SIEMENS

SCE Training Curriculums

Siemens Automation Cooperates with Education | 02/2016

CNC Technology Module 700-010

DIN programGuide Basics

Cooperates with Education

Automation

SIEMENS

Suitable SCE trainer packages to accompany these training curriculums

- **SinuTrain for SINUMERIK Operate V4.7 Basic** – free download with no time limit
  www.siemens.com/sinutrain-downloads
- **SinuTrain Classroom License for SINUMERIK Operate V4.5 –** 6 single licenses + 40 student licenses;
  Order no.: 6FC5870-1TC41-0YA0
- **SinuTrain Student Licenses for SINUMERIK Operate V4.5 –** 300 hours **–** 20 student licenses;
  Order no.: 6FC5870-1SC41-0YA0
- **SinuTrain Classroom License for SINUMERIK Operate V4.4 –** 16 single licenses + 32 student licenses; Order no.: 6FC5870-1TC40-1YA0
- **SinuTrain Student Licenses for SINUMERIK Operate V4.4 –** 300 hours **–** 32 student licenses;
  Order no.: 6FC5870-1SC40-1YA0

Note that these trainer packages will be replaced with successor packages when required.
You can find an overview of the currently available SCE packages at: siemens.com/sce/tp

In-service training courses

Get in touch with your regional SCE contact for information on regional Siemens SCE in-service training courses.
siemens.com/sce/contact

Additional information on SinuTrain

In particular: Downloads, Getting Started, Videos, Tutorials, Manuals and Programming Guides.
siemens.com/sce/sinutrain

Further information on SCE

siemens.com/sce

Notes on use

The SCE training curriculum for CNC Technology has been created for the "Siemens Automation Cooperates with Education (SCE)" program especially for educational purposes for public educational and R&D institutions. Siemens AG assumes no responsibility for the content.

This curriculum may be used only for initial education with respect to Siemens products/systems. That is, it may be copied in part or in whole and handed out to trainees for use within the framework of their education. Transmission and reproduction of this curriculum as well as communication of its content is permitted within public educational institutions for educational purposes.

Exceptions require written consent from the Siemens AG. Send all related requests to scesupportfinder.i-ia@siemens.com.
Offenders will be held liable. All rights including translation are reserved, particularly if a patent is granted or a utility model or design is registered.

Use for industry customers is expressly prohibited. Commercial use of the curriculum is not permitted.

We would like to thank Michael Dziallas Engineering, MOSER CNC Training and all those involved for their support in creating this curriculum.

Contents

1. Objective

In this module, you will learn the basics of the "OPERATE DIN programGUIDE programming" using two examples each for turning and milling.

2. Basics

This section explains some general basics on the geometry and technology for the programming of milling and turning.

2.1 Basics of geometry for milling and turning

The basics of geometry presented here mainly relate to the graphic SINUMERIK contour computer. The screenshots used here serve to support the theory.
If you want to simulate the theory examples on the controller in advance:
Operating area "Program" > Create new part program > In text editor: horizontal softkey [Contour] > vertical softkey [Create contour] > ...

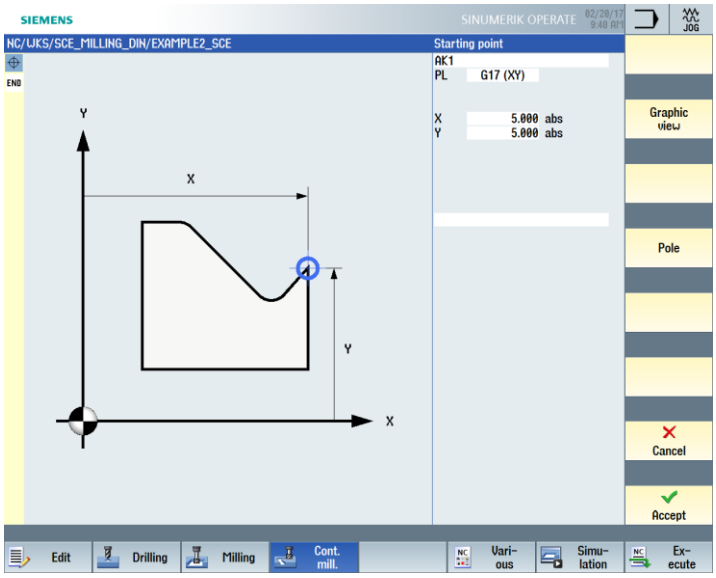
You can find a practical example in which this contour calculator is presented in context in section "Programming turning".

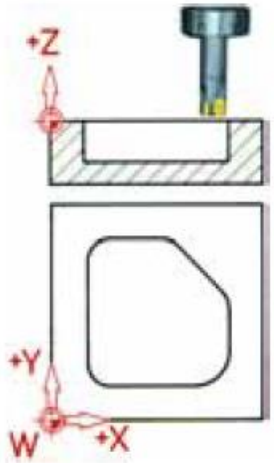
2.1.1 Tool axes and machining planes

MILLING

On universal milling machines, the tool is installed parallel to the main axes in most cases. These perpendicular axes are aligned to the main guideway of the machine according to DIN 66217 or ISO 841. The mounting position of the tool yields the corresponding machining plane.
For milling, the tool axis is usually the Z axis.

Tool axis Z - Plane G17





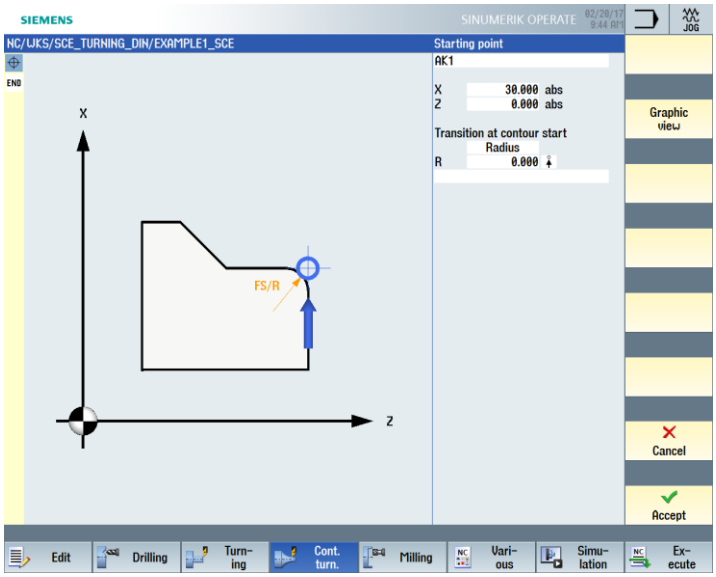
If the coordinate system shown is rotated accordingly, the axes and their directions in the respective machining plane (DIN 66217) will change.

TURNING

On universal turning machines, the tool is installed parallel with the main axes in most cases.
These perpendicular axes are aligned to the main guideway of the machine according to DIN 66217 or ISO 841.
For turning, the workpiece axis is the Z axis.

Rotational axis Z - Plane G18 *

Since the diameters of the turned workpieces are relatively easy to check, the dimensions of the transverse axis are specified based on the diameter. This allows the skilled worker to compare the actual dimensions directly to the dimensions in the drawing.

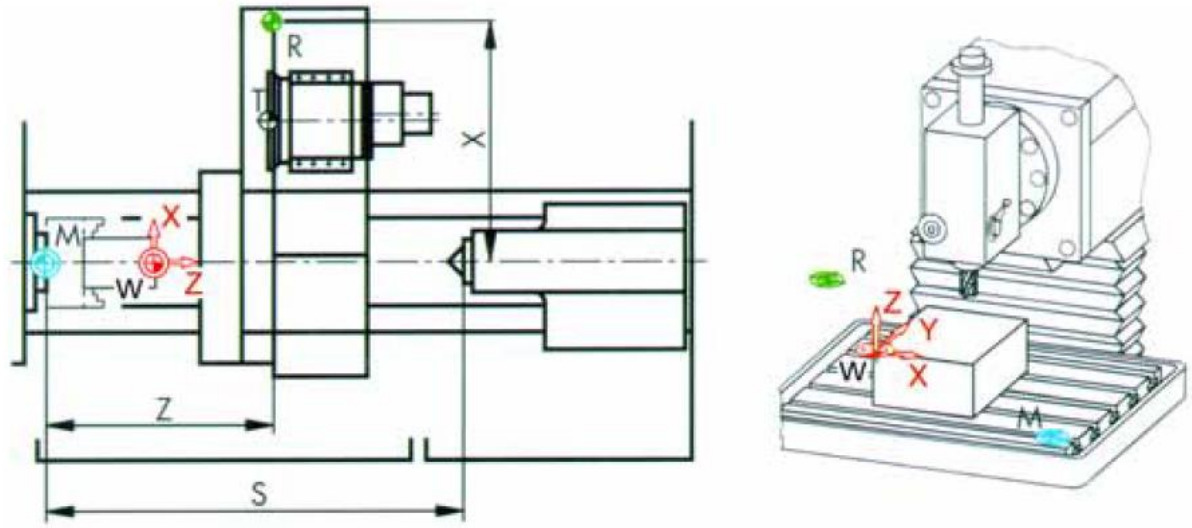


* All **turning operations** are programmed in plane **G18**.

Drilling and milling operations on the **front face** of the turned part are programmed in plane **G17**.

Drilling and milling operations on the **peripheral surface** of the turned part are programmed in plane **G19**.

A few important reference points are available so that a CNC – such as the SINUMERIK 840D sl – can orient itself in the existing work area by way of the measuring system.



Machine zero (M)

	The machine zero (M) is specified by the manufacturer and cannot be changed. It is located at the origin of the machine coordinate system for milling and at the contact surface of the spindle nose for turning.

Workpiece zero (W)

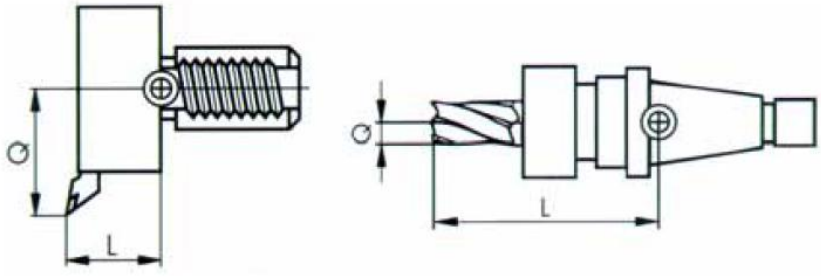
	The workpiece zero (W) - also called program zero - is the origin of the workpiece coordinate system. It can be selected freely. For drilling it should be located at the point from which the most dimensions start in the drawing. For turning the workpiece zero always lies on the axis of rotation and in most cases on the right end face

Reference point (R)

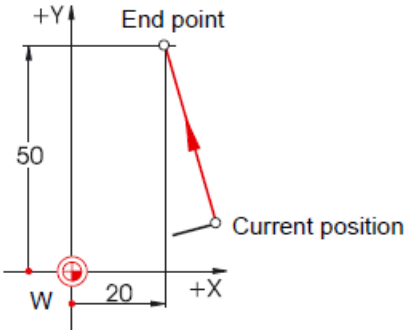
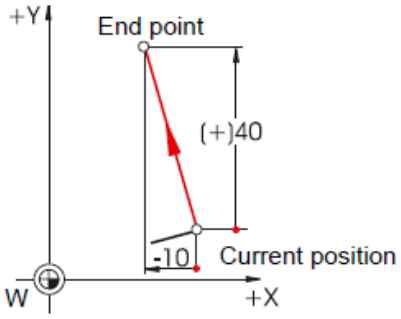
	The reference point (R) is approached for setting the position measuring system to zero, as the machine zero cannot be approached in most cases. This is how the controller finds its reference point in the position measuring system.

Tool carrier reference point (T)

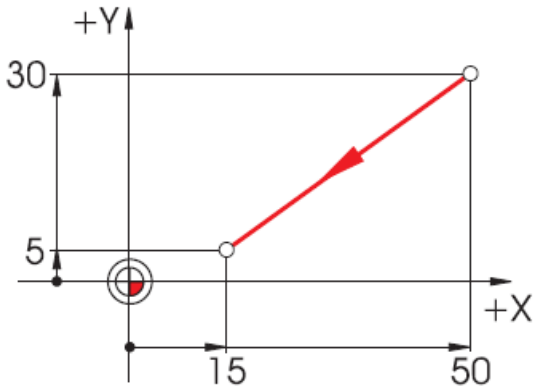
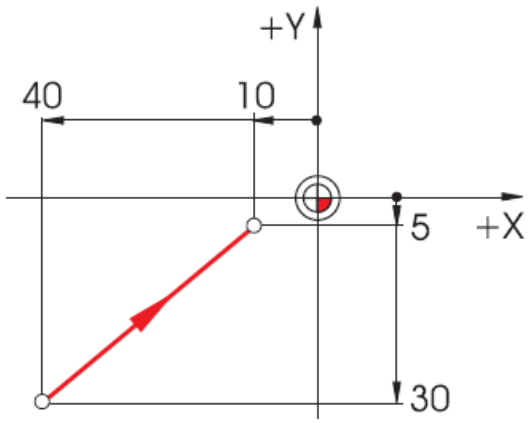
	The tool carrier reference point (T) is relevant for the setup with preset tools. The lengths L and Q values shown in the screenshot serve as tool offset values and are entered in the tool memory of the controller.



2.1.2 Absolute and incremental dimensions (milling)

Absolute inputs: The entered values are relative to the workpiece zero.		**Incremental inputs:** The entered values are relative to the current position.
*G90 Absolute dimensions		*G91 Incremental dimensions
		
G90 G0 or G1 X20 Y50		G91 G0 or G1 X-10 Y40
With absolute specifications, the **absolute** coordinate values of the **end point** must always be entered in the active coordinate system (the current position is not considered).		With incremental inputs, the **difference** values between the **current position** and the **end point** must always be entered while taking the direction into account.

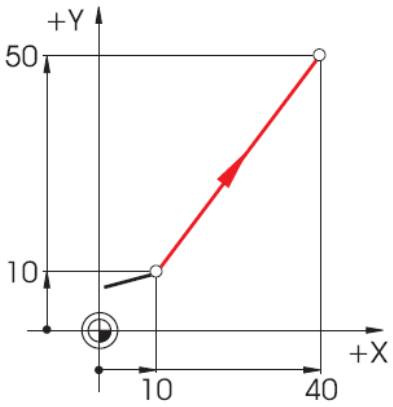
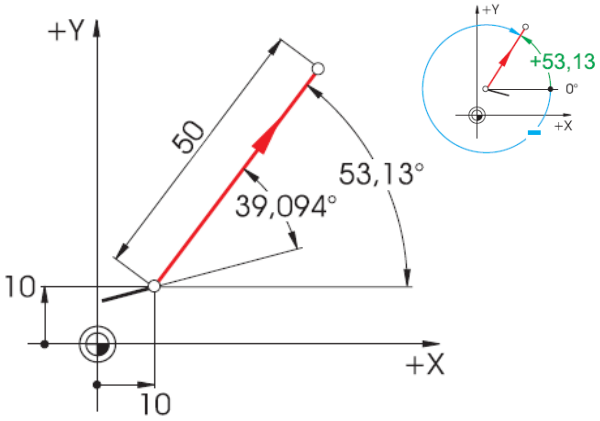
Two examples combining absolute and incremental dimensions can be found below:

	
G90 G0 or G1 X15 Y = IC(-25)	G90 G0 or G1 X = IC(30) Y-5

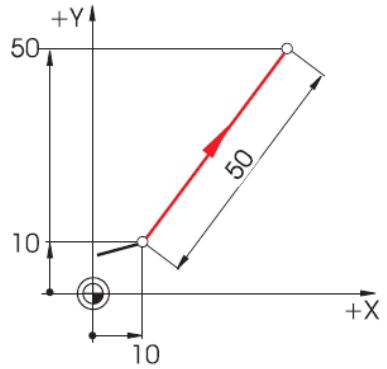
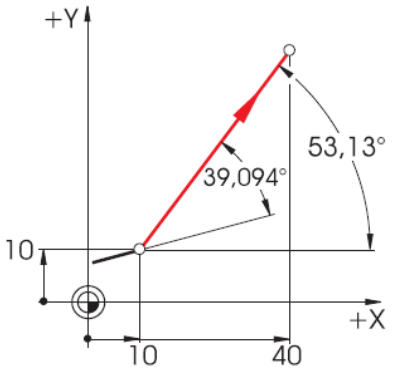
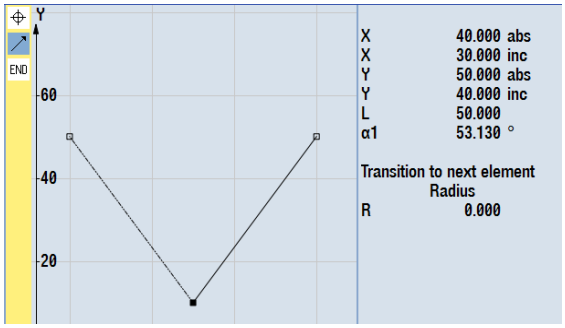
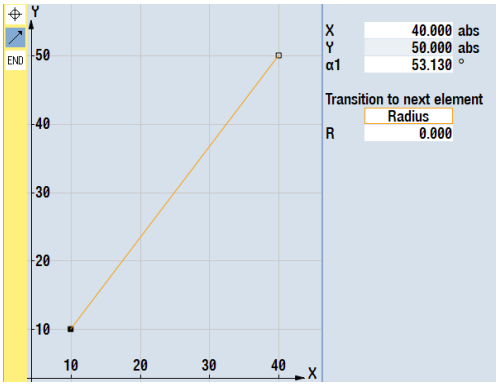
Explanation: IC = Incremental Count ➔ incremental dimension input

2.1.3 Cartesian and polar dimensions (milling)

To define the endpoint of a straight line it is necessary to specify two things. These may look like this:

Cartesian: Specification of the X and Y coordinates	**Polar:** Specification of the length and an angle
	
	Angle 53.13° = Starting angle relative to the positive X axis
G90 G0 or G1 X10 Y10 G0 or G1 X40 Y50	G90 G0 or G1 X10 Y10 G110 ; current position as POLE G0 or G1 AP = 53.13 RP = 50

Cartesian and polar inputs can be combined, e.g.:

Specification of the end point in Y and the length	Specification of the end point in X and an angle
	
 Two solutions possible with "Dialog select" and "Dialog accept"	

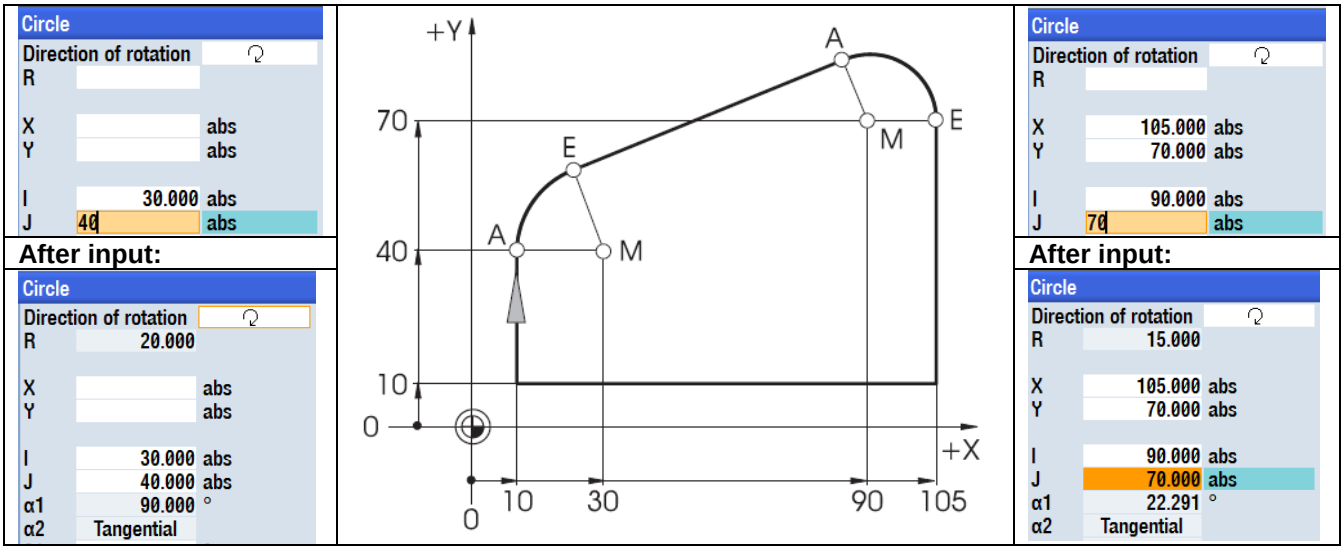
2.1.4 Circular motions (milling)

According to DIN, for circular arcs the end point of the arc (X and Y coordinates in the G17 plane) is specified along with the center point (I and J in the G17 plane).

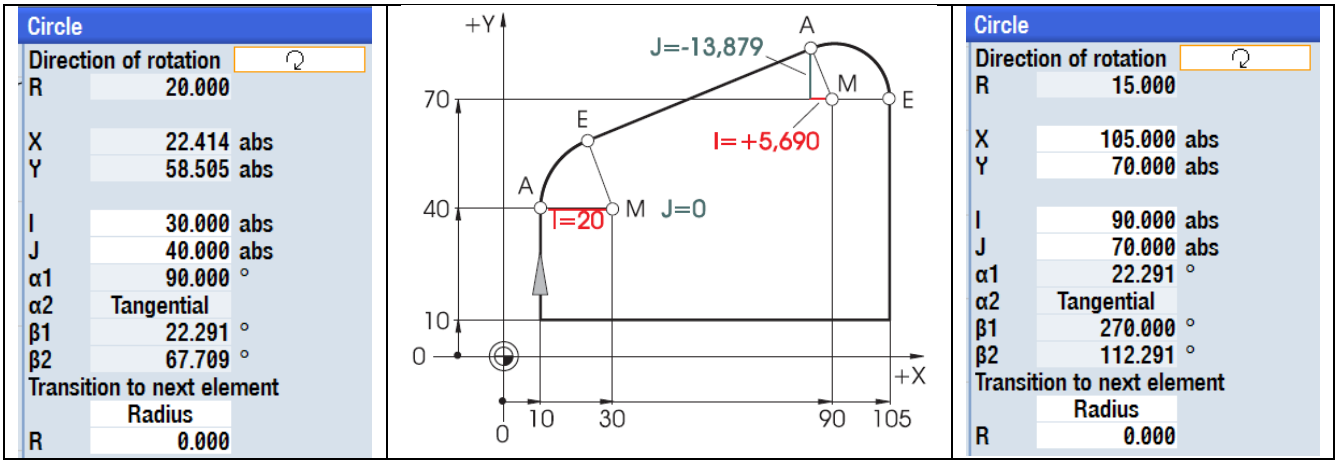
The SINUMERIK contour calculator gives you the freedom to use any dimension from the drawing for circular arcs, without having to carry out conversions.

The following example shows two – initially only partially defined – circular arcs.

Specification of the center point (absolute):



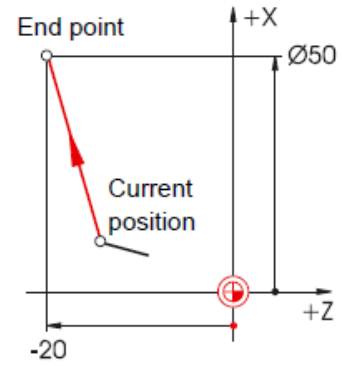
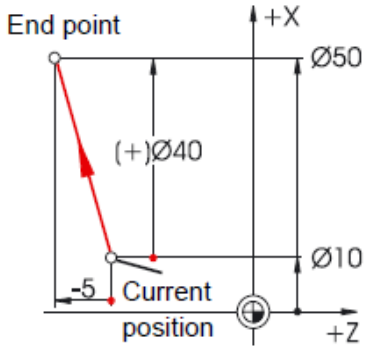
The following displays of values appear when you have entered all known dimensions and pressed the softkey in the input window of the respective arc.



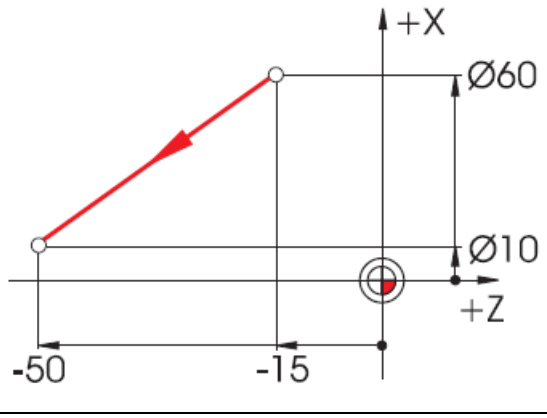
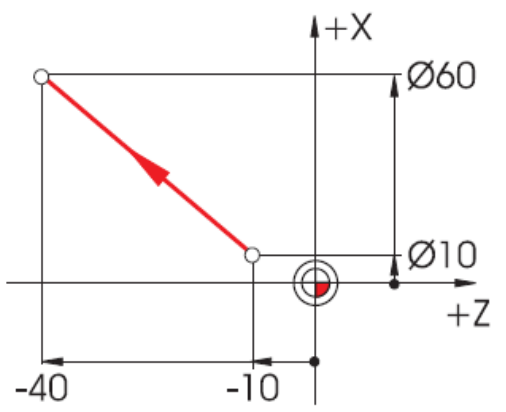
The inputs of the arcs for known starting and end coordinates in the text editor would be:

G2 X22.414 Y58.505 I20 J0 **G2 X105 Y70 I = AC(90) J = AC(70)**

2.1.5 Absolute and incremental dimensions (turning)

Absolute inputs: The entered values are relative to the workpiece zero.		**Incremental inputs:** The entered values are relative to the current position.
*G90 Absolute dimensions		*G91 Incremental dimensions
	Caution: As a deviation from DIN 66025, the I values are also entered and displayed relative to the diameter for the "DIAMON" setting applicable here.	
G90 G1 X50 Z-20		G91 G1 X40 Z-5
With absolute specifications, the absolute coordinate values of the end point must always be entered in the active coordinate system (the current position is not considered).		With incremental inputs, the difference values between the current position and the end point must always be entered while taking the direction into account.

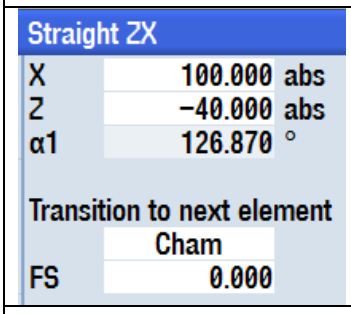
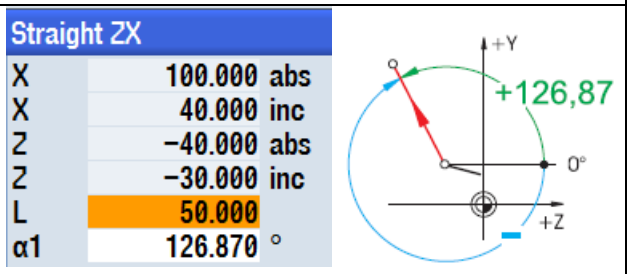
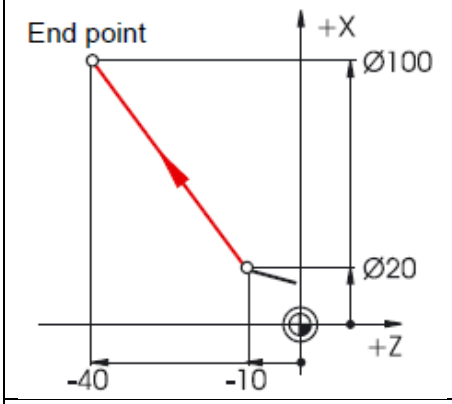
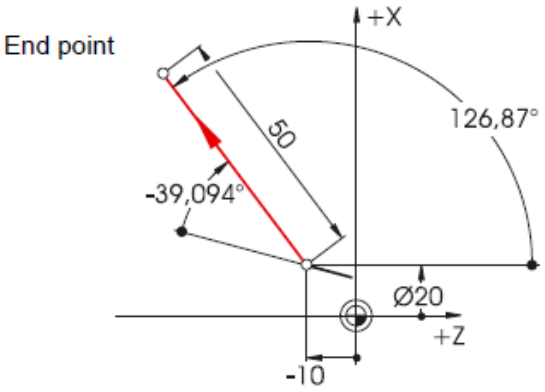
Two examples combining absolute and incremental dimensions can be found below:

	
G90 G1 X10 Z = IC(-35)	G90 G1 X = IC(50) Z-40

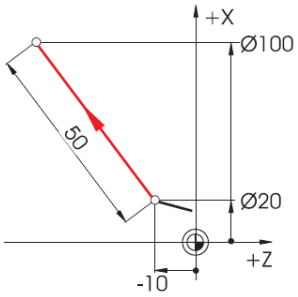
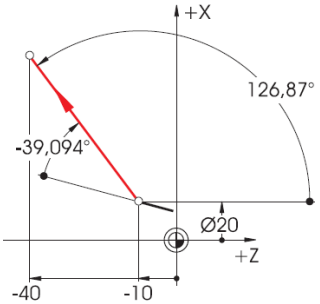
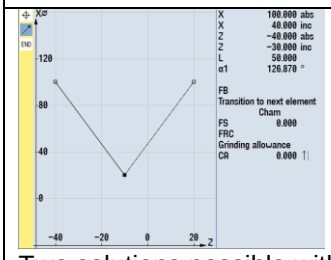
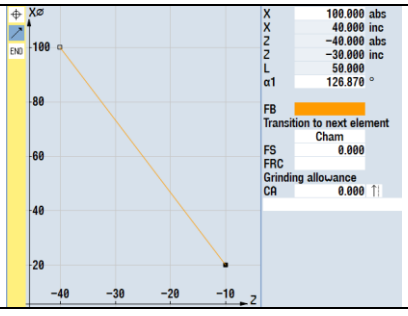
Explanation: IC = Incremental Count ➔ incremental dimension input

2.1.6 Cartesian and polar dimensions (turning)

To define the endpoint of a straight line it is necessary to specify two things. These may look like this:

Cartesian: Specification of the X and Y coordinates	**Polar:** Specification of the length and an angle
	
	
	Angle 126.87° = Starting angle relative to the positive Z axis or Angle -39.094° = Angle relative to previous element (39.094° = 360° – 320.906°)

Cartesian and polar inputs can be combined, e.g.:

Specification of the end point in X and the length	Specification of the end point in X and an angle
	
 Two solutions possible with "Dialog select" and "Dialog accept"	

2.1.7 Circular motions (turning)

According to DIN, for circular arcs the end point of the arc (X and Z coordinates in the G18 plane) is specified along with the center point (I and K in the G18 plane).

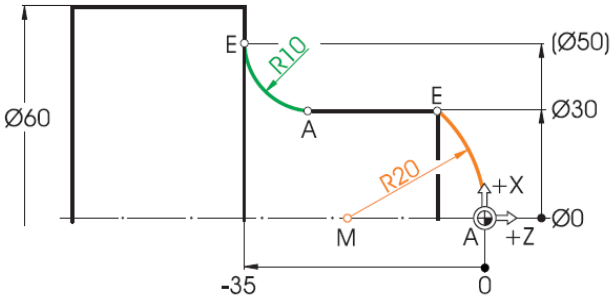
The SINUMERIK contour calculator gives you the freedom to use any dimension from the drawing for circular arcs, without having to carry out conversions.

The following example shows two – initially only partially defined – circular arcs.

Specification of the R10 arc:

After input:

Direction of rotation		
R	10.000	
X	50.000	abs
Z	−35.000	abs
I	50.000	abs
K	−25.000	abs
α1	180.000	°
α2	Tangential	
β1	90.000	°
β2	90.000	°
Transition to next element		
	Cham	
FS	0.000	



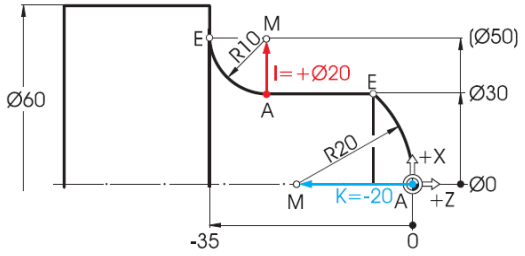
Specification of the R20 arc:

After input:

Direction of rotation		
R	20.000	
X	30.000	abs
Z	−6.771	abs
I	0.000	abs
K	−20.000	abs
α1	90.000	°
β1	138.590	°
β2	48.590	°
Transition to next element		
	Cham	
FS	0.000	

The following displays of all values appear when you have entered all known dimensions and pressed the All para-meters softkey in the input window of the respective arc.

