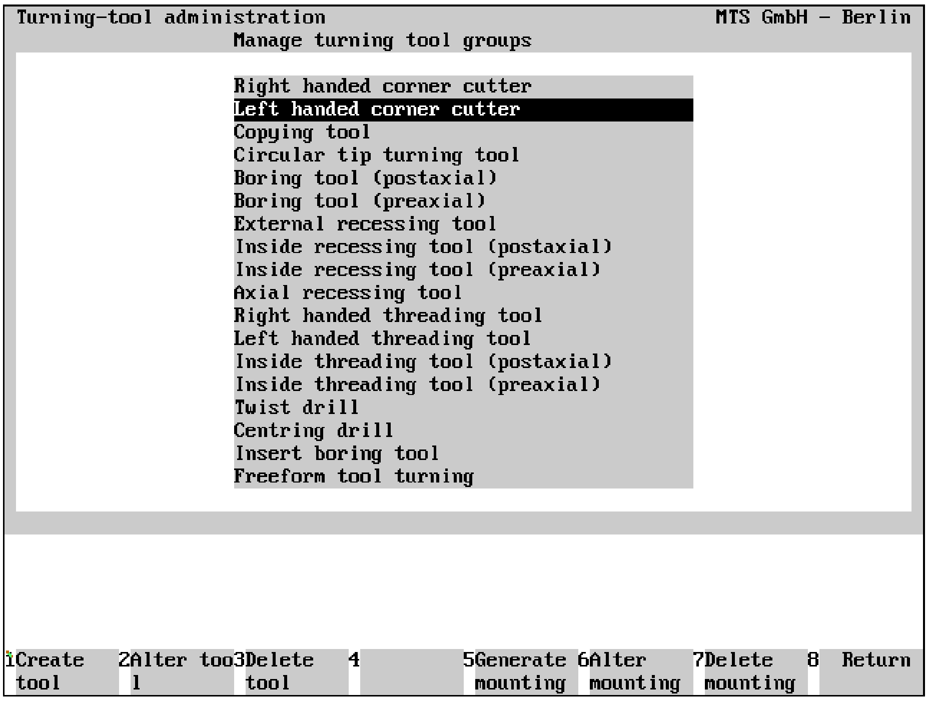


Figure 19
CNC Turning, Define/Delete Tools; Main Menu.

The screen layout of the Define/Delete Tools main menu is divided into two sections: the upper screen area contains a listing of all available tool types; the field currently in use is highlighted in color. As usual, further steps for specifying or editing tool data are indicated on the function keys at the bottom of the screen.

Select the desired step only by pressing the function keys rather than with the mouse.

↑	or	↓	Use the cursor keys ↑ or ↓ to select the tool type.
F1	or	F5	Create Tool/Tool Adapter: To generate a new tool of the current tool type, select F1; to define a new tool mounting, use F5.
F8	or	ESC	Return: Use F8 or ESC to conclude the current operation

Having started in the main menu by selecting the tool type, and subsequently selecting the Create Tool function by pressing F1 , the Data Entry menu for defining the tool is loaded.

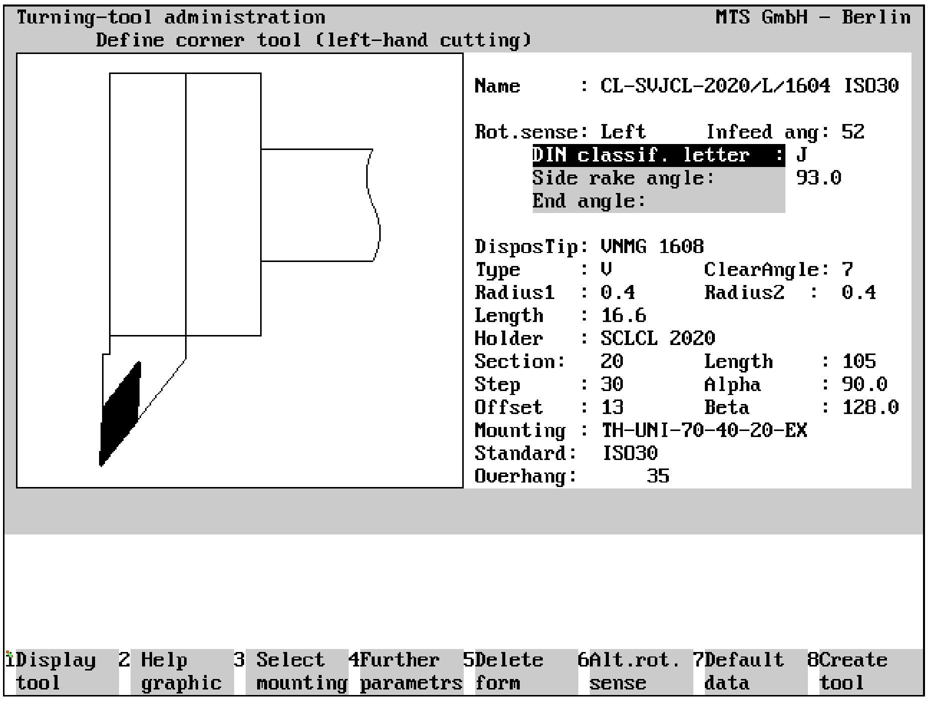


Figure 20
CNC Turning, Define/Delete Tools; defining a left-hand corner cutter.

The screen layout of the Data Entry menu is divided into three areas: the window on the left contains either a help graphic or a graphic corresponding to the data of the tool being defined (including the tool adapter). The input fields for the complete data record are located on the right.

You define a tool by manually entering the geometrical data, as well as the tool name and rotation direction. The desired tool adapter data can be automatically copied by selecting the Select Tool Mounting function. To save time, it is reasonable to define a new tool by first copying the data record of a similar tool, and then to modify the data to meet your requirements.

⇤⇥	Use the key ⇤⇥ to move from input field to input field.
→ or ←	Use the cursor keys → or ← to move the cursor within the input field.
INS or DEL	Use the key INS to insert a character, and the key DEL to delete one.
↵	If you confirm the entry in the input field with the ↵ key, the cursor moves automatically to the next input field.

[Tool Name]	Enter the tool name or number in this input field.
[Parameter]	The entries required for a tool depend on the tool type. Use the help graphics to obtain information on the parameters.
F8	Create tool: When the data entry for all tool and tool adapter parameters has been completed, you save the tool under a certain name by pressing F8 .
ESC	Use ESC to conclude the operation, and to return to the Define/Delete Tools main menu.

1.4 Special functions of the software

The CNC Simulator incorporates some special functions which effectively support processing and NC programming:

- 3D representation
- Programming aids for ISO commands
- Setting-up automatics, set-up sheet
- Status management

1.4.1 3D representation

A function supporting CNC training is given by the option to display, at any time, 3D Views of the work part, seen from different viewing angles. The program features 3D displays in Turning Simulators. To display machining inside the work part, any work part can be cut out.

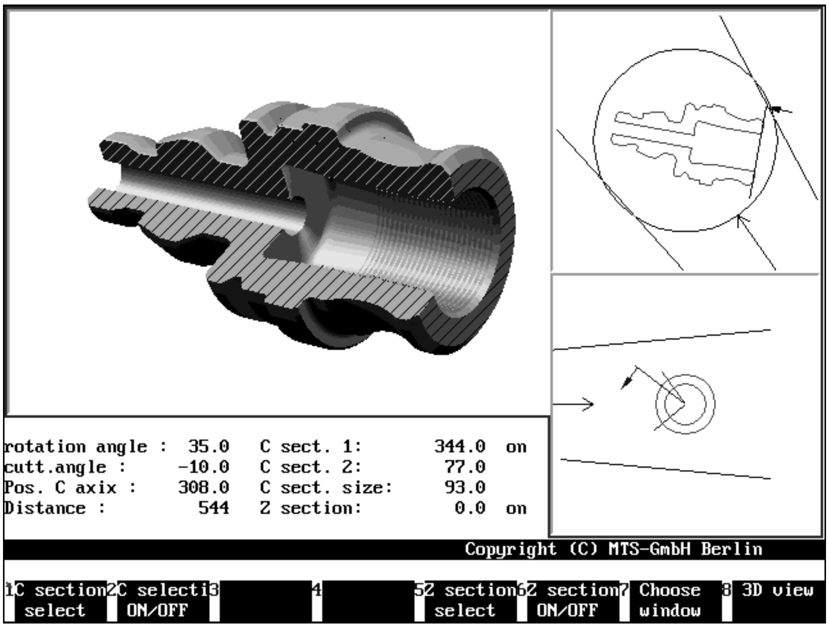


Figure 21
CNC Turning, 3D View

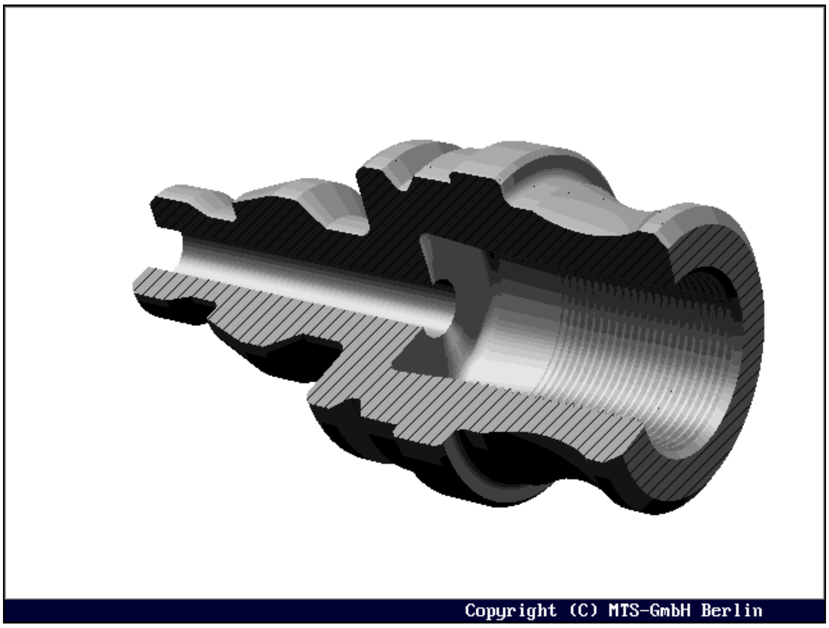


Figure 22
CNC Turning, 3D Display, full part with intersections

1.4.3 Setting-up automatics, set-up sheet

A Set-up Sheet contains all the information needed to set-up the machine by the operator. This sheet is used by the MTS-Software for an automatic set-up of the simulated machine tool when starting an NC program. This information includes:

- blank/work part geometry
- clamping fixture and method
- tool in working position and magazine configuration
- offset values of the tools used

A Set-up Sheet can be created for every current machine tool situation. It is prefixed to the NC program for which the set-up sheet was created. During the NC program load in Automatic Mode or for interactive programming the CNC Simulator is set-up automatically with the Setup Sheet Interpreter according to the stored information, but the Set-up Sheet Interpreter must be active.

To have a machine tool status loaded automatically during the CNC Simulator start, you can specify the Set-up Sheet describing that status in the configuration.

F4	Automatic Setup: this function is activated by pressing the function key F4 from the main menu. The CNC Simulator is then set-up automatically.

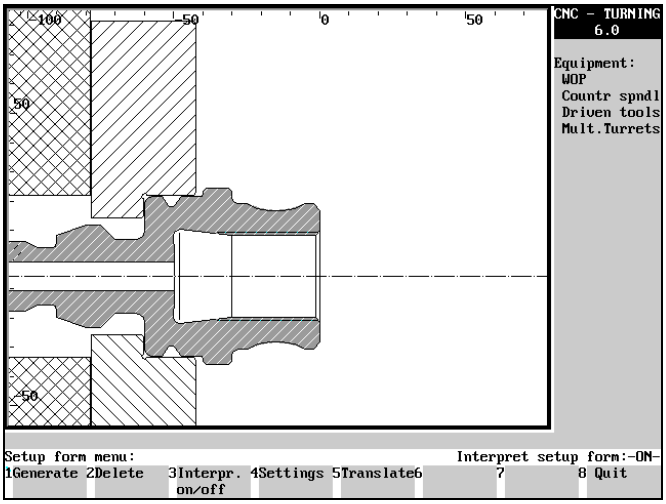
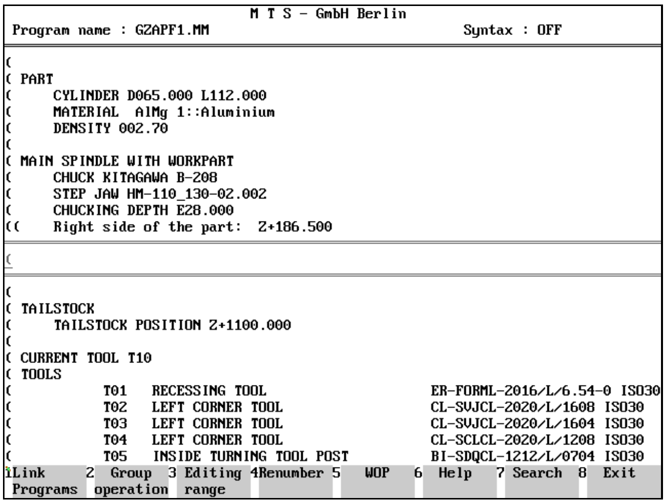


Figure 24
CNC Turning, Set-up Sheet menu


M T S - GmbH Berlin
Program name : GZAPF1.MM Syntax : OFF

(
(PART
(CYLINDER D065.000 L112.000
(MATERIAL AlMg 1::Aluminium
(DENSITY 002.70
(
(MAIN SPINDLE WITH WORKPART
(CHUCK KITAGAWA B-208
(STEP JAW HM-110_130-02.002
(CHUCKING DEPTH E28.000
((Right side of the part: Z+186.500

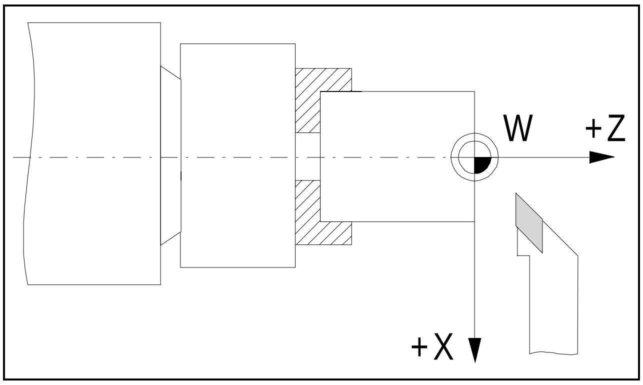
(

(
(TAILSTOCK
(TAILSTOCK POSITION Z+1100.000
(
(CURRENT TOOL T10
(TOOLS
(T01 RECESSING TOOL ER-FORML-2016/L/6.54-0 ISO30
(T02 LEFT CORNER TOOL CL-SVJCL-2020/L/1608 ISO30
(T03 LEFT CORNER TOOL CL-SVJCL-2020/L/1604 ISO30
(T04 LEFT CORNER TOOL CL-SCLCL-2020/L/1208 ISO30
(T05 INSIDE TURNING TOOL POST BI-SDQCL-1212/L/0704 ISO30
1Link Programs 2 Group operation 3 Editing range 4Renumber 5 WOP 6 Help 7 Search 8 Exit

Figure 25
CNC Turning, example of a Set-up Sheet (excerpt)

2.3 Specifying the necessary location of the work part zero point

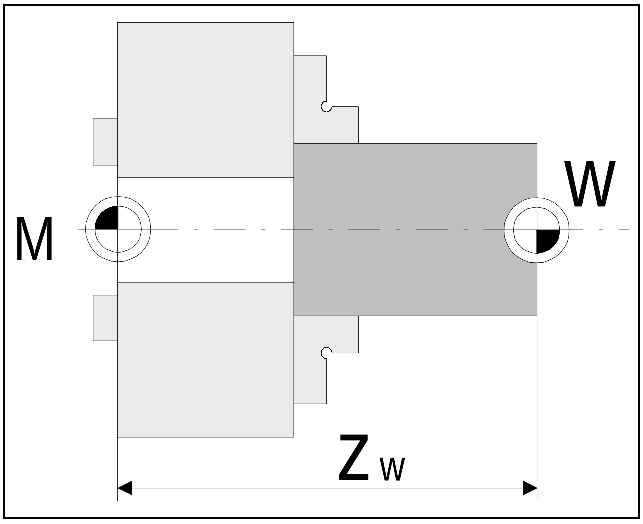
The work part zero point W is the origin of the work part-referenced coordinate system. Its location is specified by the programmer according to practical criteria. The ideal location of the work part zero point allows the programmer to take the dimensions directly from the drawing.



Work part zero point

For practical reasons the work part zero point W is selected in turning in the right-hand plane surface and in the rotation axis.

The work part zero point is set with reference to the machine zero point M or to the predefined work part zero point by setting the system variables.



Using the operation functions described below the distance in the Z-direction between the machine zero point M and the work part zero point W is specified.

This value z_W, also called the zero point shift, is then entered into the CNC control.

Procedure

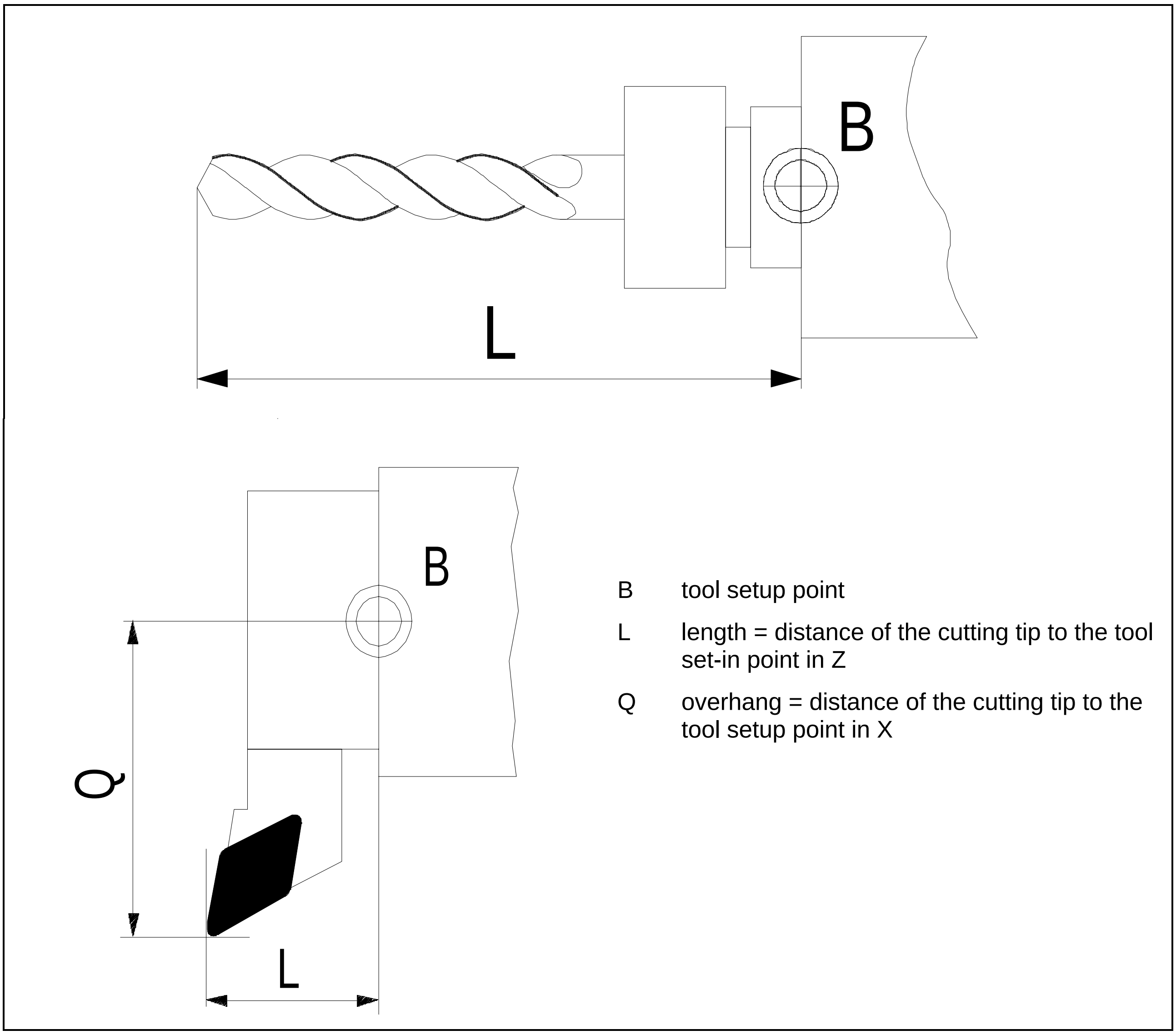
Starting situation: All machining tools have been measured and are available on the turret head.
The clamping device is prepared and the work part has been correctly clamped.

1. Switch on the spindle (counterclockwise rotation).
2. Change the tool to set the work part zero point, i.e. rotate the turret head to the corresponding position, for instance T02.

   Note: The rotation area of the turret has to be checked first to avoid collision during rotation.
3. Touch the front plane area of the work part:
   move carefully with the tool using the hand wheel
   or using the corresponding arrow keys of the keyboard of the CNC control
   until the cutting edge reaches a marking on the work part.
4. Enter the desired plane area allowance (e.g. 0.5 mm) on the CNC control.
   Actuate with the zero key.
   (The dimensions are used to face the front surface in z=0)
5. The CNC control then stores the value of the zero point shift z_W.
   The work part zero point W is clearly specified since the X coordinate zero is located on the rotation axis.
6. Because of eventual allowance the front side needs to be faced. This needs to be considered when programming the NC program.

3.3 Tool Offset Compensation

Using the tool offset compensation values it is easy to program a work part without consideration of the actually applicable tool lengths or overhangs. The available work part drawing data can be directly used for programming. The tool data, lengths as well as overhangs of the turning machines are automatically considered by the CNC control.



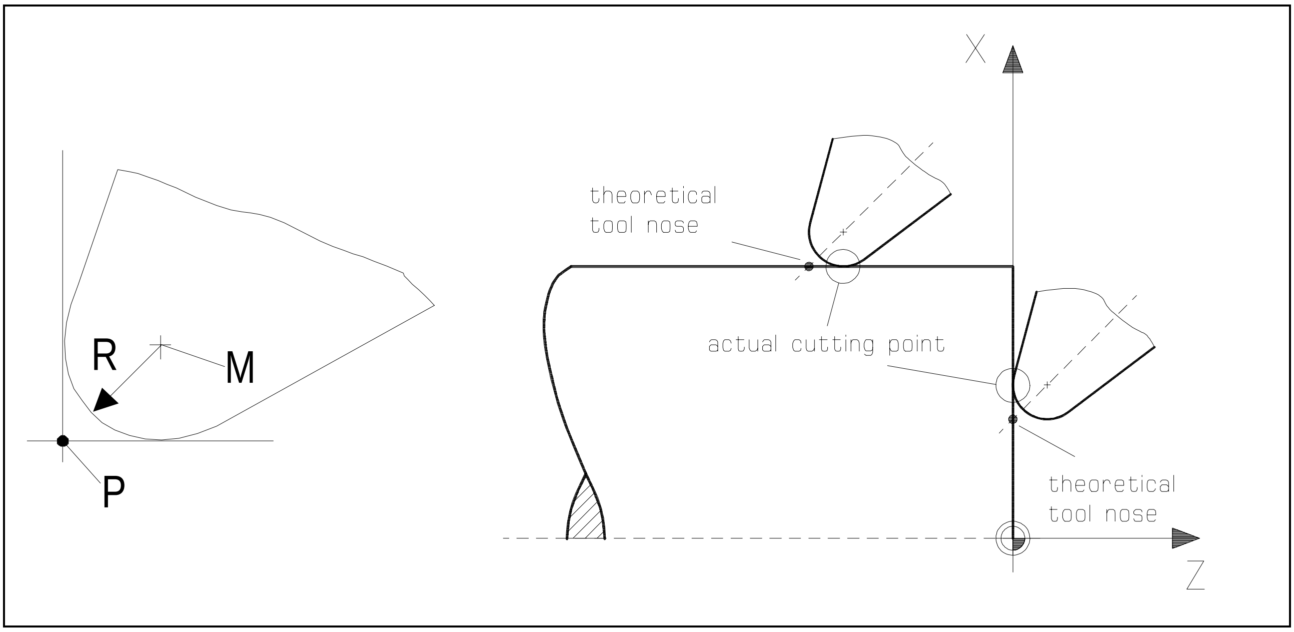
Tool offset compensation values

In computing the tool movements the control system relates all programmed coordinates to the tool setup point which is situated at the stop face of the tool mounting.

It follows that the distance between the theoretical cutting point of the tool nose and the tool setup point must be determined for every tool, so that the actual tool path can be computed. Each of these differential values is stored as a tool offset compensation value in a corresponding compensation value storage. When a programmed tool change is to be executed in the course within NC program, the system reads in the applicable compensation value storage, to account for the tool geometry in computing the tool path.

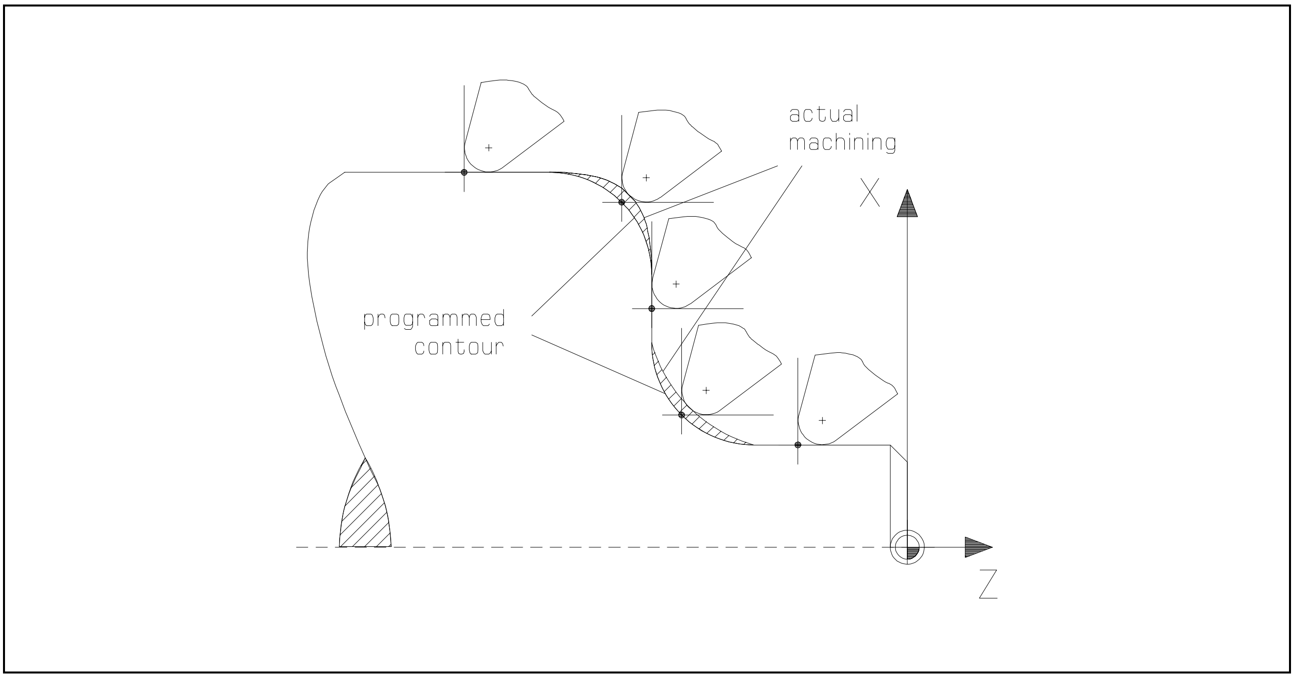
3.4 Tool Nose Compensation

The actual cutting point of the reversible tip changes during the course of machining, according to the tool movement direction.



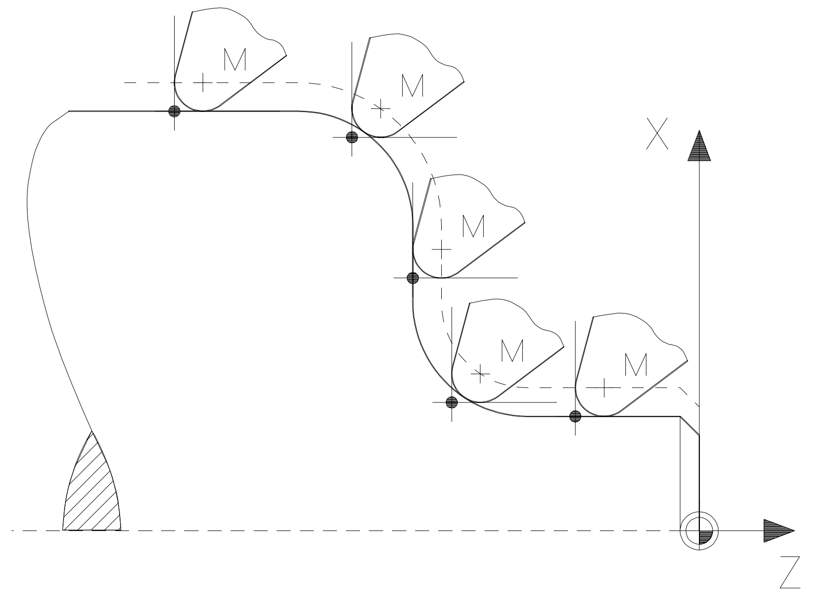
P Theoretical tool nose M Tool nose Centre

In computing the tool motion the control system assumes the movement of the theoretical cutting point of the tool nose along the programmed contour. Every time the tool executes a programmed movement which is not parallel to either the X- or Z-axis, deviations from the desired contour and the corresponding dimensions are unavoidable, due to the radius of the tool tip employed.



If tool nose compensation is not selected, the actual machining will deviate from the programmed contour on the rising and falling segments of a contour, due to the radius of the tool tip.

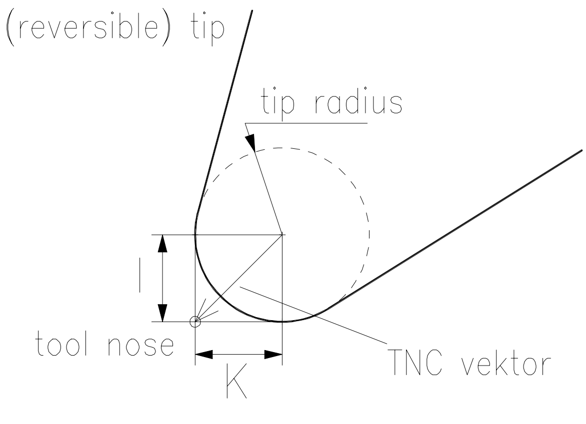
When tool nose compensation is activated, the control system computes the path of the centre of the tool nose, equidistant to the contour, accounting for the radius.



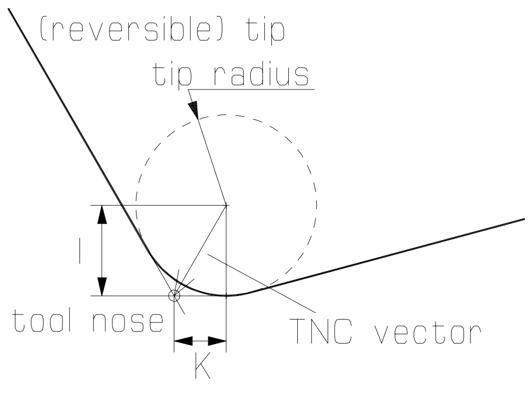
- - - Offset Path M Tool Nose Centre

If the tool nose compensation is selected the system computes the motion of the tool nose centre on an offset path equidistant to the contour, i.e. the actual cutting point moves exactly along the programmed contour of the workpiece.

With each tool the theoretical cutting point of the tool nose must be defined by the tool nose compensation vector to make sure that the control system can compute the path of the actual cutting point in the execution of a cycle. The tool nose compensation vector defines the theoretical position of the tool nose (in the directions X and Z) relative to its centre.

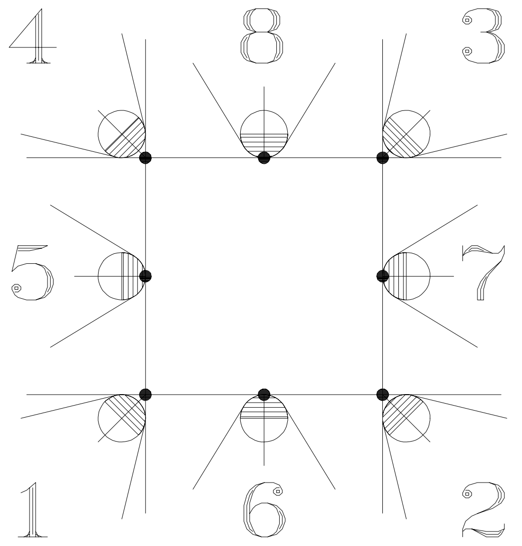


Example: Radius 0,4
I =-0,400
K=-0,400

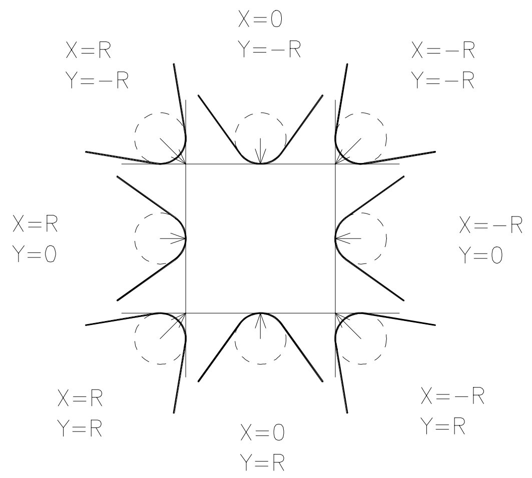


Example: Radius 0,4
I =-0,400
K=-0,231

Alternatively the tool nose compensation vector can be determined by eight tooling quadrants. This is common practice and applicable to standard cases.



tooling quadrants



tool nose compensation vectors

The tool management (see Simulator Operation Manual) predefines a TNC vector for every tool available in the Simulator system.

3.4.1 Selection of Tool Compensation Values T

For programming with tool offset and tool nose compensation it is necessary to select the tool compensation values of the actual tool by using the T command.

Command: **T**

Tool selection command

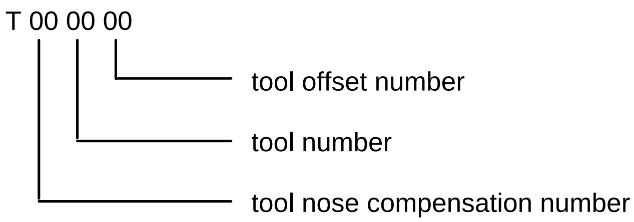
Function: Select the tool on the specified turret position with or without the tool nose compensation.

NC-Block: ... T00 00 00...

Depending on the quantity of the subsequent digits the tool nose compensation is activated or not.

4 digit command	T00 00	without the tool nose compensation
6 digit command	T00 00 00	with the tool nose compensation

The digits describe the number of the tool and the number of the compensation storage.

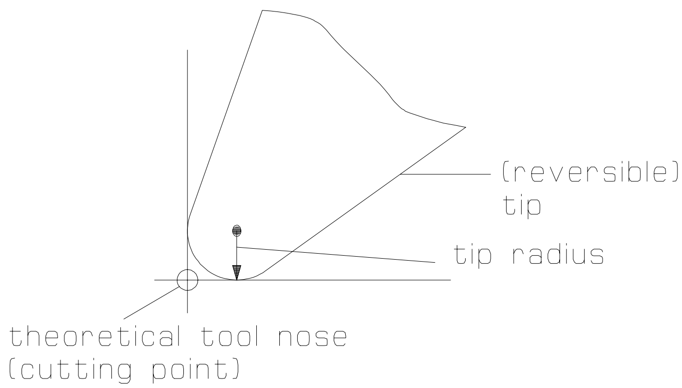


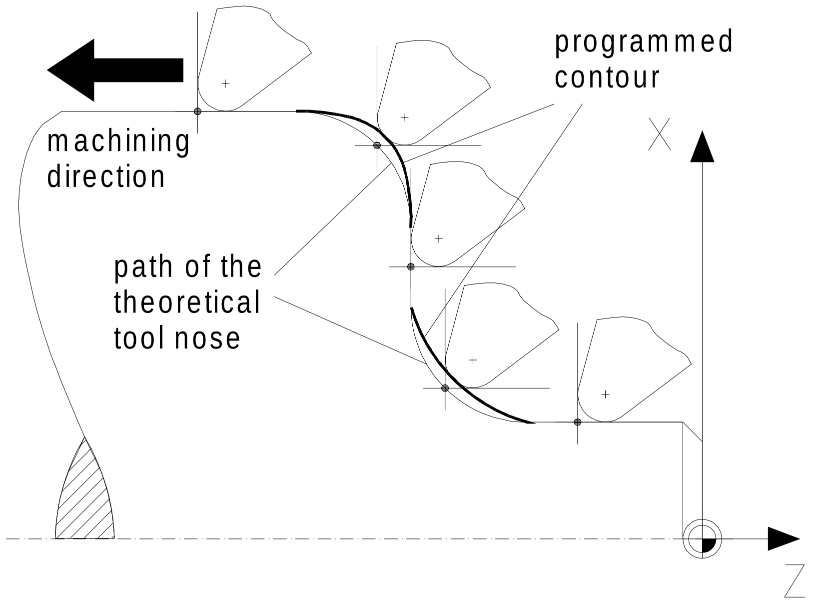
3.4.4 Tool Nose Compensation Right G42

Command:	**G42**
	Compensation to the right of the contour (in the cutting direction)
Function:	When the tool nose compensation is operative, only the work part contour points are programmed and the control system must be informed whether the tool shall move left or right of the programmed contour. The choiceof left or right applies to the direction in which the tool travels along the contour
NC-Block:	G42 G01 [X...] [Z...] [F...]
Optional Addresses:	X X-Coordinate of the Target Point
	Z Z-Coordinate of the Target Point
	F Feedrate
Note:	The command of the NC-block specifying G42 should be G00 or G01. When G42 is specified by the commands G02 or G03 an alarm message is displayed. For using the tool nose compensation the actual tool must be selected with the 6 digit Tool command

Programming Example:

```
N25 T030303
...
N100 G42
N105 (contour description)
...
N170 G40
```





3.7 Thread Cutting G33

Command: **G33**

Thread cutting

Function: The G33 cycle serves to program thread cutting parallel to the Z-axis.

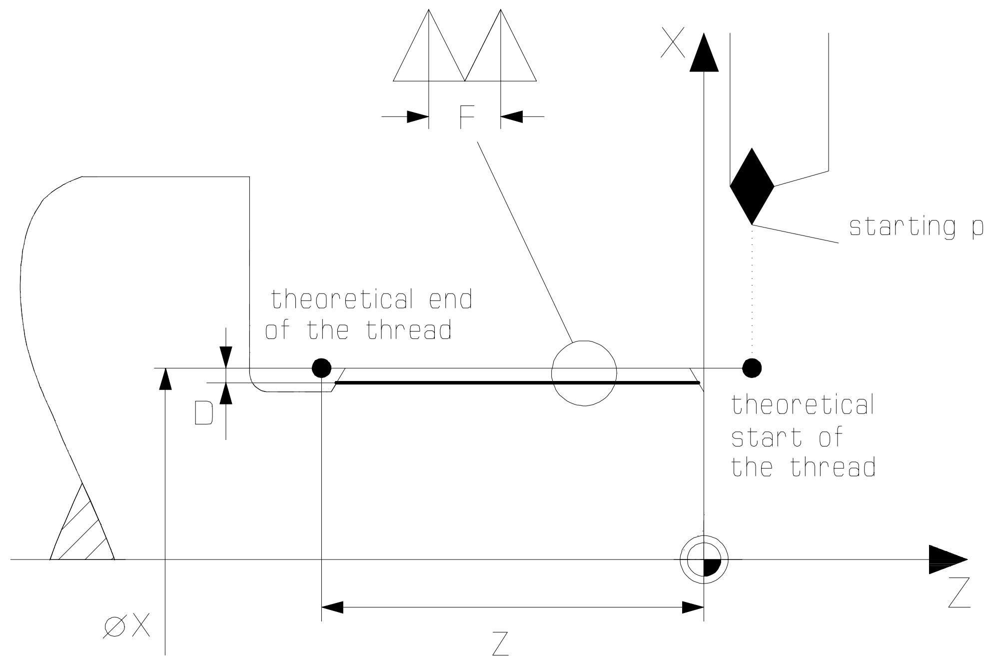
NC-Block: G33 [X...] [Z...] [F...]

Optional Addresses:

X diameter of each thread cutting cycle

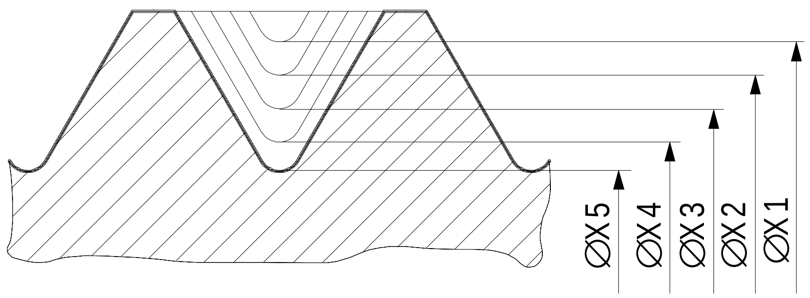
Z end point of thread in longitudinal direction

F thread lead



X nominal diameter of the thread F thread lead

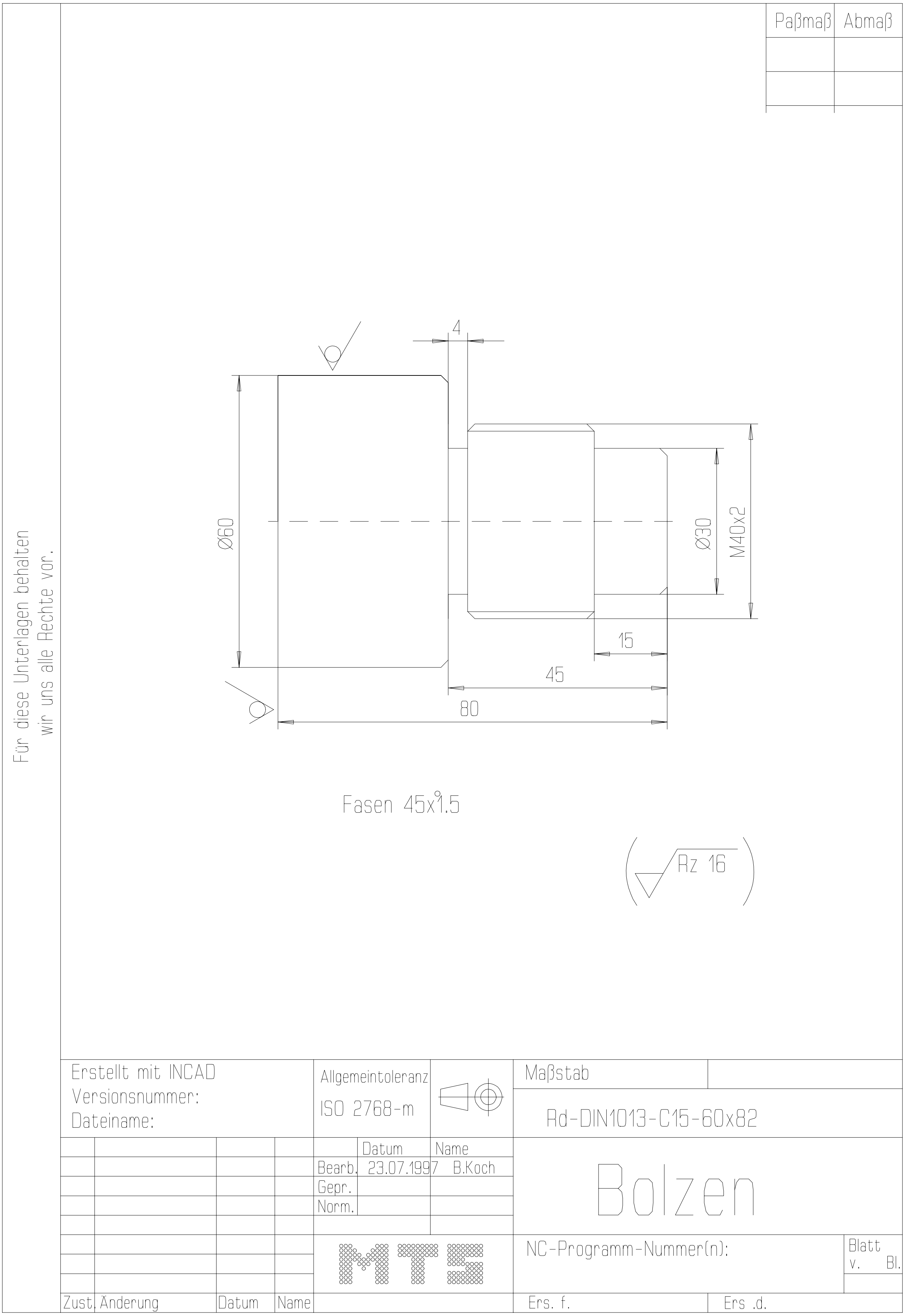
D depth of the thread relative to the radius Z Z-Coordinate of the theoretical end point of the thread



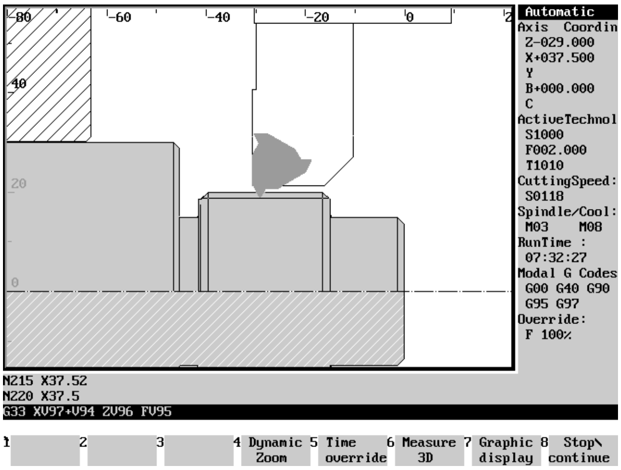
Note: The successive infeeds must be programmed seperately by using the different diameter X1, X2, X3 and so on.

Never change the spindle speed during the thread cutting cycle.

Example:
Create an NC-program for the following bolt with a thread.

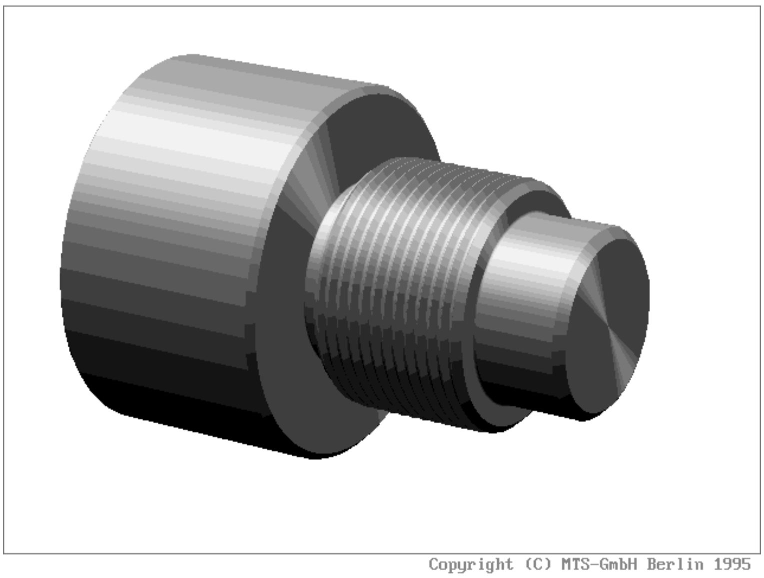


N120	G0 X500 Z500 M9
N125	G96 S280 T020202 M3 M42 M63
N130	G50 S4000
N135	G87 NLAP1 U0 W0
N140	G0 X500 Z500 M9
N145	G96 S100 T080808 M3 M42 M63
N150	G50 S1500
N155	G0 X70 Z-45. M8
N160	G73 X30 Z-41. I4 K2.5 E0.5 D4 F0.12 T11
N165	G0 X62
N170	X500 Z500 M9 M5
N175	T101010 M3 M42 M63 G97 S1000
N180	G0 X50 Z5
N185	G33 X38.5 Z-42 L1 F2
N190	X38
N195	X37.8
N200	X37.7
N205	X37.6
N210	X37.55
N215	X37.52

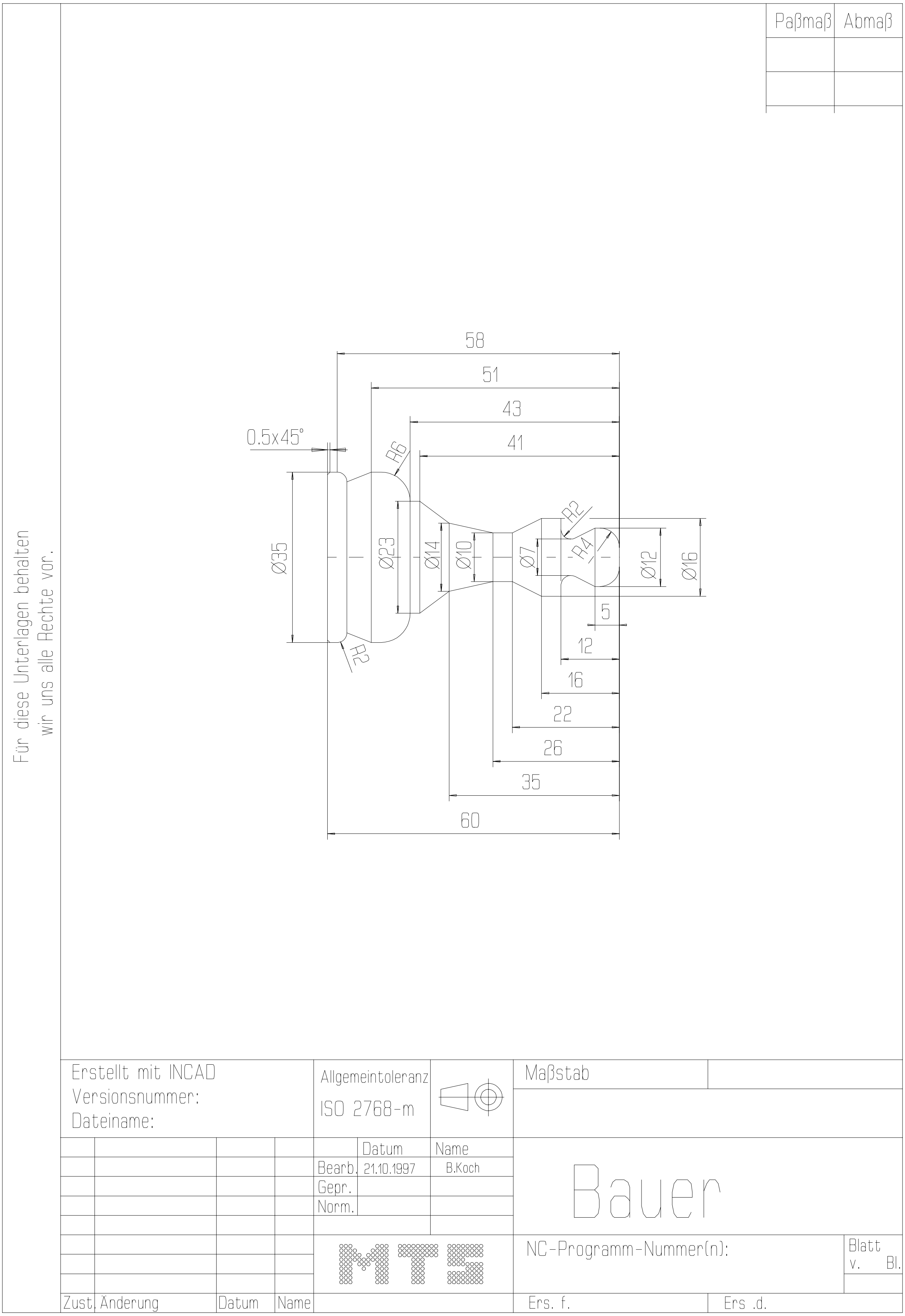


N220	X37.5
N225	G0 X500 Z500 M5 M9
N230	M2

Finished part:



Example for using subprograms:
NC-program for the following chess figure with a subprogram by using absolute input values.