Circle		
Direction of rotation		
R	10.000	
X	50.000	abs
X	10.000	inc
Z	−35.000	abs
Z	−10.000	inc
I	50.000	abs
I	10.000	inc
K	−25.000	abs
K	0.000	inc
α1	180.000	°
α2	Tangential	
β1	90.000	°
β2	90.000	°
FB		
Transition to next element		
	Cham	
FS	0.000	
FRC		
Grinding allowance		
CA	0.000	



Circle		
Direction of rotation		
R	20.000	
X	30.000	abs
X	15.000	inc
Z	−6.771	abs
Z	−6.771	inc
I	0.000	abs
I	0.000	inc
K	−20.000	abs
K	−20.000	inc
α1	90.000	°
β1	138.590	°
β2	48.590	°
FB		
Transition to next element		
	Cham	
FS	0.000	
FRC		
Grinding allowance		
CA	0.000	

The inputs of the arcs for known starting and end positions in the text editor would be:

G2 X50 Z-35 CR=10 **G3 X30 Z-6.771 I0 K-20**

2.2 Basics of technology for milling and turning

2.2.1 Cutting rate and speeds (milling)

The appropriate optimum speed of a tool depends on the cutting material of the tool, the material of the workpiece, and the tool diameter. In practice, this speed is often entered directly without a calculation, even if based on many years of experience. However, it is better to calculate the speed using the cutting rate taken from the relevant tables.

Determination of the cutting rate:
First, the optimum cutting rate is determined using either the manufacturer catalogs or a handbook.

Cutting material of the **tool**: **Hard metal**		Material of the **workpiece**: **C45**
		
V_c = 80 - 150 m/min: The mean value **v_c = 115 m/min** is selected		

Calculation of the speed:
This cutting rate and the known tool diameter are used to calculate the speed n.

$$n = \frac{v_c \cdot 1000}{d \cdot \pi}$$

The speed for two tools is calculated in the following example:

	d_1 = 63 mm	d_2 = 40 mm	
$n_1 = \frac{115mm \cdot 1000}{63mm \cdot \pi \cdot min}$			$n_2 = \frac{115mm \cdot 1000}{40mm \cdot \pi \cdot min}$
	$n_1 \approx 580 \frac{1}{min}$	$n_2 \approx 900 \frac{1}{min}$	

In NC coding, the speed is specified with the letter S (from "speed").

Therefore, the inputs are: S580 and S900.
The cutting rate of 115 m/min is achieved in each case with these speeds.

2.2.2 Feed per tooth and feedrates (milling)

In the previous section, you learned how to determine the cutting rate and calculate speeds. The tool can only perform machining if a feedrate is assigned to this cutting rate and speed for the tool.

The basic value required to calculate the feedrate is the characteristic "feed per tooth".

Determination of the feed per tooth:
Like the cutting rate, the value for the feed per tooth is also obtained from the handbook or the documents of the tool manufacturer.

Cutting material of the **tool**: **Hard metal**		Material of the **workpiece**: **C45**
		
Feed per tooth f_z = 0.1 - 0.2 mm: The mean value f_z = **0.15 mm** is selected		

Determination of the feedrate:
The feedrate v_f is calculated using the feed per tooth, the number of teeth and the known speed.

$$v_f = f_z \cdot z \cdot n$$

	d_1 = 63 mm, z_1 = 4	d_2 = 63 mm, z_2 = 9	
$v_{f_1} = 0,15\text{mm} \cdot 4 \cdot 580\frac{1}{\text{min}}$			$v_{f_2} = 0,15\text{mm} \cdot 9 \cdot 580\frac{1}{\text{min}}$
	$v_{f_1} = 348\frac{\text{mm}}{\text{min}}$	$v_{f_2} = 783\frac{\text{mm}}{\text{min}}$	

In NC coding, the feedrate is specified with F (from "feed").

Therefore, the inputs (rounded) are: F340 and F780.
The feed per tooth of 0.15 mm is achieved in each case with these feedrates.

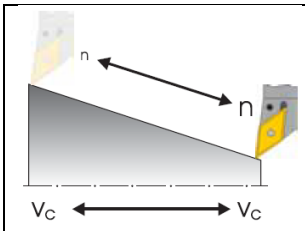
2.2.3 Cutting rate and speeds (turning)

Unlike for milling, the cutting rate is directly programmed most of the time for turning, in particular for roughing, finishing and grooving.
The desired speed is programmed only for drilling and (most of the time) for thread cutting.

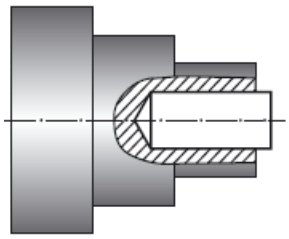
Determination of the cutting rate:
First, the optimum cutting rate is determined using either the manufacturer catalogs or a handbook.

Cutting material of the **tool**: **Hard metal**		Material of the **workpiece**: **Machining steel**
		
	v_c = 180 m/min	

Constant cutting rate v_c (G96) for roughing, finishing and grooving:

	To ensure that the selected cutting rate is maintained at every workpiece diameter, the speed is adjusted by the controller with command G96 = Constant cutting rate. This is carried out by DC motors or frequency-controlled three-phase motors. As the diameter decreases, the speed increases theoretically to infinity. To prevent danger caused by excessive centrifugal forces, a speed limit of, for example, 3000 rpm must be programmed. **The inputs are now G96 S180 LIMS = 3000.**

Constant speed n (G97) for drilling and thread cutting:

 $n = \frac{v_c \cdot 1000}{d \cdot \pi}$ d = 20mm (tool diameter) $n = \frac{120mm \cdot 1000}{20mm \cdot \pi \cdot min}$ $n \approx 1900 \frac{1}{min}$	Since a constant speed is used for drilling, the command G97 = Constant speed must be used here. The speed is dependent on the desired cutting rate (120 m/min selected here) and the tool diameter. **The inputs are now G97 S1900.**

2.2.4 Feed (turning)

In the previous section, you learned how to determine the cutting rate and calculate speeds. The tool can only perform machining if a feed is assigned to this cutting rate and speed for the tool.

Determination of the feed:
Like the cutting rate, the value for the feed is also obtained from the handbook, the documents of the tool manufacturer or practical knowledge.

Cutting material of the **tool**: **Hard metal**		Material of the **workpiece**: **Machining steel**
		
	Feed f = 0.2 - 0.4 mm The mean value **f = 0.3 mm** (often also referred to as mm per revolution in the workshop). **The input is now F0.3.**	

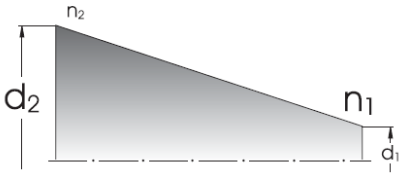
Relationship between feed and feedrate:

The constant feed f and the respective speed yield the feedrate vf.

$v_c = 180\frac{m}{min}$
$d_2 = 80mm$
$n_2 \approx 710\frac{1}{min}$

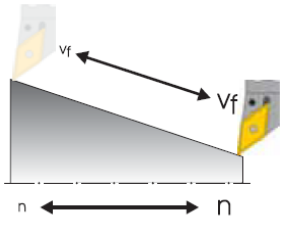
$v_{f2} = 710\frac{1}{min} \cdot 0,3mm$
$v_{f2} \approx 210\frac{mm}{min}$

$v_f = f \cdot n$



$v_c = 180\frac{m}{min}$
$d_1 = 20mm$
$n_1 \approx 2800\frac{1}{min}$

$v_{f1} = 2800\frac{1}{min} \cdot 0,3mm$
$v_{f1} = 840\frac{mm}{min}$

Since the speed is different, the feedrate is also different at the various diameters, despite having the same programmed feed.	

3. Operation

In this beginner's curriculum, the generic term "operation" is understood to mean all work sequences that take place in the direct interaction between the user and machine. After a basic introduction in section 2.1, the next subsection addresses the setup of tools and workpieces. In the third and fourth subsections, the emphasis is on production, thus on the execution of NC programs.

The 828D / 840D sl controllers are based on an open control concept that gives the machine manufacturer (and to some extent you as a user) a great deal of freedom. The controller can therefore be configured according to individual requirements. Accordingly, the actions to be taken may differ from those specified here in the details. Read the instructions of the machine manufacturer where applicable and carefully check your entries before starting the machine.

All instructions in this document relate to the operating software interface
SINUMERIK OPERATE software release 47 SP3 for SINUMERIK 828D SL / 840D sl.

3.1 Overview of the controller

In this section you will learn about the structure and use of the control components of the keyboard and screen.

Example screens:

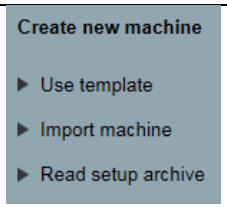
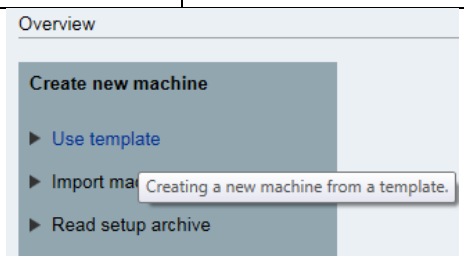
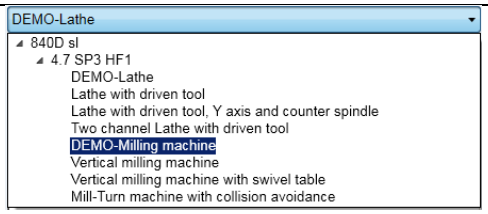
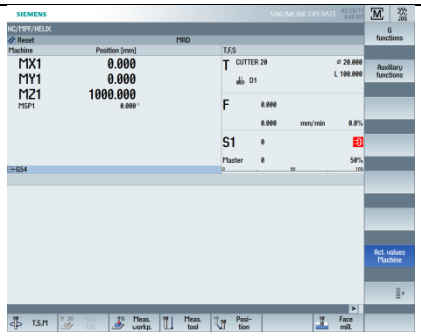
- **OP 019C Operator Panel Front**
in the picture on the left

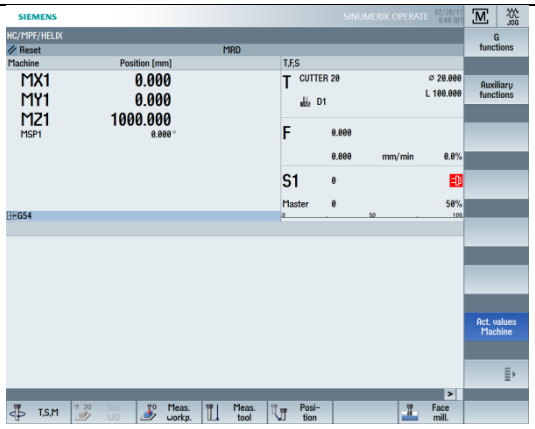
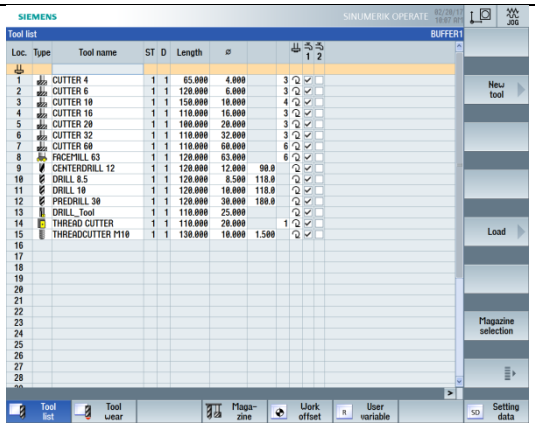
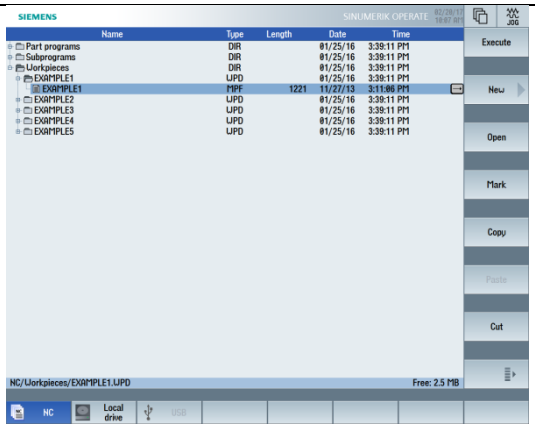
- **OP 010C Operator Panel Front**
in figure on right with TFT color screen, softkey bars (horizontal and vertical) and full mechanical CNC keyboard with 65 keys.
These components are used, in particular, for programming and handling of data.

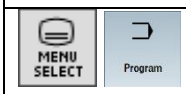
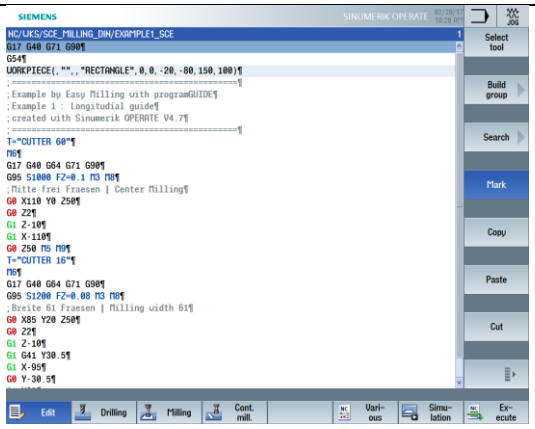
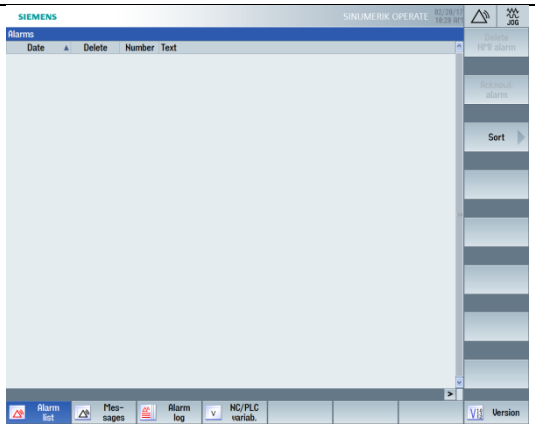
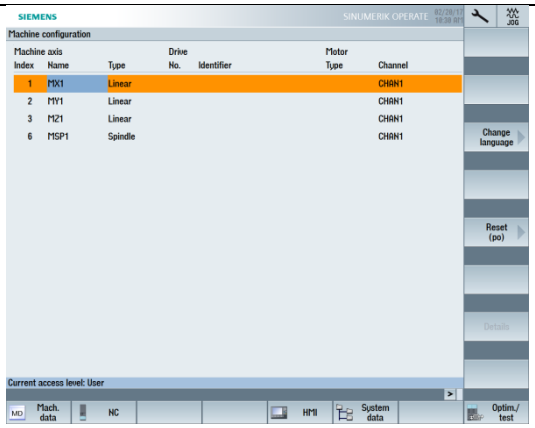
- **Machine control panel**
With override potentiometers.
The motions of the machine are directly influenced with this control panel.
It can be configured individually to some extent by the machine manufacturer.

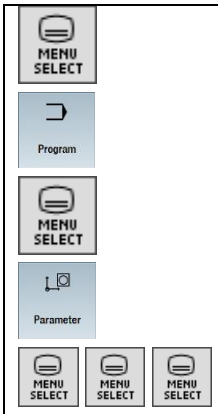
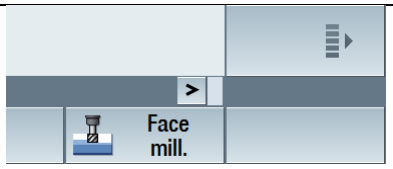
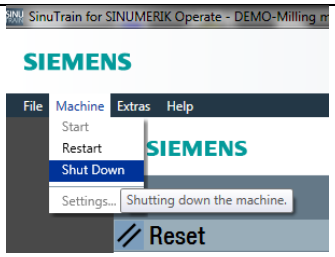
3.1.1 Switching on, switching between areas, and switching off

You start the software differently depending on whether you are training with the controller directly on the machine or are using the SinuTrain SINUMERIK training system on the PC (identical to the controller.)

Switching on			
	When you are working on the machine:		When you are working on the Windows PC:
	Switch on the machine using the main switch		Start the software using the icon on the desktop or the entry in the Start menu (Start > Programs > SinuTrain ... > SinuTrain START)
			No machines are yet created the first time you start SinuTrain.
			To create a new machine use the option “use template”.
			Choose the required machine out of a dropdown menu and afterwards click the button “create”.
			Now start the created machine with the "START" icon.
			When the selected machine starts, you are in the "JOG" operating area

Area changeover		
Keys / Inputs	Screen / Drawings	Explanation
MENU SELECT	With the <Area changeover> MENU SELECT you can – depending on the current operating situation – display the main menu with the six operating areas of the controls.	
		Active operating area "Machine – "JOG". You control the machine directly in this operating area. Here, you can move axes manually, bring tools to working position, scratch, set the zero point, and much more.
MENU SELECT Parameter		Switch to the "Parameter" operating area using the softkey. In the "Parameter" operating area, you manage your tools and the table of work offsets, among other things.
MENU SELECT Program manager		The programs are created and managed in the Program Manager. Green directory and green symbol on the file name indicates that this program is loaded in automatic mode

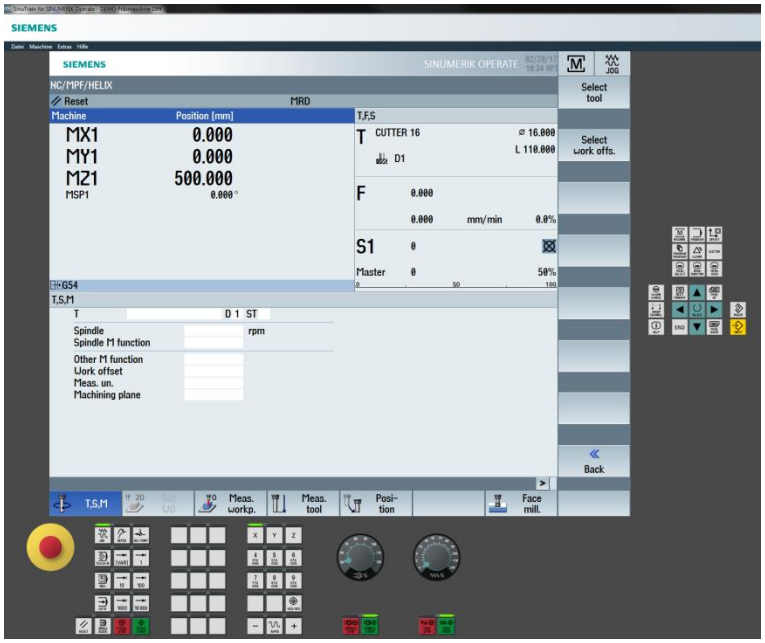
Keys / Inputs	Screen / Drawings	Explanation
		Programs are created, simulated and modified in the "Program" operating area.
		In the active "Diagnostics" operating area, alarms and messages are displayed and documented.
		The active "Setup" operating area is used for adjusting the NC data to the machine and is therefore relevant only for the system technician. It matters very little in day-to-day handling of the controller and will therefore not further considered in this document.

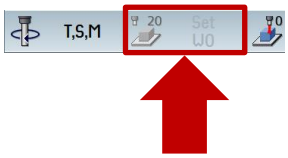
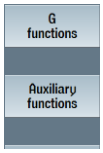
<table>
<tr><td> 
</td><td colspan="3">Through repeated pressing of the <Area changeover> key (
), you can toggle between the last two active operating areas, which is useful for programming when you want to view the tool data in parallel.
Try it out with the two operating areas "Program" and "Parameter".</td></tr>
<tr><td>  </td><td> 
</td><td colspan="2">The "etc. arrow" at the bottom right indicates that additional functions or applications are available.</td></tr>
<tr><td></td><td></td><td colspan="2"></td></tr>
<tr><td colspan="4">Switching off</td></tr>
<tr><td></td><td>When you are working on the machine:</td><td></td><td>When you are working with SinuTrain on the PC:</td></tr>
<tr><td>  </td><td>Read the information of the machine manufacturer!
Then switch off the machine with the main switch.</td><td> 
</td><td>Menu bar → Machine → Shut down</td></tr>
</table>

3.1.2 Keyboard and screen layout

During your first encounter with the controller interface, you have become familiar the <Area changeover> key (), the <etc.> key () and the horizontal softkeys of the main menu.

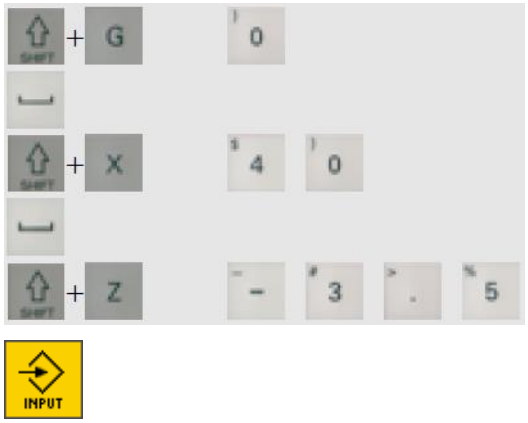
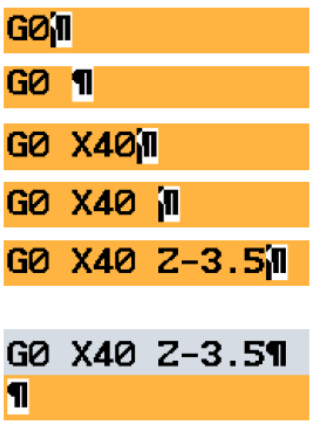
Next, you will be introduced systematically to other important keys using the SinuTrain screen keyboard as an example.



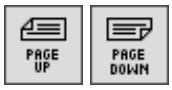
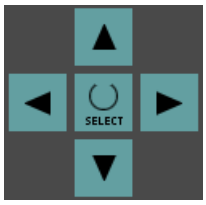
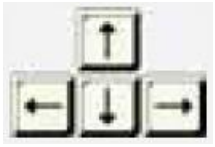
Icons	PC keys	Explanation
	**<F1> ... <F8>**	You use the horizontal softkeys (consecutively numbered from left to right) to navigate between operating areas. Within an operating area, you use these softkeys to access secondary menu areas and functions that can be called using the vertical softkeys.
	**Shift + <F1> ... <F8>**	You use the vertical softkeys (consecutively numbered from top to bottom) to call functions or to branch to other subfunctions when necessary, which in turn can be called via the vertical softkey bar.
	**<F10>**	The <Area changeover> key is used to display the main menu with the operating areas.
	**Shift + <F10>**	You can use the <Machine operating area> key to jump directly to the "Machine" operating area.
	**Shift + <F9>**	You expand the horizontal softkey bar with the <etc.> softkey.
	**<F9>**	You use the <Recall> key to close the window in the foreground and jump back to the higher-level menu. This function is always available when the key symbol is displayed above the first horizontal softkey.

Icons	PC keys	Explanation
		You use the numeric keypad to enter numbers and basic arithmetic functions. When combined with the <Shift> key, special characters (?, & ...) can be entered.
		You use the "QWERTY" keyboard to enter, for example, names of part programs and NC commands. (The name "QWERTY" originates from the key arrangement. A so-called "DIN" keyboard with an alphabetical key arrangement is often used on turning machines. The function is identical.)
		<Spacebar> for creating spaces
		When the <Shift> key is pressed, you can activate the top characters on keys with two characters and write uppercase letters (see above).
		You use the <Input> key to apply an edited value, open a directory or a file or select the end of the program line in the editor and jump with the cursor to the next, new line.

Practical example: You want to enter the following NC block on the control. G0 X40 Z-3.5

		Depending on the configuration of your controller ... • ... Uppercase letters are always written (even without <Shift> key • ... Unlike on the PC, the <Shift> key can be released again before pressing the letter key Every NC block is applied with <Input>.

The use of uppercase letters and the clear structuring of the inputs with spaces is customary and recommended. However, the controller also "understands" this input: **g0x40z-3.5**

	**<Esc>**	You use this key to acknowledge and delete the alarm that is marked with this symbol.
	**<F12>**	With the HELP key, you receive direct information/help for the current input. For example, the online help for certain commands is especially useful.
	**< Pos1>**	If multiple windows are displayed on the screen, only one of these has the focus, which can be recognized by the colored window frame. You can use this key to move from window to window (alternative: click in the window). Key inputs always relate to the window with the focus!
	**<Page Up> <Page Down>**	You use the <Page Up> and <Page Down> keys to move the scrollbar of a window. In this way, you can "page through" long part programs, for example.
	**<END>**	With this key you jump with the cursor to the line end.
  **Numeric keypad <5>**		You can move the cursor with the four <arrow keys>. You use the <Selection key> or <Toggle key> ( or 5 on the numeric keypad when "NUM LOCK" is off) to activate or deactivate a field or select between different selection options in text boxes (when the toggle symbol appears) (alternative: mouse click).
	**<Del>**	You use the <Delete> key to delete the selected character or the value of a text box in the editor.
	**<Backspace>**	You use the <Backspace> key to delete the character to the left of the cursor.

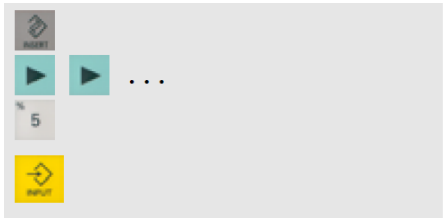
Practical example: You have written the NC block **G1 X0 F0.2** and finished with <Input>.

Now you want to change the feed to 0.3. Different ways to accomplish this:

	G1 X0 F0.2¶ G1 X0 F0.2¶ G1 X0 F0.¶ G1 X0 F0.3¶ G1 X0 F0.3¶ ¶	1st option: Because the last character is to be replaced, it is possible to use <END> to jump directly to the line end and use <Backspace> to delete the "2" (character left of the cursor).
	G1 X0 F0.2¶ G1 X0 F0.3¶ G1 X0 F0.3¶ ¶	2nd option: Alternatively, you can move the cursor to the right character by character and when the cursor stands on the "2", delete it with <DEL>.

	**<Insert>**	You use the <Edit> or <Undo> key to switch to editing mode in text boxes (see practical example). If you want to undo an accidental entry in editing mode, press  again. The overwritten entry is then restored.

Practical example: You want to change the value -82.47 to -82.475 in a text box without completely reentering the number. The value to be changed is selected (-82.470).

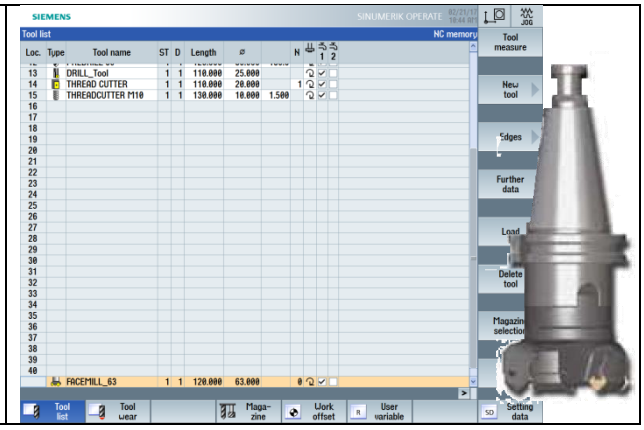
	Activate editing mode Position cursor Add the numeral 5 Accept the changed value (orange marking changes to the next text box)

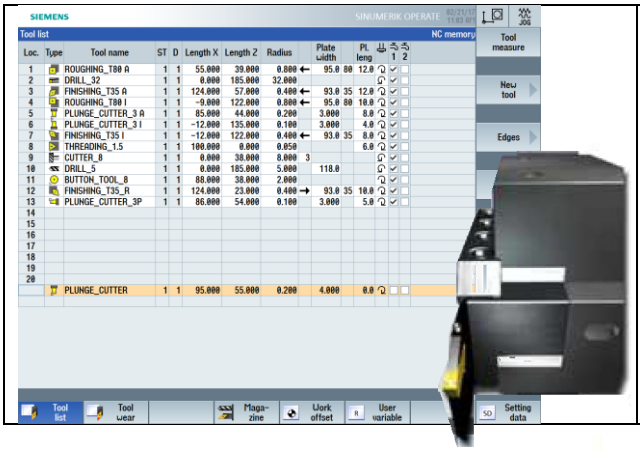
Screen layout

1. The current operating area (Machine, Parameter, etc.) is displayed here.
2. This area displays alarms and messages together with a number under which additional explanations can be looked up in the documentation.
3. Path and program name of the selected program
4. Channel status (Reset, Interrupted, Active)
5. Channel status display (e.g.: ROV: the correction for the feed also acts on the rapid traverse feed, SBL1: single block with Stop after each machine function block
6. Channel operating messages (e.g.: "Stop: EMERGENCY OFF active" or "Wait: Dwell time active")
7. Depending on the operating area, the middle part of the screen contains work windows (e.g. Program editor) and/or, as here, NC displays (Position, Feed, etc.).
8. Only one work window at a time has the focus. It is highlighted in color. Any inputs take effect in this window (see also key NEXT WINDOW).
9. The "Recall" symbol indicates that you are in a submenu and it can be exited with the key.
10. This area contains operator information, when available.
11. Horizontal softkeys: The operating areas or main functions are located here.
12. The "etc." symbol indicates that there are additional functions that you can display in the horizontal softkey bar with this key.
13. Vertical softkeys: Submenus and functions are located here.

3.2 Setup

In this section, you will learn the basic actions for the setup with the SINUMERIK 828D and 840D sl controllers. Based on a milling machine in "with **tool management**"* configuration, you will learn how to:
…Create a new tool in the tool management
…"Incorporate" this new tool in the real magazine and in the magazine image in the controller (section 2.2.1)





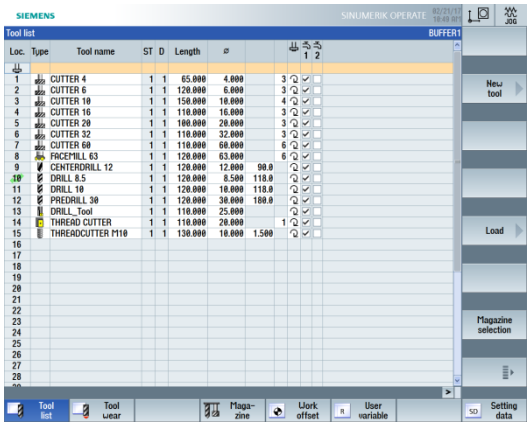
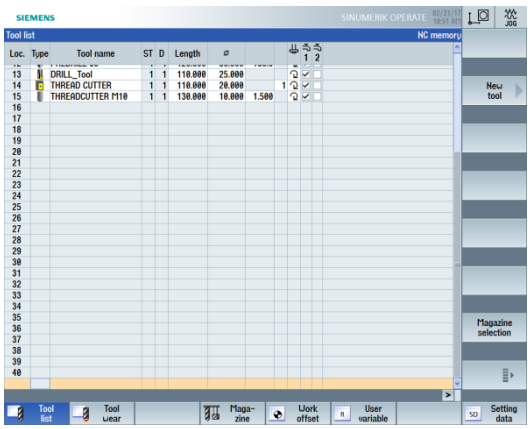
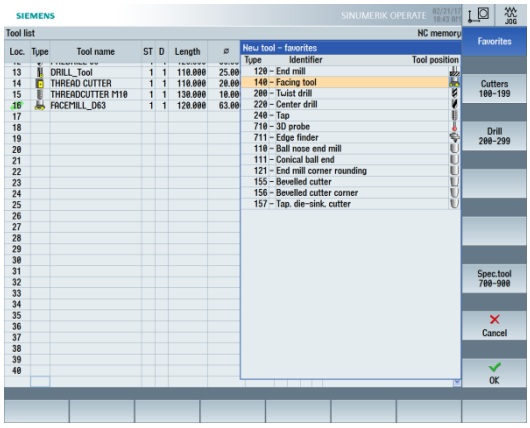
For machines with a simple "Tool compensation", tools are also managed, but with T numbers instead of names.
For turning machines, in particular, where all tools on the turret can be easily surveyed, this simpler configuration meets practical requirements.
This "with **tool compensation**" configuration is described in section 2.2.2.*

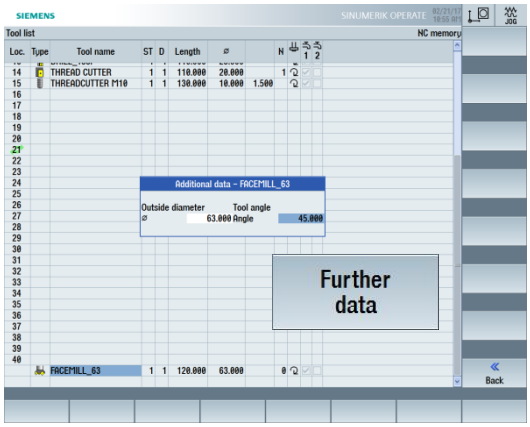
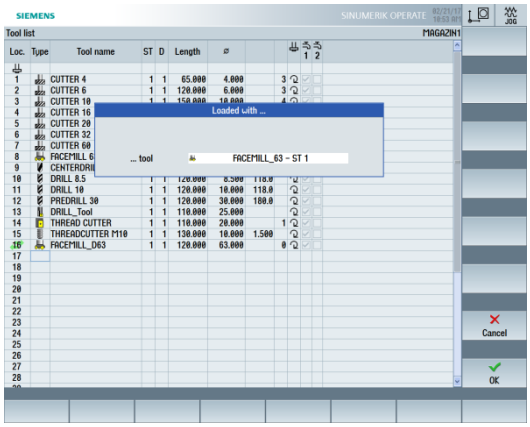
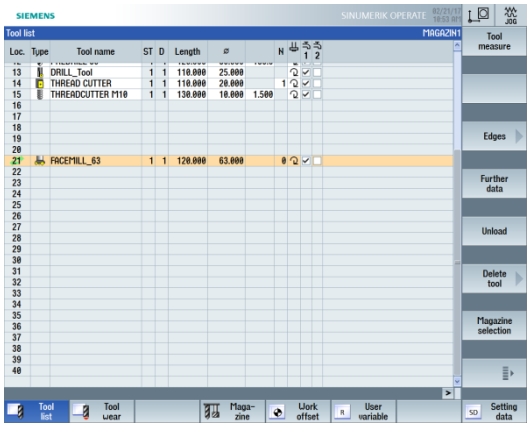
Section 2.2.3 contains a list of all tools that will be used in the subsequent example programs. Section 2.2.4 covers scratching and zero point setting.