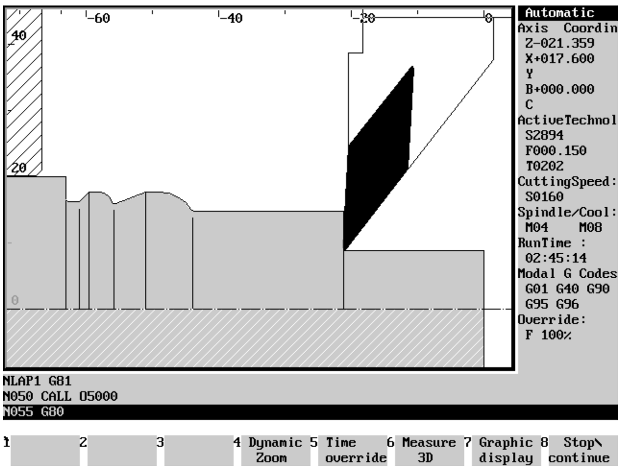
Programming Example

Subsequently, the program sequence with the subprogram call is shown.

	main program baucall
	$G54 Z177
	O10022
N010	G0 X500 Z500 T020202 M3 M42 M63
N015	G96 S160
N020	G50 S3000
N025	G0 X42 Z0
N030	G1 X-1 F0.15 M8
N035	G0 X42 Z4
N040	G85 NLAP1 D6 U0.4 F0.25
	NLAP1 G81
N050	CALL O5000 →

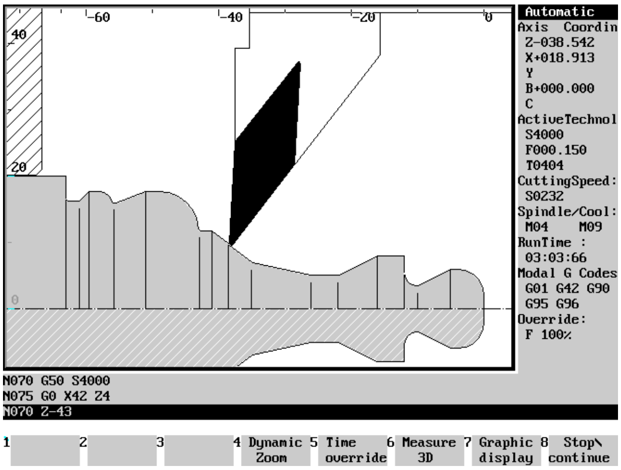
	subprogram O5000.mm
N010	G0 G42 X0 Z2
N015	G1 X0 Z0
N020	G76 X12 L-4
N025	G1 Z-5
N030	X7 Z-10
N035	G2 X11 Z-12 L2
N040	G1 X16
N045	Z-16
N050	X10 Z-22
N055	Z-26
N060	X14 Z-35
N065	X23 Z-41
N070	Z-43
N075	G3 X35 Z-49 L6
N080	G1 Z-51
N085	X31 Z-56
N090	G3 X35 Z-58 L2
N095	G1 Z-59.5
N100	X32 Z-61
N105	Z-63
N110	X42
N115	G40
N120	RTS

↓	
N055	G80

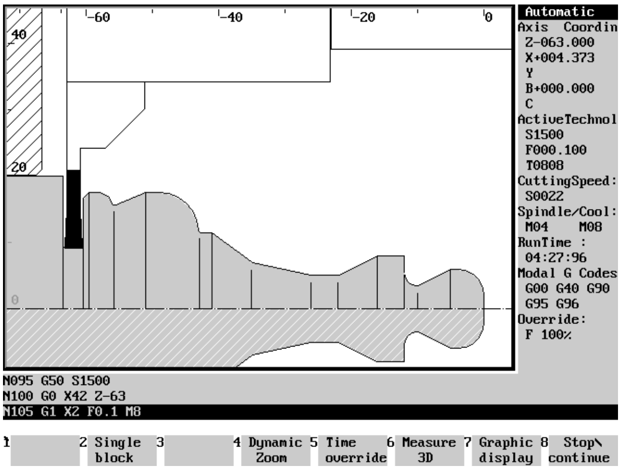


N060	G0 X500 Z500 M9

N065	G96 S280 T040404 M3 M42 M63
N070	G50 S4000
N075	G0 X42 Z4
N080	G87 NLAP1 U0 W0

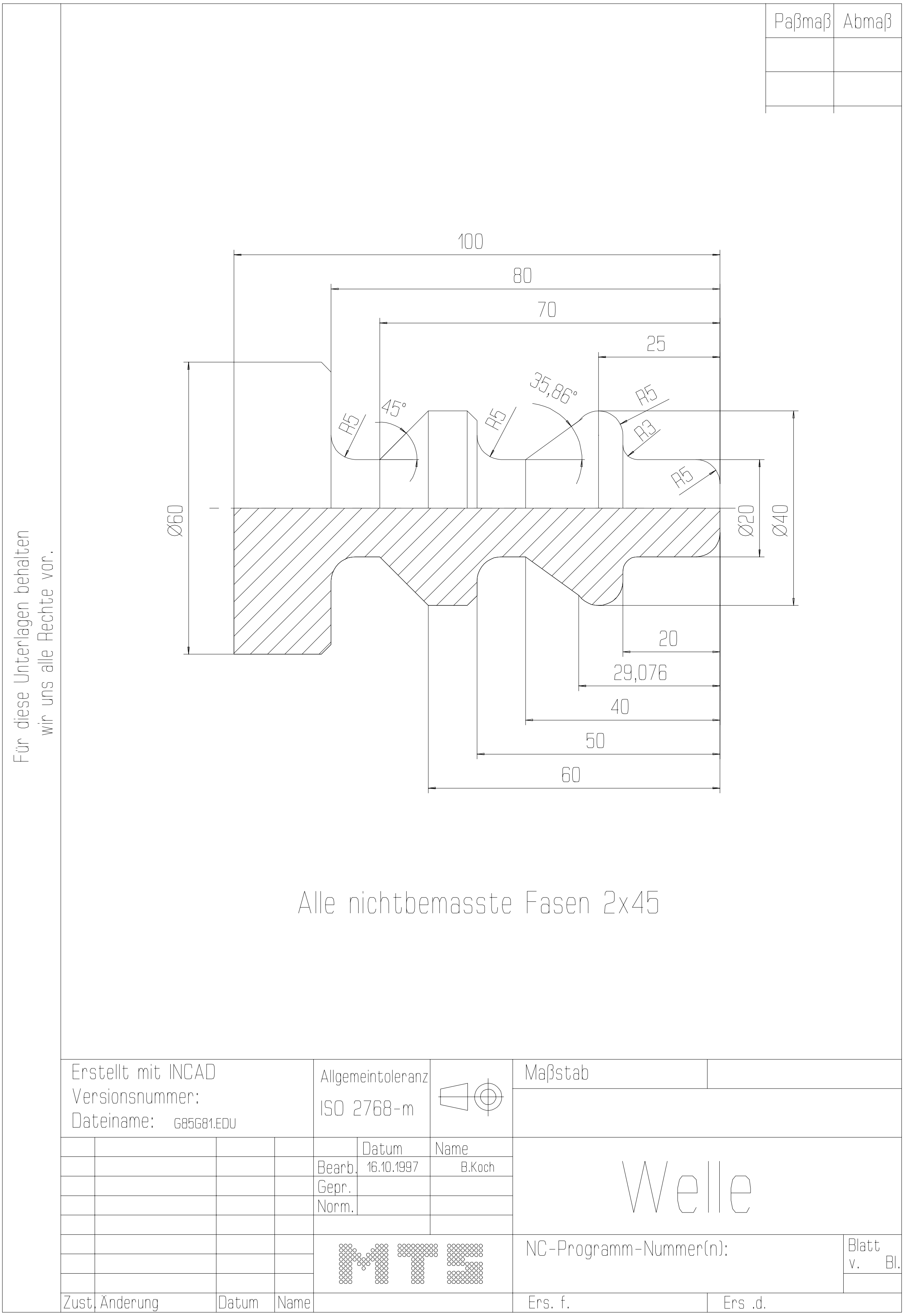


N085	G0 X500 Z500 M9
N090	G96 S100 T080808 M3 M42 M63
N095	G50 S1500
N100	G0 X42 Z-63
N105	G1 X2 F0.1 M8



N110	G0 X50
N115	X500 Z500 M9
N120	M30

Example:
Create an NC-program for the following figure with the G86 longitudinal LAP-function by using the pre-fabricated blank.



MAIN SPINDLE WITH WORKPART		
	CHUCK KITAGAWA B-208	
	STEP JAW WM-KIT_01.002	
	TYPE OF CHUCK EXTERNAL CHUCK OUTSIDE STEP JAW	
	CHUCKING DEPTH E18.000	
	RIGHT SIDE OF THE PART: Z+196.000	

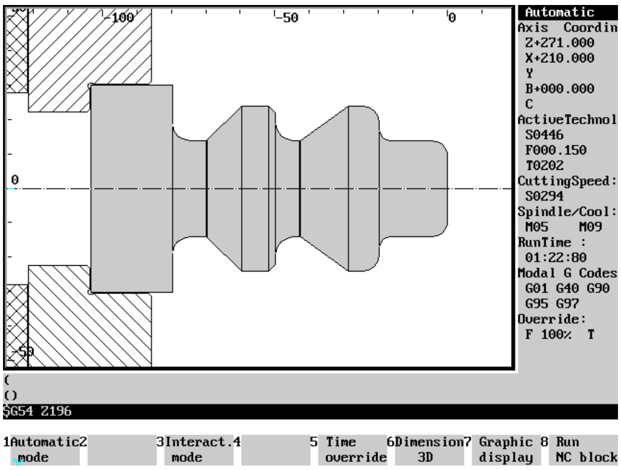
TAILSTOCK	
	TAILSTOCK POSITION Z+1100.000

CURRENT TOOL	
	T02

TOOLS		
T01	LEFT CORNER TOOL	CL-SVJCR-2020/R/1604 ISO30
T02	LEFT CORNER TOOL	CL-SVJCR-2020/R/1604 ISO30
T03	FRONT GROOVING TOOL	RA-MBS-E5N-2.5/16/040-050/R ISO30
T04	LEFT CORNER TOOL	CL-SVJCR-2020/R/1604 ISO30
T05	INSIDE TURNING TOOL POST	BI-SDQCL-1212/L/0704 ISO30
T06	INTERN. THREADING TOOL POSTAX	TI-ITTR-2016/R/60/1.50 ISO30
T07	TWIST DRILL	DR-14.00/108/R/HSS ISO30
T08	RECESSING TOOL	ER-SGTFR-2012/R/03.0-0 ISO30
T09	CENTER DRILL	CD-04.00/056/R/HSS ISO30
T10	LEFT CORNER TOOL	CL-MVJCL-2020/L/1604 IS030
T11	INSIDE TURNING TOOL POST	BI-SDUCL-1212/L/0704 ISO30
T12	REVERSIBLE TIP DRL	DI-22.00/051/R/HMT ISO30
ACCURATE OFFSET		

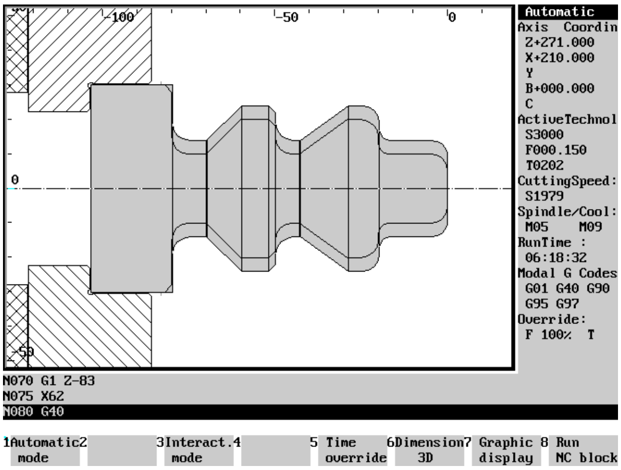
Solution:

NC-program	
	$G54 Z196
	O10086

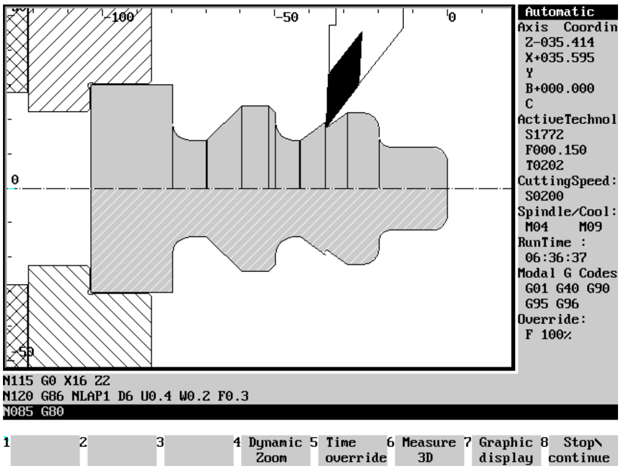


	NLAP1 G81
N005	G42 G0 X6 Z2
N010	G1 X8 Z0
N015	G76 X20 L-5
N020	G76 Z-20 L3
N025	G76 X40 L-5
N030	G3 Z-29.076 K0 I-5
N035	G1 X20 A215.86
N040	G76 Z-50 L5
N045	G75 X40 L-2

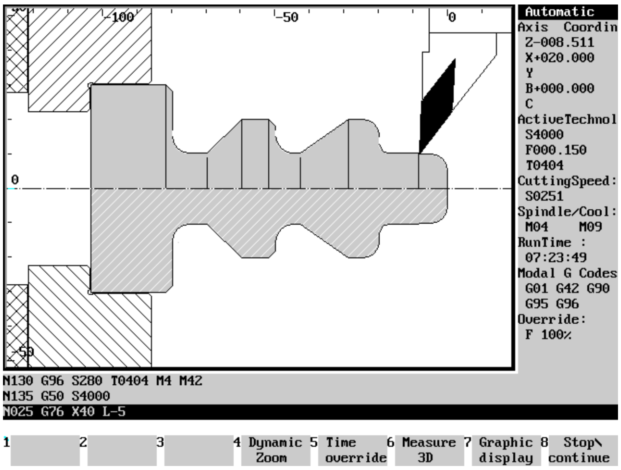
N050	G1 Z-60
N055	G1 X20 A225
N060	G76 Z-80 L5
N065	G75 X60 L-2
N070	G1 Z-83
N075	X62
N080	G40
N085	G80



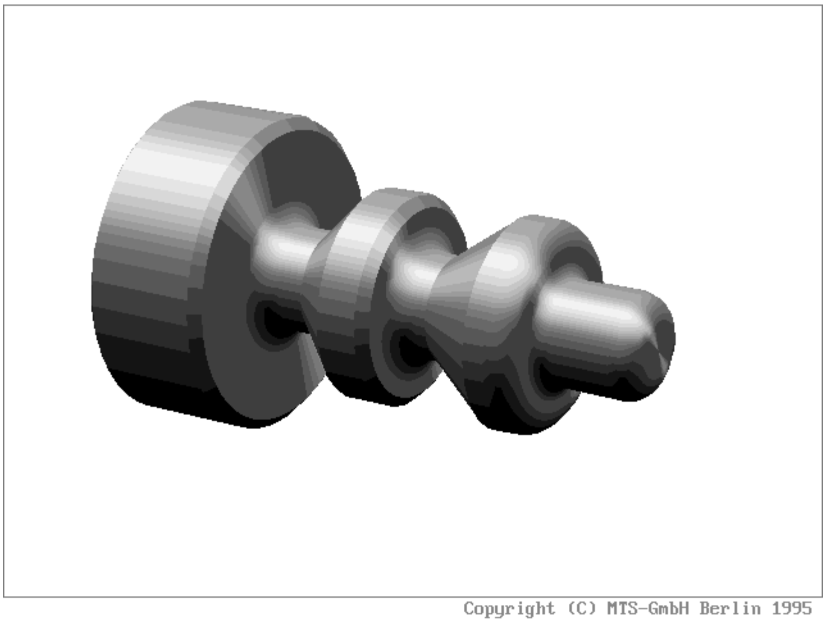
N090	G0 X500 Z500 T020202 M3 M42 M63
N095	G96 S200
N100	G50 S3000
N105	G0 X62 Z0
N110	G1 X-1 F0.15 M8
N115	G0 X16 Z2
N120	G86 NLAP1 D6 U0.4 W0.2 F0.3



N125	G0 X500 Z500 M9
N130	G96 S280 T040404 M3 M42 M63

N135	G50 S4000
N140	G87 NLAP1 U0 W0
	
N145	G0 X500 Z500 M5 M9
N150	M2

Finished part:



CNC-Milling - Excerpt

MTS TeachWare Student's Book

MTS Mathematisch Technische Software-Entwicklung GmbH • Kaiserin-Augusta-Allee 101 • D-10553 Berlin
Phone: +49 / 30 / 349 960 0 • Fax: +49 / 30 / 347 960 25 • World Wide Web: http://www.mts-cnc.com • email: mts@mts-cnc.com

1.1.1 CNC milling machine

The CNC Milling Simulator simulates a 3-axis milling machine with vertical spindle position. In the CNC simulation all positioning and feed movements appear to be made by the tool carrier, so the machine table and the work part have a fixed position and the tool moves in all three coordinates.

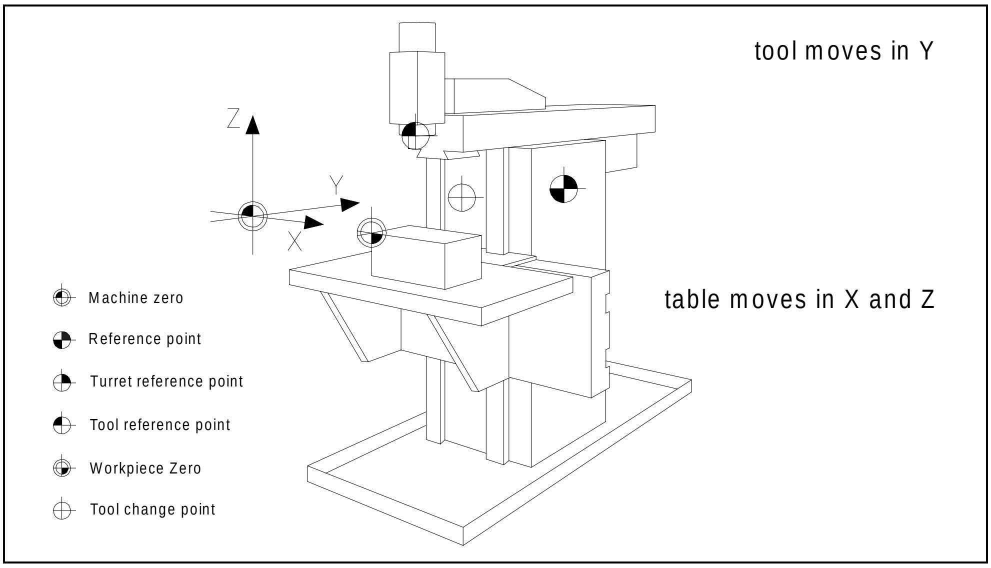


Figure 3
Schematic of the machine configuration

In the MAKINO CNC Milling machine the tool moves in Y- and Z-direction and the machine table moves in X-direction.

The work part can be clamped by using:·

- jaws,
- magnetic plate·or
- modular clamping.

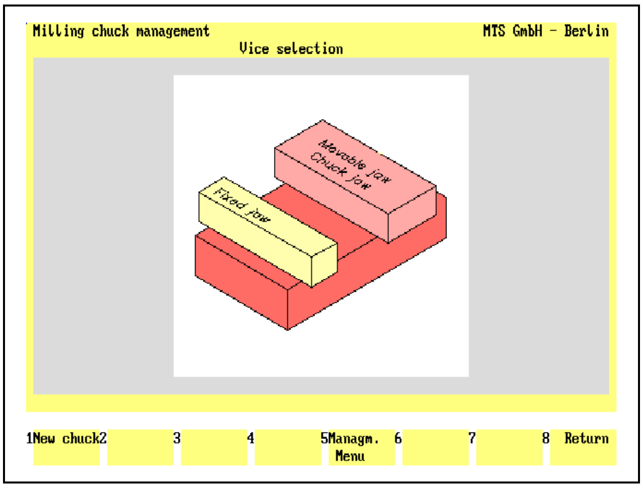


Figure 4
jaws

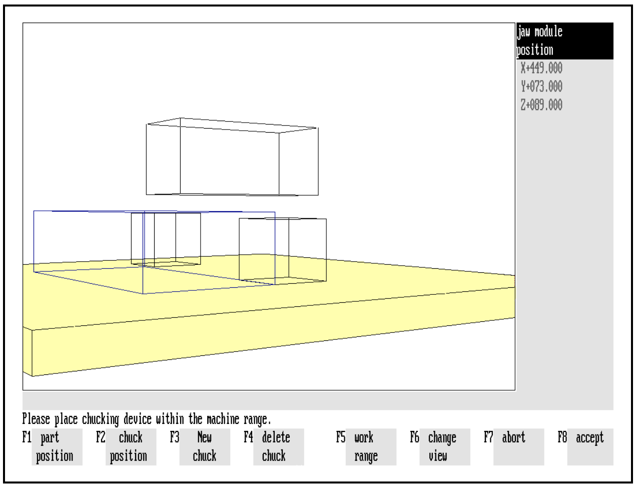
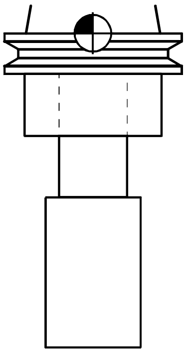
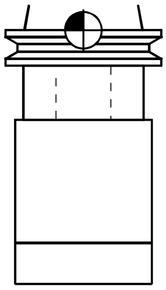
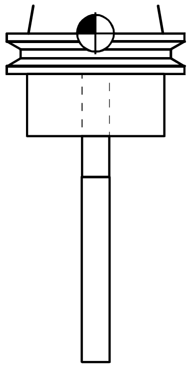
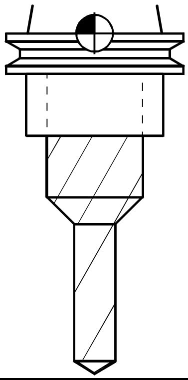
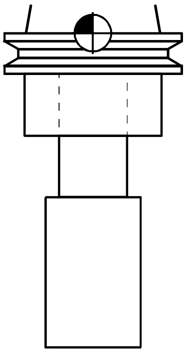
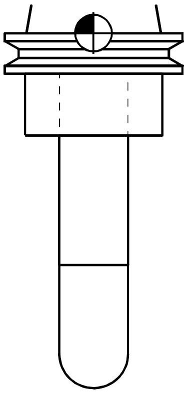
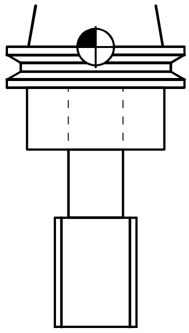
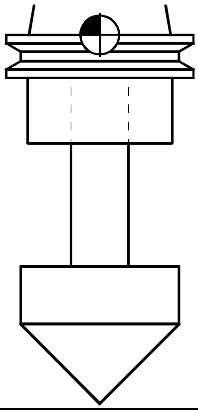
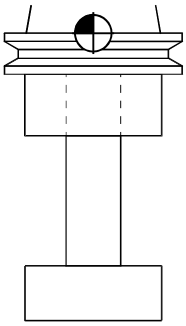
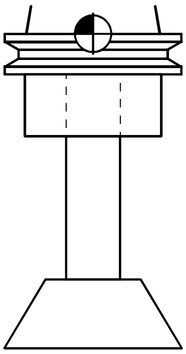
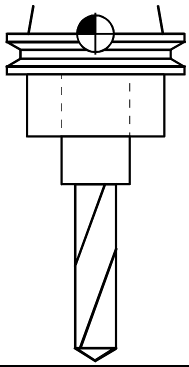
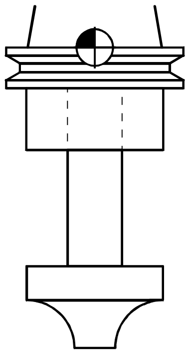
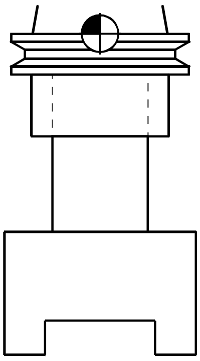
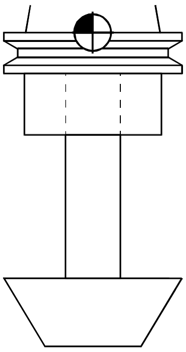
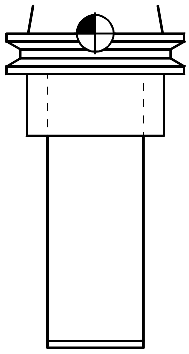
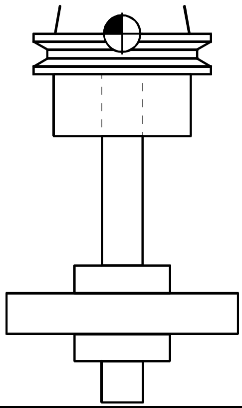


Figure 5
modular clamping

The magazine holds may up to 99 tool positions (pockets) in which the tools are inserted from the tool manager. In the actual configuration we use 16 tools.