* The procedure can be readily transferred to the other technology in each case!

3.2.1 Tool management: Creating a milling tool

Let us assume you want to create a 63 mm milling head in the tool management and load it into any free magazine location. Follow the steps below:

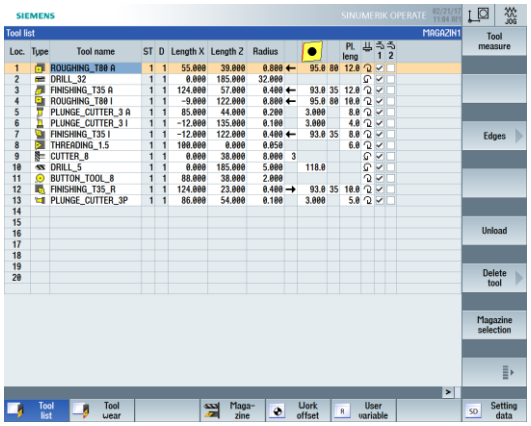
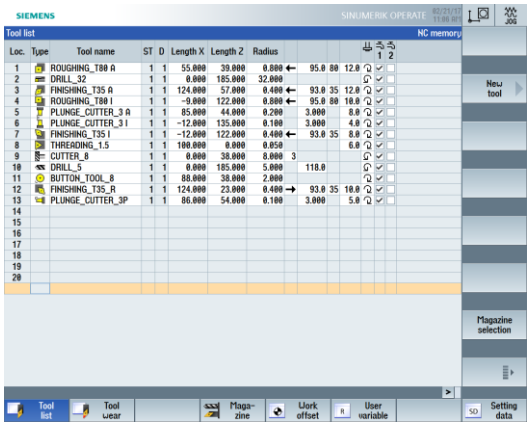
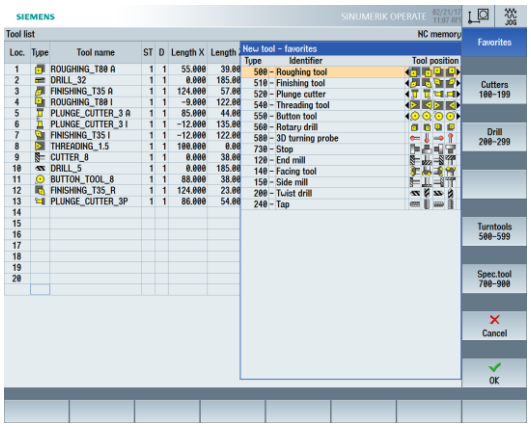
Creating a tool		
Keys / Inputs	**Screen / Drawing**	**Explanation**
		Open the "Parameter" operating area in the main menu. The tools are displayed in order of ascending location number in the "Tool list" by default.
		Place the active input line at the end of the tool list. There, the vertical softkey "New tool" is always active.
		Now select your desired tool using the different categories. In our case, this is the "Facing tool". Accept this with “OK”.

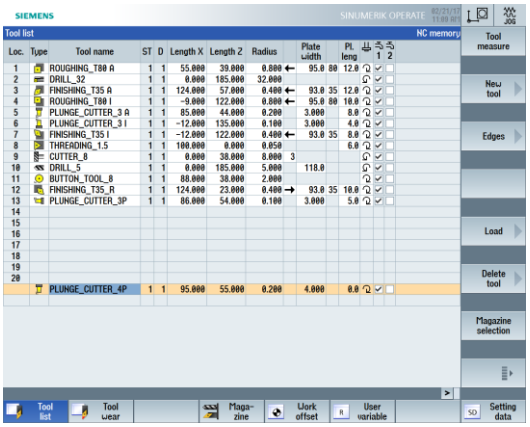
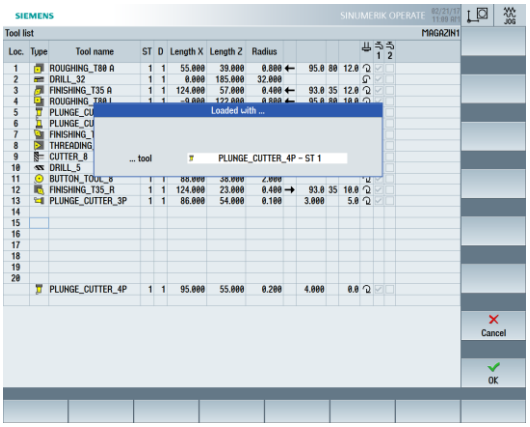
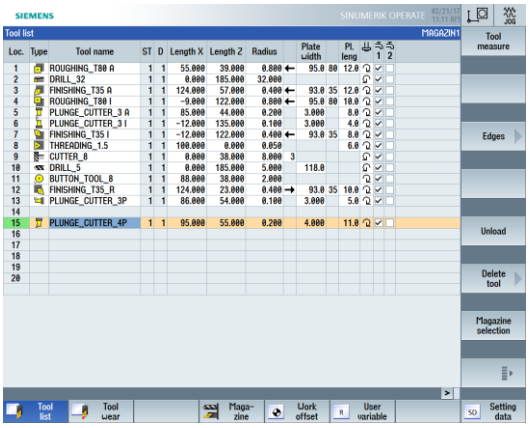
New tool		After selection of the tool, the next step is to enter the name and the tool data, such as length, diameter, etc. For some tool types, there is also "Further data" for the tool description. Be sure to take notice of this.
Load		Select the desired magazine location where you want to load the tool and press the vertical softkey "Load". You can now use the Select key  to select your desired tool and then accept it with “OK”.
		The tool is now loaded at the desired location in the tool list by the software. The physical loading depends on the method of accessing the tool magazine. Refer to the machine manual of the machine manufacturer for this.

3.2.2 Tool management: Creating a turning tool

Let us assume you want to create a plunge cutter with 4 mm width in the tool management and load it into any free magazine location. Follow the steps below:

Creating a tool

Keys / Inputs	Screen / Drawing	Explanation
MENU SELECT Parameter		Open the "Parameter" operating area in the main menu. The tools are displayed in order of ascending location number in the "Tool list" by default.
New tool		Place the active input line at the end of the tool list. There, the vertical softkey "New tool" is always active.
New tool		Now select your desired tool using the different categories. In our case, this is the "Plunge cutter". Accept this with “OK”.

		After selection of the tool, the next step is to enter the name and the tool data, such as Length X, Length Z, etc.
		Select the desired magazine location where you want to load the tool and press the vertical softkey "Load". You can now use the Select key  to select your desired tool and then accept it with “OK”.
		The tool is now loaded at the desired location in the tool list by the software. The physical loading depends on the method of accessing the tool magazine. Refer to the machine manual of the machine manufacturer for this.

3.2.3 Tools of the example programs

In the preceding sections, you have created one milling tool and one turning tool by way of example. In the example programs of sections 3 and 4, the tools listed below will be used. In order to simulate these programs using the simulation graphic, you must also create these tools in the "Parameter" operating area beforehand. (You can also use "your own" tools of the same type with a different name. Make sure to use this name when programming the tool call.)	

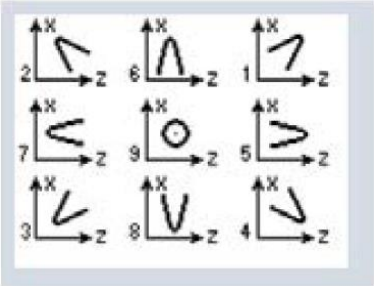
Tools in the milling programs

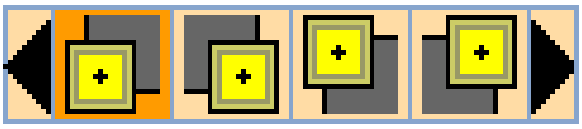
Type	Name	Cutting edge data (excerpt)	
140 Facing tool	**CUTTER 60**	D1	Ø 60
120 End mill	**CUTTER 20**	D1	Ø 20
120 End mill	**CUTTER 16**	D1	Ø 16
120 End mill	**CUTTER 10**	D1	Ø 10
220 Center drill	**CENTERDRILL12**	D1	Ø 12
200 Twist drill	**DRILL 8_5**	D1	Ø 8.5
200 Twist drill	**DRILL 10**	D1	Ø 10
240 Tap	**THREADCUTTER M10**	D1	Ø 10

A large number of tool types are available for milling.

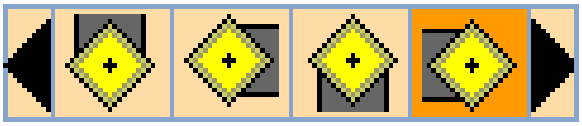
Tools in the turning programs

When turning tools are created, a key role is also played by the cutting edge position, in addition to the cutting edge radius and length compensation that you can determine by scratching or using a tool presetting unit.
Below is the graphic representation of the selection of the cutting edge position.
Classic selection graphic:

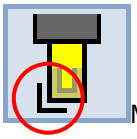




Side positions



Neutral positions



Measure left corner of plunge cutter!

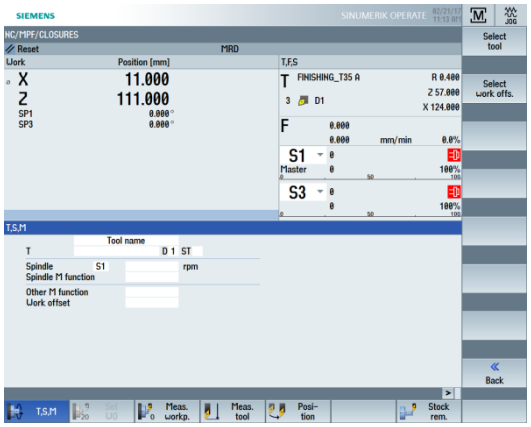
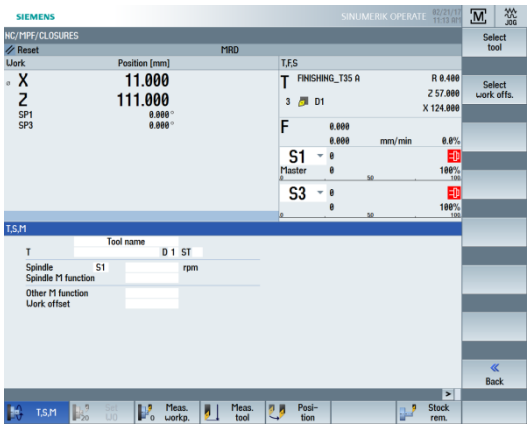
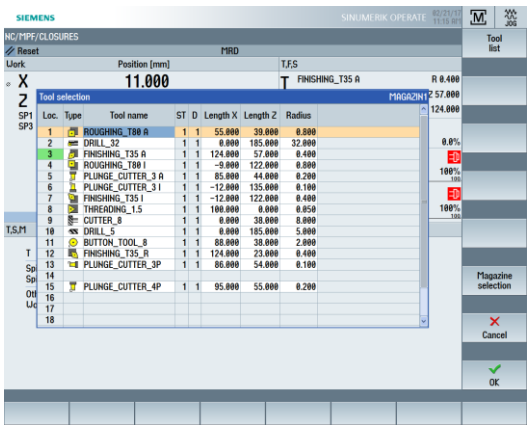
Type	Name	Cutting edge data (excerpt)	
500 Roughing tool	**ROUGHING_T80 A**	D1	Radius 0.8, cutting edge position 3
510 Finishing tool	**FINISHING_T35 A**	D1	Radius 0.4, cutting edge position 3
540 Threading tool	**THREADING_3.5**	D1	Radius 0.28, cutting edge position 8
520 Plunge cutter	**PLUNGE_CUTTER_3 A**	D1	Radius 0.2, cutting edge position 3 width 3
200 Twist drill	**DRILL_5**	D1	Diameter 5, cutting edge position 7
205 Solid drill	**SOLIDDDRILL_16**	D1	Diameter 16, cutting edge position 7

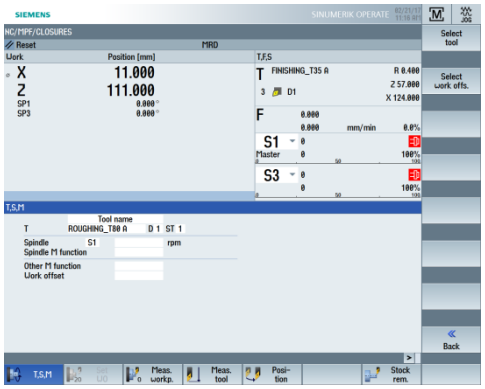
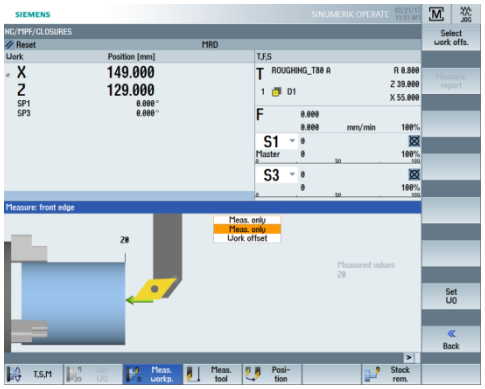
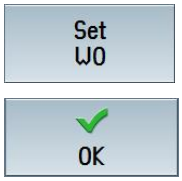
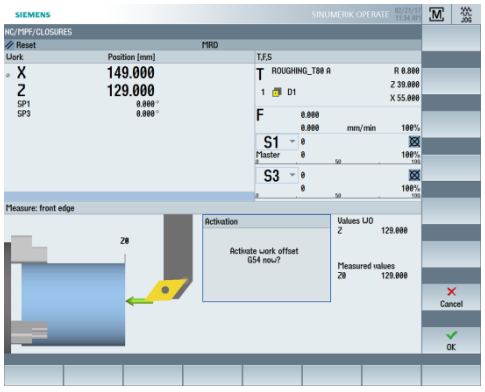
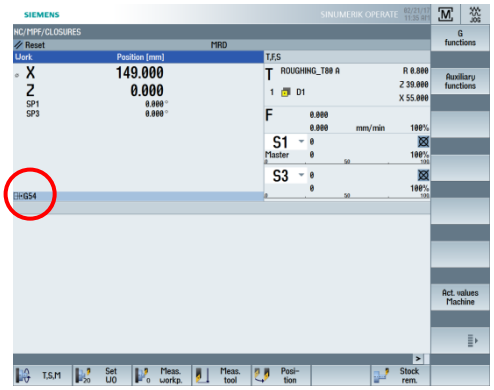
As for milling, a large number of tool types are also available for turning.

3.2.4 Scratching the workpiece and setting the zero point

With scratching, you carefully move a previously measured tool to the workpiece until the tool "scratches" the workpiece. Based on the compensation data of the tool and the current position of the tool carrier, the controller can calculate the zero offset that which the coordinates of the NC program relate to.

The scratching and workpiece zero setting is therefore a direct interaction of the controller and machine or of the tool and clamped workpiece.

Keys / Inputs	Screen / Drawing	Explanation
		Select "JOG" mode and first retract the turret with the axis direction keys to prevent a collision when the turret is rotated.
		Then select the "T,S,M" softkey and the "Tool" text box. Whether you call the tool with the tool name or location number is up to you.
		Then use the "Select tool" softkey and select the "Roughing_T80A" roughing tool. Accept with “OK”.

		Complete the text boxes for the spindle, e.g. 500 rpm and the corresponding direction of rotation. SPINDLE Enable → FEED Enable → CYCLE START
 		Carefully traverse the tool with the axis keys, a separate handheld device or electronic handwheels until it touches the workpiece.
		Press the "Meas. workp." softkey and enter the set position for the current position of the tool in the approached axis. In the example: G54 Z = 0
		Now press the "Set WO" softkey and confirm the next dialog window with “OK”.
		The first adjustable zero offset G54 is now activated and the workpiece zero in the Z-axis is set.

3.3 Managing and executing programs

This section describes the execution of an existing program. Assuming that an executable and tested program is already available (see sections 3 and 4 for programming) you will learn here how to import it, if necessary, from a USB stick to the controller, load it from the program manager and execute it.	

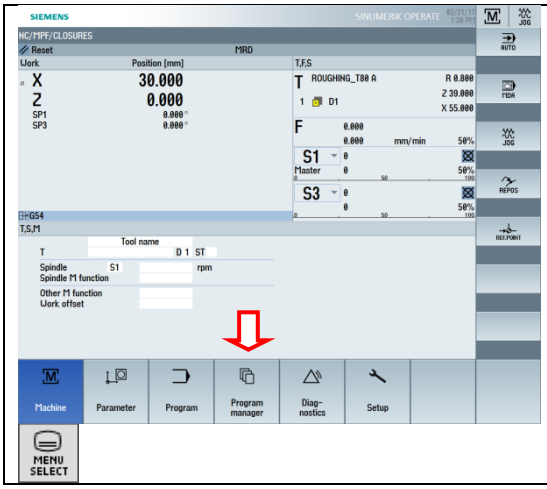
3.3.1 Storing and reading in data on a USB stick

Your SINUMERIK controller provides you with different possibilities for reading out and reading in data.

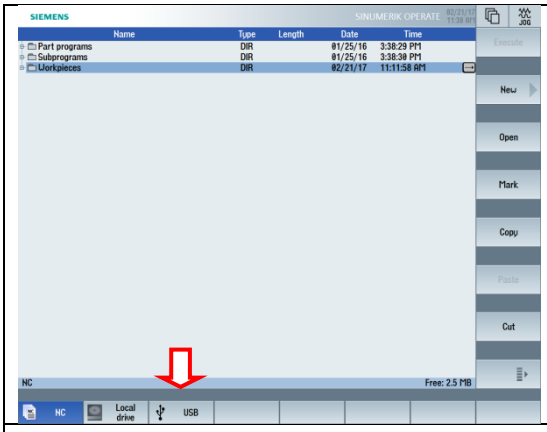
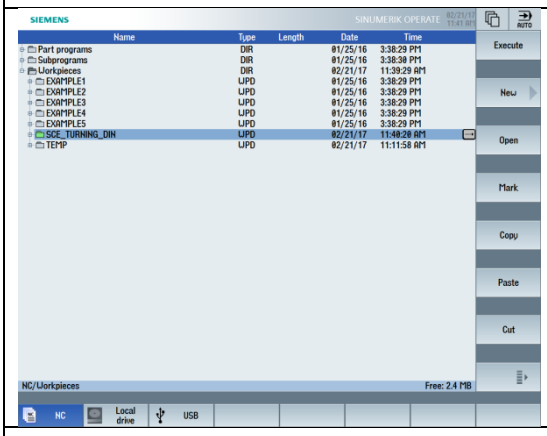
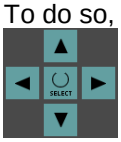
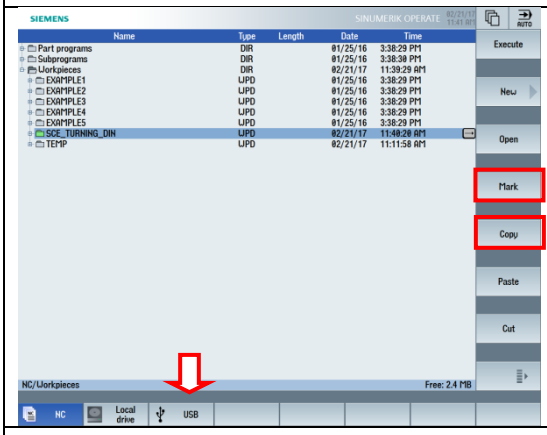
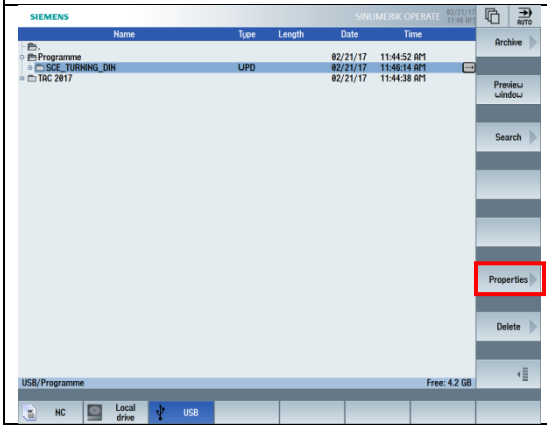
Here, for example, you will learn how to exchange data between the controller and a USB stick.
To do so, insert a USB stick in the socket on your SINUMERIK.

Fig.: SINUMERIK 828D

SINUMERIK controller ➔ USB stick (read)

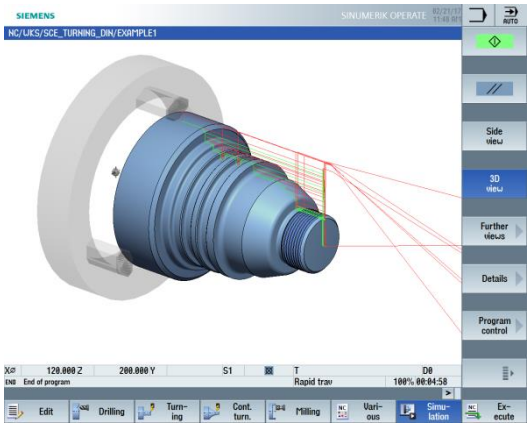
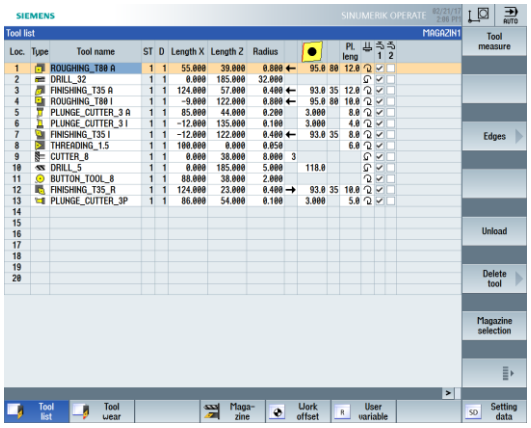
	First, select the "Program Manger" (NC memory) of your SINUMERIK using "Menu Select" and the "Program manager" main menu command.

You can find a detailed example of creating workpiece directories and programs in section 3.1.

	You now see the directories provided by SINUMERIK: Part programs, Subprograms and Workpieces. If the USB stick is not yet inserted, the softkey is also still deactivated (grayed out).
	Your task is now to copy (read out) the entire "SCE_TURNING_DIN" directory to the USB stick. To do so, navigate with the navigation arrows  to the desired directory.
	Insert the USB stick into its socket. The "USB" softkey activates after a short time. Press the "Mark" softkey followed by the "Copy" softkey. This saves the data to the clipboard.
	Use the "USB" softkey to change to the USB stick, and paste the data with the "Paste" softkey.

3.3.2 Selecting and executing programs

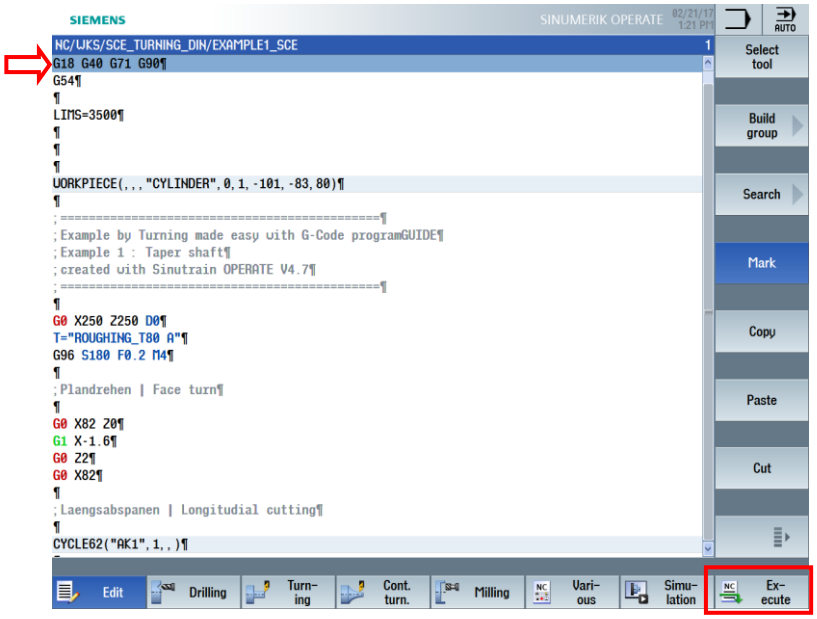
Before you start a program, you should take note of the following:

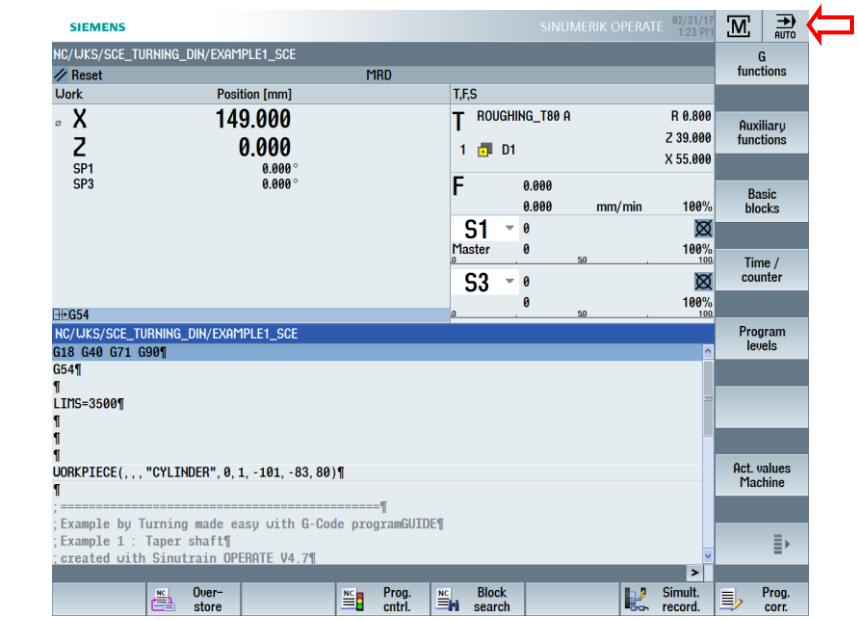
		Use the simulation to carefully check your program to determine whether it is error-free. **We assume no liability for the sample programs executed in this curriculum.** In particular, the cutting data (speed, feed, cut width) must be adapted as required to the conditions on your machine.
		Make certain that all tools used in the program are present in the magazine or turret and correctly measured.
		Make certain that the workpiece is reliably clamped and the zero point is correctly set. It may be advisable to first conduct a "dry run" of the program, i.e., without workpiece, in order to test all programmed movements for collision.
		Before the first test run of a program, rotate the feed override to ZERO so that you have time to intervene later in case the rapid traverse paths are incorrectly programmed.
		At especially critical points you should also switch to single block mode.

Now for the specific example: You have two options for selecting and executing.

1. Option:

You have written a program, tested it with the simulation and now want to implement it for cutting. To do so, position the active line to the topmost line in your program editor and select the "Execute" softkey. This loads the program automatically in "Automatic" mode, where it is ready for execution.





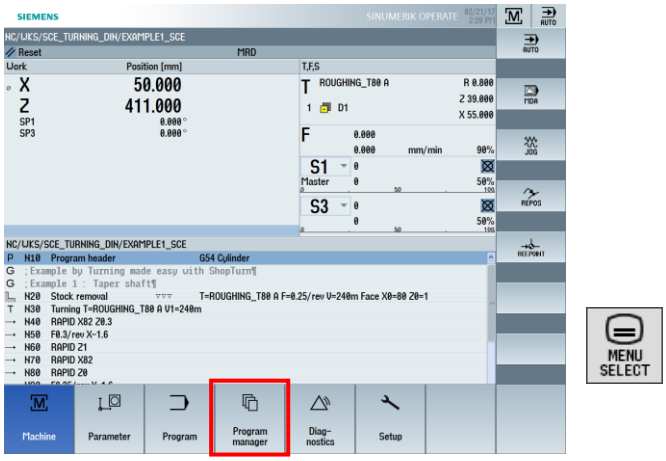
With "CYCLE START", "SPINDEL" and "FEED" Enable, the program is enabled for execution and started.

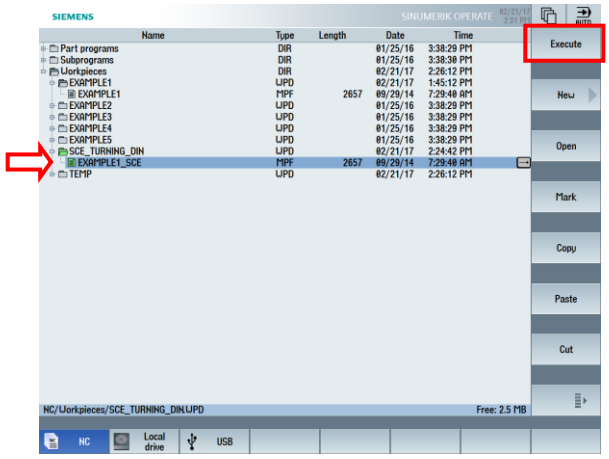
2. Option:

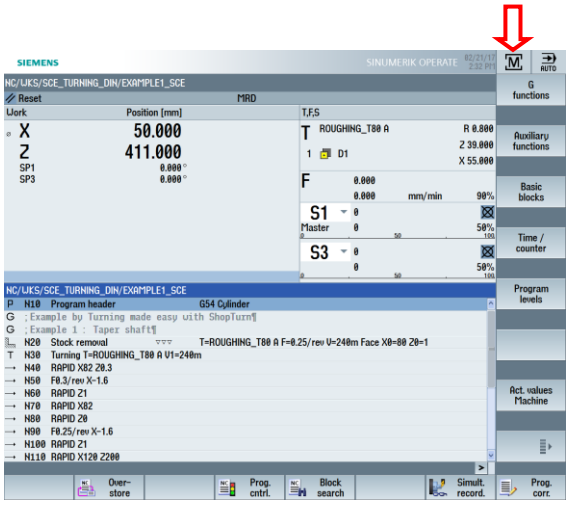
You want a previously written program that you have already executed multiple times and in which all inputs are known to be OK to be executed in "Automatic mode".

Please note the following: A collision may occur during machining if the chosen reference point or the tool data are incorrect.

Now implement the program for cutting. Follow the steps below: Select the Program Manager using "Menu Select" and the "Program manager" softkey, navigate with the navigation arrows to the desired program, and select the "Execute" softkey.





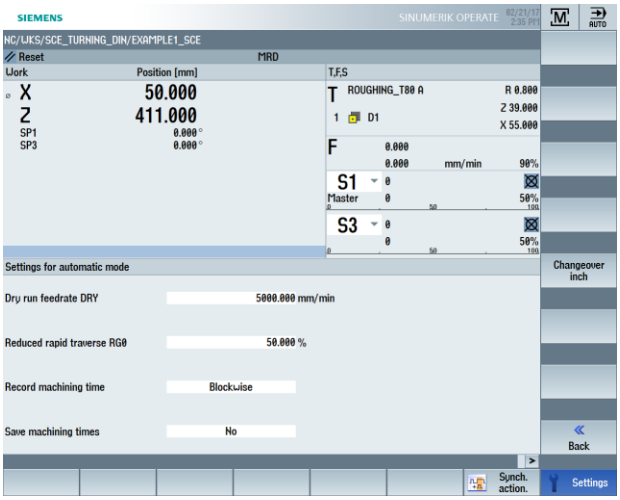


You have various options for influencing the program flow using the "Prog. cont." softkey.

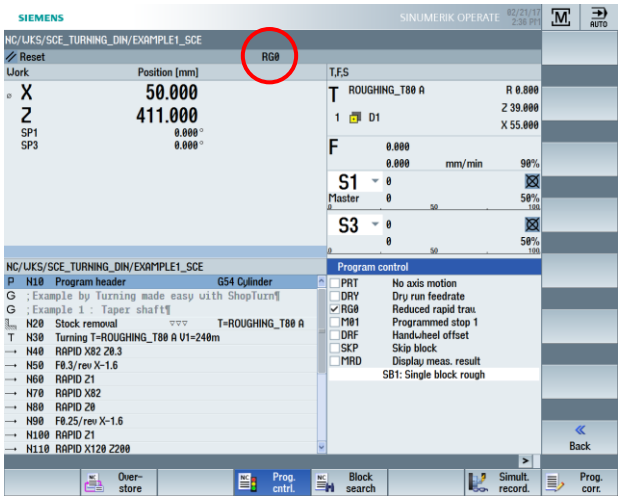
The status is displayed in the status line in the top half of the screen.

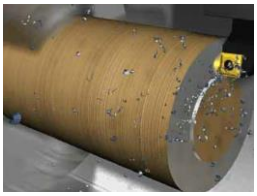
You can activate and deactivate single block mode (SBL1, SBL2 or SBL3) at any time with the <SingleBlock> key on the machine control panel.

To move in at reduced rapid traverse RG0, the percentage of the maximum rapid traverse of the machine, in the second level of the horizontal softkey bar, must be set under Settings, e.g. 50%.



Once the percentage is set, the reduced rapid traverse must also be selected under "Program control" with the toggle key.



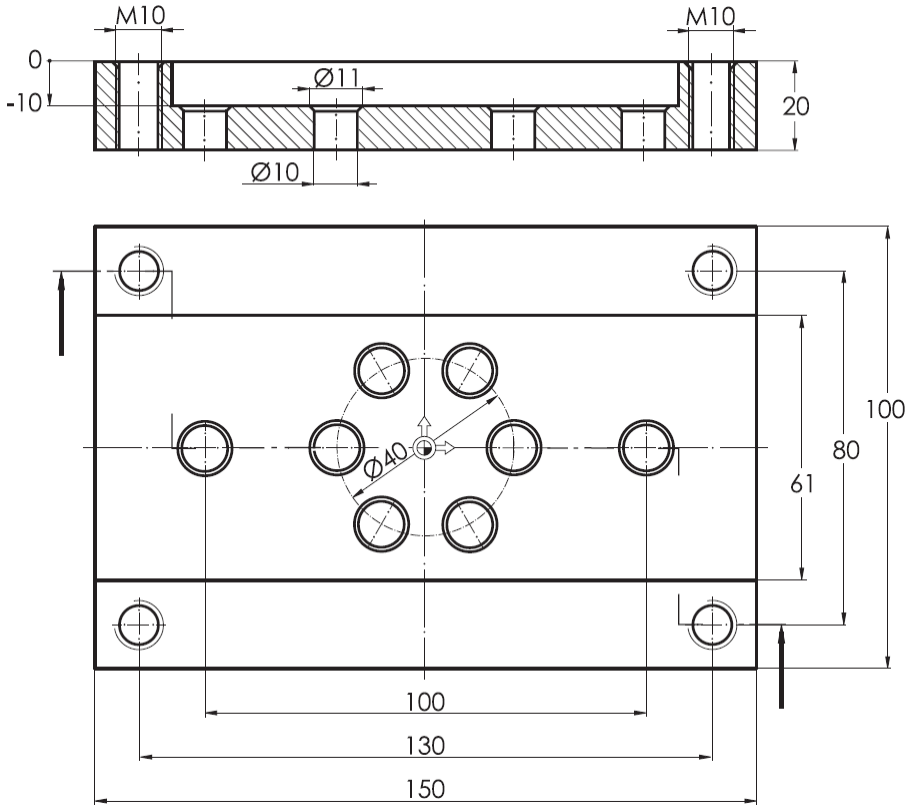
		Finally, start the program. Carefully turn the Feed override when you execute the program the first time. In critical situations:  or in an extreme case 

4. Programming MILLING

In this section, you will learn the programming of the SINUMERIK 828D / 840D sl controllers based on two simple sample workpieces.	
	Of course, not every feature of these powerful controls is covered here. But when you have reprogrammed these two workpieces, you will be prepared to continue learning on your own.

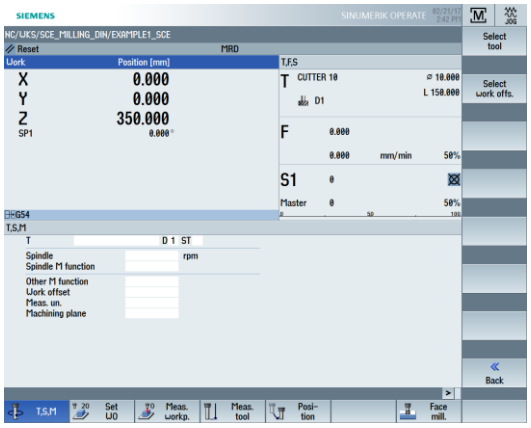
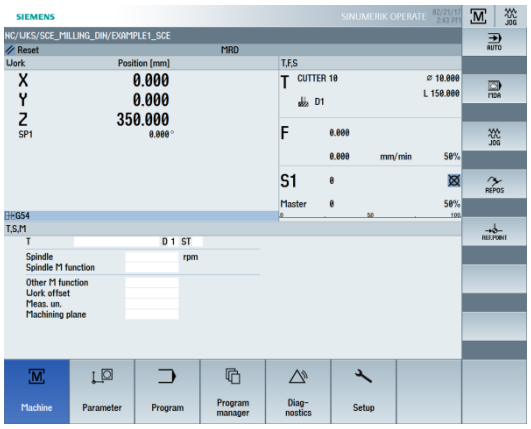
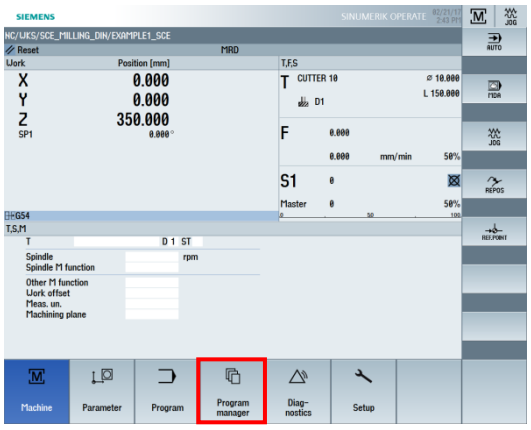
4.1 "Longitudinal guide" workpiece

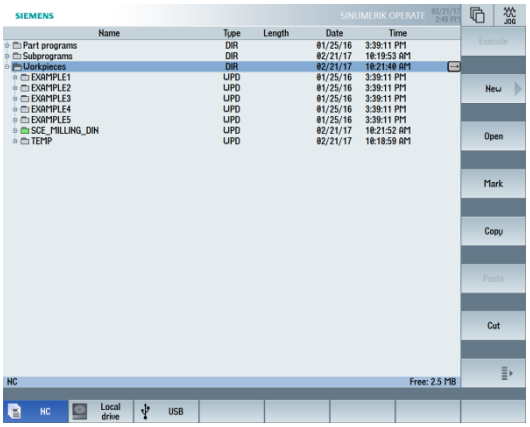
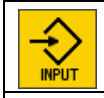
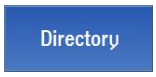
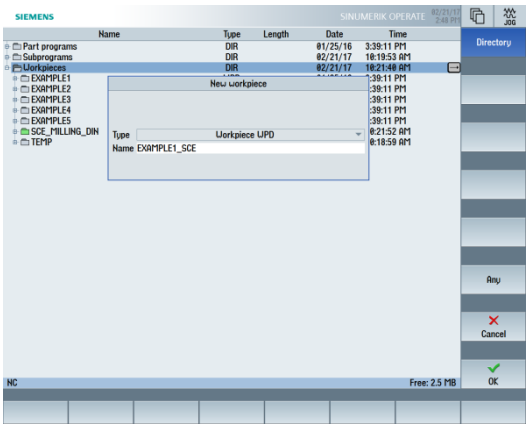
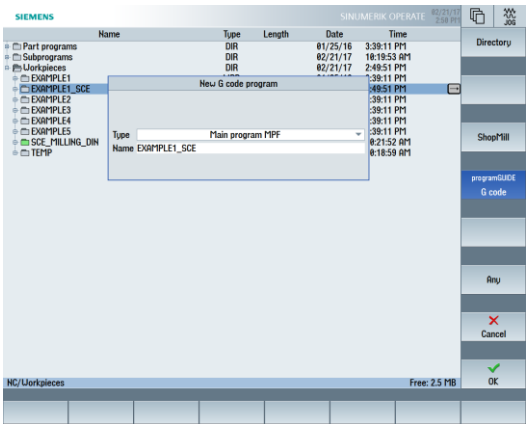
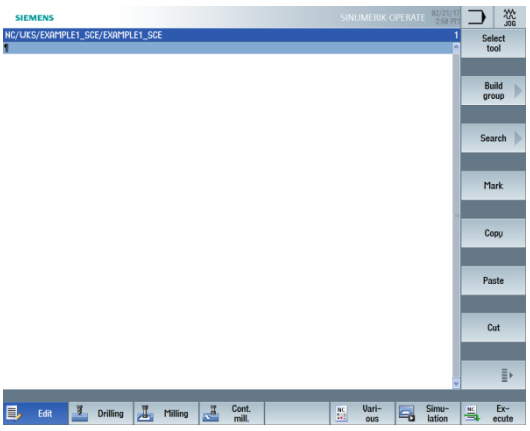
You will learn how to go from the drawing to the finished NC program with complete keystroke by keystroke instructions based on the "Longitudinal guide" workpiece. The following topics will be covered



- Creating the program
- Tool call and tool change
- Basic functions
- Technological functions (cutting data)
- Simple traversing paths with and without cutter radius compensation
- Drilling with cycles and subprogram technique
- Simulation for checking the programming

4.1.1 Creating the workpiece and part program

Keys / Inputs	Screen / Drawing	Explanation
		Initial status: Any operating area (here, "Machine") and operating mode ("JOG")
		Switch to the main menu.
		Switch to the "Program Manager" operating area using the softkey. There are various directory and file types. The "Workpieces" (WPD) directory type is a directory into which all relevant data of a machining task (part programs, subprograms, etc.) can be stored. This allows all files to be clearly organized.

		Create a new workpiece directory for the longitudinal guide "EXAMPLE1_SCE". Note that each name can be used only once. (You therefore may have to select a different name.)
	You apply text and numeric entries on the control keyboard with the yellow <Input> key or on the PC with <Enter>.	
 		Select the "NEW" softkey and then the "Directory" softkey. In the following dialog window, enter the desired directory name "EXAMPLE1_SCE" and except the name with “OK”.
 		In communication with the controller, a suggestion follows for program creation in "programGUIDE G-CODE" with the same name as the directory. This suggestion is “OK" and you therefore accept it.
		This takes you to the editor for the programGUIDE programming.

4.1.2 Tool call and tool change

Either	When you are using a controller that manages tools with plain text names (see section 2.2.1)	Or	When you are using a controller that manages tools with T numbers (see section 2.2.2)
T = “CUTTER 60“ ; porcupine cutter D60 mm		T7 ; porcupine cutter D60 mm	
 The tool (T = Tool) is selected with its plain text name, which was assigned in the Tool management ("Parameter operating area).		The tool (T = Tool) is selected with its T-number, which was assigned in the Tool management ("Parameter operating area).	
Caution: Different cases depending on the tool management approach will not be repeated in the following. You must change the tool call, if necessary, on your own!			
M6	M6 calls the tool change on machines with a tool changer.		

4.1.3 Basic program functions

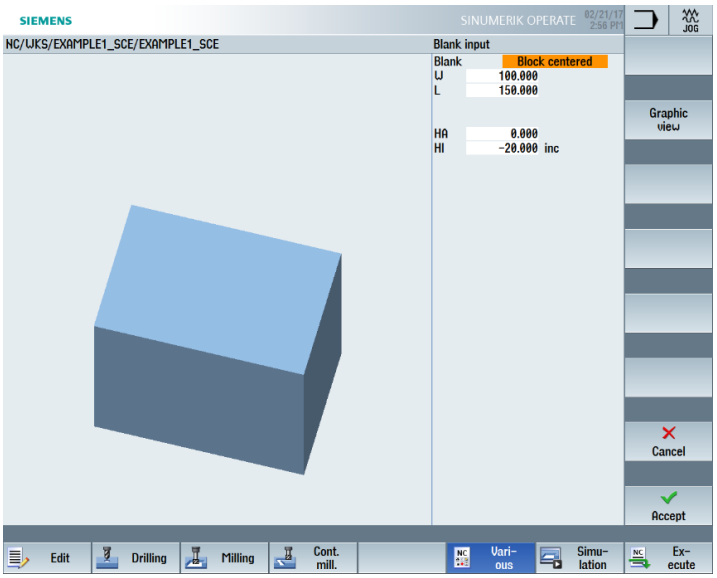
G17 G40 G64 G71 G90 G54	These are basic functions and are explained in more detail in the overview below. Often these functions are valid for an entire program. For the sake of safety, however, it is recommended to execute these functions at every tool change.
Explanation of the functions	**Functions of the same group**
G17 – Plane selection XY plane (milling)	G18 – Plane selection ZX plane (turning) G19 – Plane selection YZ plane (cylinder surface)
G40 – Cutter radius compensation deselection	G41 – Cutter radius compensation ON in machining direction LEFT of the contour in the machining direction G42 – Cutter radius compensation ON in machining direction RIGHT of the contour in the machining direction
G64 – Round contour The destination of a traversing block is not approached exactly. Instead, there is a small rounding to the subsequent traversing path.	G60 – Exact stop The destination is approached exactly. All axis drives are braked to a standstill for this.
G71 – Millimeter dimension unit	G70 – Inch dimension unit
G90 – Absolute dimensions	G91 – Incremental dimensions
G54 – Activation of the first adjustable work offset	G55, G56, G57 – Other work offsets G53 – Cancellation of all work offsets (acts block-wise) G500 –Deactivation of all work offsets

Functions of a group cancel each other out. You can "look up" which function is currently active in the "Machine" operating area via softkey. 

The start of the program also includes the option of defining the blank for the simulation, regardless of whether milling or turning technology is selected.

Five different blank shapes are possible:

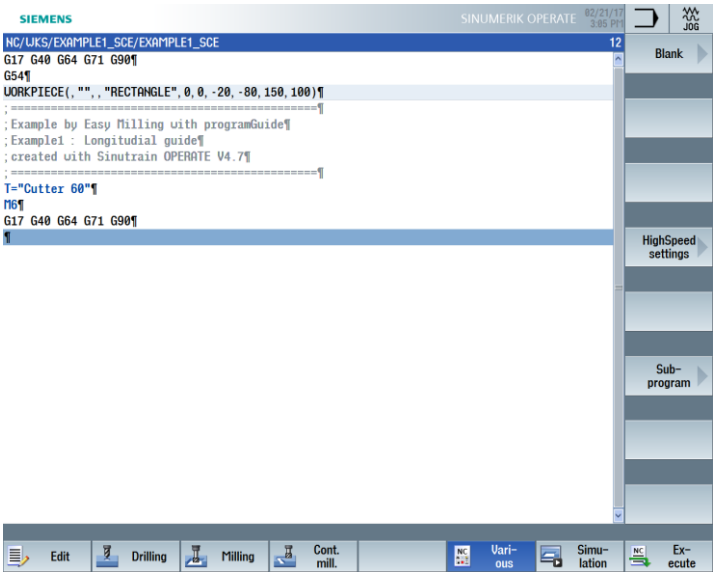
Cylinder, Pipe, Block centered, Block and N corner



The blank is programmed using the horizontal softkey "Various" 

and the vertical softkey 

.

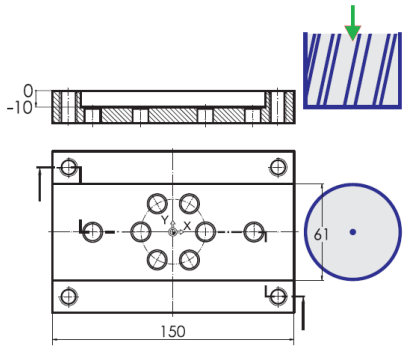
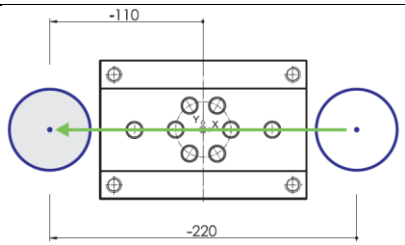


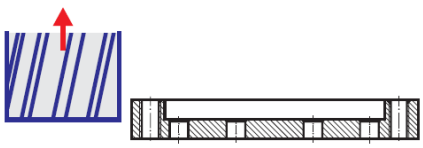
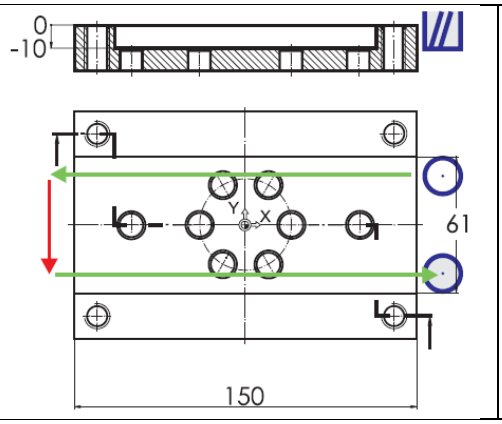
The first lines of the program are complete.

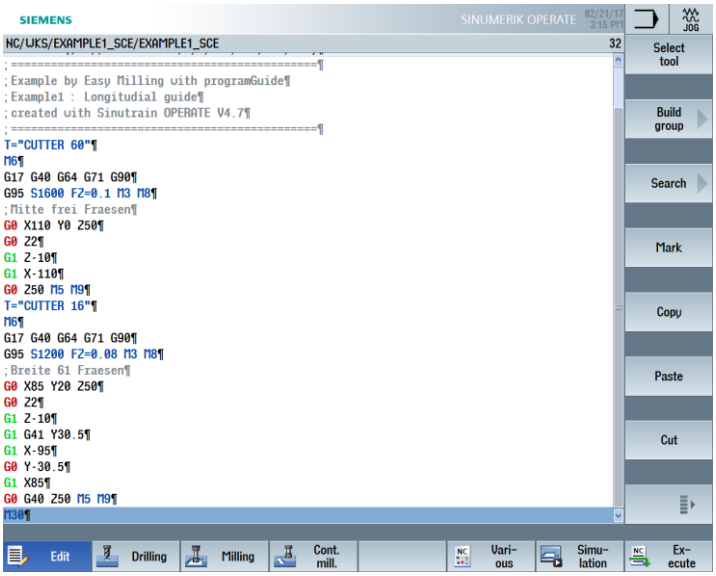
The first tool has been loaded and important general basic settings specified.

The 61 mm wide groove will now be rough-cut with this 60 mm wide tool.

4.1.4 Simple traversing paths with/without cutter radius compensation

G95 S1000 FZ = 0.1 M3 M8	G95 Speed in 1/min FZ = 0.1 feed per tooth (the number of teeth must be specified in the tool list for this.) M3 Spindle ON clockwise M8 External cooling lubricant ON In this block, the spindle can also be started up and the coolant switched on.* * Caution: All given technological specifications only serve as an example. Use your own empirical values for the machine and observe the information in the tool catalog!	
;Cut center free	Insert comment line with semicolon	
G0 X110 Y0 Z50	At rapid traverse (G0) the tool first moves to its start position. 110 = X value of the workpiece edge + cutter radius + safety clearance = 150/2+60/2+5 (To improve readability, the INPUT key for entering a program line will, starting from now, no longer be mentioned. Accept each line on your own with INPUT)	
G0 Z2	Before the milling cutter is moved to milling depth, it is positioned at a safety clearance 2 mm above the workpiece surface (Z2). This provides safety when run-in the program (if the workpiece zero or the tool compensation was incorrectly set by accident).	
G1 Z-10		During work feed G1 the tool is moved to the machining depth.
G1 X-110		The milling cutter is fed at 0.1 mm feed per tooth on a straight line (G1) **to** the destination point X-110 (absolute dimension relative to zero point). With G91 (incremental dimension), the destination point X-220 would have had to be programmed because the milling cutter moves **a distance of** 220 mm in the negative axis direction.

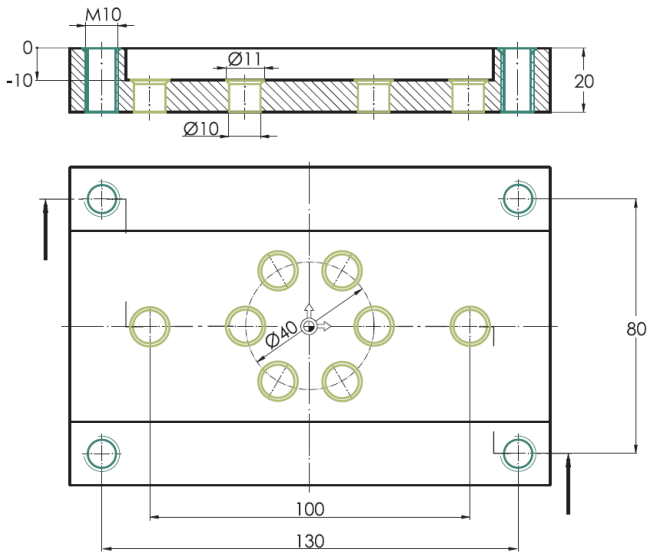
G0 Z50 M5 M9		At rapid traverse (G0) the milling cutter is moved away from the workpiece in the Z direction. At the same time the spindle is stopped with M5 and the coolant is switched off with M9.
T = "CUTTER 16" M6	The 16 mm end mill will be used to mill the two edges of the groove (rough-cut to a width of 61 mm with porcupine cutter ø60) to size.	
G17 G40 G64 G71 G90	The same G functions used for the first machining are also the basis for machining with the end mill.	
G95 S1600 FZ = 0.08 M3 M8	Set technology data for "CUTTER 16", Spindle “ON” clockwise and external coolant “ON”	
;Mill width 61	;Insert comment line with semicolon	
G0 X85 Y20 Z50 G0 Z2 **G1 Z-10** **G1 G41 Y30.5** **G1 X-95** **G0 Y-30.5** **G1 X85**		In this section, finish-cutting of the contour with automatic offset of the cutter radius; G41 Left of the contour in the machining direction
G0 G40 Z50 M5 M9	At the end, the workpiece is exited again in rapid traverse, the spindle is stopped and the coolant is switched off.	
M30	End of program	



You can check your entries at any time using a simulation.

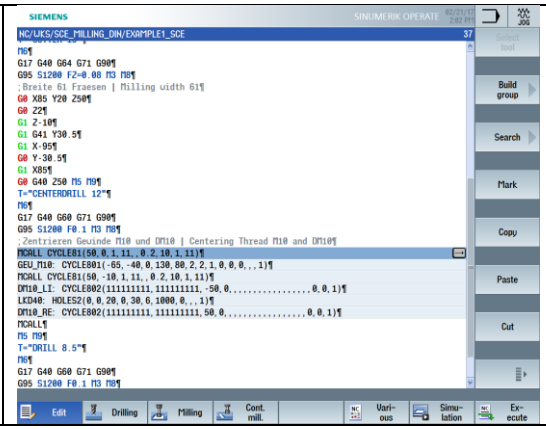
4.1.5 Drilling with cycles and subprogram technique

Centering		
T = "CENTERDRILL 12" M6	All 12 drill holes will be centered first.	
G17 G40 **G60** G71 G90	**G60 (exact stop)** is used for drilling in order to achieve optimal positioning accuracy for all drill holes.	
G95 S1600 F0.1 M3 M8	Set technology data for "CENTERDRILL 12", Spindle ON clockwise and external coolant ON	
;Center M10 and DM10	;Insert comment line with semicolon	



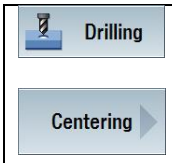
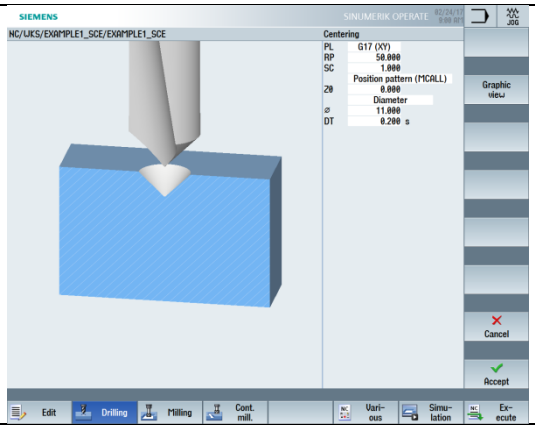
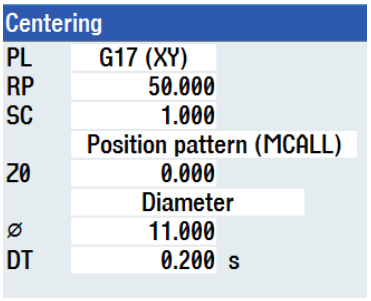
The drill holes can be divided into two groups:
- 4 x M10 thread at the corners
- 2 individual drill holes and
- 1 circle of holes in the groove

The positions of the first group will be entered later in a subprogram named **GEW_M10**, and the positions of the other drill holes in subprograms **DM10_LI, LKD40 and DM10_RE**.
Subprograms are useful here because the positions are approached both for centering as well as for drilling and thread cutting.



Select horizontal softkey for opening the "Drilling" main menu.

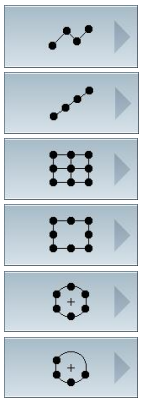
The associated submenus now appear on the vertical softkey bar.

<table>
<tr>
<td> 
</td>
<td> 
</td>
<td>The "Centering" vertical softkey is used to open the dialog window for the input dialog.</td>
</tr>
<tr>
<td></td>
<td> 
</td>
<td>PL ➔ Machining plane G17 (XY)
RP ➔ Retraction plane 50 mm
SC ➔ Safety clearance 1 mm
Machining various position patterns with modal cycle call (MCALL)
Z0 ➔ Coordinate surface
Start of machining in Z direction
"Diameter" input selected, "Tip" would be possible as an alternative
DT ➔ Dwell time in seconds</td>
</tr>
<tr>
<td colspan="3">"Modal" means "latching". This means that a command (for example a G function, a programmed axis position or, as in the case here, a complete cycle) is active beyond the block in which it is located. In the case of drilling cycles, it means that the command is executed again after each subsequent programmed traversing path.</td>
</tr>
</table>

Now let us continue with the positioning of the previously defined "Centering" machining.

The various input options of positions can be found under the "Drilling" softkey, "Positions" softkey.

Positioning options are:



Single Positions cartesian and polar

Linear hole pattern - Straight

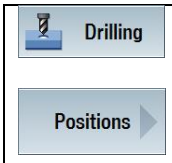
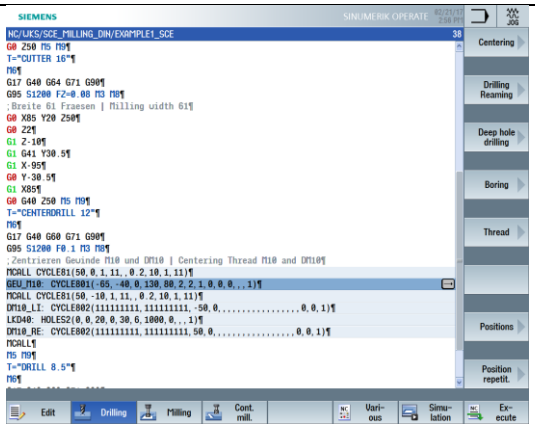
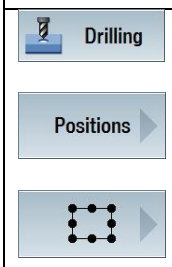
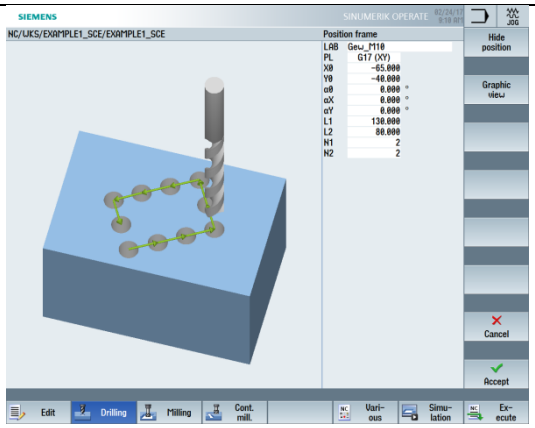
Linear hole pattern - Grid

Linear hole pattern - Frame

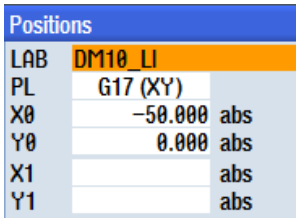
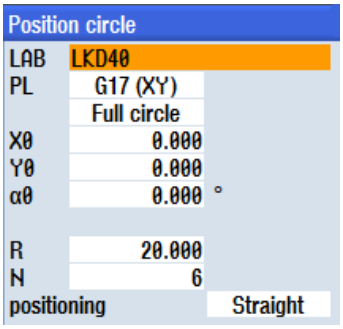
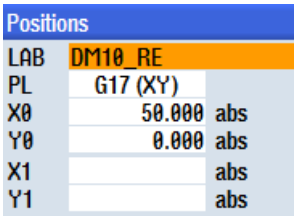
Circular hole pattern - full circle

Circular hole pattern - divided circle

Certain positions can be hide during the entering of the positions.

		Select horizontal softkey for opening the "Drilling" main menu. We now continue with the position input in the "Positions" submenu command on the vertical softkey bar.
		Select the softkey for "Linear hole patterns" to start entering the position data for the four threaded holes M10. LAB ➔ Name of the jump label for program section repetition PL ➔ Machining plane G17 (XY) FRAME one of the three linear position pattern options X0 Start position in X Y0 Start position in Y α0 Angle of rotation αX Shear plane angle X αY Shear plane angle Y L1 Distance between columns L2 Distance between rows N1 Number of columns N2 Number of rows

Now repeat the input sequence "Centering" and "Positioning" for the drill holes diameter 10 at Z-10.