The following tool types are available in the Tool Manager:

End mills 	Face milling cutters 	Reamers 	Step drills 
Slot milling tools 	Radius cutters 	Taps 	Core drills 
T-slot cutters 	Corner tool (Type A) 	Drills 	Concave type cutters 
Shell end mills 	Corner tool (Type B) 	Insert tip drills 	Side milling tools 

1.3.4 Data management

The internal data management functions provide a convenient means for documenting and backing up all work results. These functions include:

- NC Program Manager;
- Tool Manager;
- Clamping Fixture Manager;
- Saving created work parts;
- Saving current editing progress;
- Generating various set-up sheets and
- Managing configuration files.

Example: The CNC Simulator has its own tool management function. The program provides almost all ISO tool types and tools as standard options, and allows all common tools to be defined. Naturally, the tool management includes options for editing the available tool files, i.e. modification of existing tools and deletion of those no longer required.

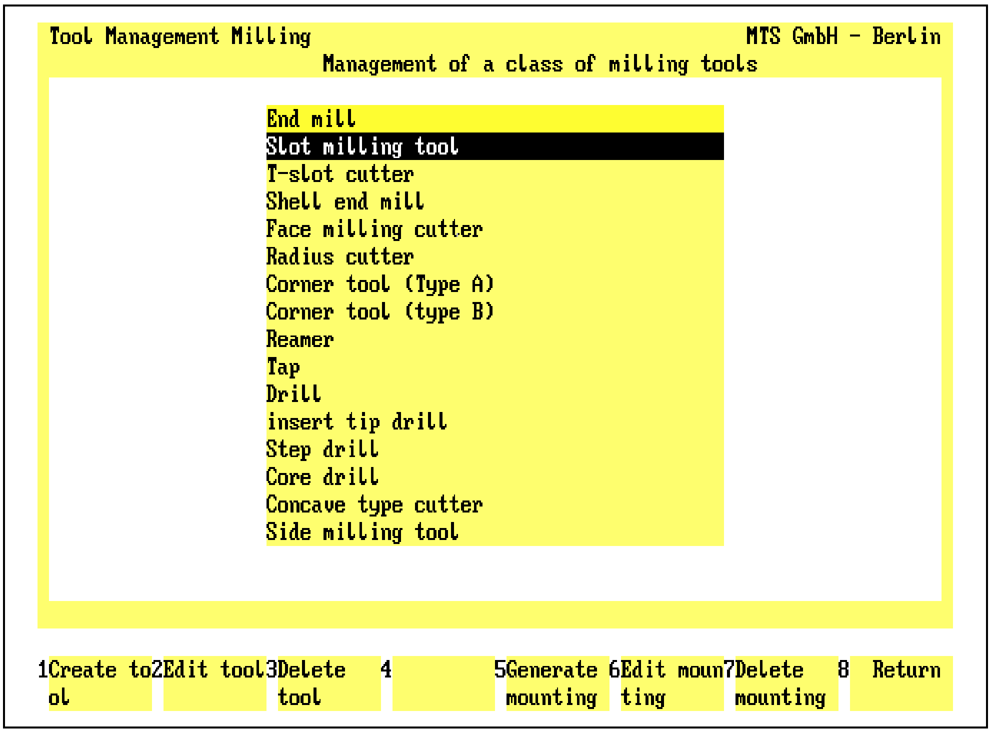


Figure 17
CNC Milling, Define/Delete Tools; Main Menu.

The screen layout of the Define/Delete Tools main menu is divided into two sections: the upper screen area contains a listing of all available tool types; the field currently in use is highlighted in color. As usual, further steps for specifying or editing tool data are indicated on the function keys at the bottom of the screen.

Select the desired step only by pressing the function keys rather than with the mouse.

↑	or	↓	Use the cursor keys ↑ or ↓ to select the tool type.
F1	or	F5	Create Tool/Tool Adapter: To generate a new tool of the current tool type, select F1; to define a new tool adapter, use F5.
F8	or	ESC	Return: Use F8 or ESC to conclude the current operation

Having started in the main menu by selecting the tool type, and subsequently selecting the Create Tool function by pressing F1 , the Data Entry menu for defining the tool is loaded.

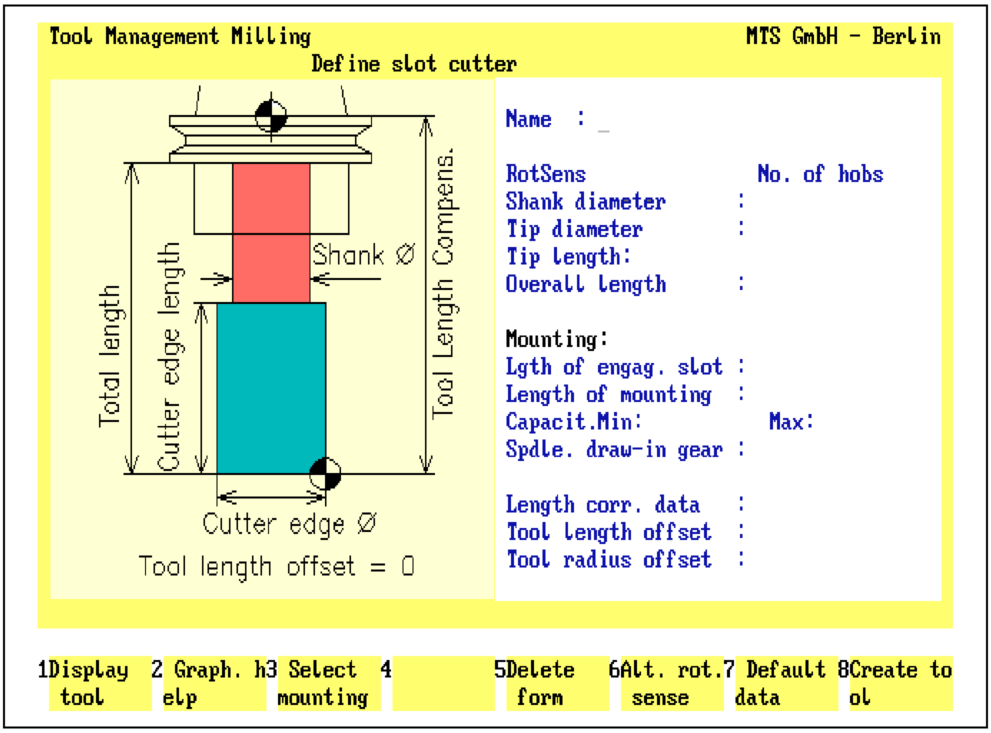


Figure 18
CNC Milling, Define/Delete Tools; defining a slot cutter.

The screen layout of the Data Entry menu is divided into three areas: the window on the left contains either a help graphic or a graphic corresponding to the data of the tool being defined (including the tool adapter). The input fields for the complete data record are located on the right.

You define a tool by manually entering the geometrical data, as well as the tool name and rotation direction. The desired tool adapter data can be automatically copied by selecting the Select Tool Adapter function. To save time, it is reasonable to define a new tool by first copying the data record of a similar tool, and then to modify the data to meet your requirements.

⇤⇥	Use the key ⇤⇥ to move from input field to input field.
→ or ←	Use the cursor keys → or ← to move the cursor within the input field.
INS or DEL	Use the key INS to insert a character, and the key DEL to delete one.
↵	If you confirm the entry in the input field with the ↵ key, the cursor moves automatically to the next input field.

[Tool Name]	[Tool Name] Enter the tool name or number in this input field.
[Parameter]	The entries required for a tool depend on the tool type. Use the help graphics to obtain information on the parameters.
F8	Create tool: When the data entry for all tool and tool adapter parameters has been completed, you save the tool under a certain name by pressing F8 .
ESC	Use ESC to conclude the operation, and to return to the Define/Delete Tools main menu.

1.4 Special functions of the software

The CNC Simulator incorporates some special functions which effectively support processing and NC programming:

- 3D representation
- Programming aids for ISO commands
- Setting-up automatics, set-up sheet
- Status management

1.4.1 3D representation

A function supporting CNC training is given by the option to display, at any time, 3D Views of the work part, seen from different viewing angles. The program features 3D displays in Milling Simulators. To display machining inside the work part, any work part quadrants can be cut out.

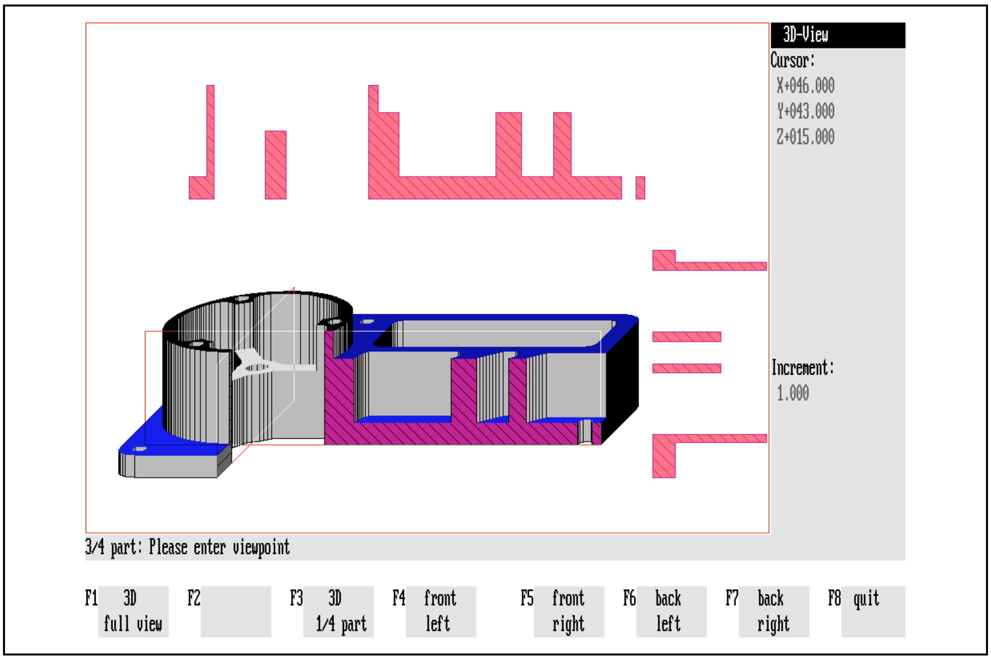


Figure 19
CNC Milling,3D View, three-quarter view with intersections

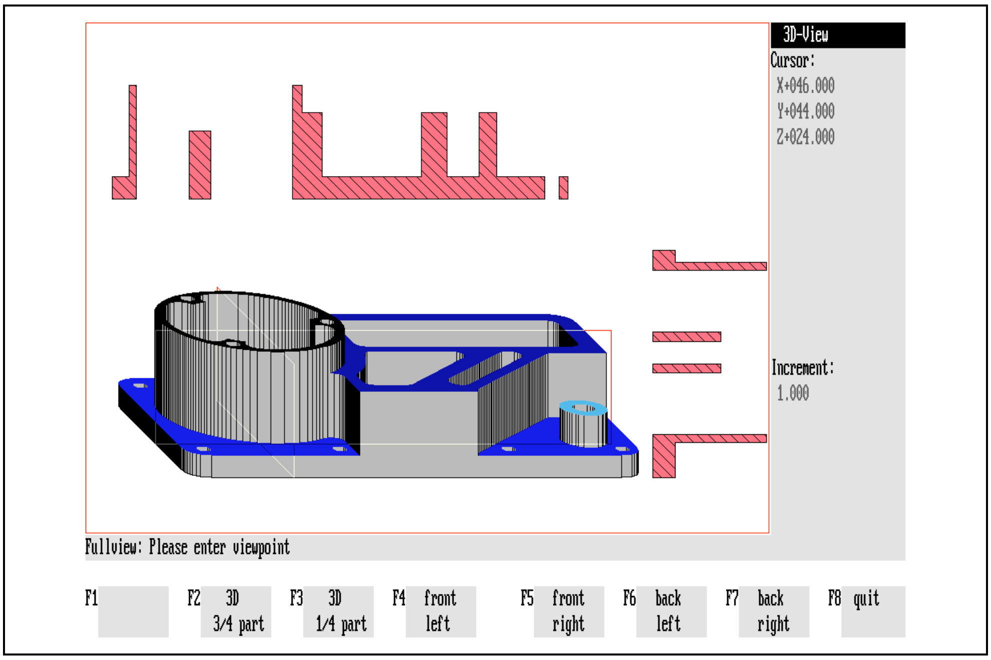


Figure 20
CNC Milling, 3D Display, full part with intersections

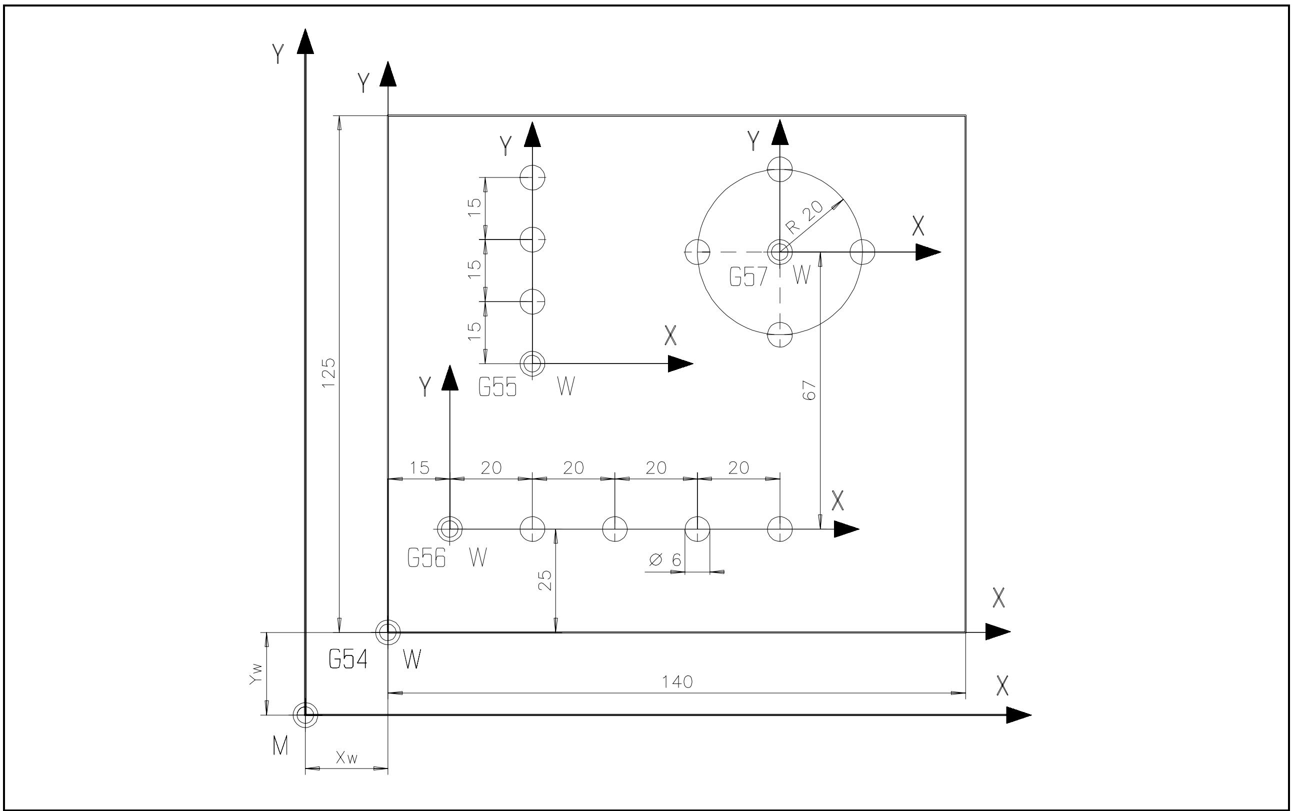
2.2.3 Setting the work part coordinate system with the commands G54 - G59

Six different work part coordinate systems can be used, for example, to program complex or repetitive contours. The coordinates of the respective zero point may measured as the distance between the reference point of the work part and the machine zero point. The value and the direction of this distance may be stored into the NC control.

Each stored zero point will be activated with the corresponding command (G54 - G59) in the NC program.

Note: Coordinate values of all zero points always relate to the machine zero point.

Exercise:
Create an NC-program for the following plate with respect to the newly defined work part zero points.



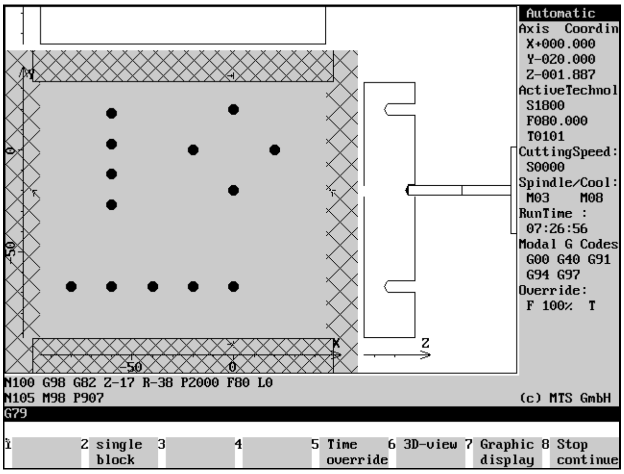
Use the following configuration:

CONFIGURATION	
	MACHINE MAKINO FX 650
	CONTROL FANUC 16M FX650

BLANK DIMENSIONS	
	X+140.000 Y+125.000 Z+025.000

VISE	
	MAKFX 160
	CHUCKED HEIGHT E+031.000
	SHIFT V+000.000
	ORIENTATION A0°

N095	G57
N100	G0 X0 Y0 M8
N105	G91
N110	G98 G82 Z-17 R-38 P2000 F80 L0
N115	M98 P907

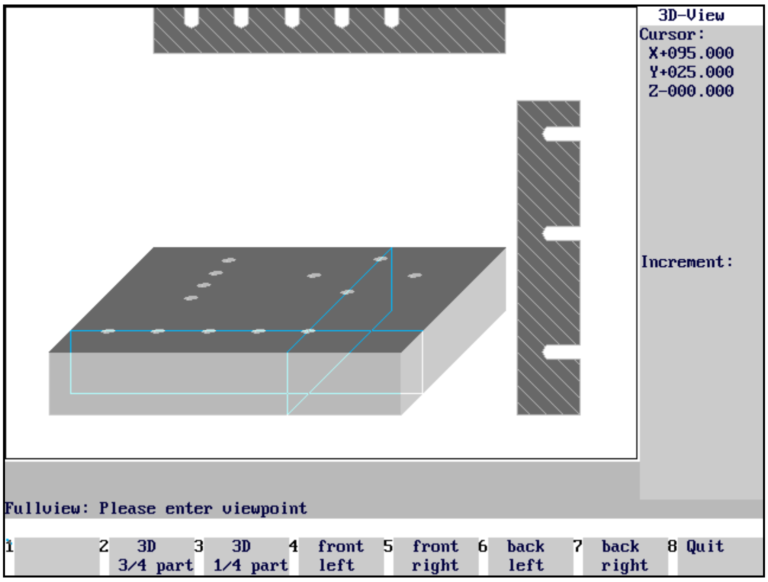


N120	G53
N125	G54
N130	G0 Z20 M5
N135	G91 G28 Z0 M9
N140	G91 G28 X0 Y0
N145	G90 G49 G80 G40
N150	M30

Subprograms

0905	0906	0907
N10 G91 G99 X0 Y0 N15 X20 N20 X20 N25 X20 N30 G98 X20 N35 G90 G80 N40 M99	N10 G91 G99 X0 Y0 N15 Y15 N20 Y15 N25 G98 Y15 N30 G90 G80 N35 M99	N010 G91 G99 X20 Y0 N015 X-20 Y20 N020 X-20 Y-20 N025 G98 X20 Y-20 N030 G90 G80 N035 M99

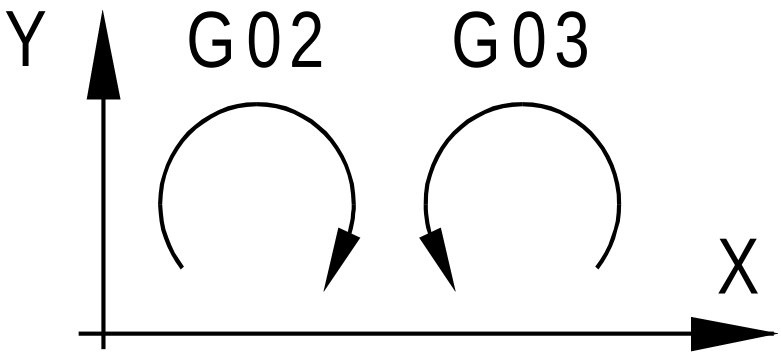
Finished part:



4.2 Circular interpolation

Circular interpolations can be moved in two opposite directions.

G02 in clockwise direction, or in
G03 counter-clockwise direction.



Directions for Circular Interpolations.

4.2.1 Circular Interpolation Clockwise G02

Command: **G02**

Circular Interpolation Clockwise G02

Function: The tool will move clockwise on a circular arc to the target position.

NC-Block: G02 [X...] [Y...] [Z...] [I...] [J...] [K...] [F...]...

Optional Addresses:

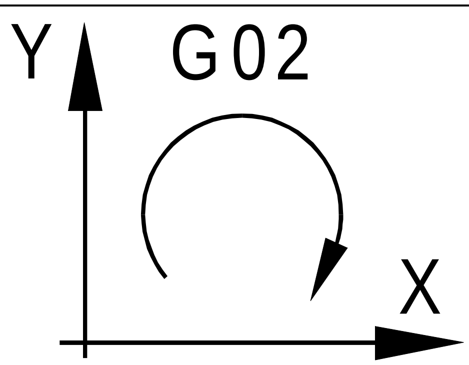
- X X-Coordinate of the Target Point
- Y Y-Coordinate of the Target Point
- Z Z-Coordinate of the Target Point
- I Circle Center Incremental (distance between the starting position and the circle center in the X-direction).
- J Circle Center Incremental (distance between the starting position and the circle center in the Y-direction).
- K Circle Center Incremental (distance between the starting position and the circle center in the Z-direction).

Note: The addresses I, J and K are always programmed in the incremental system, regardless of the selected value command system (G90 or G91).

- F Feedrate

The tool will move at the programmed feedrate clockwise on a circular arc to the target position as defined by the coordinates in X and Y.

These coordinates may either be programmed in the absolute system (G90) or in the incremental system (G91).



Command: # G02

Circular Interpolation Clockwise G02

Function: The tool will move clockwise on a circular arc to the target position.

NC-Block: G02 [X...] [Y...] [Z...] [I...] [J...] [K...] [F...]...

Programming Example

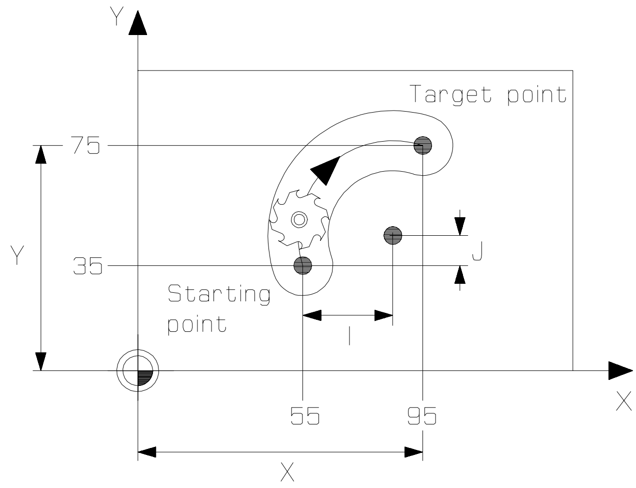
with Absolute Coordinates:

N085 G90

N090 G00 X+55. Y+35. Z+2.

N095 G01 Z-5.

N100 G02 X+95. Y+75. I+30. J+10.



Please note that in the absolute system the target points must be programmed according to their position in the coordinate system with reference to the origin of that system.

Programming Example

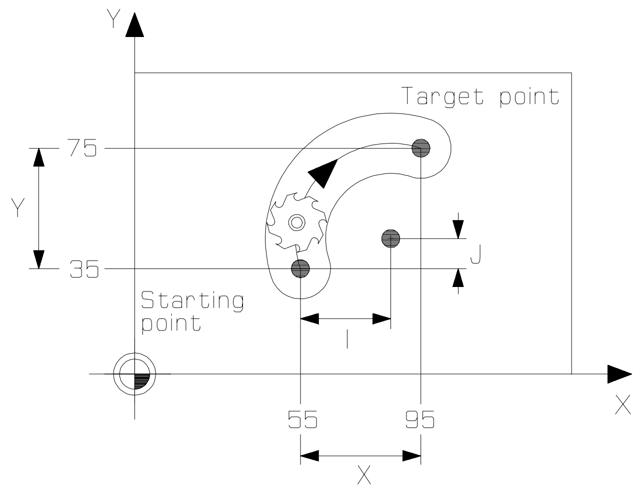
with Incremental Coordinates:

N085 G00 X+55. Y+35. Z+2.