Single position LEFT Jump label name: **DM10_LI**	Center circle of holes Jump label name: **LKD40**	Single position RIGHT Jump label name: **DM10_RE**
		
		

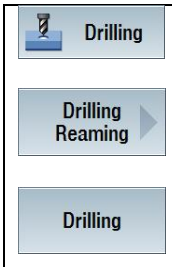
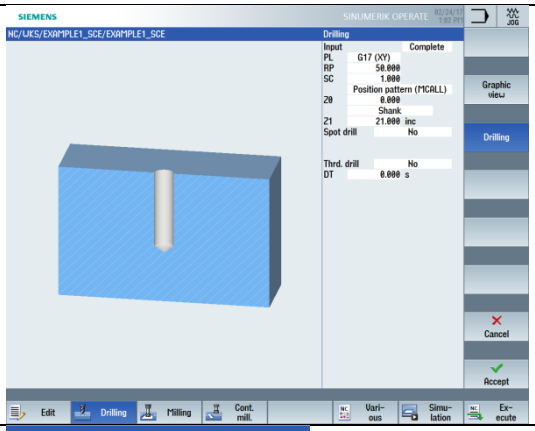
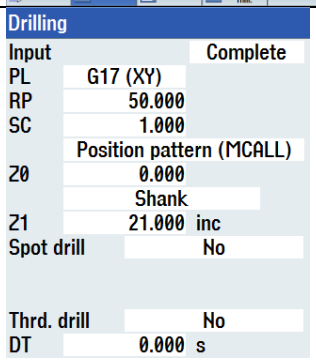
Current program excerpt:

```
MCALL CYCLE81(50,0,1,11,,0.2,10,1,11)
GEW_M10: CYCLE801(-65,-40,0,130,80,2,2,1,0,0,0,,,1)
MCALL CYCLE81(50,-10,1,11,,0.2,10,1,11)
DM10_LI: CYCLE802(111111111,111111111,-50,0,,,,,,,,,,,,,,,,0,0,1)
LKD40: HOLES2(0,0,20,0,30,6,1000,0,,,1)
```

DM10_RE: CYCLE802(111111111,111111111,50,0,,,,,,,,,,,,,,,0,0,1)

MCALL	Deselection of the modal cycle call MUST be entered!
M5 M9	Spindle STOP Coolant OFF

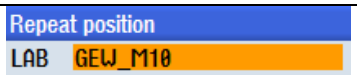
Drilling the thread tap hole for M10		
T = "DRILL 8.5" M6	The threaded holes M10 have a ø8.5 mm tap hole. A twist drill is used for drilling.	
G17 G40 **G60** G71 G90	**G60 (exact stop)** is used for drilling in order to achieve optimal positioning accuracy for all drill holes.	
G95 S1450 F0.1 M3 M8	Set technology data for "DRILL 8.5", Spindle ON clockwise and external coolant ON	
;Drill tap diameter (M10) ➔ D8.5	;Insert comment line with semicolon	

		The "Drilling" vertical softkey is used to open the dialog window for the input dialog.
		PL ➔ Machining plane G17 (XY) RP ➔ Retraction plane 50 mm SC ➔ Safety clearance 1 mm Machining various position patterns with modal cycle call (MCALL) Z0 ➔ Coordinate surface Start of machining in Z direction "Shank" input selected, "Tip" would be possible as an alternative Z1 ➔ Incremental drilling depth relative to Z0 DT ➔ Dwell time in seconds

Now we continue with the programming of the program section repetition for the positioning of the threaded holes M10.

The input options of position repetition can be found under the "Drilling" softkey, "Position repetit." softkey.

Position repetition

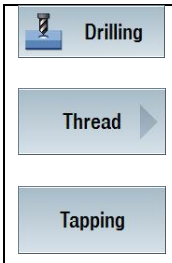
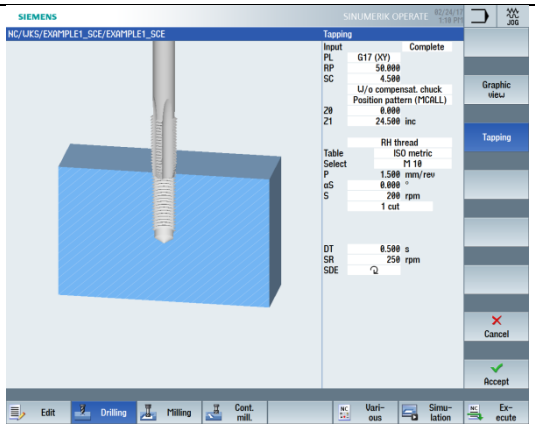
		Input: Jump label for "Repeat position" of the desired position(s) that were previously used during the preceding programming.

Current program excerpt:

REPEATB GEW_M10 ;#SM

MCALL	Deselection of the modal cycle call MUST be entered!
M5 M9	Spindle STOP Coolant OFF

Tapping M10		
T = "THREADCUTTER M10" M6	The feed rate during tapping results from the speed and the thread pitch, which is entered in the cycle.	
G17 G40 **G60** G71 G90	**G60 (exact stop)** is used for drilling in order to achieve optimal positioning accuracy for all drill holes.	
G95 S200 M3 M8	Set technology data for "THREADCUTTER M10", Spindle ON clockwise and external coolant ON	
;Tap M10	Insert comment line with semicolon	

		The "Tapping" vertical softkey is used to open the dialog window for the input dialog.
	Tapping Input Complete PL G17 (XY) RP 50.000 SC 4.500 U/o compensat. chuck Position pattern (MCALL) Z0 0.000 Z1 24.500 inc RH thread Table ISO metric Select M 10 P 1.500 mm/rev αS 0.000 ° S 200 rpm 1 cut DT 0.500 s SR 250 rpm SDE	PL ➔ Machining plane G17 (XY) RP ➔ Retraction plane 50 mm SC ➔ Safety clearance 4.5 mm Machining without compensating chuck Machining various position patterns with modal cycle call (MCALL) Z0 ➔ Coordinate surface Start of machining in Z direction "Shank" input selected, "Tip" would be possible as an alternative Z1 ➔ Incremental drilling depth relative to Z0 Selection ➔ Right thread Selection of the thread pitch from the **table** for ISO metric thread M10 Thread pitch ➔ P 1.5 mm/rev αS ➔ Start angle 0 ° S ➔ Spindle speed 200 rev/min Machining in one cut DT ➔ Dwell time in seconds SR ➔ Spindle retraction speed 250 rev/min SDE ➔ Direction of rotation after cycle end M3

Repetition of the positions for machining the threaded holes. See section:
Drilling the thread tap hole for M10

Current program excerpt:

REPEATB GEW_M10 ;#SM

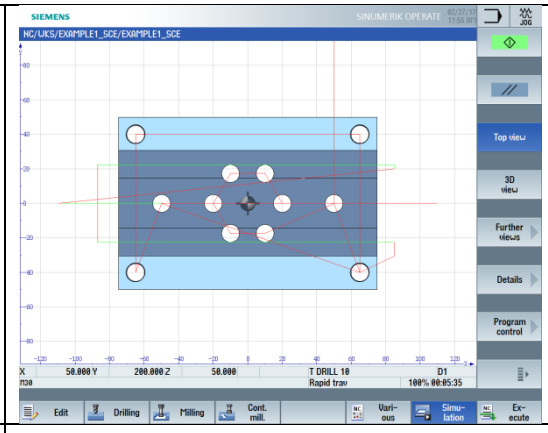
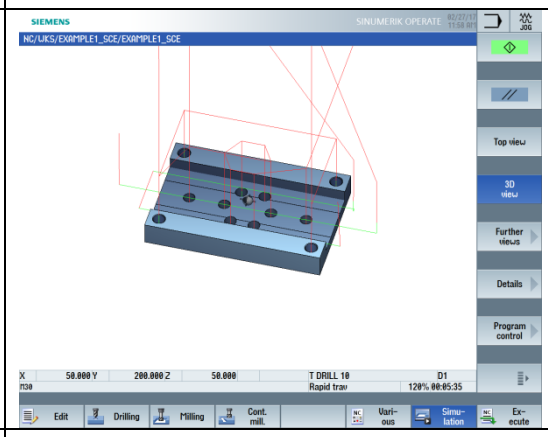
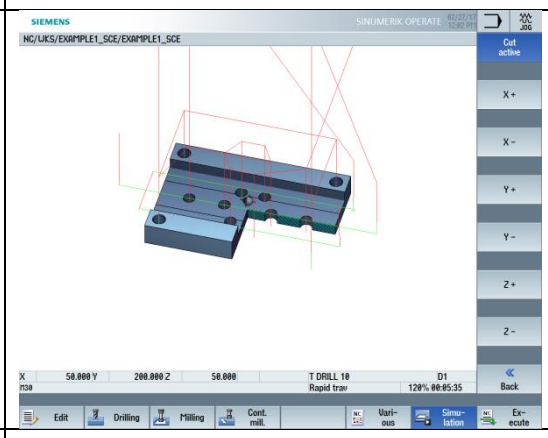
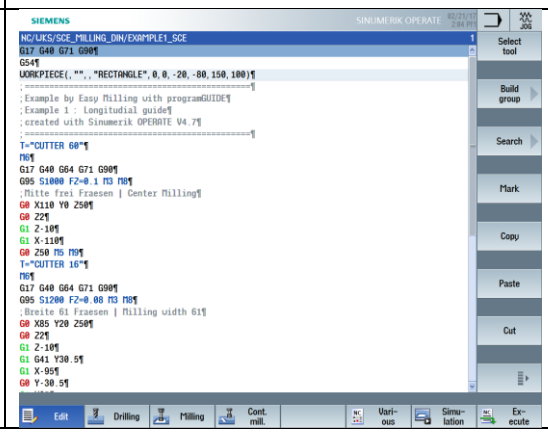
MCALL	Deselection of the modal cycle call MUST be entered!
M5 M9	Spindle STOP Coolant OFF

Drilling the through-holes Ø10		
T = "DRILL 10" M6	Program lines for the through-holes Ø10 You enter the drilling cycle again using softkeys and the input dialog.	
G17 G40 **G60** G71 G90	**G60 (exact stop)** is used for drilling in order to achieve optimal positioning accuracy for all drill holes.	
G95 S1000 F0.12 M3 M8	Set technology data for "DRILL 10", Spindle ON clockwise and external coolant ON	
;Drill diameter D10	;Insert comment line with semicolon	
MCALL CYCLE82(50,...)	Modal drilling cycle call	
REPEATB DM10_LI ;#SM REPEATB LKD40 ;#SM REPEATB DM10_RE ;#SM	Drilling position repetitions at Z-10	
MCALL	Deselection of the modal cycle call MUST be entered!	
G0 Y200 M5 M9	Retract in rapid traverse in the Y direction Spindle STOP Coolant OFF	
M30	In the program with return to the start	

Current program excerpt:

```
MCALL CYCLE82(50,-10,1,,11,0,10,1,11)
REPEATB DM10_LI ;#SM
REPEATB LKD40 ;#SM
REPEATB DM10_RE ;#SM
MCALL
G0 Y200 M5 M9
M30
```

4.1.6 Simulating the milling program

Simu-lation Top view		Simulation of the machining operations in the top view with traversing paths Red ➔ Rapid traverse motions Green ➔ Feed motions
Simu-lation 3D view		Simulation of the machining operations in the 3D view with traversing paths Red ➔ Rapid traverse motions Green ➔ Feed motions
Simu-lation 3D view Details Cut		Simulation of the machining operations in the 3D view with traversing paths Cut active Red ➔ Rapid traverse motions Green ➔ Feed motions
Simu-lation		End the simulation. Press the "Simulation" softkey again or "Edit" softkey.

4.1.7 Solution program "Longitudinal guide"

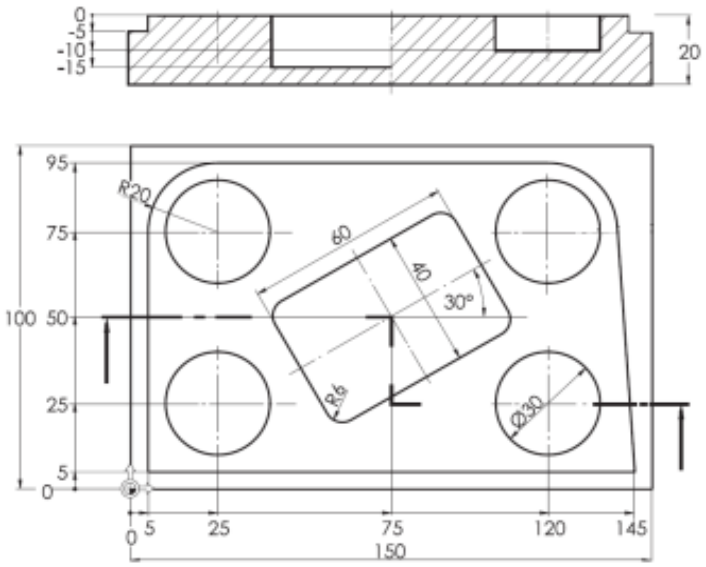
```
G17 G40 G64 G71 G90
G54
WORKPIECE(,"",,"RECTANGLE",0,0,-20,-80,150,100)
;============================================
;Example by Easy Milling with programGUIDE
;Example 1 : Longitudinal guide
;created with SinuTrain OPERATE V4.7
;============================================
T="CUTTER 60"
M6
G17 G40 G64 G71 G90
G95 S1000 FZ=0.1 M3 M8
;Cut center free
G0 X110 Y0 Z50
G0 Z2
G1 Z-10
G1 X-110
G0 Z50 M5 M9
T="CUTTER 16"
M6
G17 G40 G64 G71 G90
G95 S1200 FZ=0.1 M3 M8
;Mill width 61
G0 X85 Y20 Z50
G0 Z2
G1 Z-10
G1 G41 Y30.5
G1 X-95
G0 Y-30.5
G1 X85
G0 G40 Z50 M5 M9
T="CENTERDRILL 12"
M6
G17 G40 G60 G71 G90
G95 S1200 F0.1 M3 M8
;Center thread M10 and DM10
MCALL CYCLE81(50,0,1,11,,0.2,10,1,11)
GEW_M10: CYCLE801(-65,-40,0,130,80,2,2,1,0,0,0,,,1)
MCALL CYCLE81(50,-10,1,11,,0.2,10,1,11)
DM10_LI: CYCLE802(111111111,111111111,-50,0,,,,,,,,,,,,,,,,0,0,1)
LKD40: HOLES2(0,0,20,0,30,6,1000,0,,,1)
DM10_RE: CYCLE802(111111111,111111111,50,0,,,,,,,,,,,,,,,,0,0,1)
MCALL
M5 M9
T="DRILL 8.5"
M6
G17 G40 G60 G71 G90
G95 S1200 F0.1 M3 M8
;Drill tap diameter M10 ==> D8.5
MCALL CYCLE82(50,0,1,,21,0,10,1,11)
REPEATB GEW_M10 ;#SM
MCALL
M5 M9
```

```
T="THREADCUTTER M10"
M6
G17 G40 G60 G71 G90
G95 S200 M3 M8
;Tap M10
MCALL
CYCLE84(50,0,4.5,,24.5,0.5,3,,1.5,0,200,250,0,1,0,0,5,1.4,,"ISO_METRIC","M10",,1001,20010
01)
REPEATB GEW_M10 ;#SM
MCALL
M5 M9
```

```
T="DRILL 10"
M6
G17 G40 G60 G71 G90
G95 S1200 F0.12 M3 M8
;Drill diameter D10
MCALL CYCLE82(50,-10,1,,11,0,10,1,11)
REPEATB DM10_LI ;#SM
REPEATB LKD40 ;#SM
REPEATB DM10_RE ;#SM
MCALL
G0 Y200 M5 M9
M30
```

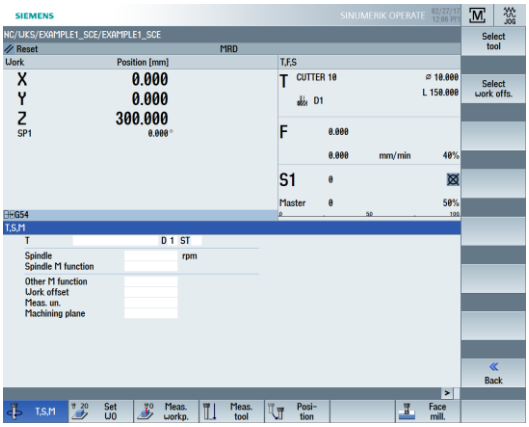
4.2 "Injection mold" workpiece

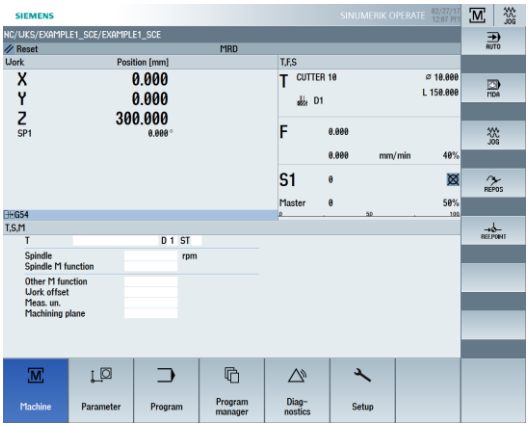
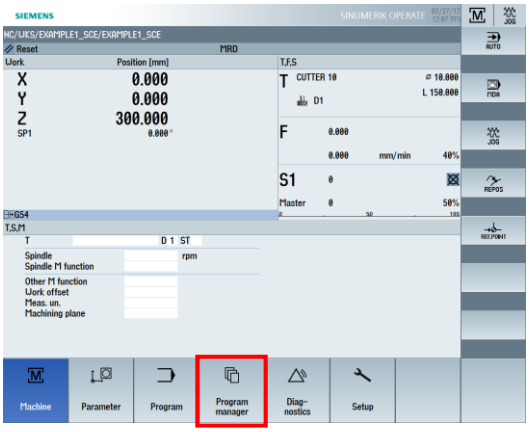
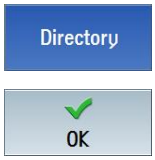
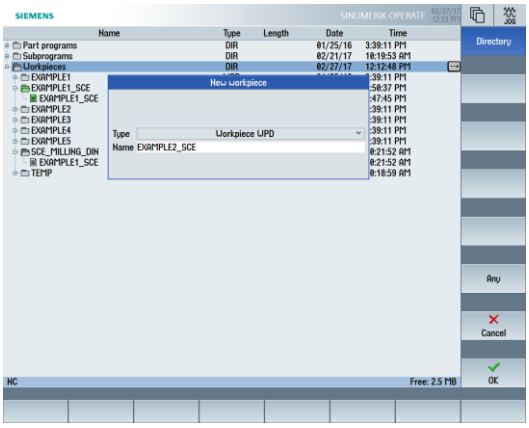
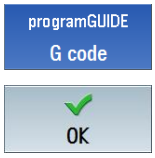
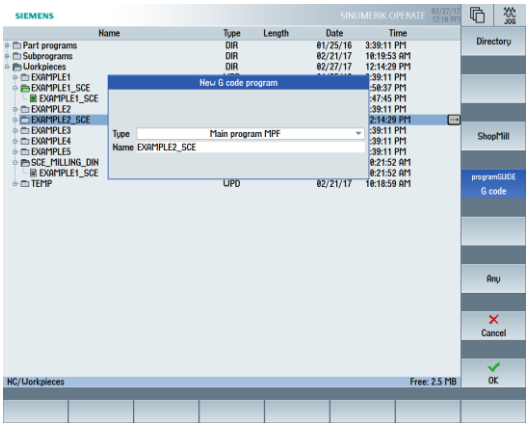
Based on the "Injection mold" workpiece, you will learn functions of the controls for path milling and pocket milling. It is assumed that you have already worked through the "Longitudinal guide" example or are familiar with the subjects covered in that example. The following subjects are newly described in this section:

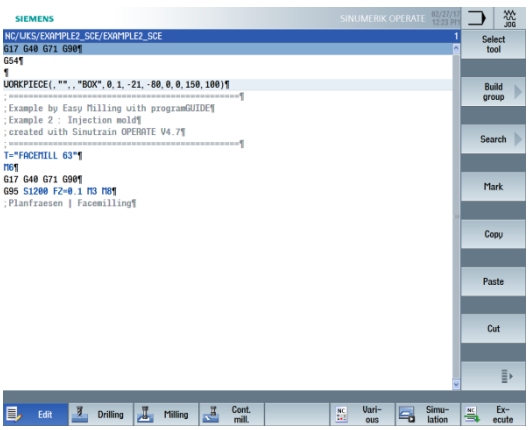
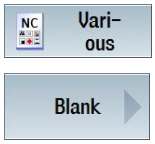
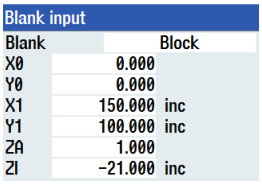


- Face milling
- Working with the contour calculator
- Milling with a path milling cycle and tool radius compensation
- Rectangular pocket (roughing)
- Circular pocket (roughing)
- Copying a program section
- Chamfering contour parts

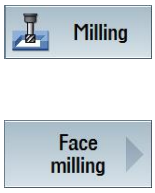
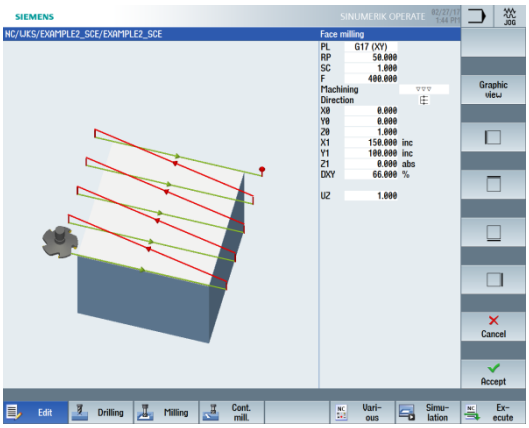
4.2.1 Creating the workpiece and part program

Keys / Inputs	Screen / Drawing	Explanation
		Initial status: Any operating area (here, "Machine") and operating mode ("JOG")

		Switch to the main menu.
		Switch to the "Program Manager" operating area using the softkey. There are various directory and file types. The "Workpieces" (WPD) directory type is a directory into which all relevant data of a machining task (part programs, subprograms, etc.) can be stored. This allows all files to be clearly organized.
		Select the "NEW" softkey and then "Directory". In the following dialog window, enter the desired directory name "EXAMPLE2_SCE" and except the name with OK.
		In the communication between operator and control, the suggestion follows for program creation in "programGUIDE G-CODE" with the same name as the directory. This suggestion is OK and you therefore accept it.

		This takes you to the editor for the programGUIDE programming. The basic G functions, blank description and tool call are made.
		Block-shaped blank description with Z allowance ZA = 1 for face milling.

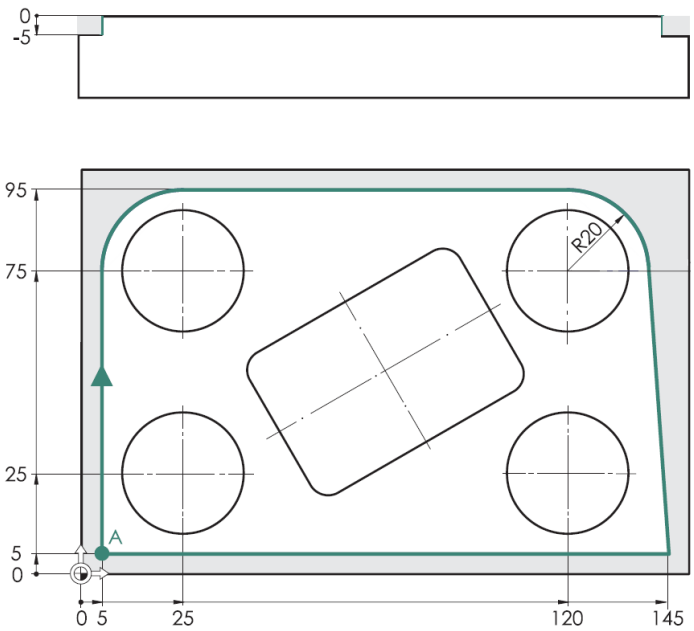
4.2.2 Face milling

		PL ➔ Machining plane G17 (XY) RP ➔ Retraction plane 50 mm SC ➔ Safety clearance 1 mm F ➔ Feed, e.g. 400 mm/min Machining Finishing Direction of machining Axially parallel in X direction X0 ➔ Start position in X0 = 0 Y0 ➔ Start position in Y0 = 0 Z0 ➔ Coordinate surface Start of machining in Z direction Z0 = 1 X1 ➔ Destination position in X1 = 150 inc Y1 ➔ Destination position in Y1 = 100 inc Z1 ➔ Destination position in Z1 = 0 abs DXY ➔ Plane infeed as percentage of the active tool e.g. 66% UZ ➔ 1 Allowance in Z

Previous program input:

```
G17 G40 G64 G71 G90
G54
WORKPIECE(,"",,"BOX",0,1,-21,-80,0,0,150,100)
;==========================================
;Example by Easy Milling with programGUIDE
;Example 2 : Injection mold
;created with SinuTrain OPERATE V4.7
;==========================================
T="FACEMILL 63"
M6
G17 G40 G64 G71 G90
G95 S1200 M3 M8
;Face milling
CYCLE61(50,1,1,0,0,0,150,100,0.5,66,1,400,12,0,1,10)
M5 M9
```

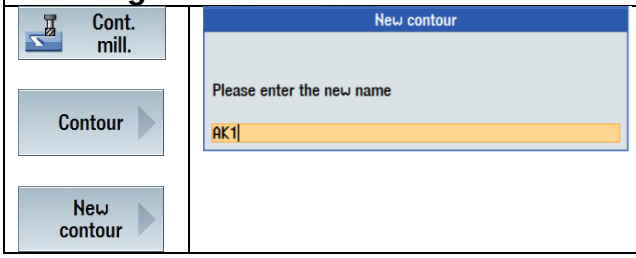
4.2.3 Path milling with the contour calculator



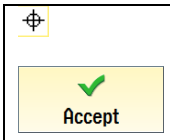
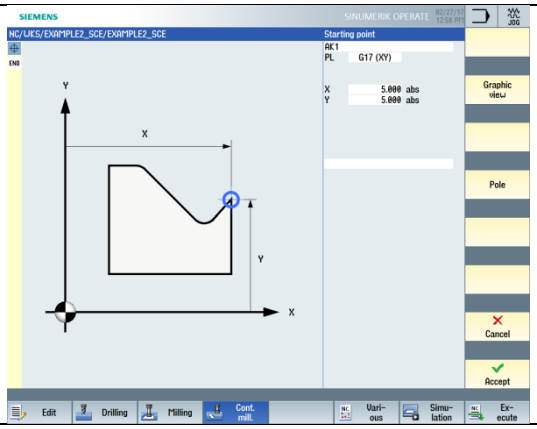
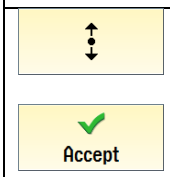
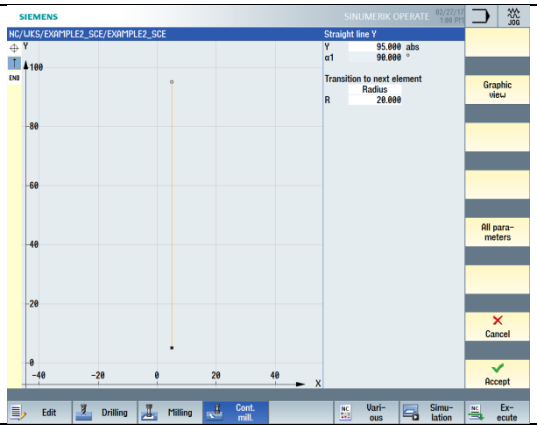
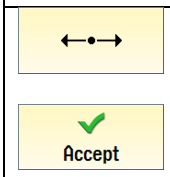
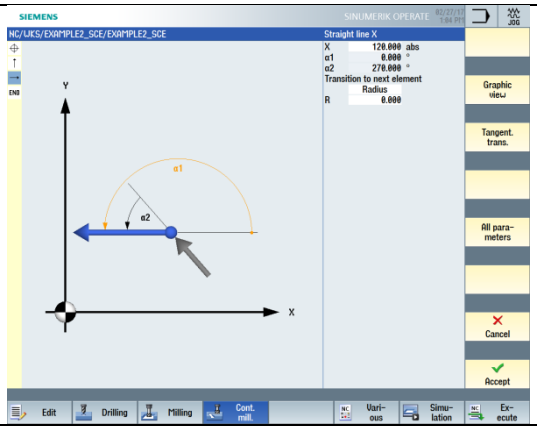
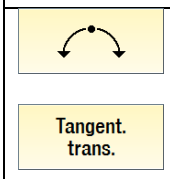
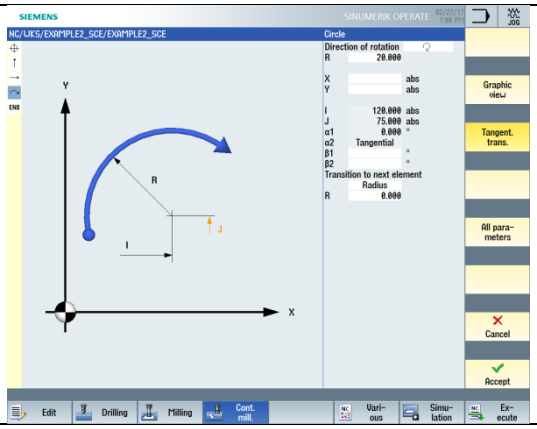
A 32 mm end mill is used to cut the outside contour along the highlighted contour.
The contour will be created starting from point A X5 Y5 using the contour calculator.
Down-cut milling will be used, i.e. the contour will be traversed clockwise with the clockwise-rotating milling cutter.

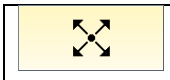
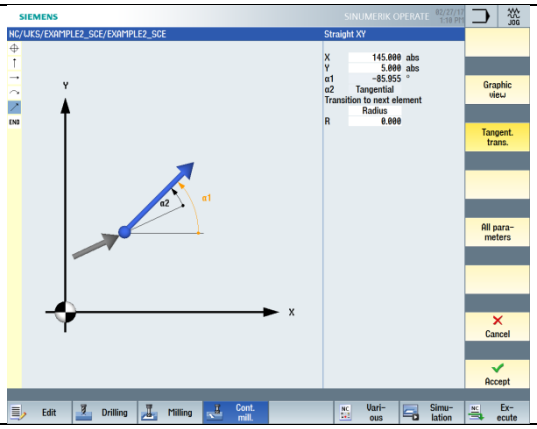
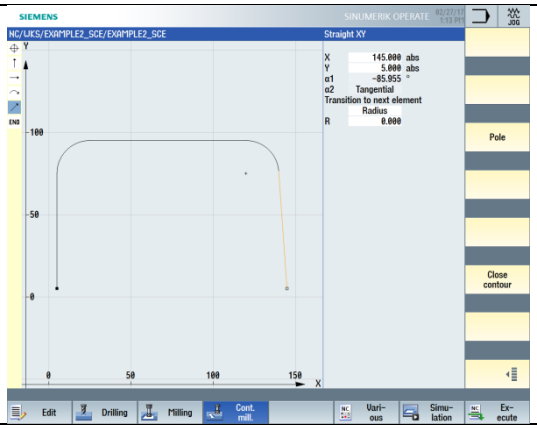
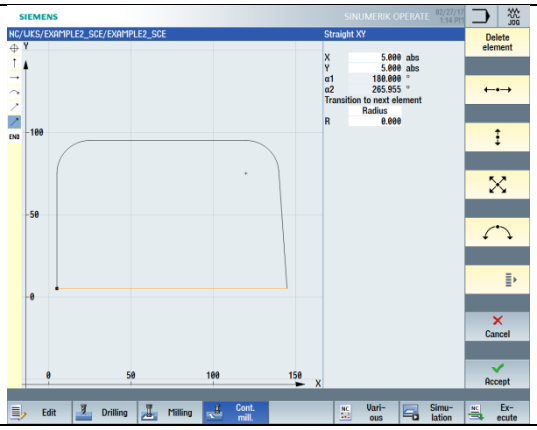
The path milling cycle "CYCLE72" is used for the milling.

Working with the contour calculator



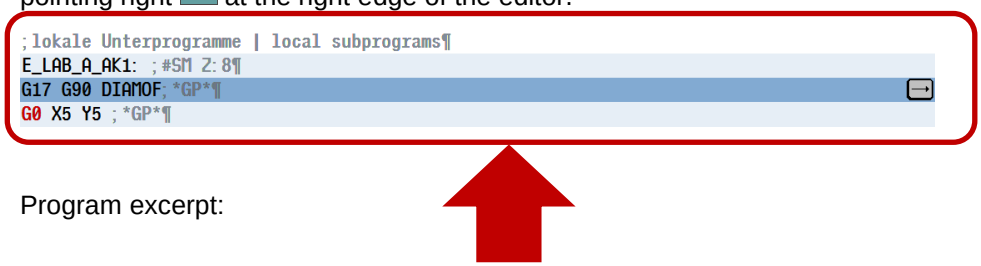
Enter a name for the contour to be created.

		Select the milling machining plane “PL G17 (XY)”, enter the starting point for the contour with X5 Y5 and apply these inputs.
		Vertical line to Y 95 mm with a corner radius R20
		Horizontal line to X 120 mm
		Circle programming in the clockwise direction with tangential connection to the predecessor element and a radius of 20 mm

 Accept		Inclined line with any angle, tangential to predecessor with an end position X145 Y5.
 Close contour  Accept		Advance to the second vertical level of the contour calculator, select Close contour; A connection line from the current position to the start position is created automatically.
 Accept		Use the "Accept" softkey to apply the created contour to the program.

The contour created with the contour calculator is placed after the end of program M30 where it does not affect the program.

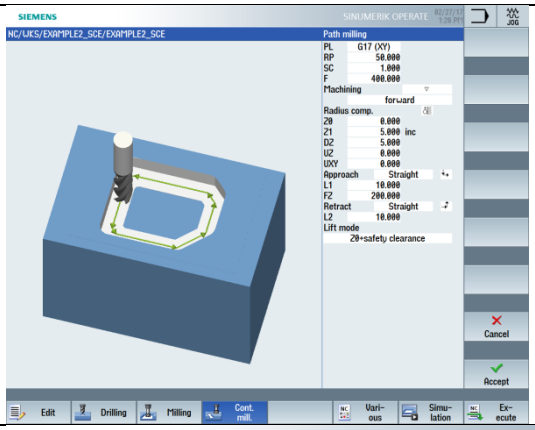
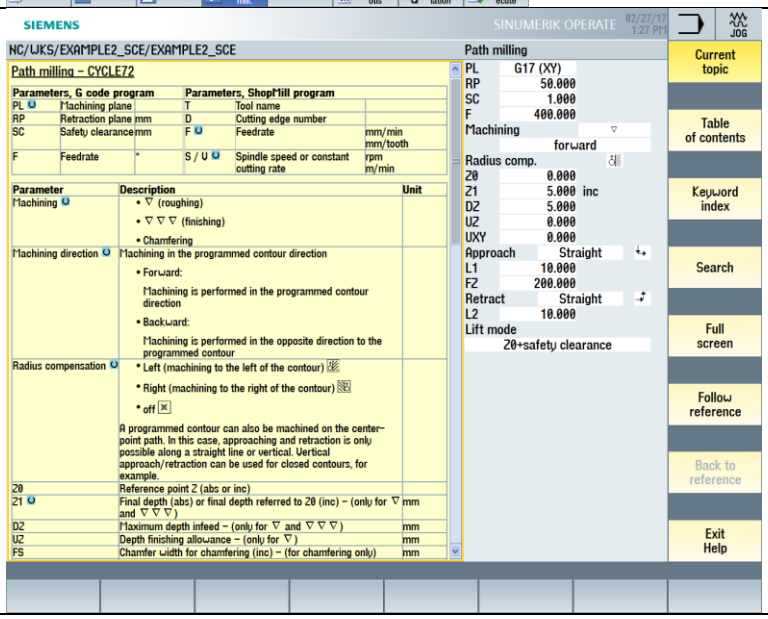
It can be edited at any time once you are in the contour description area by selecting the arrow pointing right at the right edge of the editor.



Program excerpt:

M30
;------------------------------
;Local subprograms
E_LAB_A_AK1: ;#SM Z:2
;#7__DlgK contour definition begin – Don't change!;*GP*;*RO*;*HD*
G17 G90 DIAMOF;*GP*
G0 X5 Y5 ;*GP*
G1 Y95 RND=20 ;*GP*
X120 ;*GP*
G2 X139.95 Y76.411 I=AC(120) J=AC(75) ;*GP*
G1 X145 Y5 ;*GP*
X5 ;*GP*
;CON,0,0.0000,5,5,MST:0,0,AX:X,Y,I,J,TRANS:1;*GP*;*RO*;*HD*
;S,EX:5,EY:5;*GP*;*RO*;*HD*
;LU,EY:95;*GP*;*RO*;*HD*
;R,RROUND:20;*GP*;*RO*;*HD*
;LR,EX:120;*GP*;*RO*;*HD*
;ACW,AT:0,RAD:20;*GP*;*RO*;*HD*
;LA,EX:145,EY:5,AT:0;*GP*;*RO*;*HD*
;LA,EX:5,EY:5;*GP*;*RO*;*HD*
;#End contour definition end – Don't change!;*GP*;*RO*;*HD*
E_LAB_E_AK1:

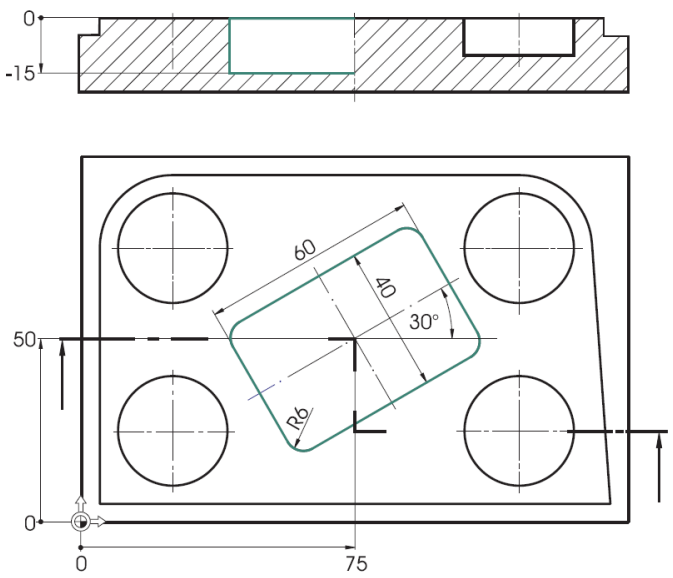
Path milling with the contour calculator

Cont. mill. Contour Contour call Accept	Contour call Contour name CON AK1	Select the contour name variant in the first text box. Enter the name for the contour to be machined.
Cont. mill. Path milling Accept		"Cont. mill." softkey "Path milling" softkey Accept the inputs as shown on the left.
Cont. mill. Path milling Accept		For detailed information, you can use the online help at any time. To do this, press the "Help" button. HELP

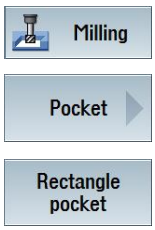
Program excerpt:

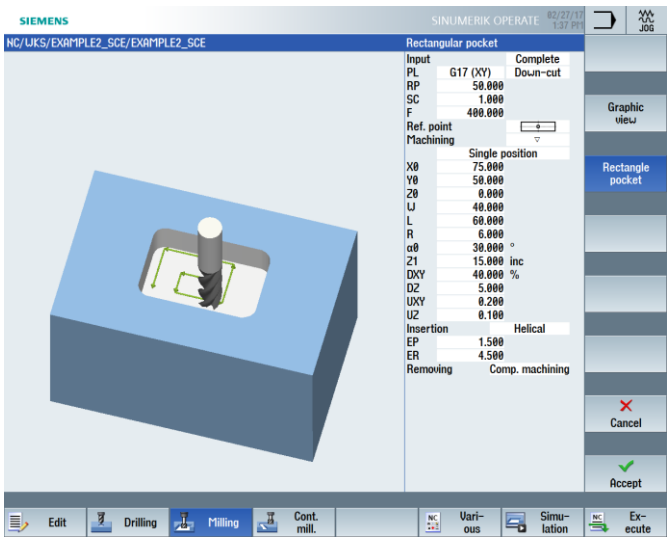
```
;Mill outer contour
CYCLE62(,2,"E_LAB_A_AK1","E_LAB_E_AK1")
CYCLE72("",50,0,1,5,5,0,0,400,200,101,41,1,10,0.1,1,10,0,1,2,101,1011,101)
```

4.2.4 Rectangular pocket (roughing)



Due to the corner radius R6, a smaller milling cutter is needed for the rectangular pocket.
The pocket will first be roughed with 0.2 mm allowance on the wall and 0.1 mm allowance on the base and then finished. Both can be implemented using the rectangular pocket cycle (POCKET3).



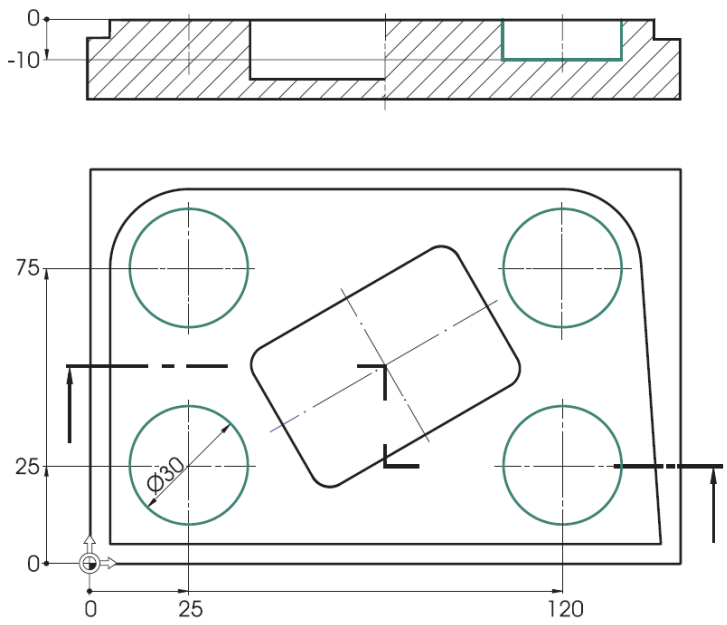


PL ➔ G17 (XY)
Down-cut
RP ➔ 50 mm
SC ➔ 1 mm
F ➔ 400 mm/min
Reference point: CENTER
Machining Roughing
Single position
X0 ➔Pocket center X0 = 75
Y0 ➔ Pocket center Y0 = 50
Z0 ➔ Coordinate surface
Start of machining in Z direction Z0 = 0
W ➔ Pocket width 40 mm
L ➔Pocket length 60 mm
R ➔ Corner radius R 6 mm
α0 ➔ Twist angle 30°
Z1 ➔ Destination position in Z1 = 15 inc reference Z0
DXY ➔ Plane infeed as percentage of the active tool e.g. 40%
DZ ➔ max. Z infeed
UXY ➔Allowance in XY
UZ ➔ Allowance in Z
Insertion strategy Helical
EP ➔ Helix pitch
ER ➔ Helix radius
Complete machining

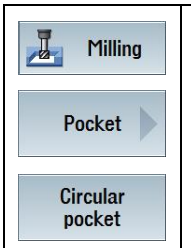
Program excerpt:

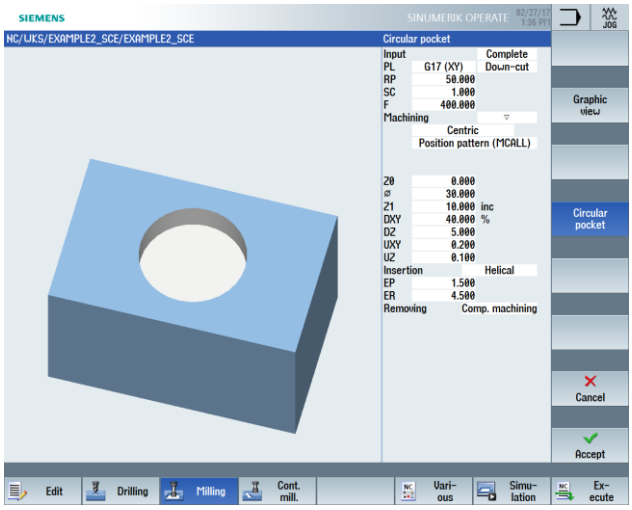
Milling rectangular pocket (ROUGHING)
POCKET3(50,0,1,15,60,40,6,75,50,30,5,0.2,0.1,400,0.1,0,21,40,8,3,15,4.5,1.5,0,1,2,11100,11,111)

4.2.5 Circular pocket



All for circular pockets have identical dimensions. Their positions are programmed using the FRAME position pattern.

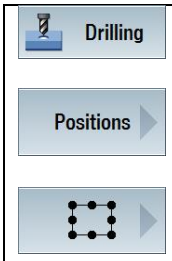
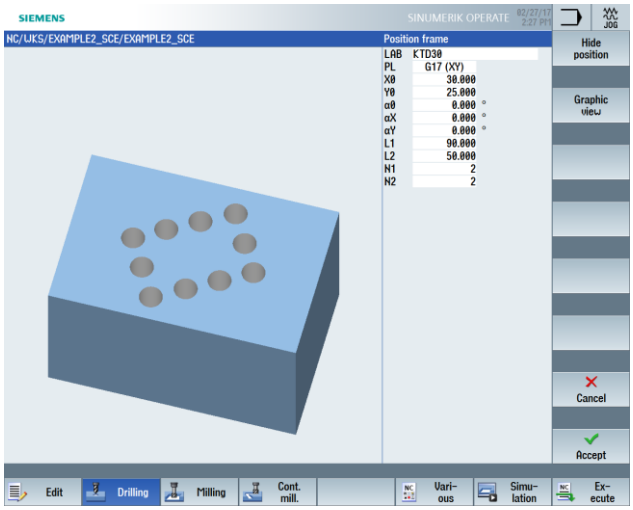




PL ➔ G17 (XY)
Down-cut
RP ➔ 50 mm
SC ➔ 1 mm
F ➔ 400 mm/min
Machining Roughing
Centric machining
Position pattern with modal cycle call
Z0 ➔ Coordinate surface
Start of machining in Z direction Z0 = 0
Diameter Ø30 mm
Z1 ➔ Destination position in Z1 = 10 inc reference Z0
DXY ➔ Plane infeed as percentage of the active tool e.g. 40%
DZ ➔ max. Z infeed
UXY ➔Allowance in XY
UZ ➔ Allowance in Z
Insertion strategy Helical
EP ➔ Helix pitch
ER ➔ Helix radius
Complete machining

Program excerpt:

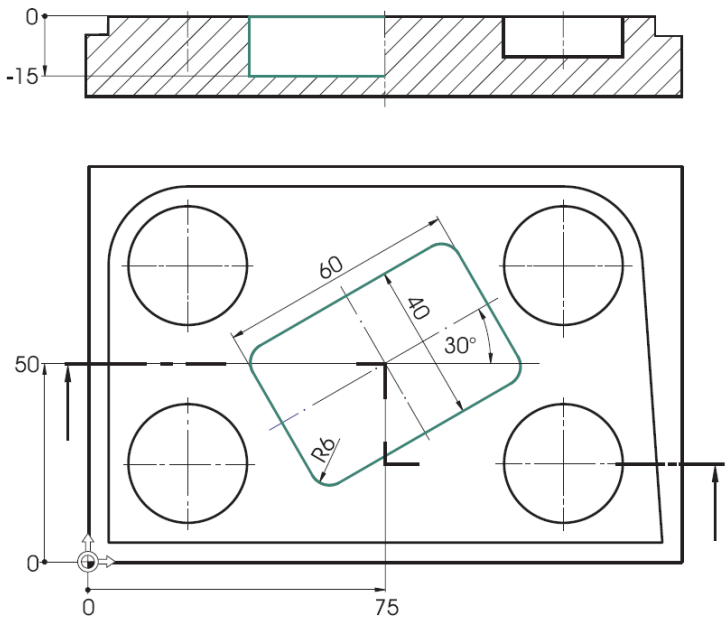
```
;Mill circular pocket ROUGHING
MCALL
POCKET4(50,0,1,10,30,0,0,5,0.2,0.1,400,0.1,0,21,40,9,15,4.5,1.5,0,1,2,10100,111,111)
```

		Select the softkey for "Linear hole patterns" to start entering the position data for the four circular pockets Ø30 mm. LAB ➔ Name of the jump label for program section repetition PL ➔ Machining plane G17 (XY) FRAME X0 Start position in X Y0 Start position in Y α0 Angle of rotation αX Shear plane angle X αY Shear plane angle Y L1 Distance between columns L2 Distance between rows N1 Number of columns N2 Number of rows

Program excerpt:

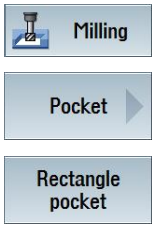
KTD30: CYCLE801(30,25,0,90,50,2,2,1,0,0,0,,,1)
MCALL

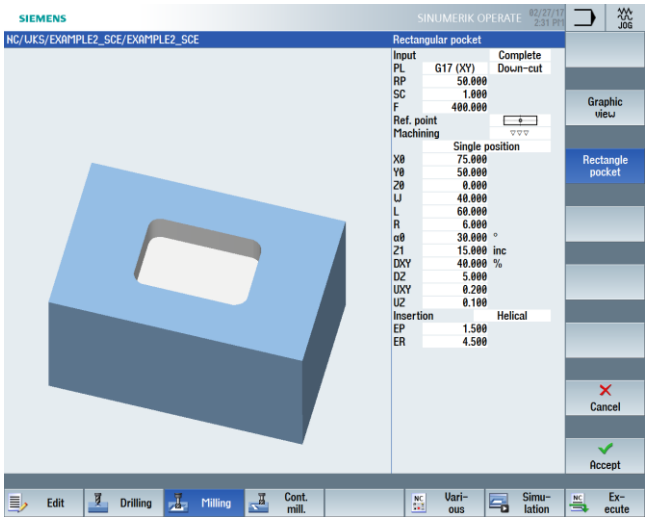
4.2.6 Rectangular/circular pocket (Finishing)



The finishing of the pockets will now be shown using the example of the rectangular pocket.

Implement the finishing operation for the circular pockets correspondingly on your own .





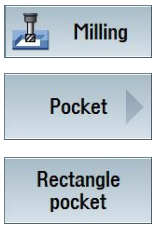
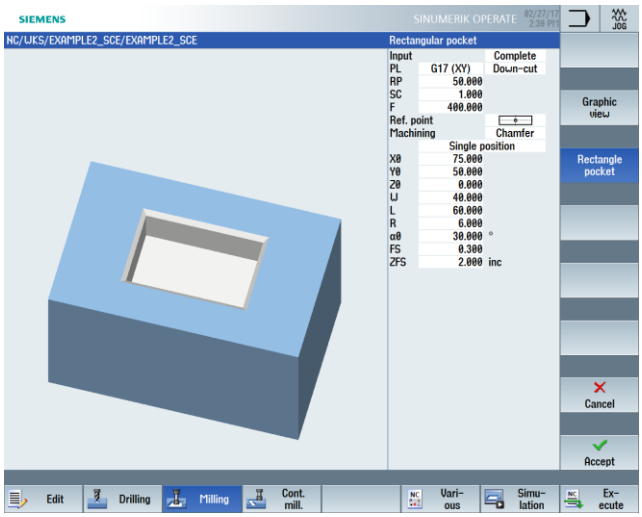
PL ➔ G17 (XY)
Down-cut
RP ➔ 50 mm
SC ➔ 1 mm
F ➔ 400 mm/min
Reference point: CENTER
Machining Finishing
Single position
X0 ➔Pocket center X0 = 75
Y0 ➔ Pocket center Y0 = 50
Z0 ➔ Coordinate surface
Start of machining in Z direction Z0 = 0
W ➔ Pocket width 40 mm
L ➔Pocket length 60 mm
R ➔ Corner radius R 6 mm
α0 ➔ Twist angle 30°
Z1 ➔ Destination position in Z1 = 15 inc reference Z0
DXY ➔ Plane infeed as percentage of the active tool e.g. 40%
DZ ➔ max. Z infeed
UXY ➔Allowance in XY
UZ ➔ Allowance in Z
Insertion strategy Helical
EP ➔ Helix pitch
ER ➔ Helix radius
Complete machining

Program excerpt:

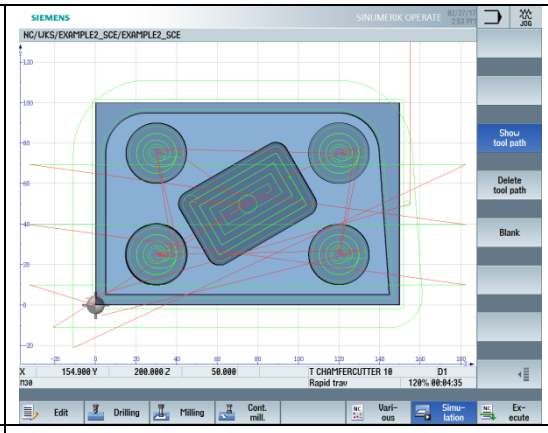
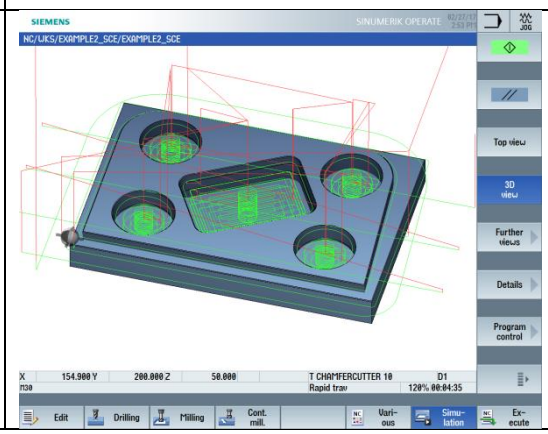
;Mill rectangular pocket ROUGHING
POCKET3(50,0,1,15,60,40,6,75,50,30,15,0.2,0.1,400,0.1,0,22,40,8,3,15,4.5,1.5,0,1,2,11100,11,111)

4.2.7 Chamfering contours

Now it will be shown how easy it is to machine chamfers with milling machining using the example of the rectangular pocket!

<table>
<tr>
<td> 
</td>
<td> 
</td>
<td>PL ➔ G17 (XY)
Down-cut
RP ➔ 50 mm
SC ➔ 1 mm
F ➔ 1200 mm/min
Reference point: CENTER
Machining Chamfer
Single position
X0 ➔Pocket center X0 = 75
Y0 ➔ Pocket center Y0 = 50
Z0 ➔ Coordinate surface
Start of machining in Z direction Z0 = 0
W ➔ Pocket width 40 mm
L ➔Pocket length 60 mm
R ➔ Corner radius R 6 mm
α0 ➔ Twist angle 30°
FS ➔ Chamfer size 0.3 mm
ZFS ➔ Infeed depth for chamfer machining 2 mm inc reference Z0</td>
</tr>
</table>

4.2.8 Simulating the milling program

Simu-lation Top view		Simulation of the machining operations in the top view with traversing paths Red ➔ Rapid traverse motions Green ➔ Feed motions
Simu-lation 3D view		Simulation of the machining operations in the 3D view with traversing paths Red ➔ Rapid traverse motions Green ➔ Feed motions

4.2.9 "Injection mold" solution program

```
G17 G40 G64 G71 G90
G54
WORKPIECE(,"",,"BOX",0,1,-21,-80,0,0,150,100)
;============================================
;Example by Easy Milling with programGUIDE
;Example 2 : Injection mold
;created with SinuTrain OPERATE V4.7
;============================================
T="FACEMILL 63"
M6
G17 G40 G64 G71 G90
G95 S1200 M3 M8
;Face milling
CYCLE61(50,1,1,0,0,0,150,100,0.5,66,1,400,12,0,1,10)
M5 M9
T="CUTTER 32"
M6
G17 G40 G64 G71 G90
G95 S1350 M3 M8
;Mill outer contour
CYCLE62(,2,"E_LAB_A_AK1","E_LAB_E_AK1")
CYCLE72("",50,0,1,5,5,0,0,400,200,101,41,1,10,0.1,1,10,0,1,2,101,1011,101)
M5 M9
T="CUTTER 10"
M6
G17 G40 G64 G71 G90
G95 S1400 M3 M8
;Mill rectangular pocket ROUGHING
POCKET3(50,0,1,15,60,40,6,75,50,30,2.5,0.2,0.1,400,0.1,0,21,40,8,3,15,4.5,1.5,0,1,2,11100,1
1,111)
;Mill circular pocket ROUGHING
MCALL
POCKET4(50,0,1,10,30,0,0,5,0.2,0.1,400,0.1,0,21,40,9,15,4.5,1.5,0,1,2,10100,111,111)
KTD30: CYCLE801(30,25,0,90,50,2,2,1,0,0,0,,,1)
MCALL
;Mill rectangular pocket ROUGHING
POCKET3(50,0,1,15,60,40,6,75,50,30,15,0.2,0.1,400,0.1,0,22,40,8,3,15,4.5,1.5,0,1,2,11100,11
,111)
;Mill rectangular pocket FINISHING
MCALL
POCKET4(50,0,1,10,30,0,0,10,0.2,0.1,400,0.1,0,22,40,9,15,4.5,1.5,0,1,2,10100,111,111)
REPEATB KTD30 ;#SM
MCALL
M5 M9
```

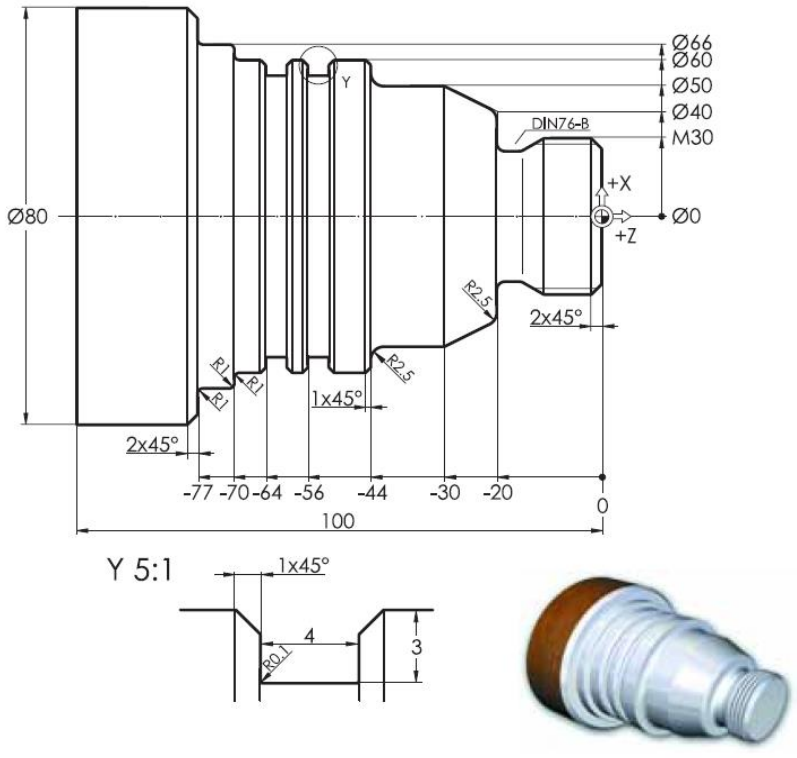
```
T="CHAMFERCUTTER 10"
M6
G17 G40 G64 G71 G90
G94 S6000 M3 M8
;Chamfer 0.3x45 degree various contours
POCKET3(50,0,1,15,60,40,6,75,50,30,15,0.2,0.1,1200,0.1,0,25,40,8,3,15,4.5,1.5,0,0.3,2,11100
,11,111)
MCALL
POCKET4(50,0,1,10,30,0,0,10,0.2,0.1,1200,0.1,0,25,40,9,15,4.5,1.5,0,0.2,2,10100,111,111)
REPEATB KTD30 ;#SM
MCALL
CYCLE72("",50,0,1,5,5,0,0,1200,200,105,41,1,5,0.1,1,5,0,0.3,2,101,1011,101)
CYCLE76(50,-5,1,,5,150,100,0,0,0,0,0.5,0.1,0.1,1200,200,0,5,12,6,0.3,2,2100,1,101)
G0 Y200 M5 M9
M30
;-------------------------------
;Local subprograms

E_LAB_A_AK1: ;#SM Z:2
;#7__DlgK contour definition begin – Don't change!;*GP*;*RO*;*HD*
G17 G90 DIAMOF;*GP*
G0 X5 Y5 ;*GP*
G1 Y95 RND=20 ;*GP*
X120 ;*GP*
G2 X139.95 Y76.411 I=AC(120) J=AC(75) ;*GP*
G1 X145 Y5 ;*GP*
X5 ;*GP*
;CON,0,0.0000,5,5,MST:0,0,AX:X,Y,I,J,TRANS:1;*GP*;*RO*;*HD*
;S,EX:5,EY:5;*GP*;*RO*;*HD*
;LU,EY:95;*GP*;*RO*;*HD*
;R,RROUND:20;*GP*;*RO*;*HD*
;LR,EX:120;*GP*;*RO*;*HD*
;ACW,AT:0,RAD:20;*GP*;*RO*;*HD*
;LA,EX:145,EY:5,AT:0;*GP*;*RO*;*HD*
;LA,EX:5,EY:5;*GP*;*RO*;*HD*
;#End contour definition end – Don't change!;*GP*;*RO*;*HD*
E_LAB_E_AK1:
```

5. Programming TURNING

In this section, you will learn the programming of the SINUMERIK 828D / 840D sl controllers based on two simple turned parts. As in the milling section, the following also applies here: The sample programs are intended as an introduction to give you an initial overview of the programming possibilities of the control.	
	When you have practice, you can optimize the programs later to reflect your own ideas. With the second shaft, you will get to know the SINUMERIK contour calculator and functions for complete machining.

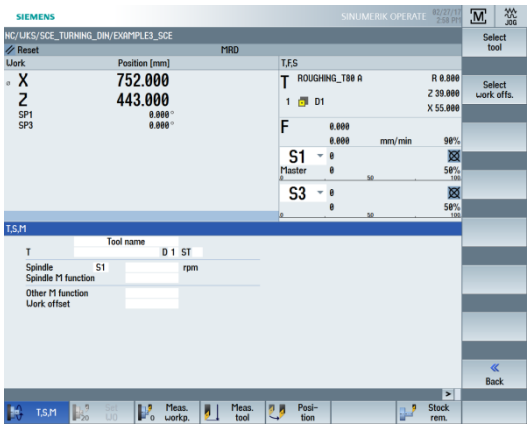
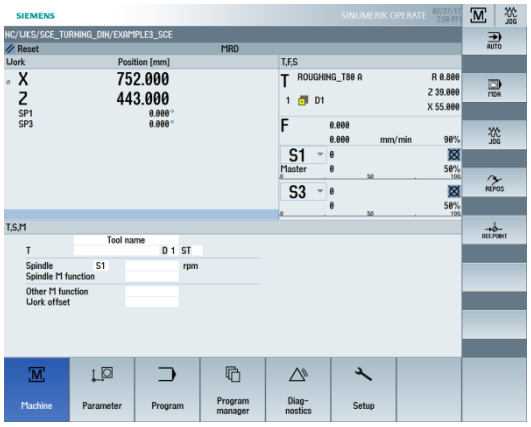
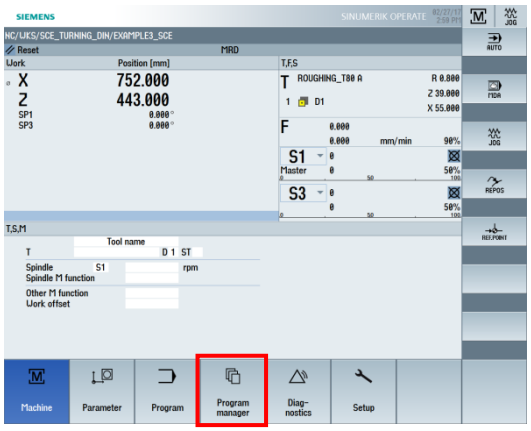
5.1 "Shaft" workpiece

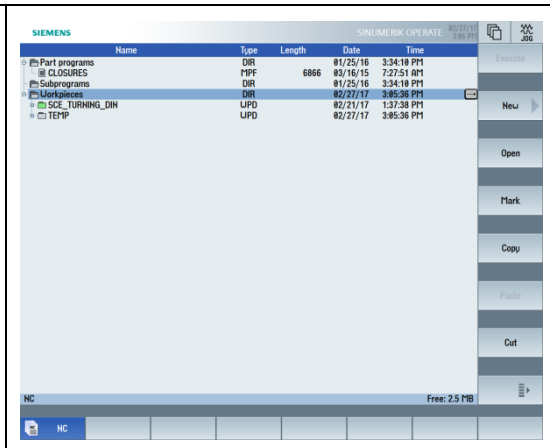
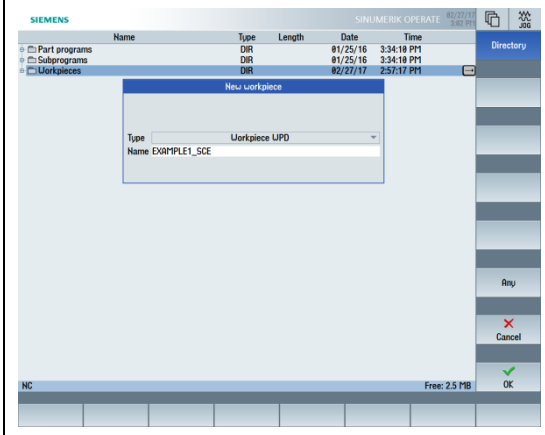
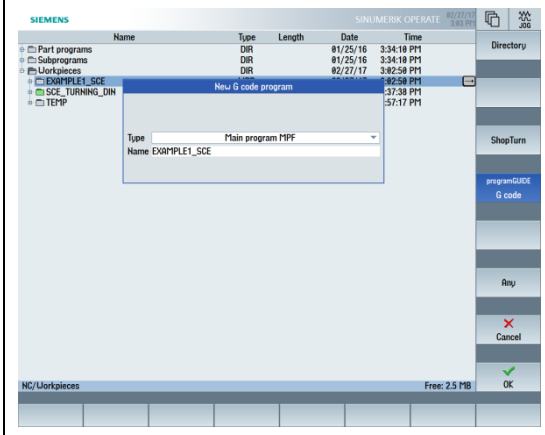
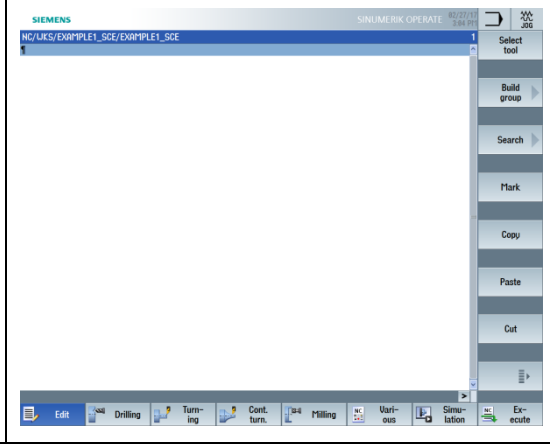
You will learn how to go from the drawing to the finished NC program with complete keystroke by keystroke instructions based on the "shaft" workpiece (blank ø80, length 101). The following topics will be covered



- Local subprogram technique for the contour description
- Tool call, cutting speed, basic functions
- Face turning with G0 / G1
- Contour call CYCLE62
- Stock removal cycle CYCLE962 against contour Roughing and finishing
- Thread undercut cycle CYCLE940
- Thread cycle CYCLE99
- Grooving cycle CYCLE930

5.1.1 Creating the workpiece and part program

Keys / Inputs	Screen / Drawing	Explanation
		Initial status: Any operating area (here, "Machine") and operating mode ("JOG")
		Switch to the main menu.
		Switch to the "Program Manager" operating area using the softkey. There are various directory and file types. The "Workpieces" (WPD) directory type is a directory into which all relevant data of a machining task (part programs, subprograms, etc.) can be stored. This allows all files to be clearly organized.