N090 G91

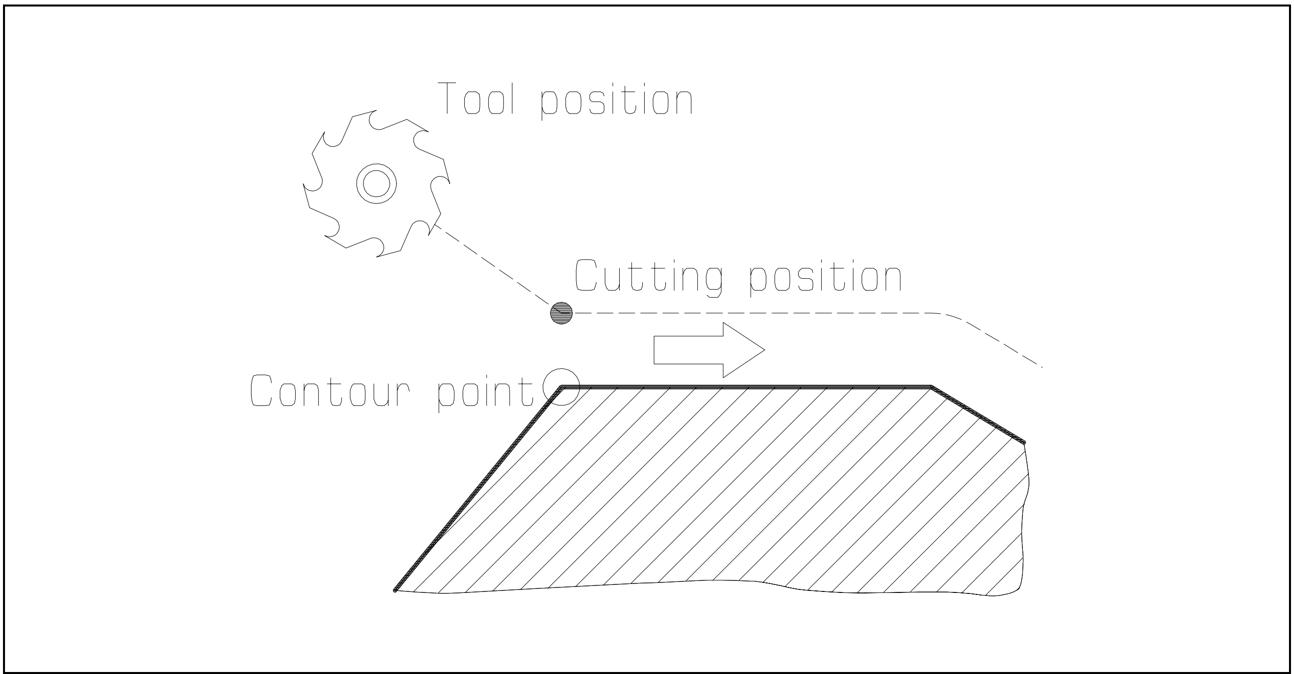
N095 G01 Z-7.

N100 G02 X+40. Y+40. I+30. J+10.



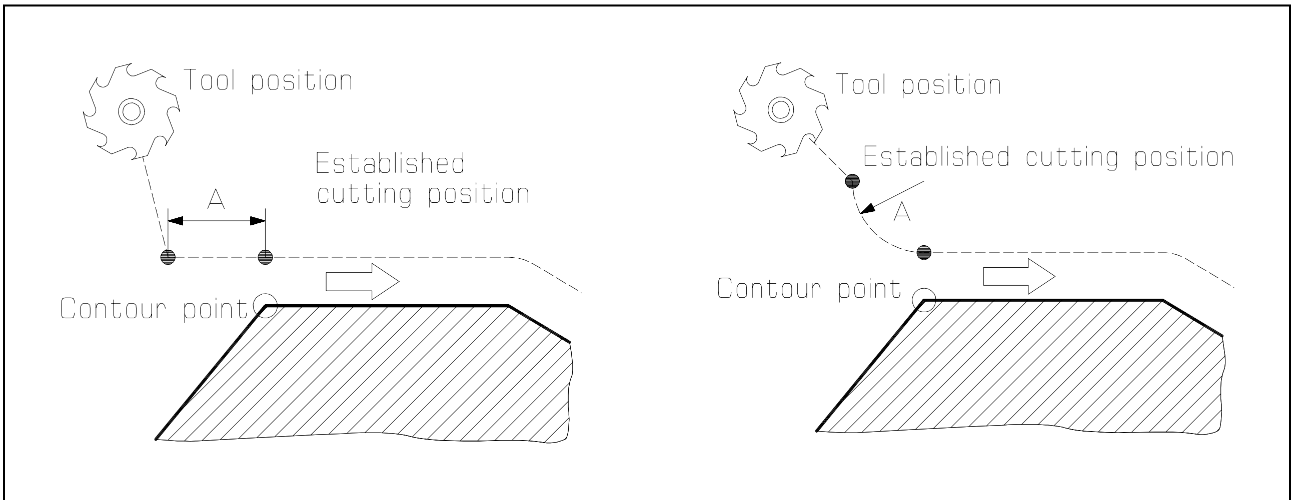
Tool approach and retreat movements

The cutter radius compensation is activated within a block. This means that the cutter radius compensation must at the latest be selected when the first contour point is approached.



Activate Cutter Radius Compensation

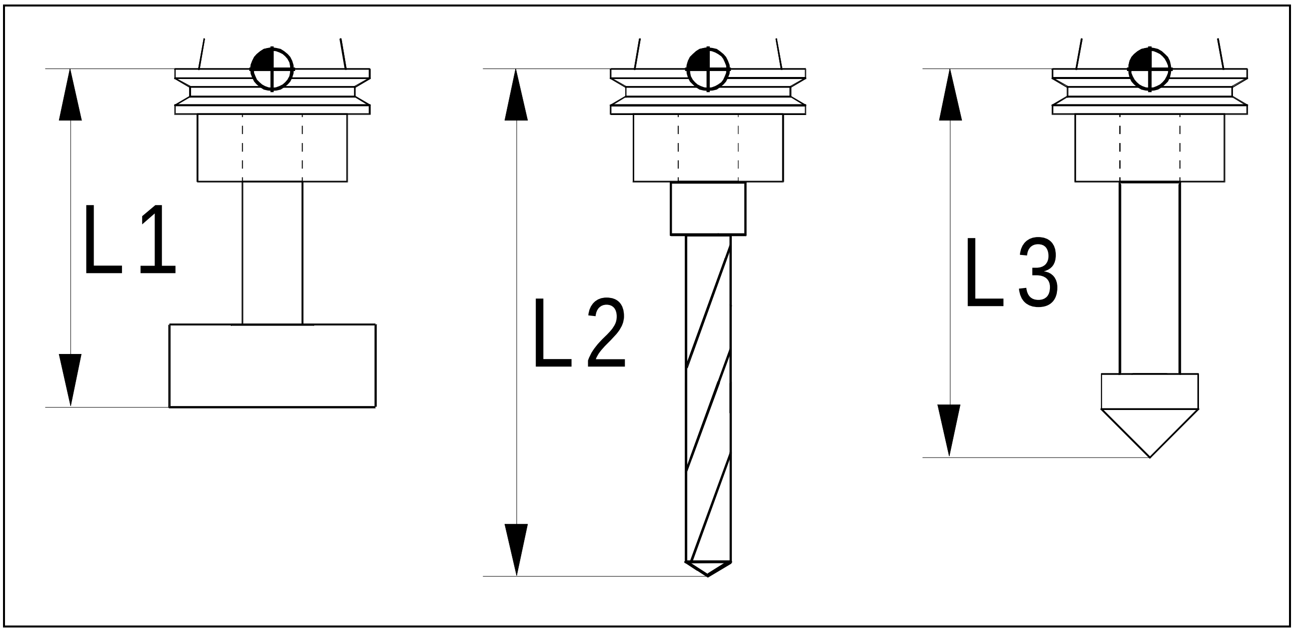
Additionally, contour-parallel or tangential approaching motions are also often programmed.



Contour-parallel Approach — Tangential Approach in a Quadrant

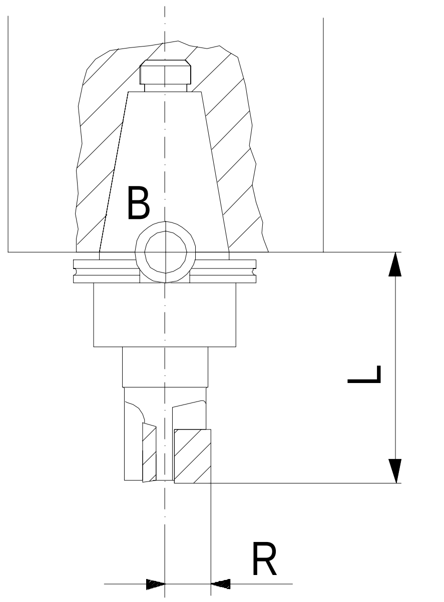
4.5 Tool length compensation

Using the tool compensation values it is easy to program a work part without directly considering the applicable tool lengths or tool radii. The available work part drawing data can be directly used for programming. The tool data, lengths as well as radii of the milling machines or indexable inserts are automatically considered by the CNC control.



T-slot cutter drill core drill

When programming an NC-program in absolute dimensioning, the control requires a coordinate system as well as information on the lengths of all employed tools. For this it is necessary to measure the length L, i.e. the distance between the tool setup point B and the cutting tip, and to enter it into the control.



B tool setup point

L length = distance of the cutting tip to the tool setup point in Z

R radius of the milling tool

Tool compensation values

A tool length compensation with reference to the reference point enables the adjustment between the set and actual tool length, as in the case of finishing the tool. This tool length value has to be available to the control.

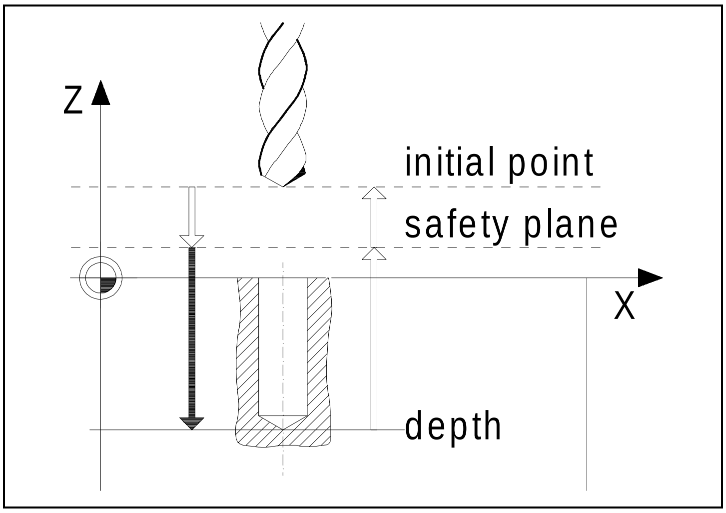
5 Cycles

5.1 Function and use of cycles on a CNC milling machine

In CNC-controls, predefined machining cycles are available which can be invoked with specific commands. Similar to subprograms, they contain prevalent command sequences. These machining cycles can be divided into three different types:

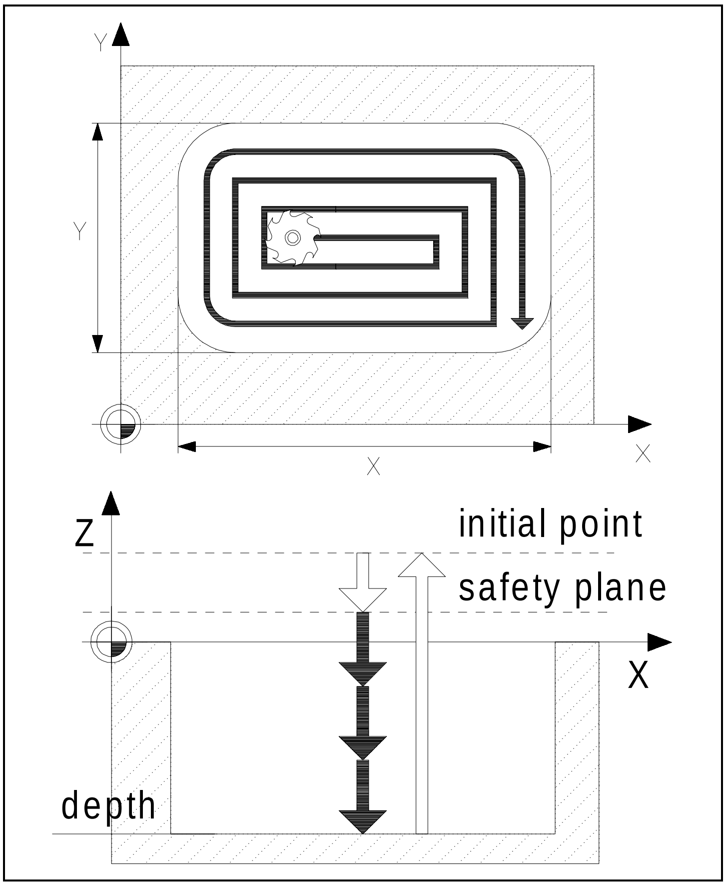
- drilling cycles
- milling cycles
- special cycles

Drilling cycles



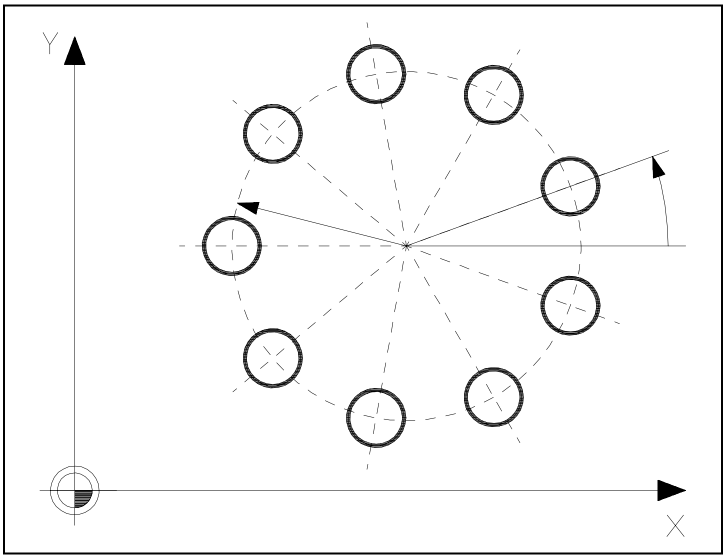
For drilling cycles, also called canned cycles in the FANUC-control, specific drilling, reaming or threading tasks are programmed by a command in conjunction with information on the required parameter. The CNC-control then executes all operations, e.g. for threading.

Milling cycles



For milling cycles, also called macro in the FANUC-control, specific milling operations, e.g. circular or rectangular pockets, are executed. For these cycles, the CNC-control must perform extensive calculations, e.g. to generate the individual travel motions for a rectangular pocket.

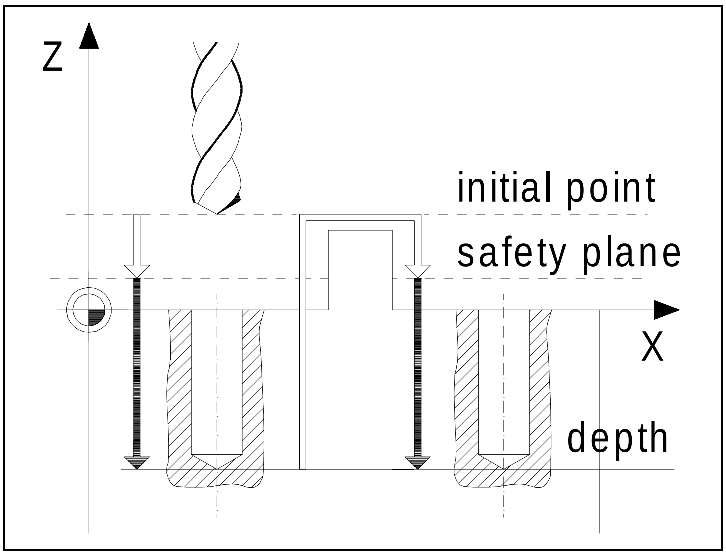
Special cycles



Belonging to the special cycles are e.g. various drill patterns. Combined with drilling cycles, e.g. holes on a circle or in a row can thus be easily programmed.

Safety planes

Multiple repetition of these cycles is common e.g. with drilling holes on a divided circle or on a straight line.



In the execution of a repeated cycle the tool will be retracted to the initial point before moving (in rapid traverse motion) to the next target position.

Programming the Z-coordinate of this initial point (the Y- or X-coordinate accordingly, if G18 or G19 have been programmed in the machining plane selection) is not necessary, it will be established from the actual tool position at the moment of the cycle invocation.

Please make sure that the Z-coordinate of this initial point (i.e. the position of the retracted tool) is sufficiently defined above the work part contour. After the cycle is invoked, the tool must be positioned to the Z-coordinate of this initial point. Subsequently the tool will be moved in the rapid traverse mode from this Z-position down to the safety plane.

After completion of the cycle the tool is retracted in a rapid motion to the Z-coordinate of the initial point.

5.3.4 finishing inside of circle macro P9110

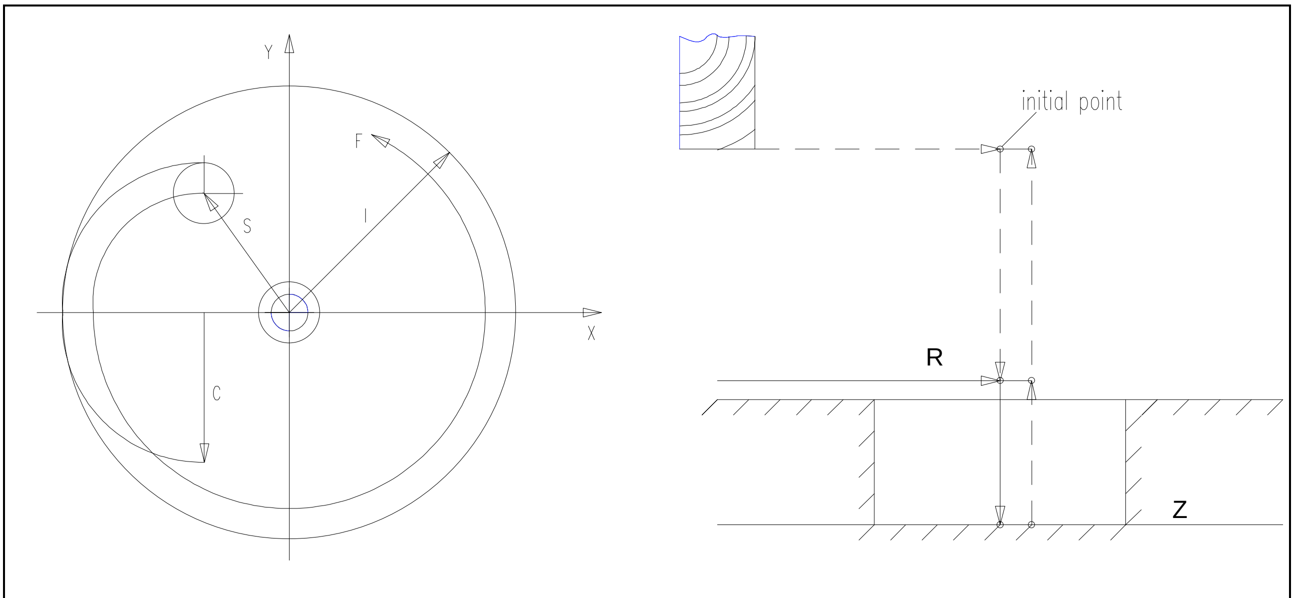
Command: **G65 P9110**

finishing inside of circle

NC-Block: **G65 P9110 I... D... R... Z... F... C... S... Q... M...**

Optional Addresses:

I	cutting circle radius
D	cutter radius offset number
R	Z-position of the safety plane
Z	Z-position of the bottom of the pocket
F	feedrate
C	approach circle radius
S	approach feedrate
Q	cutting direction
M	setting mode for R and Z



- - - - : rapid traverse	R : safety plane
——— : cutting feed	Z : depth

Note: The offset value must be less than the approach circle radius.

The cutter radius compensation is used.

Specify Q1. for counterwise cutting direction and omit Q for counter-clockwise cutting direction.

Specify M1. for incremental values of R and Z.
Omit M for absolute values of R and Z.

5.3.5 deep cutting of circular pocket macro P9120

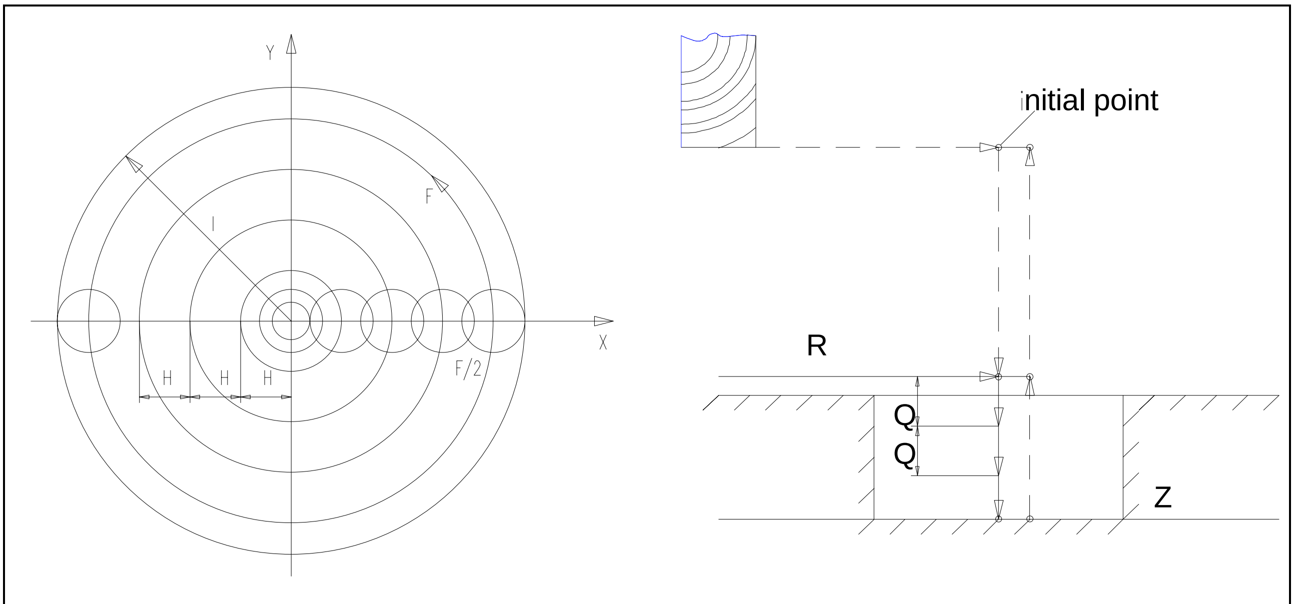
Command: **G65 P9120**

deep cutting of circular pocket

NC-Block: **G65 P9120 I... D... H... R... Z... F... S... Q... M...**

Optional Addresses:

I	cutting circle radius
D	cutter radius offset number
H	cutting width per pass
R	Z-position of the safety plane
Z	Z-position of the bottom of the pocket
F	feedrate
S	approach feedrate
Q	infeed per pass
M	setting mode for R and Z



- - - - : rapid traverse	R : safety plane
——— : cutting feed	Z : depth

Note:

Specify H so that it is less than the cutter diameter.

Only the counter-clockwise cutting direction is available..

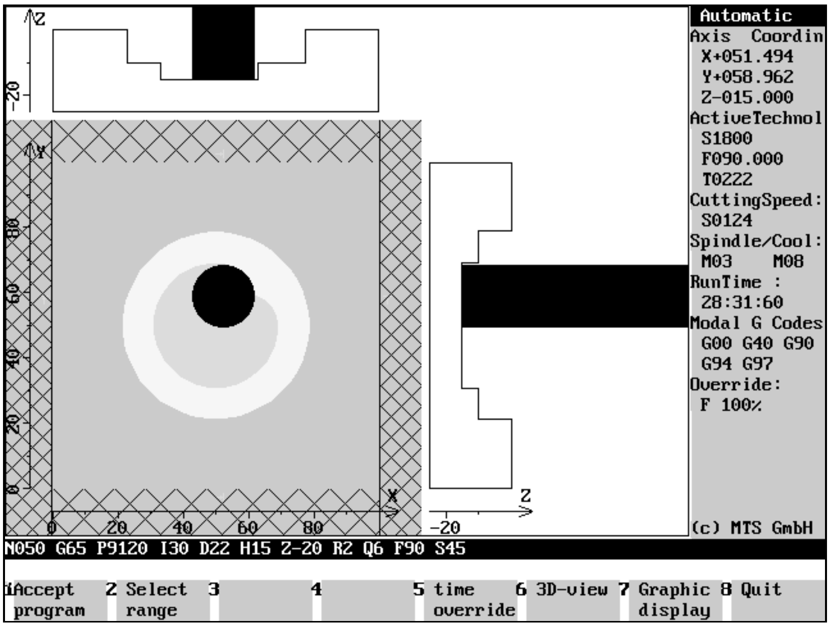
Specify Q1. for counterwise cutting direction and omit Q for counter-clockwise cutting direction.

Specify M1. for incremental values of R and Z.
Omit M for absolute values of R and Z.