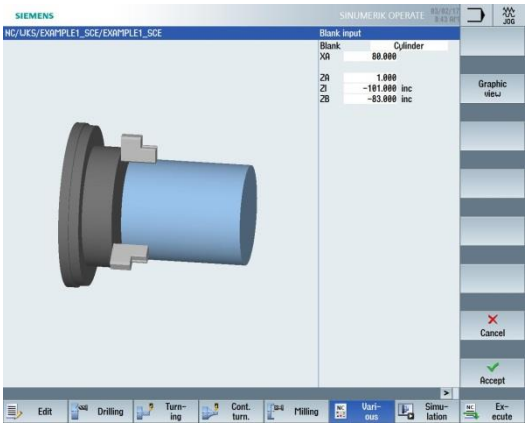
New		Create a new workpiece directory for the longitudinal guide "EXAMPLE1_SCE". Note that each name can be used only once. (You therefore may have to select a different name.)
INPUT	You apply text and numeric entries on the control keyboard with the yellow <Input> key or on the PC with <Enter>.	
Directory OK		Select the "NEW" softkey and the "Directory" softkey. In the following dialog window, enter the desired directory name "EXAMPLE1_SCE" and except the name with “OK”.
programGUIDE G code OK		In the communication between operator and control, the suggestion follows for program creation in "programGUIDE G-CODE" with the same name as the directory. This suggestion is “OK” and you therefore accept it.
		This takes you to the editor for the programGUIDE programming.

5.1.2 Beginning of turning program

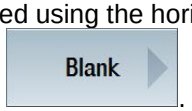
G18 G40 G64 G71 G90  G54 	These are basic functions and are explained in more detail in the overview below. Often these functions are valid for an entire program. For the sake of safety, however, it is recommended to execute these functions at every tool change.
LIMS = 3000	Speed limitation for G96 e.g. capped at 3000 rpm
Explanation of the functions	**Functions of the same group**
G18 – Plane selection ZX plane (turning)	G17 – Plane selection XY plane (milling) G19 – Plane selection YZ plane (cylinder surface)
G40 – Cutter radius compensation deselection	G41 – Cutter radius compensation ON in machining direction LEFT of the contour in the machining direction G42 – Cutter radius compensation ON in machining direction RIGHT of the contour in the machining direction
G64 – Round contour The destination of a traversing block is not approached exactly. Instead, there is a small rounding to the subsequent traversing path.	G60 – Exact stop The destination is approached exactly. All axis drives are braked to a standstill for this.
G71 – Millimeter dimension unit	G70 – Inch dimension unit
G90 – Absolute dimensions	G91 – Incremental dimensions
G54 – Activation of the first adjustable work offset	G55, G56, G57 – Other work offsets G53 – Cancellation of all work offsets (acts block-wise) G500 – Deactivation of all work offsets

The beginning of the program also includes the option of defining the blank for the simulation.

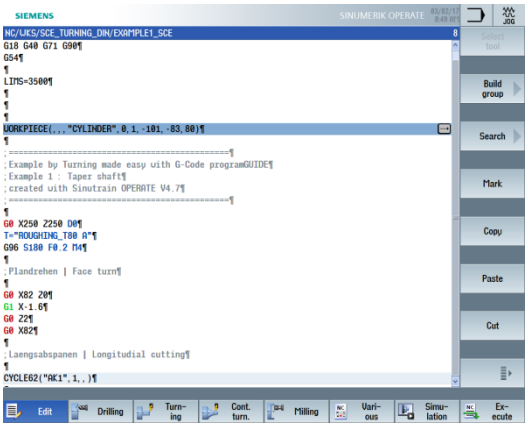
Five different blank shapes are possible, irrespective of which technology (milling or turning) is selected: Cylinder, Pipe, Block centered, Block and N corner.



The blank is programmed using the horizontal softkey "Various" 

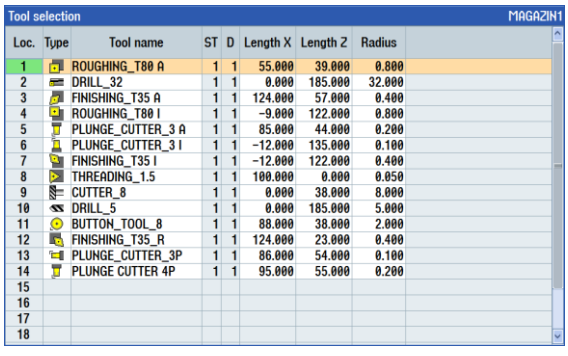
and the vertical softkey 

.



The tool is called using the 

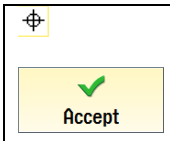
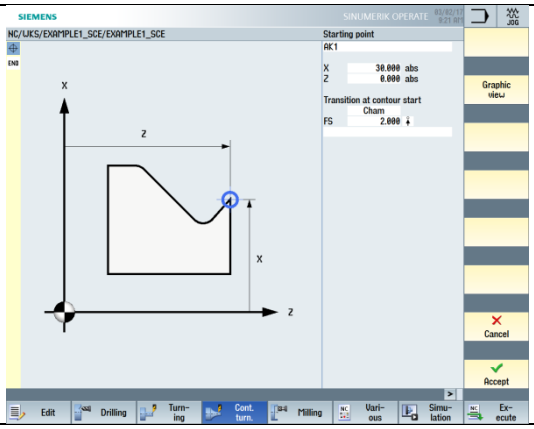
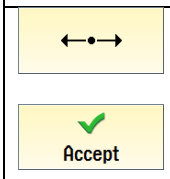
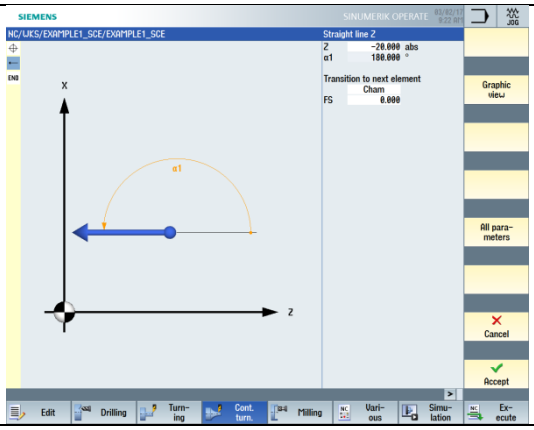
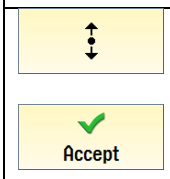
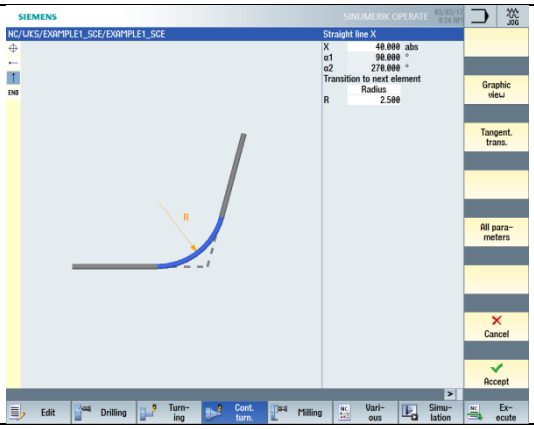
softkeys followed by selection of the desired tool from the tool list dialog window.

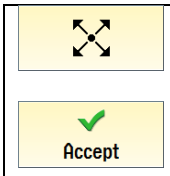
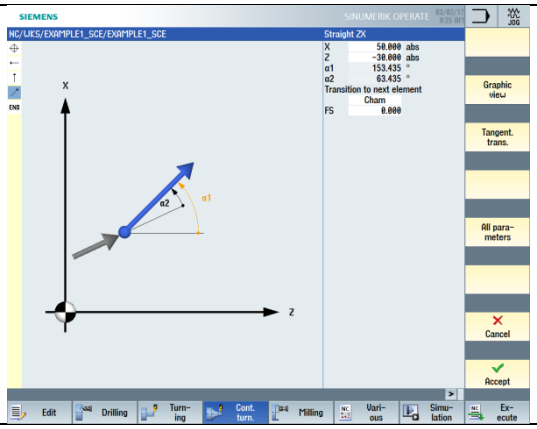
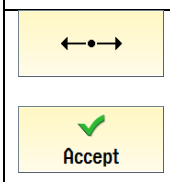
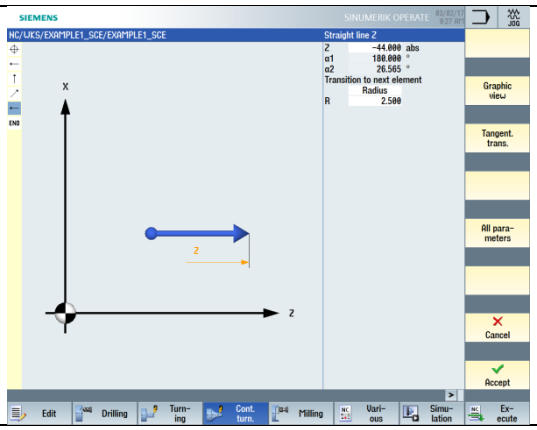
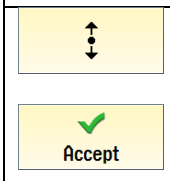
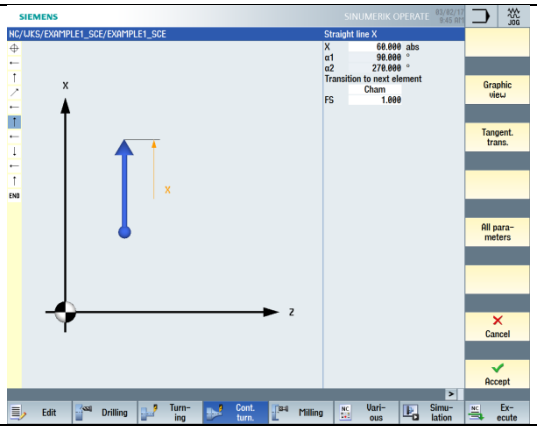
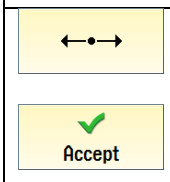
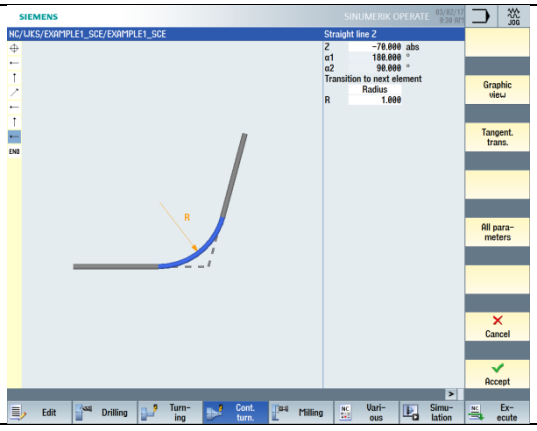


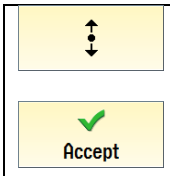
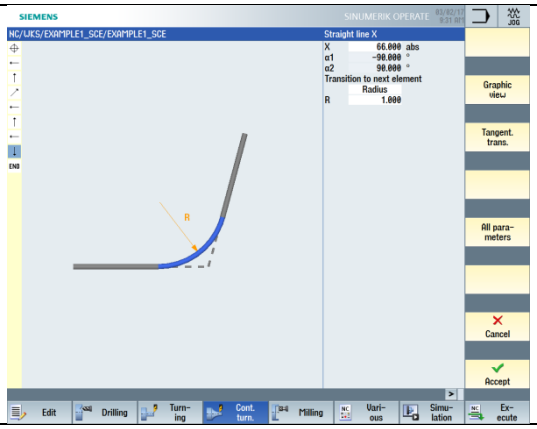
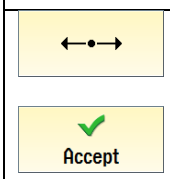
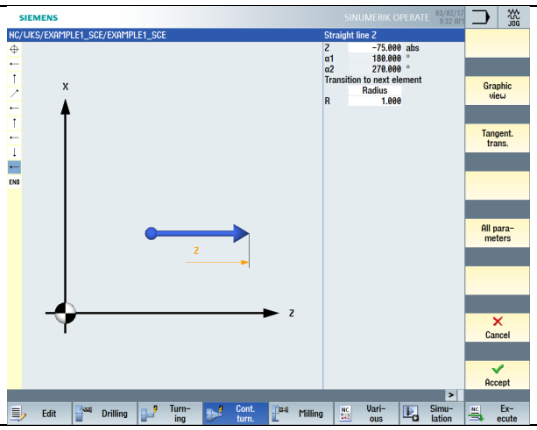
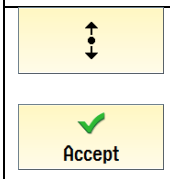
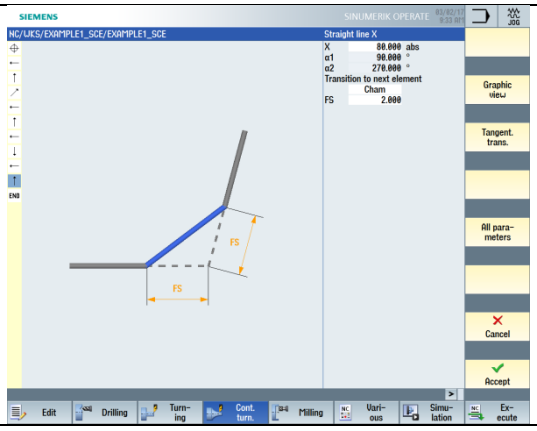
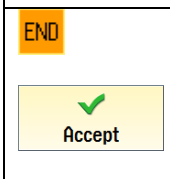
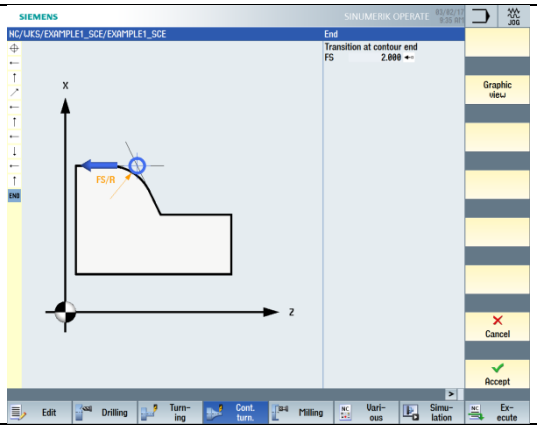
Loc.	Type	Tool name	ST	D	Length X	Length Z	Radius
1		ROUGHING_T80 A	1	1	55.000	39.000	0.800
2		DRILL_32	1	1	0.000	185.000	32.000
3		FINISHING_T35 A	1	1	124.000	57.000	0.400
4		ROUGHING_T80 I	1	1	-9.000	122.000	0.800
5		PLUNGE_CUTTER_3 A	1	1	85.000	44.000	0.200
6		PLUNGE_CUTTER_3 I	1	1	-12.000	135.000	0.100
7		FINISHING_T35 I	1	1	-12.000	122.000	0.400
8		THREADING_1.5	1	1	100.000	0.000	0.050
9		CUTTER_8	1	1	0.000	38.000	8.000
10		DRILL_5	1	1	0.000	185.000	5.000
11		BUTTON_TOOL_8	1	1	88.000	38.000	2.000
12		FINISHING_T35_R	1	1	124.000	23.000	0.400
13		PLUNGE_CUTTER_3P	1	1	86.000	54.000	0.100
14		PLUNGE CUTTER 4P	1	1	95.000	55.000	0.200
15							
16							
17							
18							

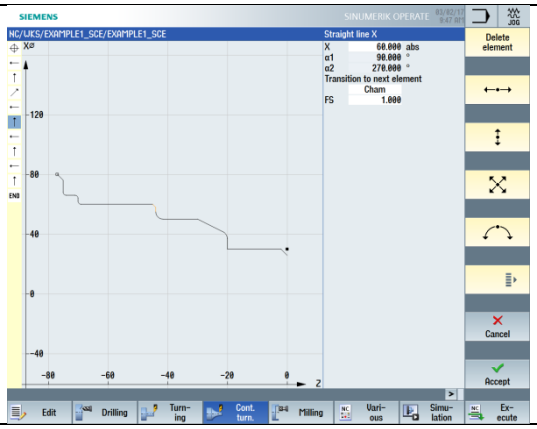
The face turning operation is programmed in G0 and G1 blocks.

5.1.3 Contour turning with the contour calculator

Working with the contour calculator		
		Enter a name for the contour to be created.
		Enter the starting point for the contour with X30 Z0 and a chamfer-type transition at the contour start with size 2 and direction  .
		Horizontal line to Z-20 mm Note: You make the thread undercut DIN 76-B required here later in the program with a standard turning cycle.
		Vertical line to X 40 mm and a rounding with radius R 2.5 mm.

		Inclined line with known X and Z end position X 50 mm Z -30 mm.
		Horizontal line to Z-44 mm and a rounding with radius R 2.5 mm.
		Vertical line to X 60 mm and a chamfer with a width of 1 mm.
		Horizontal line to Z-70 mm and a rounding with radius R 1 mm.

		Vertical line to X 66 mm and a rounding with radius R 1 mm.
		Horizontal line to Z-75 mm and a rounding with radius R 1 mm.
		Vertical line to X 80 mm and a chamfer with a width of 2 mm.
		In this text box, you give the previously programmed chamfer its left pointing direction .

Accept		Use the "Accept" softkey to apply the created contour to the program. ☺

The contour created with the contour calculator is placed after the end of program M30 where it does not affect the program.

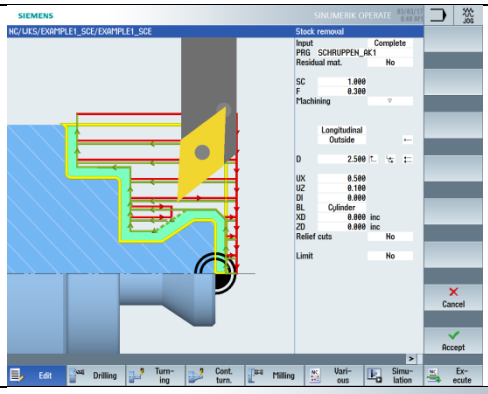
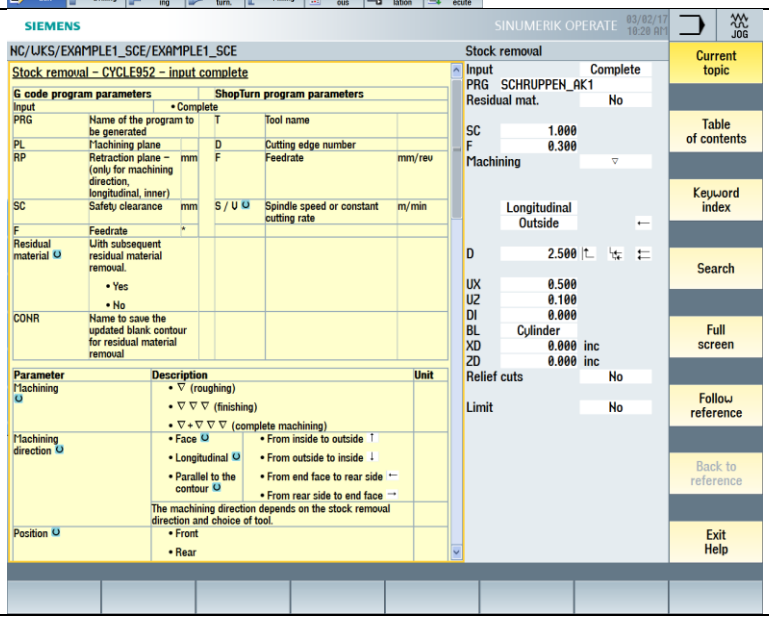
It can be edited at any time once you are in the contour description area by selecting the arrow  pointing right  at the right edge of the editor.

```
;-------------------------------------------------¶
;lokale Unterprogramme | local Subprogram¶
¶
;AUSSEN-Kontur | OUTSIDE-Contour¶
E_LAB_A_AK1:  ;#SM Z:22¶
G18 G90 DIAM90;*GP*¶
G0 Z0 X26 ;*GP*¶
G1 X30 CHR=2 ;*GP*¶
Z-20 ;*GP*¶
X40 RND=2.5 ;*GP*¶
```

Program excerpt:

```
M30
;---------------------------------------------
;Local subprograms

;OUTER contour
E_LAB_A_AK1: ;#SM Z:6
;#7__DlgK contour definition begin – Don't change!;*GP*;*RO*;*HD*
G18 G90 DIAM90;*GP*
G0 Z0 X28 ;*GP*
```

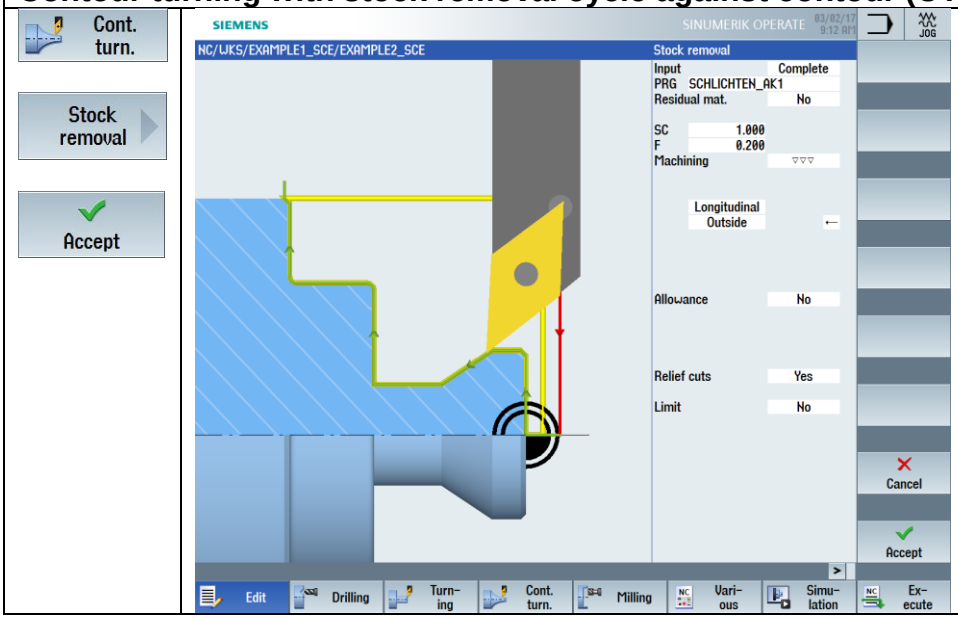
Contour turning with stock removal cycle against contour (CYCLE952) ROUGHING

Cont. turn. Contour Contour call Accept	Contour call Contour name CON AK1	Select the contour name variant in the first text box. Enter the name for the contour to be machined.
Cont. turn. Stock removal Accept	 	Softkey "Cont. turn." Softkey "Stock removal" Accept the inputs as shown on the left.
Cont. turn. Stock removal Accept		For detailed information, you can use the online help at any time. To do this, press the "Help" button. HELP

Program excerpt:

;Contour turning
CYCLE62("AK1",1,,)
CYCLE952("SCHRUPPEN_AK1",,"",2101311,0.3,0,0,2.5,0.1,0.1,0.5,0.1,0.1,0,1,0,0,,,,,2,2,,,0,1,,0,12,1100010,1,0,0.1)

Contour turning with stock removal cycle against contour (CYCLE952) FINISHING



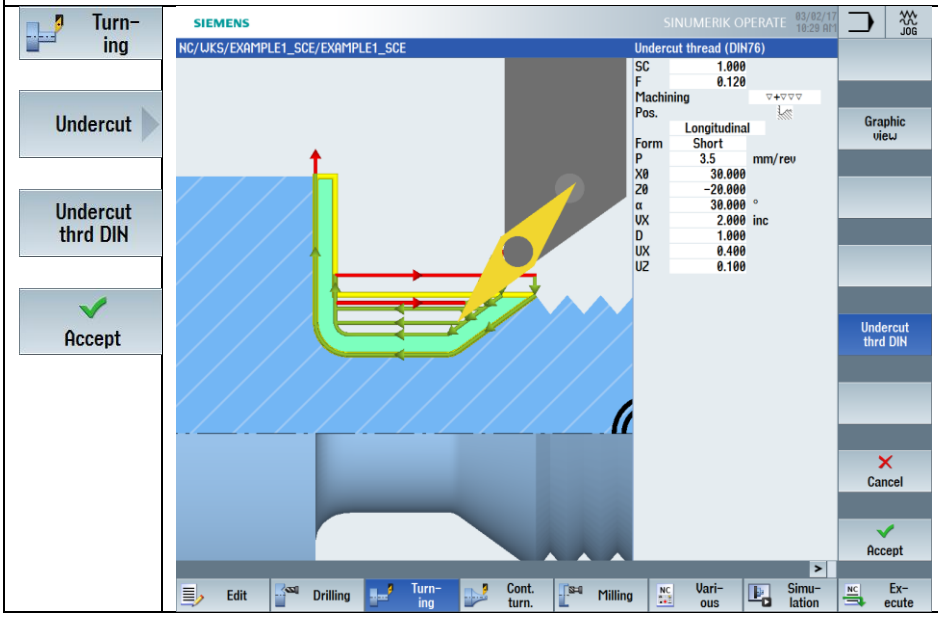
Once you have approached the tool change position in the finishing tool has been brought to working position, the finishing of the outer contour follows. A contour call is NOT necessary because the current call is still valid.

Program excerpt:

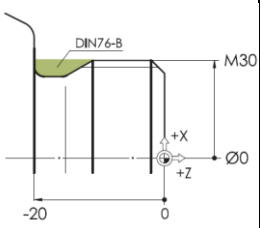
```
T="FINISHING_T35 A"
G96 S200 F0.2 M4
;Contour turning Finishing
CYCLE952("SCHLICHTEN_AK1",,"",2101321,0.2,0,0,2.5,0.1,0.1,0.5,0.1,0.1,0,1,0,0,,,,,2,2,,,0,1,
,0,12,1100010,1,0,0.1)
```

5.1.4 Thread undercut DIN 76-B

Thread undercut DIN 76-B



Select the contour name variant in the first text box. Enter the name for the contour to be machined.

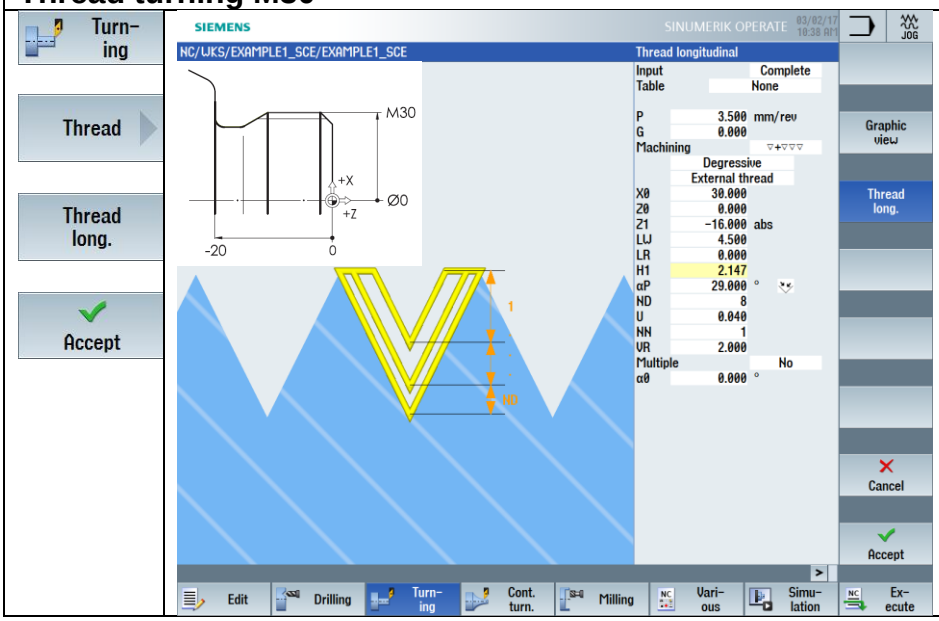


Program excerpt:

```
;Thread undercut DIN 76-B
CYCLE940(30,-20,"B",1,1,0.12,13,,,,,30,2,1,0.1,0.4,0.1,18,,,2,1100)
G0 X250 Z250 D0
```

5.1.5 Thread turning nominal diameter M30

Thread turning M30



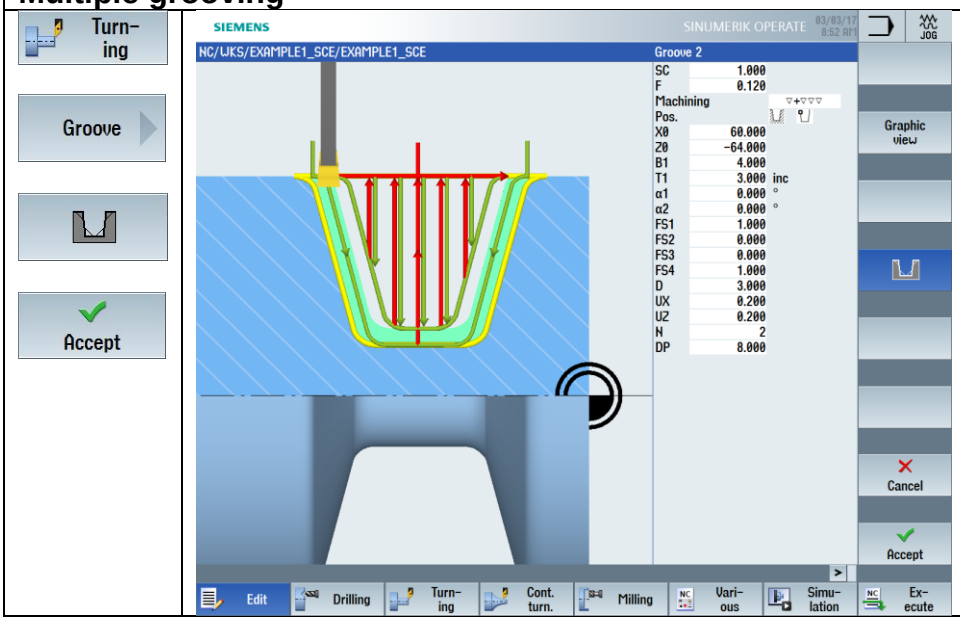
You can find the thread longitudinal cycle in the standard turning cycles. The function
"Table: None" means that you defined the thread using its pitch P.
The correct thread depth H1 is calculated automatically based on this.
Refer to the online help for more details.

Program excerpt:

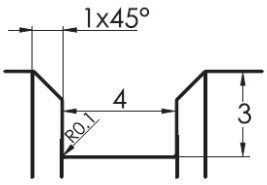
```
T="THREADING_3.5"
;Comment: Tool in overhead position clamped behind the turning center!!!
G95 S800 M3
CYCLE99(0,30,-
16,,4.5,0,2.14704,0.04,29,0,8,1,3.5,1300203,4,2,0.398193,0.5,0,0,1,0,0.510053,1,,,,102,0)
```

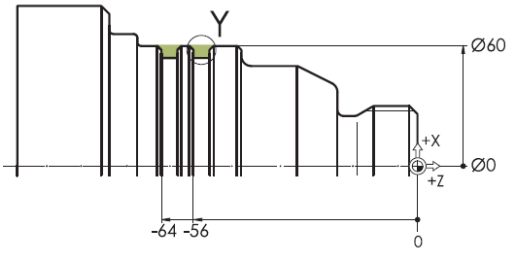
5.1.6 Multiple grooving

Multiple grooving



You can find the Groove cycle in the standard turning cycles. Choose variant 2 with transition elements.