Programming Example for the macro: G65 P9120 deep cutting of circular pocket

	$G54 X400 Y250 Z140

	O 120		
N010	G54		
N015	G90 G49 G80 G40 G17 G21		
N020	G91 G28 Z0 M9		
N025	G91 G28 X0 Y0		
N030	T02 M6		
N035	G90 S1800 M3		
N040	G0 G43 Z20 H18		
N045	X50 Y50 M8		
N050	G65 P9120 I30 D2 H15 Z-20 R2 Q6 F60 S30	P9120	deep cutting of circular pocket
		I30	cutting circle radius
		D2	cutter radius offset number
		H15	cutting width per pass
		Z-20	Z-position of the bottom of the pocket
		R2	Z-position of the safety plane
		Q6	infeed per pass
		F60	feedrate
		S30	approach feedrate



N055	G0 Z20 M9
N060	G91 G28 Z0 M5
N065	G90 G49 G80 G40
N070	M30

5.3.10 matrix maching macro P9200

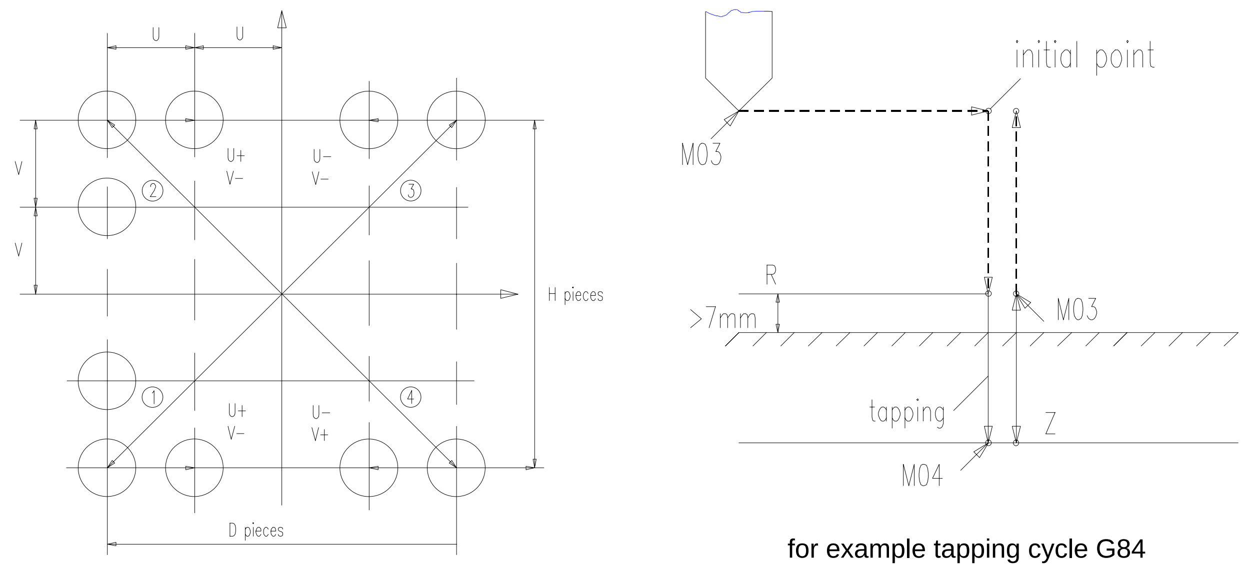
Command: **G65 P9200**

matrix maching

NC-Block: **G65 P9200 X... Y... U... D... V... H... S...**

Optional Addresses:

X	X coordinate of the first hole
Y	Y coordinate of the first hole
U	pitch in X-direction
D	number of holes in X-direction
V	pitch in Y-direction
H	number of holes in Y-direction
S	subprogram number called



for example tapping cycle G84

- - - - : rapid traverse	R : safety plane
——— : cutting feed	Z : depth

There are two possibilities to use the bolt hole circle:

1) for drilling:
```
G90 G98 G84 Z-30 R7 P1000 F1000 L0
G65 P9200 X... Y... U... D... V... H...
G80 X... Y...
```

2) for multi-block machining
```
G65 P9200 X... Y... U... D... V... H... S...
```

Note: Use the absolute input value (G90) for positioning.

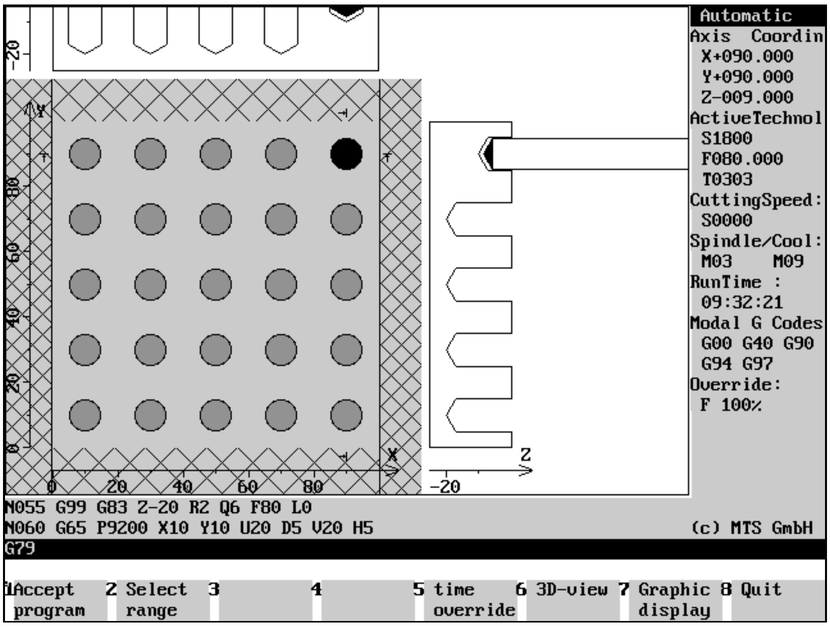
Don´t specify S by programming a canned cycle (first possibility).

The subprogram must be programmed with incremental value input (second possibility).

Programming Example for the macro: G65 P9200 matrix machining for drilling

	$G54 X400 Y250 Z135

	O 200	
N010	G54	
N015	G90 G49 G80 G40 G17 G21	
N020	G91 G28 Z0 M9	
N025	G91 G28 X0 Y0	
N030	T03 M6	
N035	G90 S1800 M3	
N040	G0 G43 Z20 H19 M8	
N045	G99 G83 Z-20 R2 Q6 F80 L0	definition of a peck drilling cycle
N050	G65 P9200 X10 Y10 U20 D5 V20 H5	P9200 matrix machining
		X10 X coordinate of the first hole
		Y10 Y coordinate of the first hole
		U20 pitch in X-direction
		D5 number of holes in X-direction
		V20 pitch in Y-direction
		H5 number of holes in Y-direction



N055	G80	
N060	G0 Z20 M9	
N065	G91 G28 Z0 M5	
N070	G90 G49 G40	
N075	M30	

MATHEMATISCH TECHNISCHE
SOFTWARE-ENTWICKLUNG GMBH

CAD/CAM Turning & Milling

with MTS INCAD

MTS TeachWare Student's Book - Excerpt

MTS Mathematisch Technische Software-Entwicklung GmbH • Kaiserin-Augusta-Allee 101 • D-10553 Berlin
Phone: +49 / 30 / 349 960-0 • Fax: +49 / 30 / 349 960-25 • World Wide Web: http://www.mts-cnc.com • email: mts@mts-cnc.com

3 CAD/CAM Milling

3.1 From a drawing to a finished work part:

In contrast to manual NC-programming the CAD-CAM-system supports the programmer in many aspects. The system does some of the preparing work for example the computing of not measured contour points. The system directly takes the geometries, so the input (coordinates etc.) is automatically right.
Using a cutting value table cutting speed etc. can be set automatically.

For automatic programming the sequence of operations for generating an NC-program is as follows:

1. First the work part must be geometrically defined. A representation of the finished part as well as the blank is necessary.
2. Subsequently, the individual machining operations are specified. The programming system assists the programmer in selecting the appropriate tool and automatically calculates the necessary cutting data.
3. Finally a NC-program for a specific CNC-machine tool with a specific CNC-control is generated and can then be transferred to the machine.

It follows a description of these steps.

3.1.1 basic concept on the use of CAD data for NC production in milling

The main goal of a CAD-system has been a simplified generation of technical drawings. Advantages lie in the possibility of making changes easily, copying and printing several times the drawing.

Using the CAD-system only the drawing itself was generated, other information like measures or tolerances had to be set manually.

The CAD-CAM-system has a much wider range of tasks. The system should generates directly from the drawing NC-blocks for the production of the part. This means all points and contour elements have to be in the system with required tolerance.

From this follows:.

the input of each coordinate has to be done with the highest precision possible.

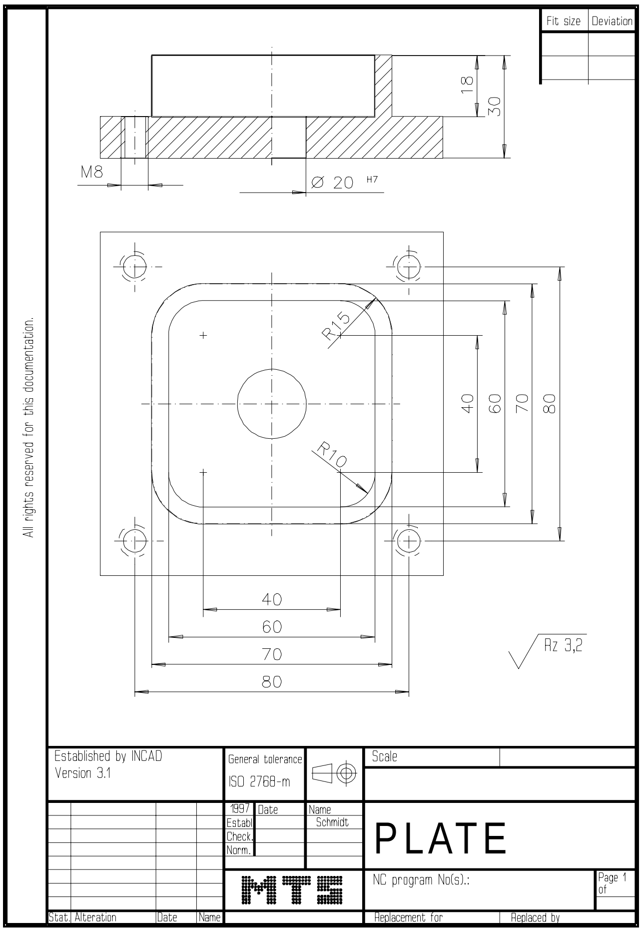
You should use numerical input or the other help functions like auxiliary contours or trapping functions.

Contours can be used in the CAD-CAM-system for automatically generation NC-programs, if the construction has been done appropriate. Every contour has to be a „contour string“. The INCAD-system supplies the function „contour string“ and a „contour tracing“ function, if elements of a contour are not generated using „contour string“.

3.1.5 CAD/CAM drawing

We want describe the function and the using of the INCAD-system milling by programming the NC-program for the following plate. It is to be manufactured as individual workpart on a CNC milling machine.

blank	100 x 100 x 30 mm
clamping height	70 mm
material	AlMgSi1

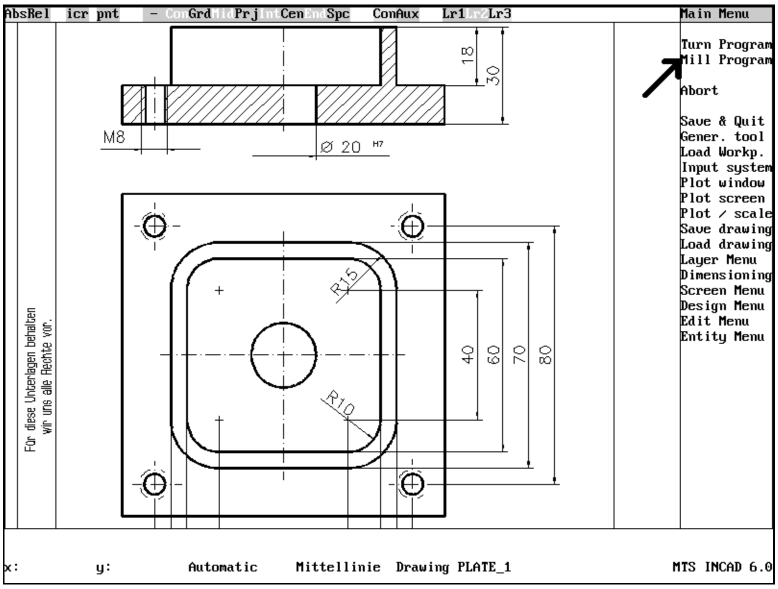


WORK PLAN			
contour milling	T01	shell end mill	MW-040/032 HSS ISO 2586
pocket milling	T11	slot milling tool	MS-18.0/063L HSS ISO 1641
core hole drilling ∅ 6.8	T12	drill	DR-06.80/069 HSS ISO 235
threading M8	T02	tap	TA-M08.0/1.25 HSS ISO 2857
predrilling ∅ 19.8	T04	step drill	DS-19.8/11.5-118 HSS ISO 3439
reaming 20H7	T08	reamer	RE-D20.0/H7 HSS ISO 521

3.2.1 Starting NC programming system milling

Before starting the mill program you should choose a zoom window, so that only necessary elements are visible.

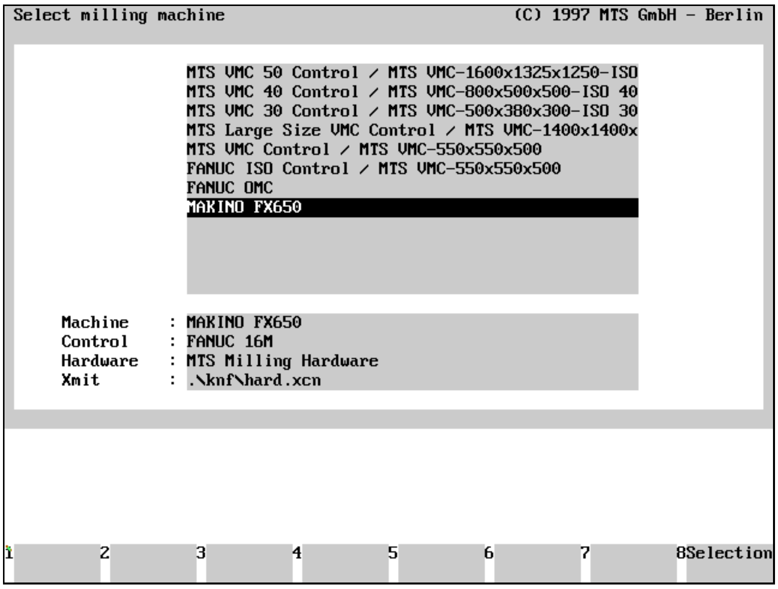
activating CADCAM-System milling



3.2.2 Selecting CNC machine

Starting the mill program some steps have to be done (only once) as a basis for what follows. They are necessary and cannot be changed during generation of the NC-program.

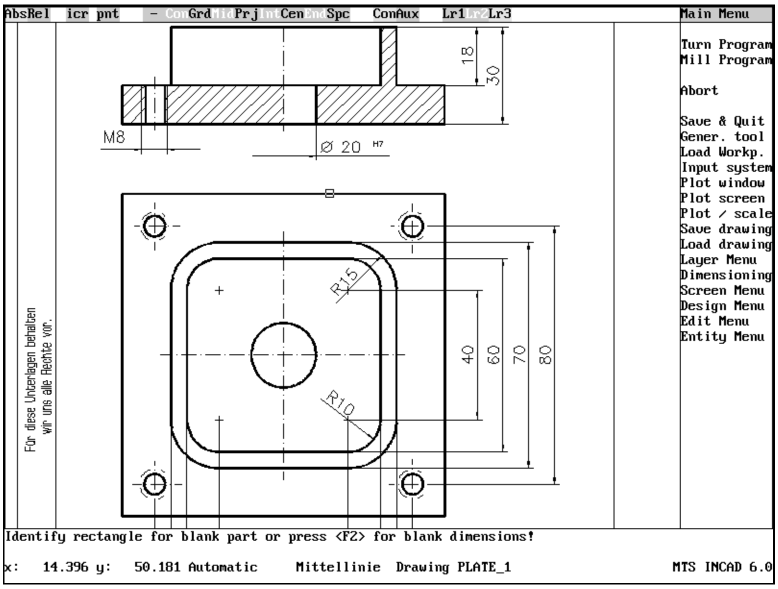
selection of milling machine „MAKINO FX 650“



3.2.4 Defining the location of workpart zero point

Every NC-program requires a workpart zero point, to which measures relate. You choose this point before the programming, if necessary, it can be changed later by using zero point shifts.

defining the blank



You identify the blank part in a dialog:

Identify rectangle for blank part or press <F2> for blank dimensions!

Click with the mouse to identify the blank part in the top view. Then you have to enter the height or identify it in the front view.

Enter height numerically : <F1>, define by 2 points : <F2> !

Pushing the <F1> key you see the following prompt:

Enter blank height !

Now you can enter the height, in this example 30mm.

Selecting the work part zero point the clamping situation has to be taken into account. In this example there are some go through holes, so there has to be a distance to the machine table. The procedure is the same as with the height:

<F1> for clamping height numerically, <F2> for defining by two points !

Pushing the <F1> key you see the following prompt:

Clamping height :

In our example we use the clamping height 70 mm.

Note: With the machine MAKINO FX650 we have a special machine configuration: The machine range in the Z-axis begins with 150mm. This value has to be added to the clamping height. Therefore the clamping height = 0 is not allowed!

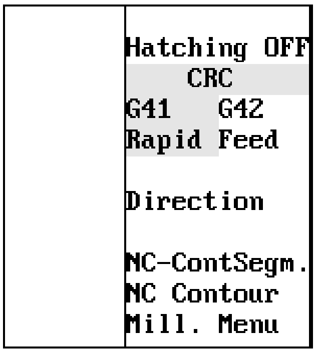
In our example we enter a clamping height of 220mm!

NC program generation: contour milling
Activating the function „Contours“ starts the generation of NC-commands.

cutter radius compensation
Before selecting the contour you can activate cutter radius compensation and choose the position of tool relative to the contour. To do this, activate the appropriate switches with the mouse.

G41 tool left to contour
G42 tool right to contour

switches for contour milling

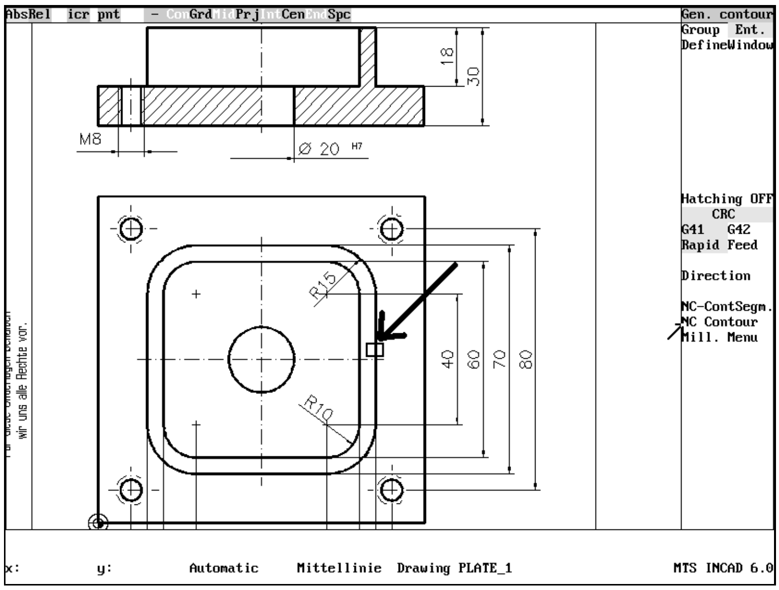


The positioning movement can be in rapid speed or in infeed. In most cases the tool will move in rapid speed, to do so, set the switch „Rapid/Feed“ with the mouse to „Rapid“.

selecting a NC contour to be machined
After activating the function „NC Contour“ you start with the selection of the contour to be milled through the following dialog.

Identify NC-entity at the start point !

select a contour



With the mouse select the contour, which is marked then by colour.

This entity <F2>, Next entity <F1>

With <F2> yyou confirm this contour. Pushing <F2> a second time marks the passing direction of the tool. If this direction is not correct, you can change it with the function „Direction“.

Press <F1> to generate NC-program , <F3> to abort !

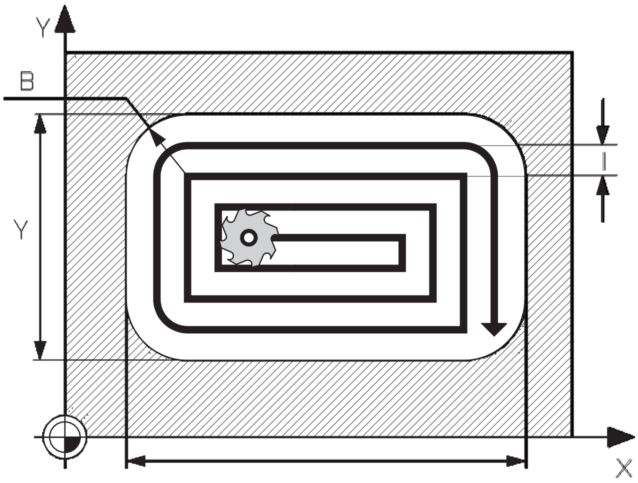
With <F1> you confirm every input.

3.2.7 Pockets

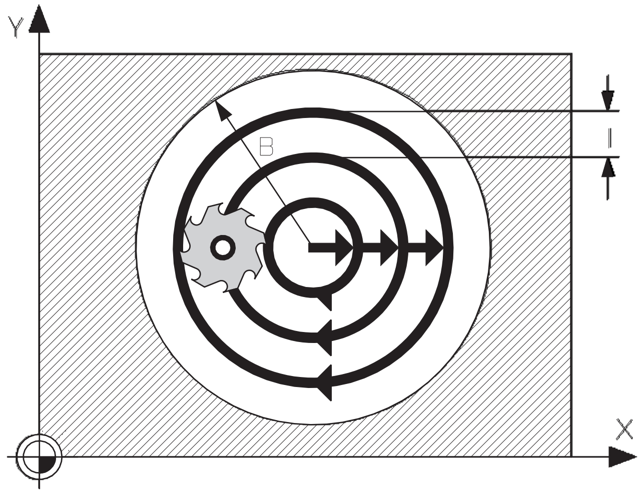
With the INCAD-system you can create NC-commands for manufacturing pockets. You have to enter the necessary technological data or activate existing ones. The geometric data are automatically created through the graphical selection in INCAD. You have to consider, that a contour is a rectangle, circle or „contour string".

Following different pocket/pin cycles are available:

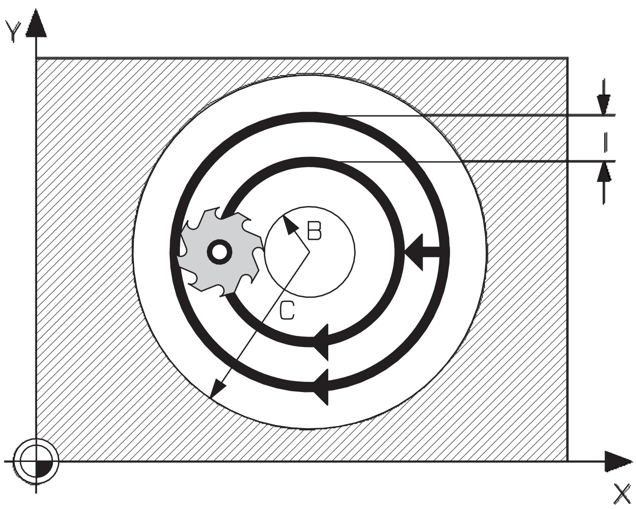
G87- Rectangular pocket cycle

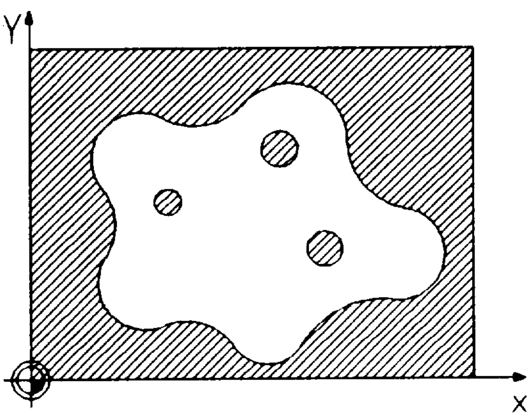


G88 - Circular pocket cycle



G89 - Pin cycle





G01/G02/G03-G41/G42

Contour pocket with islands
Alternative clockwise / counterclockwise
Starting point for downfeed

tool selection

First we choose a slot milling tool with a diameter of 18 mm and place it at position nr. 2 in the magazine.

We click the menupoint „tools“ with the mouse.

First we select with the cursor the second position, so that its frame is marked then. Then we open the window for selecting tool groups with <F1> „Equipmagazine“ and mark the chosen class of milling tools.

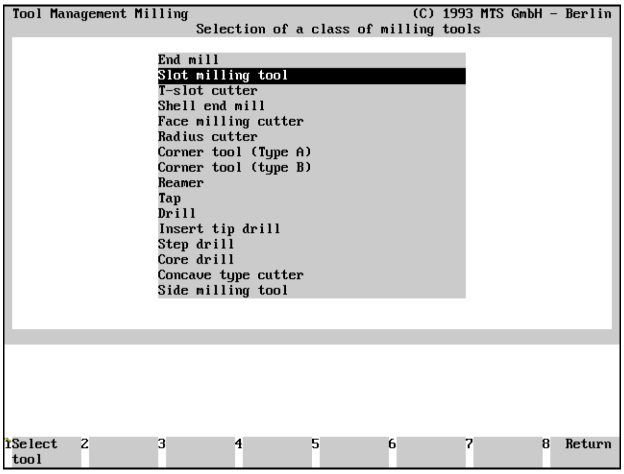
select „Equip turret“

with <F1>

select with the arrow taste the class of milling tool

„slot milling tool“

and confirm with<F1>

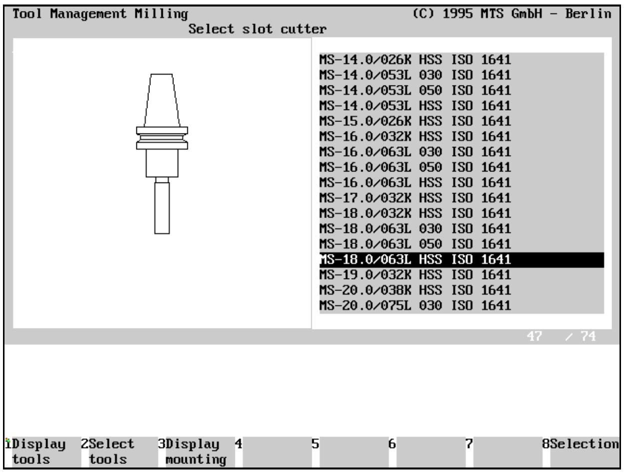


In the menu for slot milling tools select one tool and confirm with <F8>.

Select with the arrow taste the slot milling tool

„MS-18.0/063L HSS ISO 1641“

and confirm with<F8>

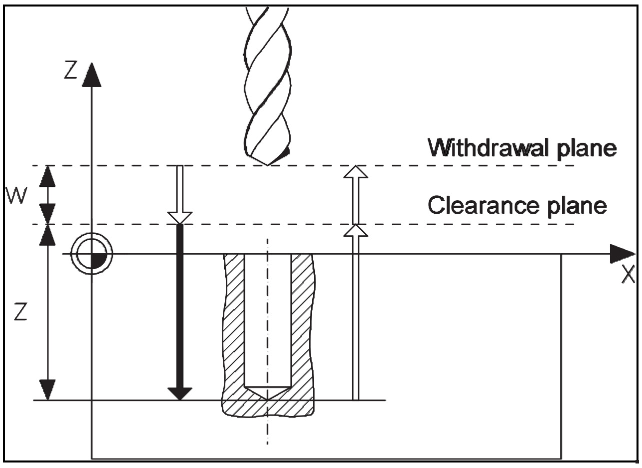


3.2.8 Drilling

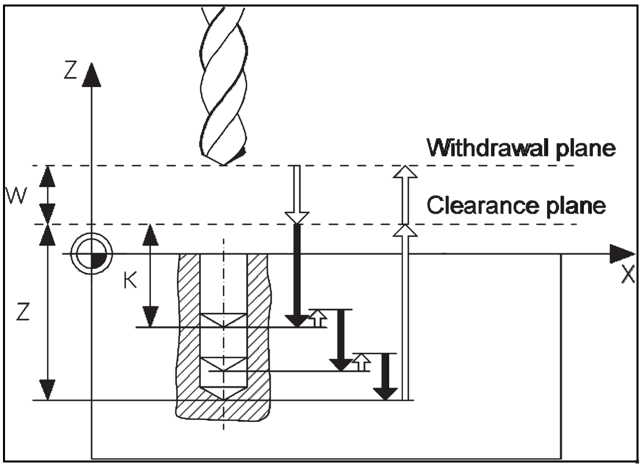
With INCAD you can create NC-commands for drilling, trapping and reaming. You have to enter all necessary technological data or activate existing ones. All geometric data are automatically generated by INCAD after related shapes are selected. Take into consideration that all boreholes are represented by circles!

Following different drilling cycles are available:

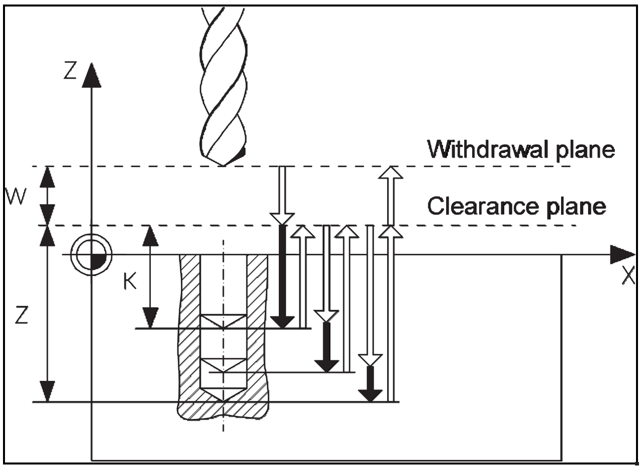
G81 - Drilling cycle



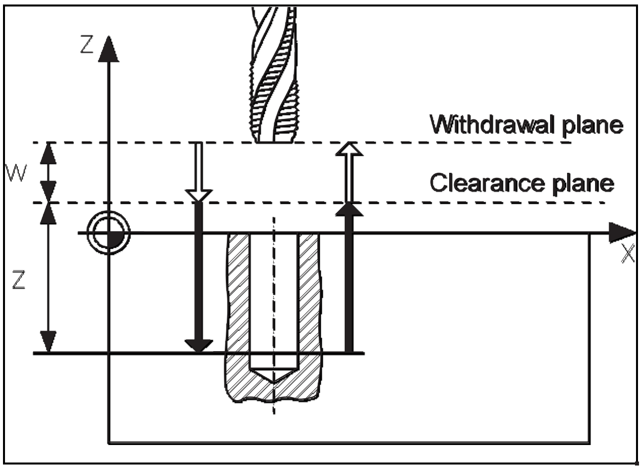
G82 - Drilling cycle with chip breaking



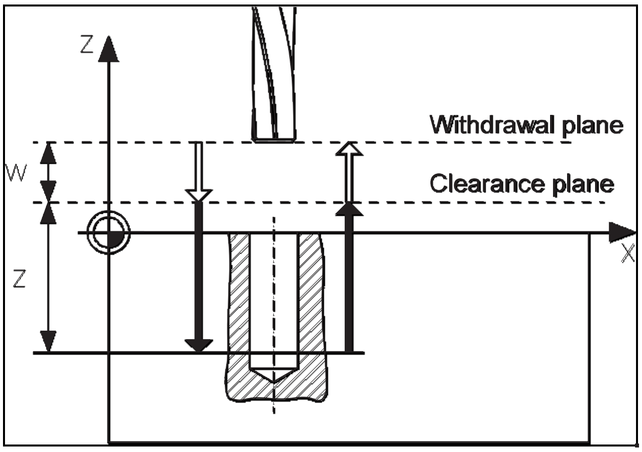
G83 - Drilling cycle with chip breaking and chip-removal



G84 - Tapping cycle



G85 - Reaming cycle



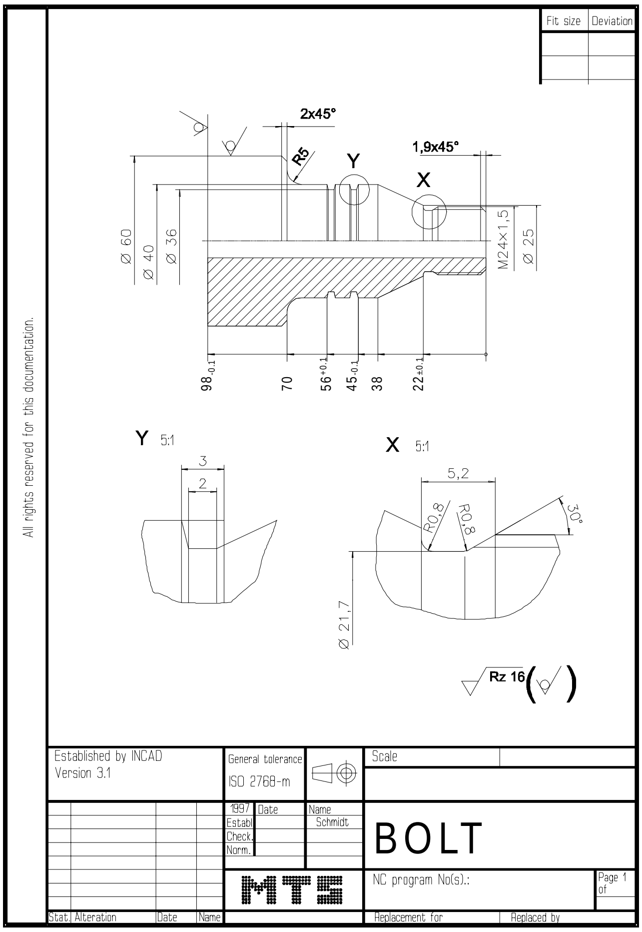
4 CAD/CAM Turning

4.1 NC program generation turning

4.1.1 CAD/CAM drawing

We want describe the function and the using of the INCAD-system turning by programming the NC-program for the following bolt. It is to be manufactured as individual workpart on a CNC turning machine.

blank	∅65mm x 102mm
chucking depth	22mm
material	AlMgSi1

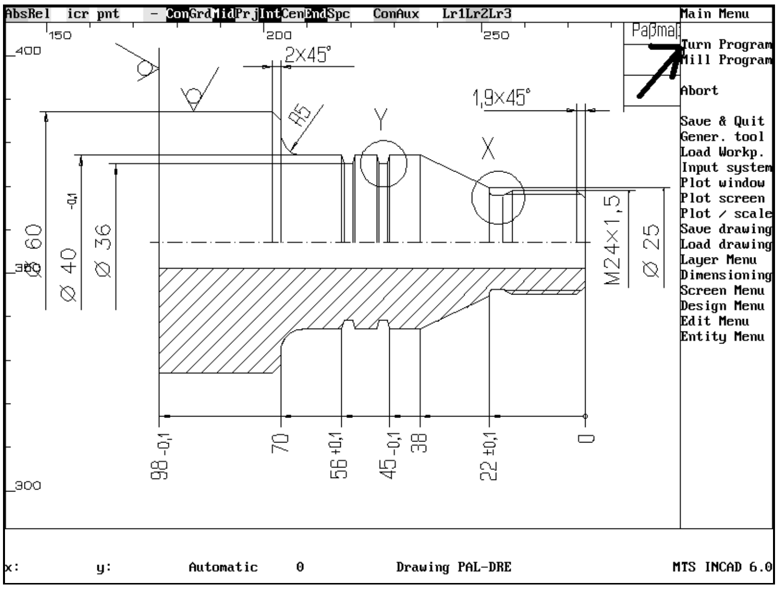


WORK PLAN		
face turning	T01	left handed corner cutter
centring	T11	centring tool
drilling	T12	twist drill
contour roughing	T02	left handed corner cutter
contour finishing	T04	left handed corner cutter
recessing	T06	external recessing tool
threading	T08	left handed threading tool

4.1.2 Starting the NC programming system turning

Before starting the turn program you should choose a zoom window, so that only necessary elements are visible.

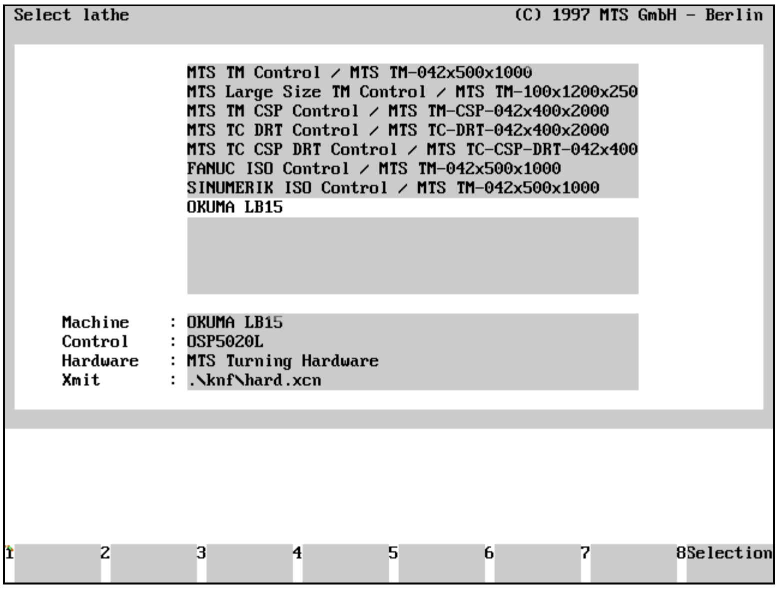
calling CADCAM-System turning



4.1.3 Selecting the CNC machine

Starting the turn program some steps have to be done (only once) as a basis for what follows. They are necessary and cannot be changed during generation of the NC-program.

selecting the turning machine „OKUMA LB15“

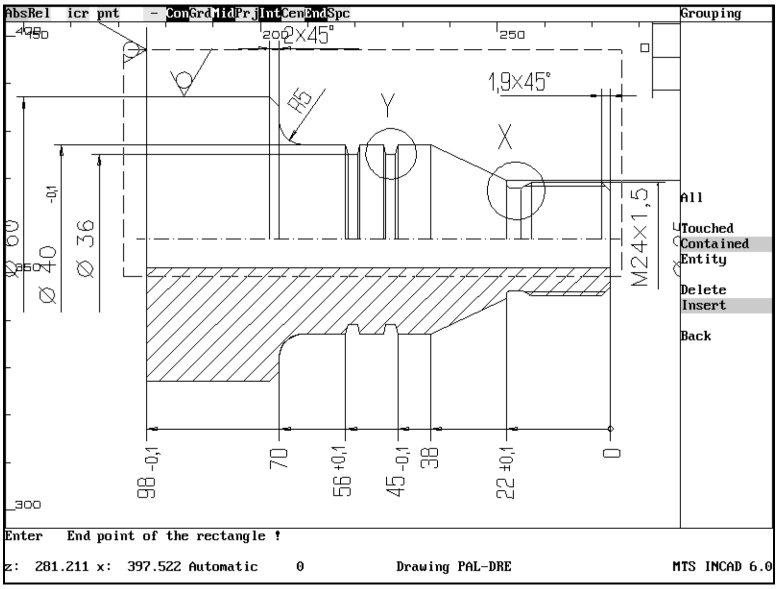


4.1.5 selecting drawing elements for NC programming

For the following programming steps the necessary grafical entities must be selected.

<F1> : Select entities , <F2> entire drawing, <F3> abort !

selecting entities



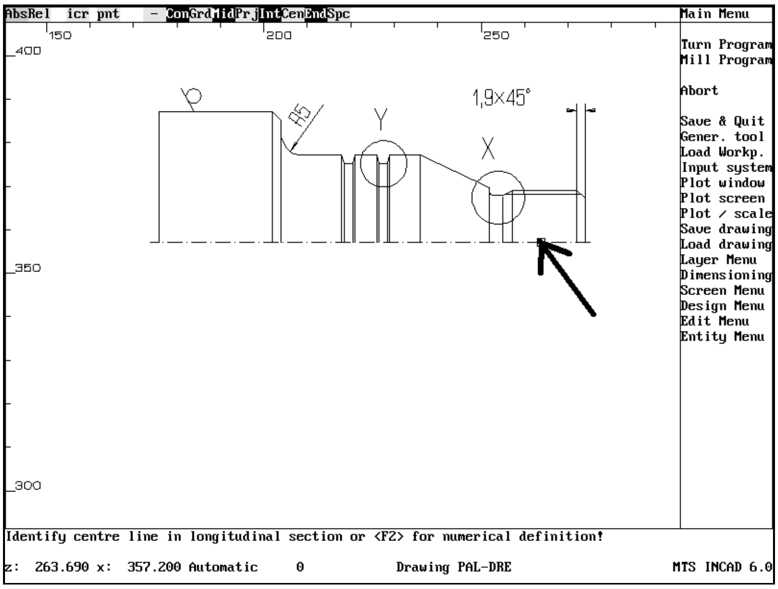
By presssing the <F1> key the following dialog appears:

Enter start point of the rectangle !

Enter end point of the rectangle !

Use the mouse to create a rectangle which contains all necessary entities. The selected entities are marked with another colour. Confirm the selection by calling the „back“ function with the mouse.

selected entities



The centre line must be selected with the mouse.

Identify centre line in longitudinal section or <F2> for numerical definition

Finally the workpart must be selected with the mouse and confirmed with<F1>.

Identify workpart in longitudinal section !

<F1>: Accept entity as workpiece, <F2>: another piece !

4.1.6 defining blank

The INCAD-system hides all unselected entities, generates a new view from the workpart and shows information about his volume and his weight.

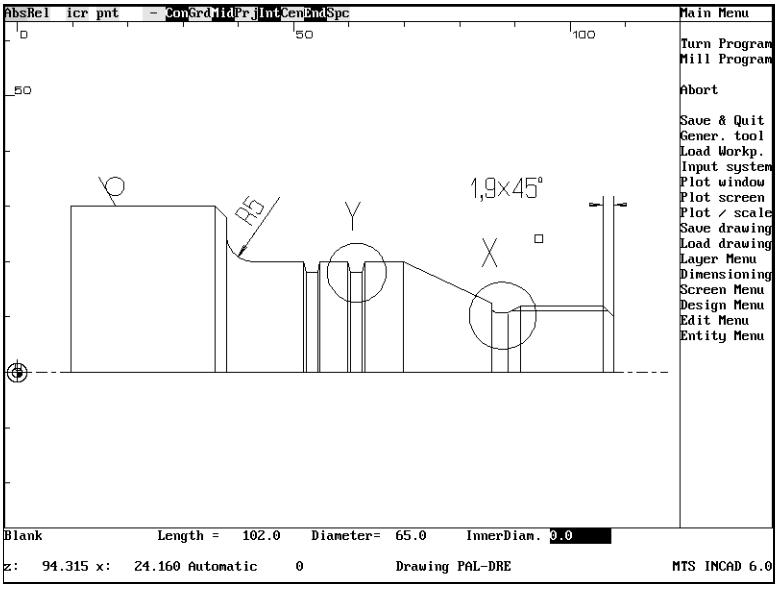
Part data: (<F1> to continue)

Volume : 141.520 ccm Weight : 382.102 grams

By pressing the <F1> key the following dialog for the blank dimension appears:

Blank Length = 102.000 Diameter = 65.000 InnerDiam. 0.0

defining blank



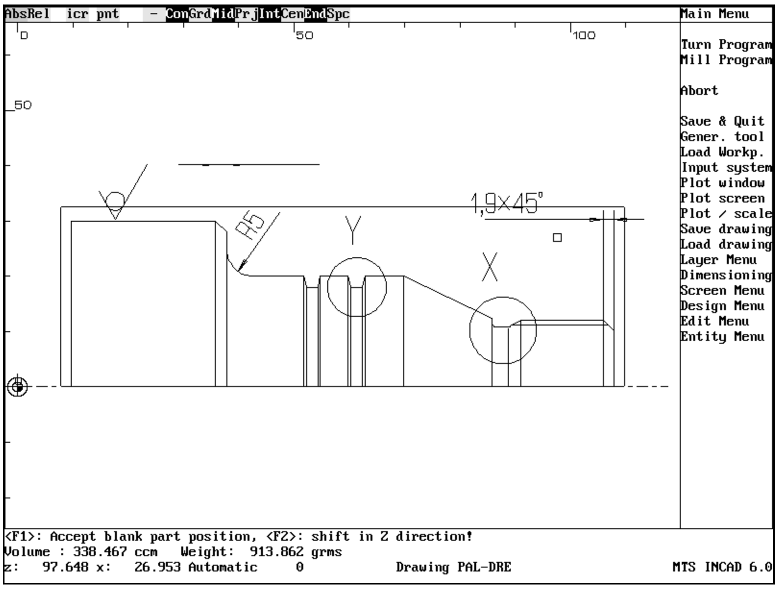
Use the tastatur to write the desired dimensions or confirm with the <Enter> key. The INCAD-system asks for a centring of the blank by the following prompt.

Centring ? (Y/N) N

The standard answer from the system is No. Confirm it by pressing <Enter> .
Finally a shift of the blank can be entered by the following dialog

<F1>: Accept blank part position, <F2>: shift in Z-direction!

blank part

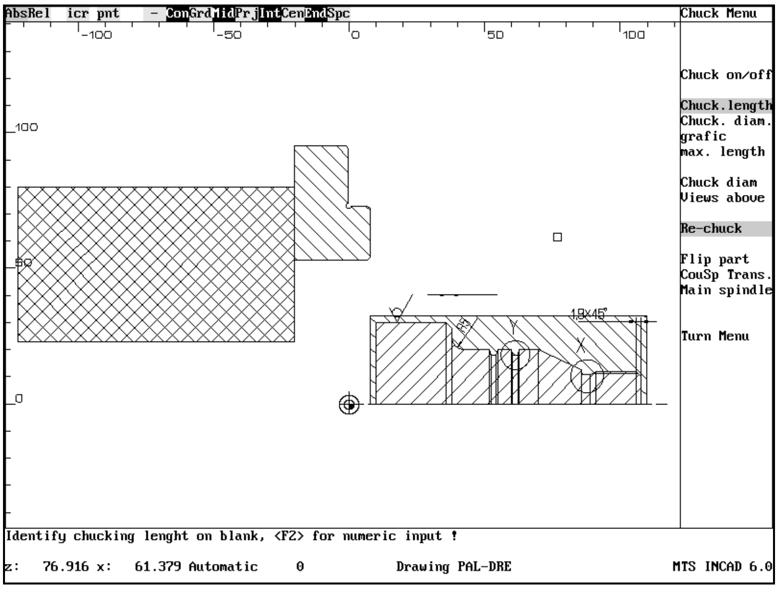


Confirm the blank part position with the <F1> key.

4.1.7 Selecting clamping devices

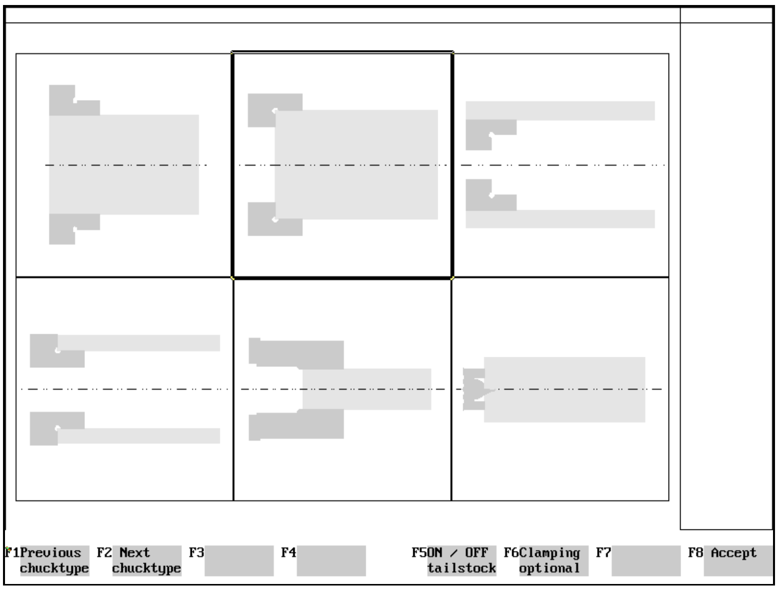
The INCAD-system offers a chuck.

offered chuck



You can choose the chuck by calling the „Main spindle“ function with the mouse. Use the <F1> or <F2> key to select the desired chucktype.

selecting a chuck



Press <F8> to confirm your choice.

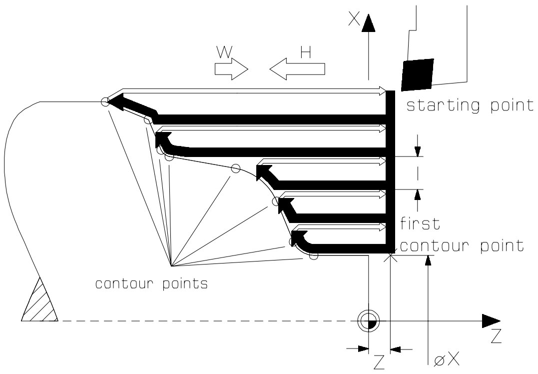
Select the chuck „ KFD-HS 160“ by pressing <F8> (Return).

4.1.11 survey over possible machining sequences

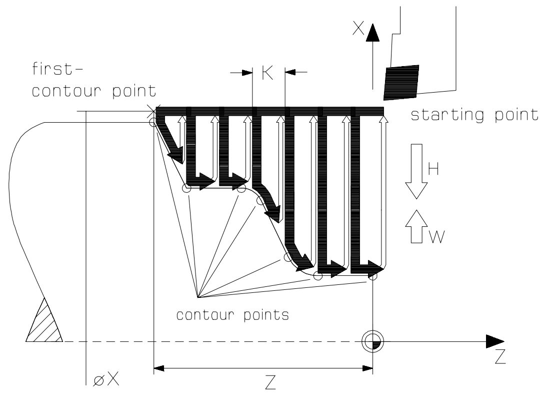
Machining sequences and procedures you have to use in MTS-Code independant of your selectedmachine and control. With a postprocessor the system generates an NC-program in the control language of your control for example OKUMA.

SURVEY OF PROCESS CAPABILITIES		
Cycles	G81	Straight Roughing Cycle / Optional contour
	G82	Cross Roughing Cycle / Optional contour
	G83	Contouring cycle - Multipass cycle
	G84	Deep drilling cycle
	G31	Threading cycle
	G79	Recessing cycle

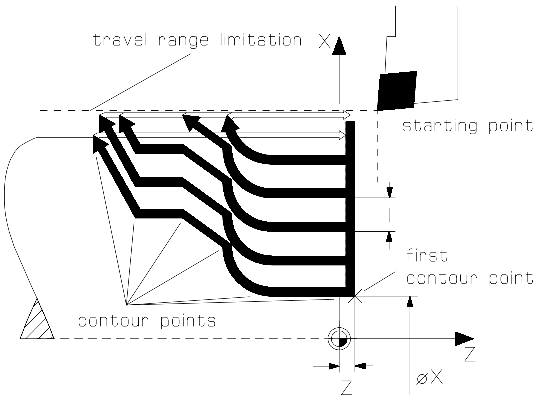
G81 Straight Roughing Cycle / Optional contour



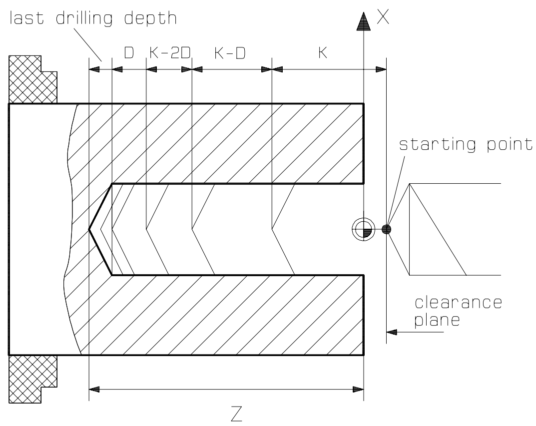
G82 Cross Roughing Cycle / Optional contour



G83 Contouring cycle - Multipass cycle



G84 Deep drilling cycle



4.2 programming the machine sequences

In principle this is the procedure for generating NC-programs:

1. loading
   a qualified CAD file
2. starting
   NC programming system turning
3. selecting
   the CNC-machine
4. selecting
   the blank material
5. defining the location
   of workpart zero point
6. programming
   the machining sequences

programming a machining sequence
tool selection defining technological values selecting contours for geometrical data defining approach and withdrawal strategy

7. saving
   the generated NC-program
8. simulating
   the generated NC-program

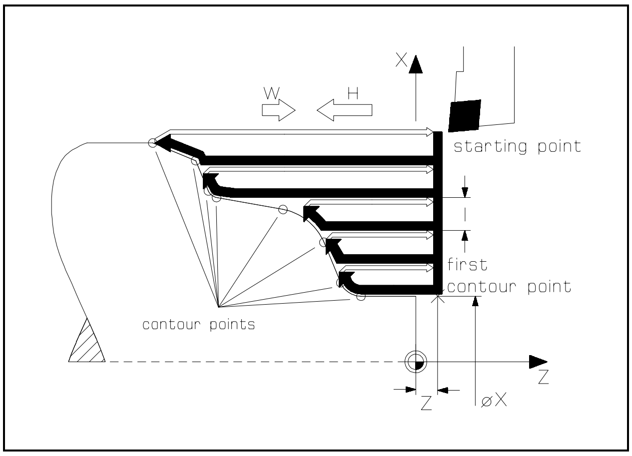
With the following machining sequences we´ll describe the possibilities of the CAD-CAM-system.

WORK PLAN		
face turning	T01	left handed corner cutter
centring	T11	centring tool
drilling	T12	twist drill
contour roughing	T02	left handed corner cutter
contour finishing	T04	left handed corner cutter
recessing	T06	external recessing tool
threading	T08	left handed threading tool

4.2.4 Straight roughing

The cycle G81 is a straight roughing cycle with movements parallel to the Z-axis by selecting an contour. It can be programmed for either internal or external machining.

straight roughing cycle



At first activate the function „straight roughing Cycle“ by selecting the menupoint „convnt.Tools“ with the mouse and after that the menupoint „Straight rgh“.

tool selection
In the work plan an the set-up sheet the following machining sequence and tool are described.

contour roughing	T02	left handed corner cutter
T02	left handed corner cutter	CL-SVJCL-2020/L/1604 ISO30

We click the menupoint „tools“ with the mouse. First we select with the cursor the second position, so that its frame is marked then. Go back to the turn menu by confirming with <F8>.

defining technological values
For the machine sequence „contour roughing“ cutting values are required. You can get these data automatically from the INCAD-system by activating the function „CutValuesOn“.

In the following dialog you can confirm all answers with <Enter> or change if desired the value with the keyboard. In our example we confirm all.

Coolant M08 — Feed(mm/rev): 0.25
m/min: 200 — Speed lim. 3500
Move to the tool changing point: Y — Approach opt.: 0
CRC : N
Autofinish N — Downfeed: 2.00
Finishing allowance Z :0.2 — Finishing allowance X :0.2 — Finishing allowance parallel :0.0

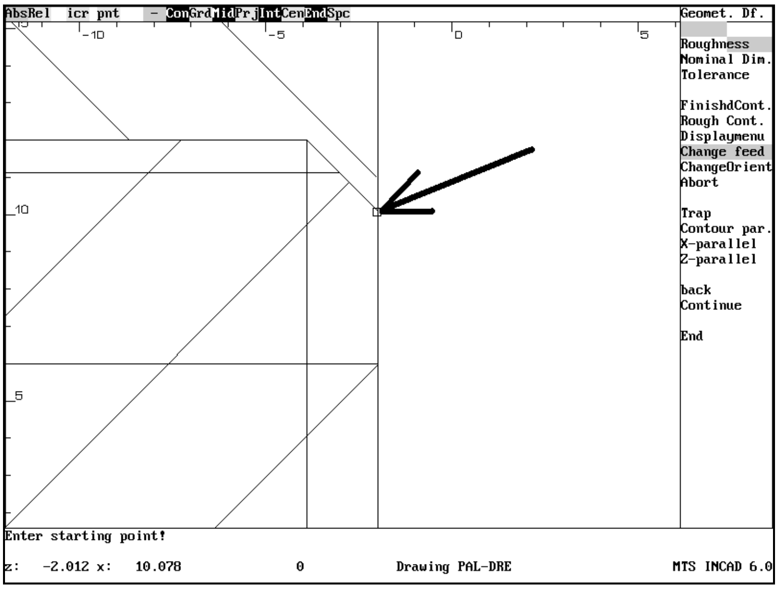
selecting the contour to be machined

The system asks for the starting point of the contour

Enter starting point !

Use the zoom function with <F6> for showing the details. Click with the mouse at the following point. Use the automatic trapping function by selecting in the swith line.

starting point

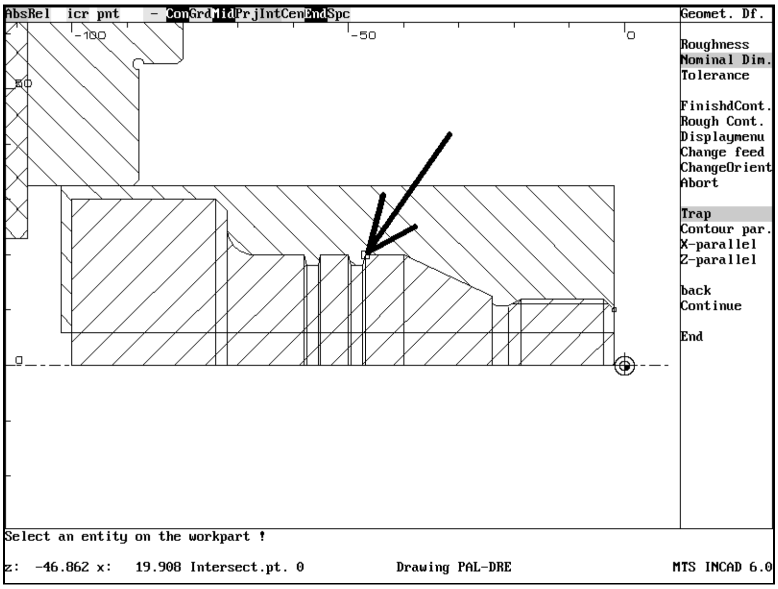


Enter next point !

The system asks for the next point. Activate the trap function with the switch „Trap“ and click with the mouse at the following point.

next point of the contour

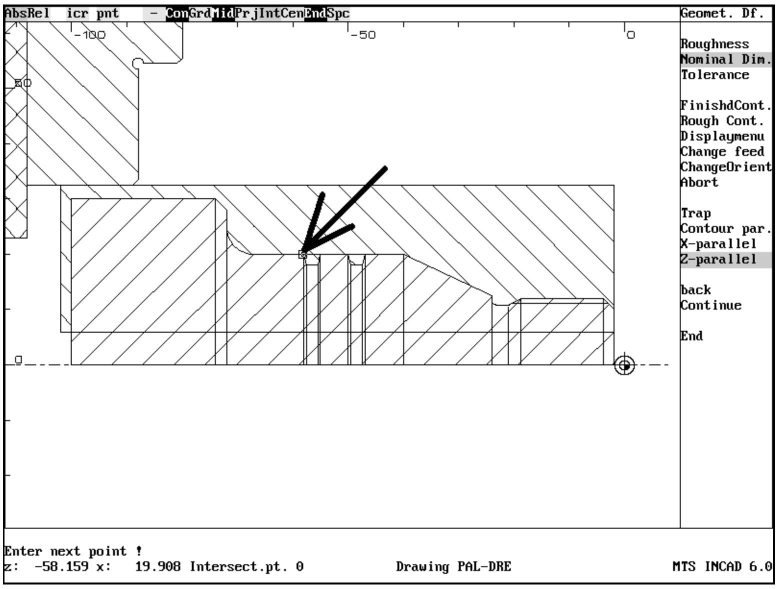
using the function „Trap“



Now activate the the switch „Z-parallel“ and click with the mouse at the following point.

next point of the contour

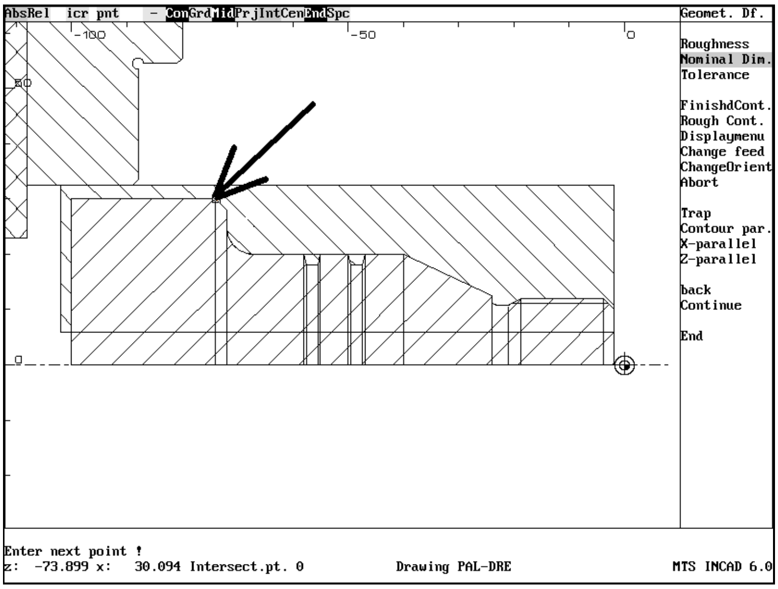
using the function „Z-parallel“



Use again the trap function with the switch „Trap“ and click with the mouse at the following end point

end point of the contour

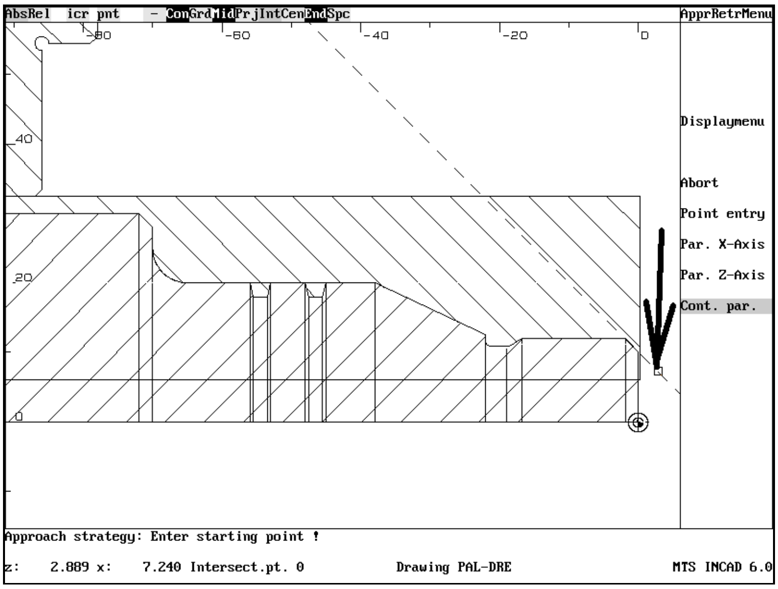
using the function „Trap“



Confirm these entries with the menupoint „End“

defining approach and withdrawal strategy
For the approach movement activate the function „Cont. par.“ and click right of the workpart.

approach movement



Enter the distance to the workpart with the keyboard

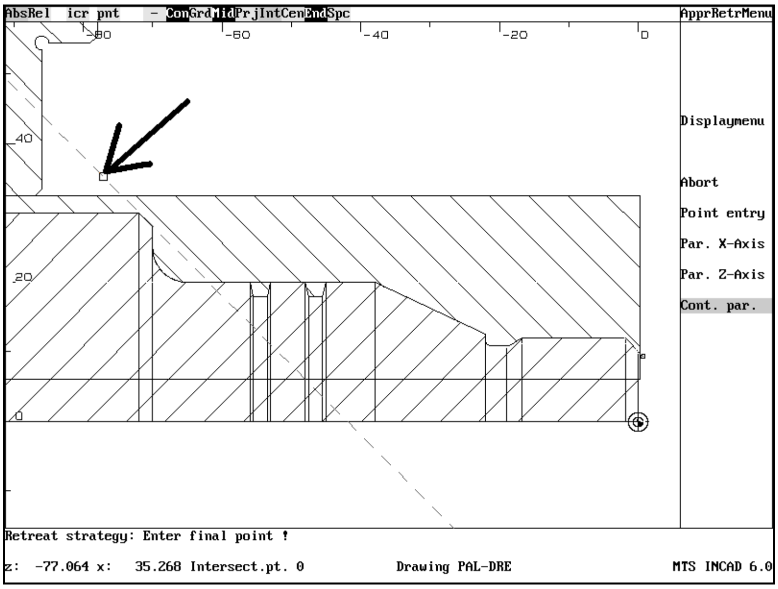
Incremental value : 1

and confirm with <Enter>. The following prompt appears:

Retreat strategy: Enter final point !

For the retrat movement activate the function „Cont. par.“ and click over the workpart.

retreat movement



Enter the distance to the workpart with the keyboard

Incremental value : 4

and confirm with <Enter>. At least confirm all with <F1> three times.

<F1> to accept starting and end points, <F2> other selection

<F2> to entr the cycle invocatn oint, <F1> to cont. !

4.2.5 Finishing contours

At first activate the function „finishing contours“ by selecting the menupoint „convnt.Tools“ with the mouse and after that the menupoint „Finishing“.

tool selection
In the work plan and the set-up sheet the following machining sequence and tool are described.

contour finishing	T04	left handed corner cutter
T04	left handed corner cutter	CL-SVJCL-2020/L/1604 ISO30

We select with the cursor the fourth position, so that its frame is marked then. Go back to the turn menu by confirming with <F8>.

defining technological values
For the machine sequence „contour finishing“ cutting values are required. You can get these data automatically from the INCAD-system by activating the function „CutValuesOn“.

In the following dialog you can confirm all answers with <Enter> or change if desired the value with the keyboard. In our example we confirm all.

Coolant M08	Feed(mm/rev): 0.10
m/min: 300	Speed lim. 3500
Move to the tool changing point: Y	Approach opt.: 0

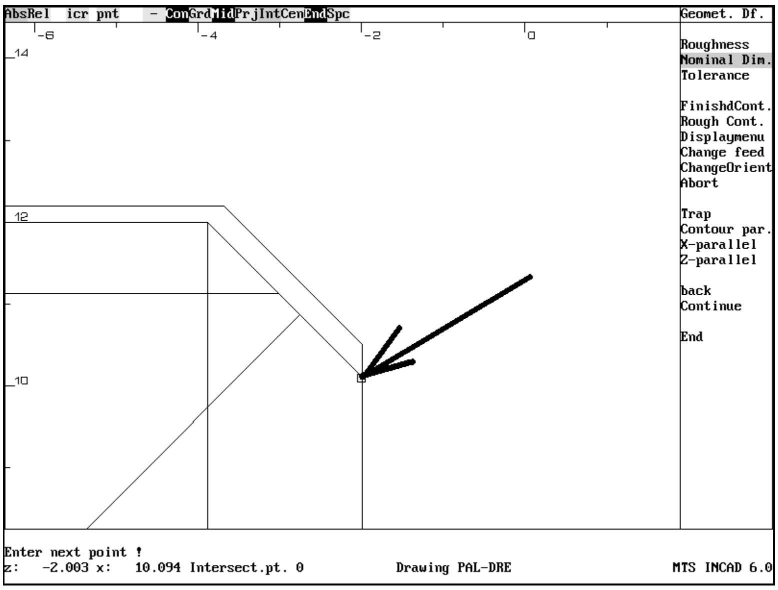
selection of the contour to be machined

The system asks for the starting point of the contour

Enter starting point !

Use the zoom function with <F6> for showing the details. Click with the mouse at the following point. Use the automatic trapping function „Int“ by selecting in the swith line.

starting point

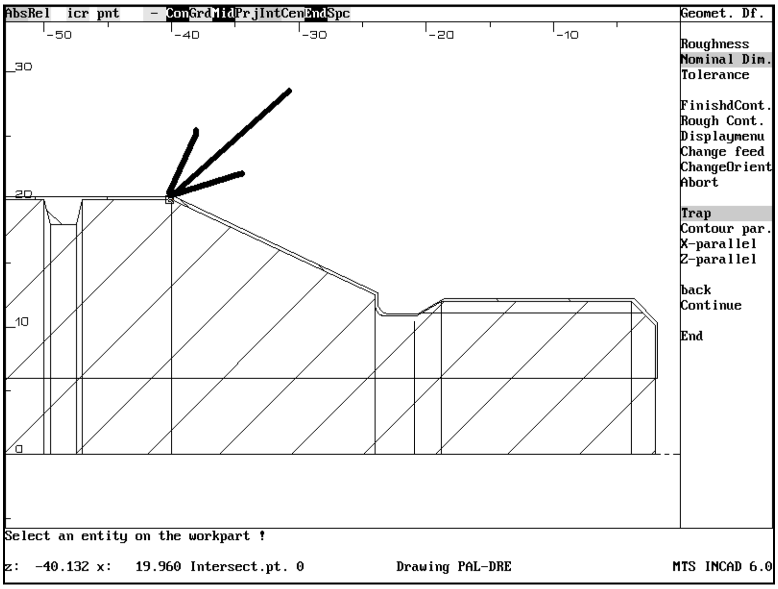


Enter next point !

The system asks for the next point. Activate the trap function with the switch „Trap“ and click with the mouse at the following point.

next point of the contour

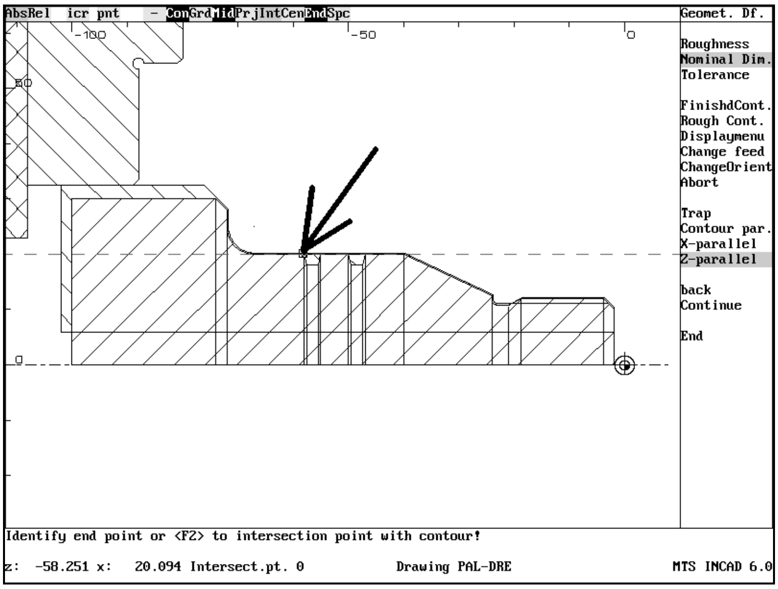
using the function „Trap“



Now activate the the switch „Z-parallel“ and click with the mouse at the following point.

next point of the contour

using the function „Z-parallel“



tool selection
In the work plan and the set-up sheet the following machining sequence and tool are described.

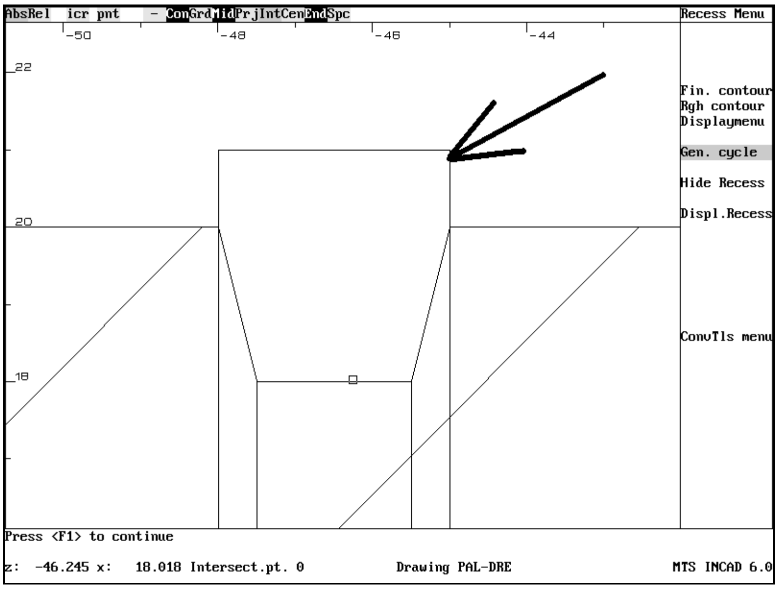
recessing	T06	external recessing tool
T06	external recessing tool	ER-SGTFL-1212/L/01.8-0 ISO30

We select with the cursor the sixth position, so that its frame is marked then. Go back to the turn menu by confirming with <F8>.

In the following dialog you can confirm all answers with <Enter> or change if desired the value with the keyboard. In our example we confirm all.

Coolant M08	Feed(mm/rev): 0.05	
m/min: 300	Speed lim. 3500	
Move to the tool changing point: Y	Approach opt.: 0	
ClearDist. 2.00	Allow in Z 0.00	Diam Allow 0.00

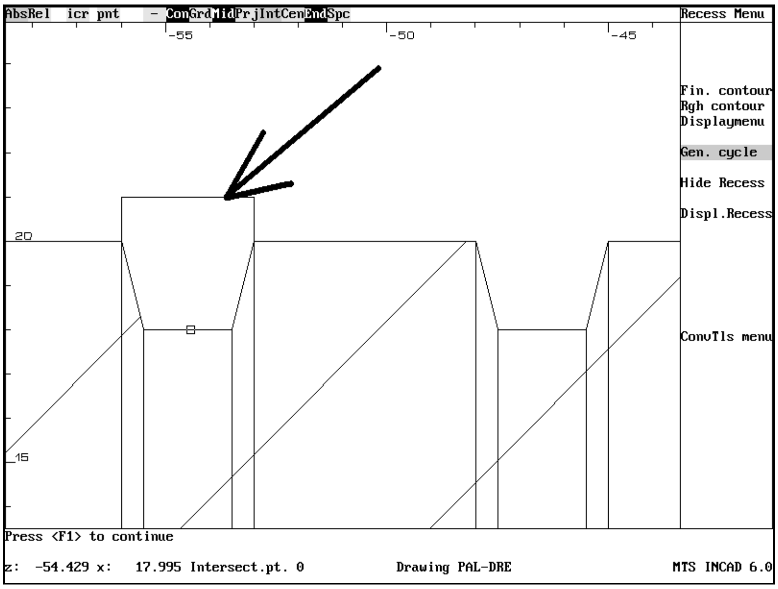
confirming the recessing cycle



Press <F1> to continue

Confirm all entries with <F1>. Do the same with the second recess:

confirming the recessing cycle for the second recess

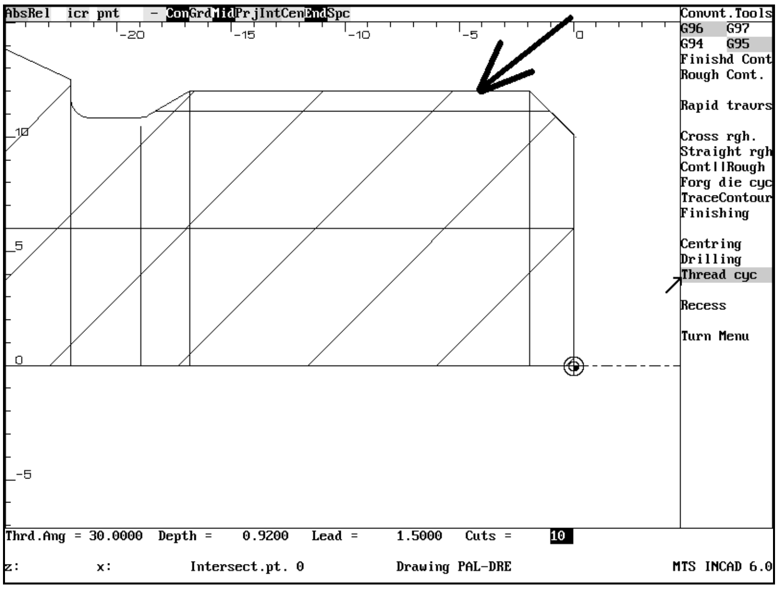


Press <F1> to continue

Confirm all entries with <F1>. Leave this menu with the menupoint „ConvTls menu“

defining outer diameter of threading and threading geometry
The system asks for the threading surface. Click on the following entity with the mouse.

selecting the threading surface



Enter the following values in this dialog:

Thrd.Ang = 30 Depth = 0.92 Lead = 1.5 Cuts = 10

Then the system asks for the start and the end point of the threading cycle.

Enter first point of cycle

Click with the mouse in the right of the thread and confirm the following prompt with <Enter>

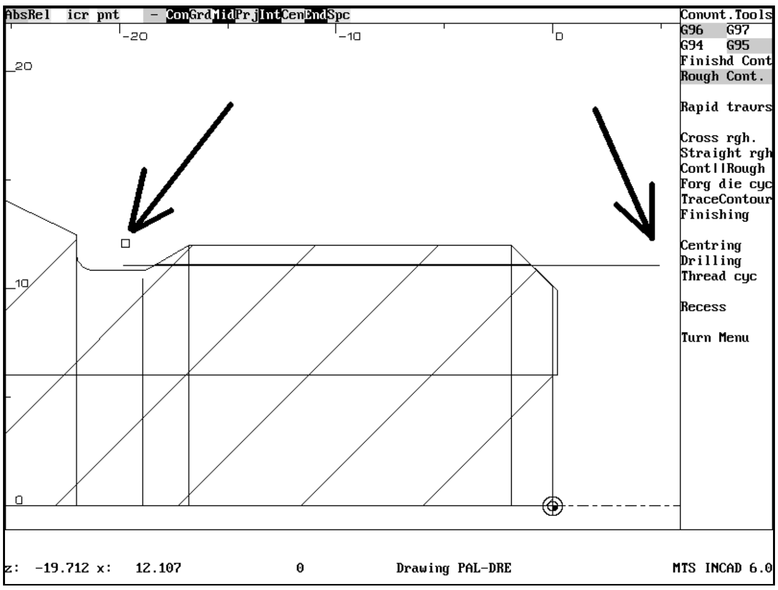
Allowance in Z: 0.0

Enter second point of cycle

Click with the mouse in the left of the thread and confirm the following prompt with <Enter>

Allowance in Z: 0.0

identifying first and second point of the thread



Leave this menu with the menupoint „Turn Menu“.

Then you can simulate or save this NC-program in the same way as in CAD-CAM-milling!