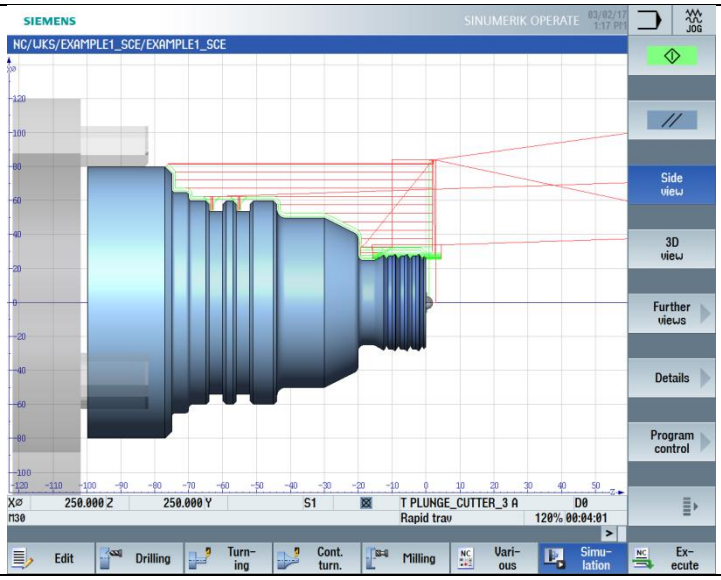
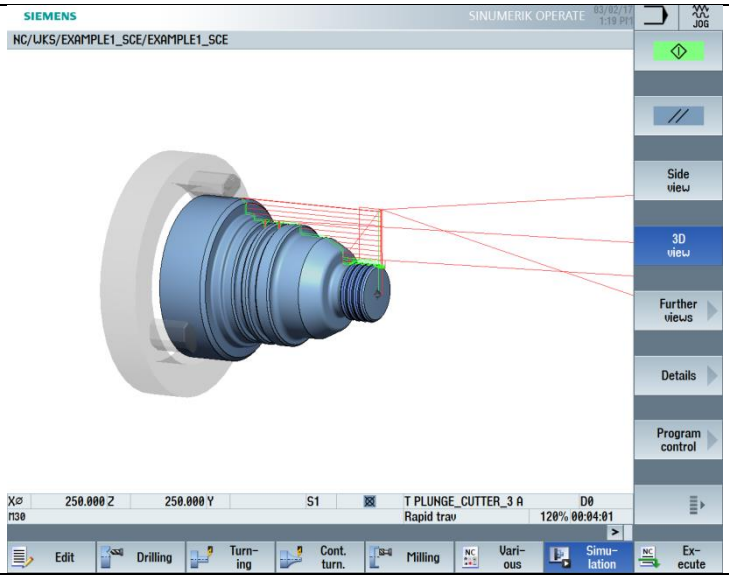




Program excerpt:

```
T="PLUNGE_CUTTER_3 A"
G96 S120 F0.12 M4
;Groove 4 wide
CYCLE930(60,-64,4,4,3,,0,0,0,1,0,0,1,0.2,3,1,10130,,2,8,0.12,1,0.2,0.2,2,1001110)
```

5.1.7 Simulating the turning program

Simu-lation Side view		Simulation of the machining operations in the side view with traversing paths Red ➔ Rapid traverse motions Green ➔ Feed motions
Simu-lation 3D view		Simulation of the machining operations in the 3D view with traversing paths Red ➔ Rapid traverse motions Green ➔ Feed motions

5.1.8 "Shaft" solution program

```
G18 G40 G64 G71 G90
G54
LIMS=3500
WORKPIECE(,,,"CYLINDER",0,1,-101,-83,80)
;================================================
;Example by Turning made easy with G-Code programGUIDE
;Example 1: Taper shaft
;created with SinuTrain OPERATE V4.7
;================================================
G0 X250 Z250 D0
T="ROUGHING_T80 A"
```

```
G96 S180 F0.2 M4
;Face turning
G0 X82 Z0
G1 X-1.6
G0 Z2
G0 X82
;Contour turning ROUGHING
CYCLE62("AK1",1,,)
CYCLE952("SCHRUPPEN_AK1",,"",2101311,0.3,0,0,2.5,0.1,0.1,0.5,0.1,0.1,0,1,0,0,,,,,2,2,,,0,1,,
0,12,1100010,1,0,0.1)
G0 X250 Z250 D0
T="FINISHING_T35 A"
```

```
G96 S180 F0.2 M4
;Contour turning Finishing
CYCLE952("SCHLICHTEN_AK1",,"",2101321,0.2,0,0,2.5,0.1,0.1,0.5,0.1,0.1,0,1,0,0,,,,,2,2,,,0,1,
,0,12,1100010,1,0,0.1)
;Thread undercut DIN 76-B
CYCLE940(30,-20,"B",1,1,0.12,13,,,,,30,2,1,0.1,0.4,0.1,18,,,2,1100)
G0 X250 Z250 D0
T="THREADING_3.5"
;Comment: Tool in overhead position clamped behind the turning center!!!
```

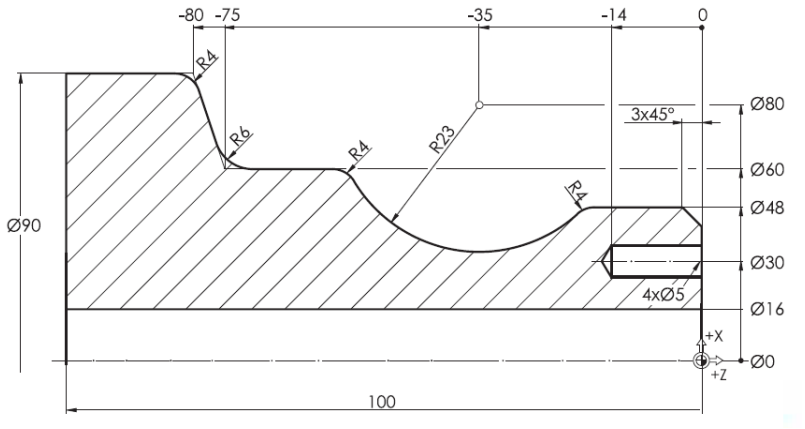
```
G95 S800 M3
CYCLE99(0,30,-
16,,4.5,0,2.14704,0.04,29,0,8,1,3.5,1300203,4,2,0.398193,0.5,0,0,1,0,0.510053,1,,,,102,0)
G0 X250 Z250 D0
T="PLUNGE_CUTTER_3 A"
```

```
G96 S120 F0.12 M4
;Groove 4 wide
CYCLE930(60,-64,4,4,3,,0,0,0,1,0,0,1,0.2,3,1,10130,,2,8,0.12,1,0.2,0.2,2,1001110)
G0 X250 Z250 D0
M30
```

```
;---------------------------------------------
;Local subprograms

;OUTER contour
E_LAB_A_AK1: ;#SM Z:6
;#7__DlgK contour definition begin - Don't change!;*GP*;*RO*;*HD*
G18 G90 DIAM90;*GP*
G0 Z0 X28 ;*GP*
G1 X30 CHR=1 ;*GP*
Z-20 ;*GP*
X40 RND=2.5 ;*GP*
Z-30 X50 ;*GP*
Z-44 RND=2.5 ;*GP*
X60 CHR=1 ;*GP*
Z-70 RND=1 ;*GP*
X66 RND=1 ;*GP*
Z-75 RND=1 ;*GP*
X80 CHR=2 ;*GP*
Z-77 ;*GP*
;CON,V64,2,0.0000,10,10,MST:3,2,AX:Z,X,K,I,TRANS:0;*GP*;*RO*;*HD*
;S,EX:0,EY:30,ASE:90;*GP*;*RO*;*HD*
;F,LFASE:1;*GP*;*RO*;*HD*
;LL,EX:-20;*GP*;*RO*;*HD*
;LU,EY:40;*GP*;*RO*;*HD*
;R,RROUND:2.5;*GP*;*RO*;*HD*
;LA,EX:-30,EY:50;*GP*;*RO*;*HD*
;LL,EX:-44;*GP*;*RO*;*HD*
;R,RROUND:2.5;*GP*;*RO*;*HD*
;LU,EY:60;*GP*;*RO*;*HD*
;F,LFASE:1;*GP*;*RO*;*HD*
;LL,EX:-70;*GP*;*RO*;*HD*
;R,RROUND:1;*GP*;*RO*;*HD*
;LU,EY:66;*GP*;*RO*;*HD*
;R,RROUND:1;*GP*;*RO*;*HD*
;LL,EX:-75;*GP*;*RO*;*HD*
;R,RROUND:1;*GP*;*RO*;*HD*
;LU,EY:80;*GP*;*RO*;*HD*
;F,LFASE:2,ASE:180;*GP*;*RO*;*HD*
;#End contour definition end - Don't change!;*GP*;*RO*;*HD*
E_LAB_E_AK1:
```

5.2 "Complete" workpiece

Besides a repetition of the "classic" turning machining previously described for the "Shaft" example, you will learn other elementary and useful aspects of the controller based on the "Complete" workpiece (blank ø90, length 101):



- CYCLE952 in combination roughing only; Residual material machining and finishing
- Centric drilling on the turning machine
- Off-center machining of the front face with the TRANSMIT function (with driven tools) CYCLE82
- Hole circle cycle HOLES2

The creation of a workpiece directory (WPD) and a program file MPF (Main Program File) is not described here.

If you need to look these up, see section 4.1.1

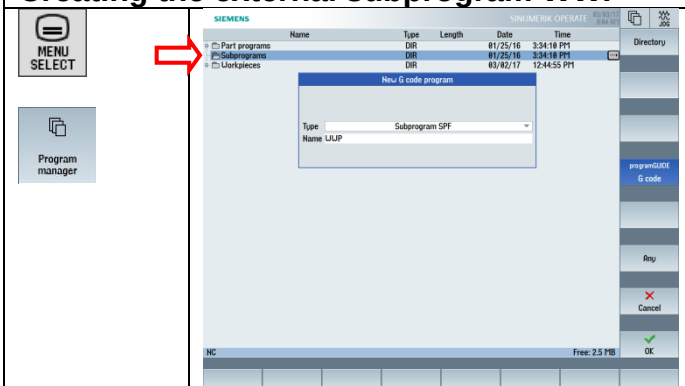
5.2.1 Creating the external subprogram

For machining the "Compete" workpiece, you want to use the option of providing the tool change point in an external subprogram in the "Subprograms" directory globally for all programs in which you want to use the subprogram.

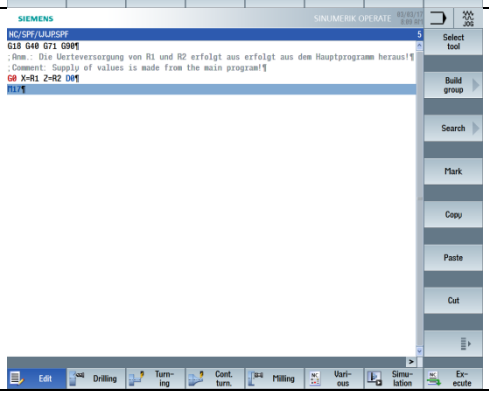
The program will be named WWP.

It is necessary for this subprogram that the tool is placed at a free position outside the workpiece.

Creating the external subprogram WWP



Open the "Program Manager" using "Menu Select", select the "Subprograms" directory and create a new file named "**WWP**" with the "NEW" softkey.



The file contains basic program functions and the travel to the tool change position with R parameters. Subprogram end M17.

Travel to tool change point

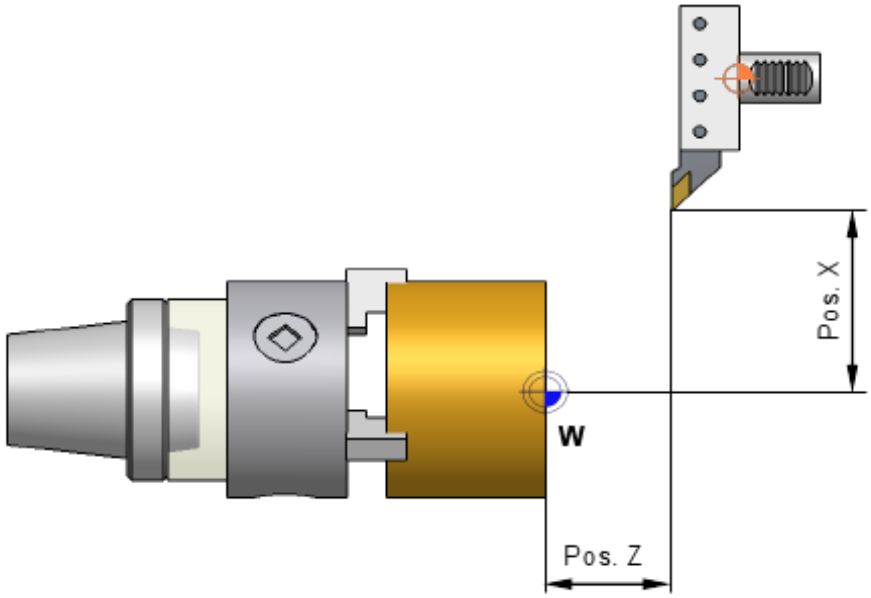
You want to use the "WWP" subprogram to achieve a variable approach of the tool change position relative to the workpiece zero without active cutting-edge data D0.

In general:

The swiveling of the turret must always be performed at a **collision-free** point in the working area of the turning machine.

For this, the tool carrier is usually withdrawn far into the positive area of the work area.

Programming example 1:	Explanation:
N50 G0 X350 Z250	;Approach of the tool change point with active cutting-edge data
N60 ----------	
N70 T1 **D1** ;Roughing tool	;Swiveling of the turret to T1, tool call. Data
N80 ……	



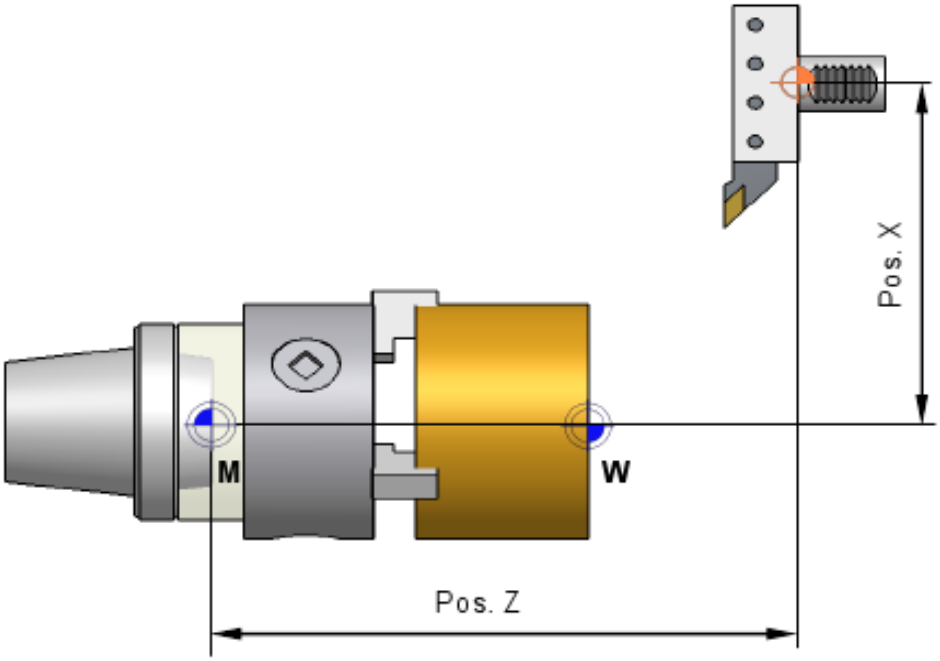
The tool carrier approaches different positions in X and Z depending on the active tool length.
The **tool tip** is always positioned.

In order to approach a **tool change point independent of** the tool lengths, the following conditions must be programmed.

Deactivating the tool lengths in X and Z Code D0

D word explanation: The D word is used to activate the cutting edge data that is stored in the tool list for the respective cutting-edge in the controller for the active tool.

Programming example 2:	Explanation:
N10 G18 G54 G…	
N20 G0 X350 Z250**D0**	Approach of the tool change point in the WCS relative to the tool carrier reference point without active tool data
N30 T1 ;Roughing tool	Swiveling of the turret to T1 in working position
N40 **D1**	Call of the tool data for T1 with D word D1 for cutting edge 1
D0	Deactivation of the tool dimensions
D1 to D9	Activation of the tool dimensions after the tool change



Nevertheless, be sure to always call the required tool cutting edge again after each tool change.

Program example:

G18 G40 G71 G90

G0 X=R1 Z=R2 D0

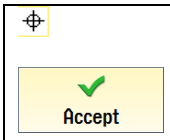
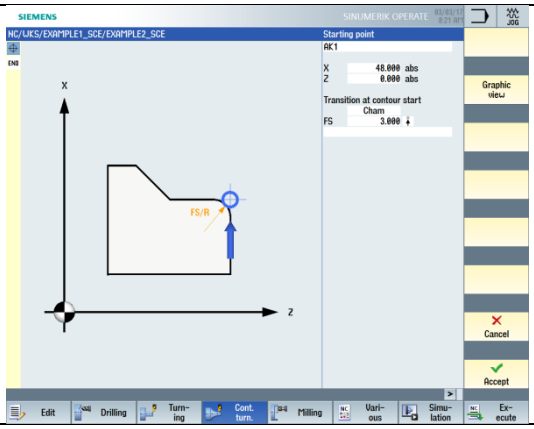
M17

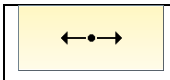
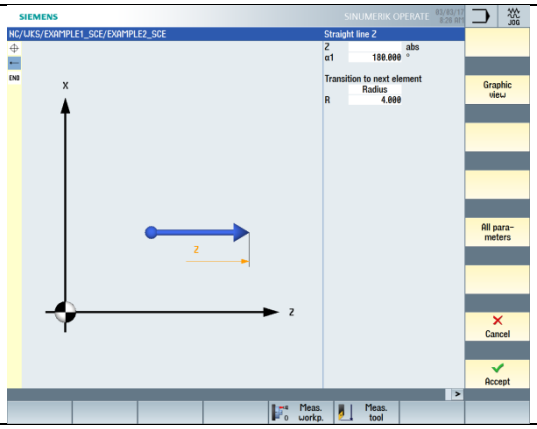
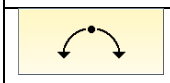
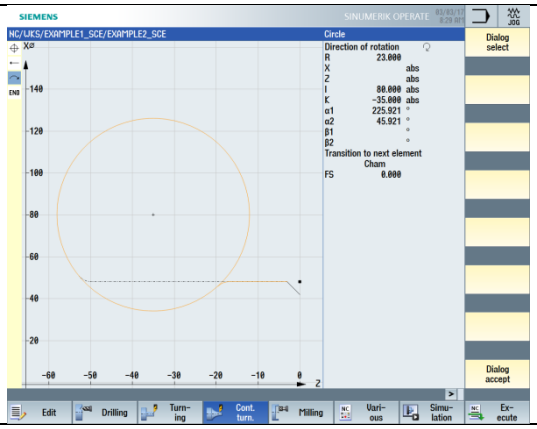
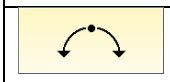
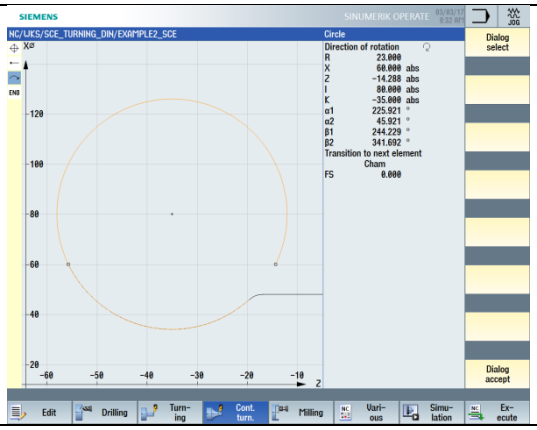
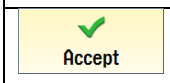
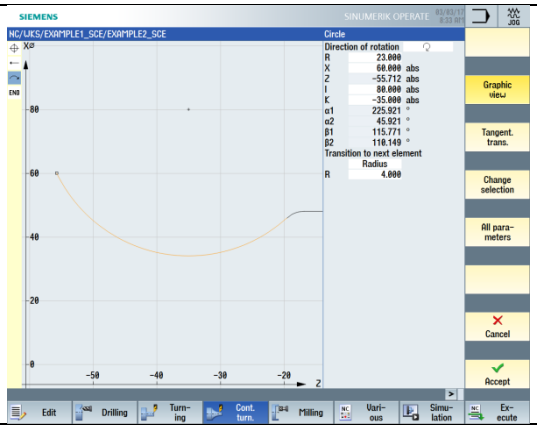
The basic program starting information and the face turning are already programmed. See current program excerpt.

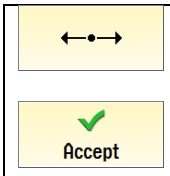
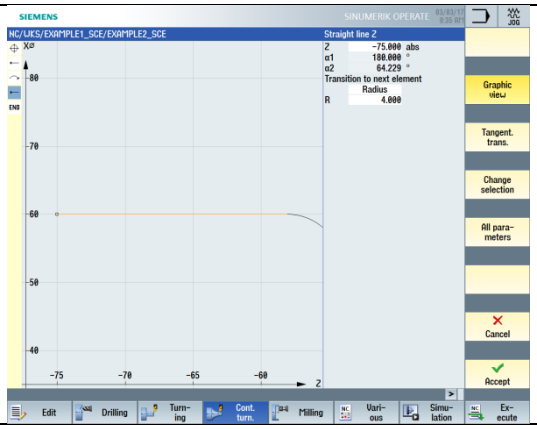
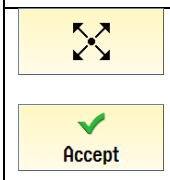
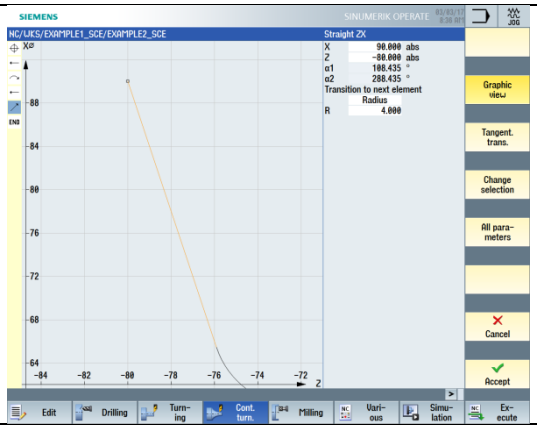
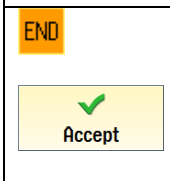
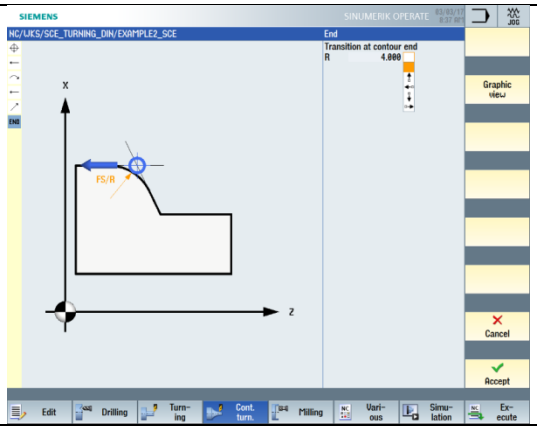
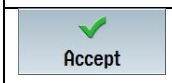
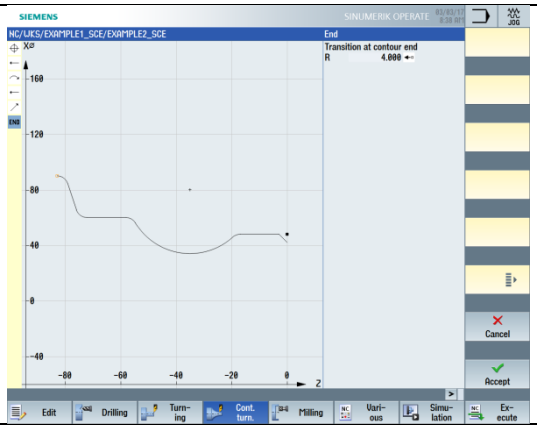
Current program excerpt:

```
G18 G40 G64 G71 G90
G54
LIMS=3500
WORKPIECE(,,,"CYLINDER",0,1,-101,-92,90)
;=====================================================
;Example by Turning made easy with G-Code programGUIDE
;Example 2 : Guide shaft
;created with SinuTrain OPERATE V4.7
;=====================================================
R1=250 ;WWP-Position X
R2=250 ;WWP-Position Z
;=====================================================
WWP
T="ROUGHING_T80 A"
G96 S180 F0.2 M4
;Face turning
G0 X92 Z0
G1 X-1.6
G0 Z2
G0 X92

M30
```

5.2.2 Contour turning with the contour calculator

Working with the contour calculator		
		Enter a name for the contour to be created.
		Enter the starting point for the contour with X48 Z0 and a chamfer-type transition at the contour start with size 3 and direction  .

 		Horizontal line with position not defined by end point at α1 = 180° and a rounding of 3 mm.
		After entering the following contour data I80 K-35, there is a selection for the length of the straight line. Select the short variant using "Dialog select" and then apply it with "Dialog accept".
		After entering the end diameter X 60 mm, another solution variant arises. In this case, it is the shorter variant that must be selected.
 		The rounding with radius R 4 mm is also entered.

		Horizontal line with an end position in Z direction of Z-75 and a rounding with radius R 6 mm.
		Now program an inclined straight line with the end position X90 Z-80 and a transition radius at the contour end with R 4 mm.
		In this text box, you give the previously programmed chamfer its left pointing direction  .
		Use the "Accept" softkey to apply the created contour to the program.

The contour created with the contour calculator is placed after the end of program M30 where it does not affect the program.

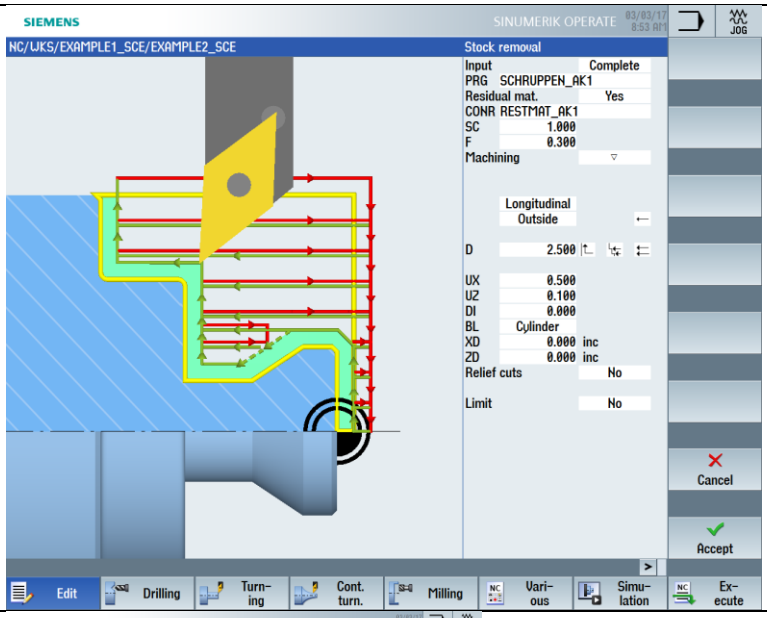
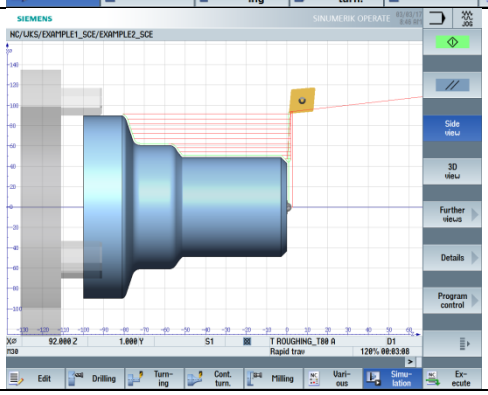
It can be edited at any time once you are in the contour description area by selecting the arrow  pointing right  at the right edge of the editor.

```
;-------------------------------------------------------¶
;lokale Unterprogramme | local subprograms¶
;Aussen-Kontur | Outside-Contour¶
E_LAB_A_AK1: ;#SM Z:16¶
G18 G90 DIAM90;*GP*¶
G0 Z0 X42 ;*GP*¶
G1 X48 CHR=3 ;*GP*¶
Z-18.477 RND=4 ;*GP*¶
```

Program excerpt:

```
M30
;---------------------------------------------
;Local subprograms
;Outer contour
E_LAB_A_AK1: ;#SM Z:3
;#7__DlgK contour definition begin - Don't change!;*GP*;*RO*;*HD*
G18 G90 DIAM90;*GP*
G0 Z0 X42 ;*GP*
G1 X48 CHR=3 ;*GP*
```

Contour turning with stock removal cycle against the contour (CYCLE952) ROUGHING

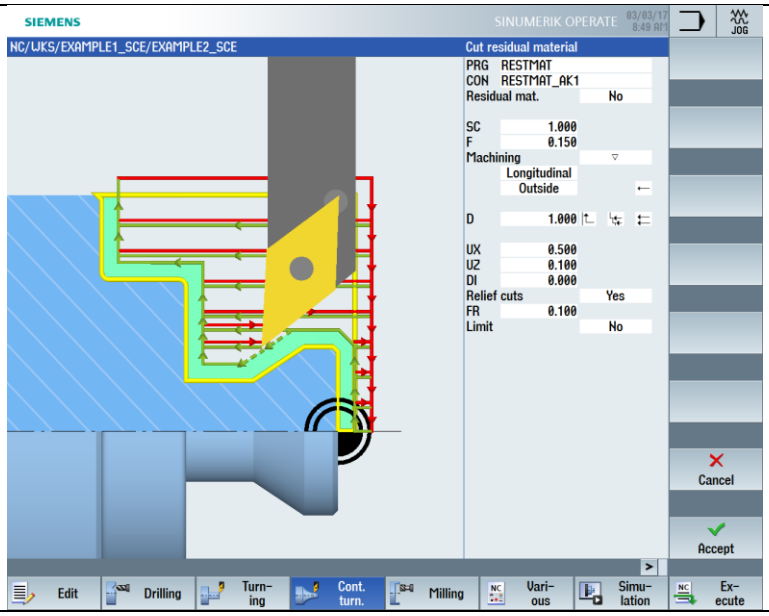
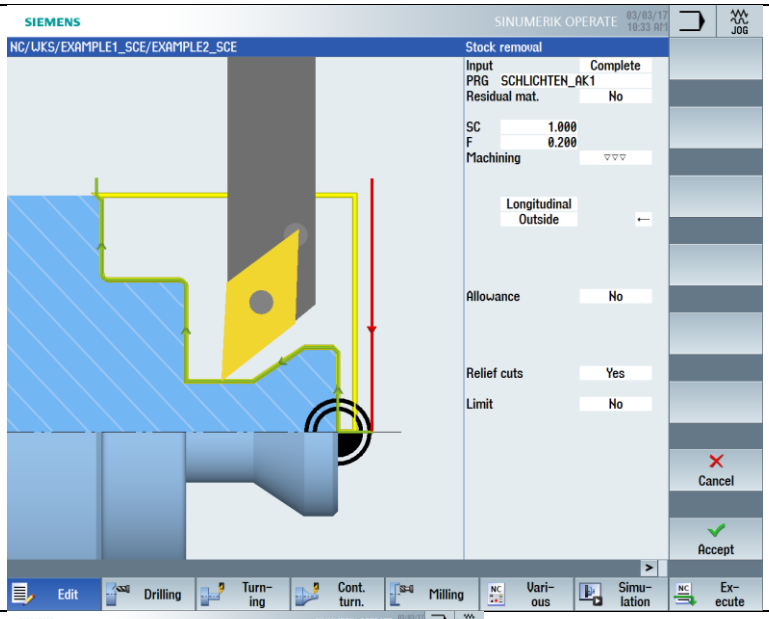
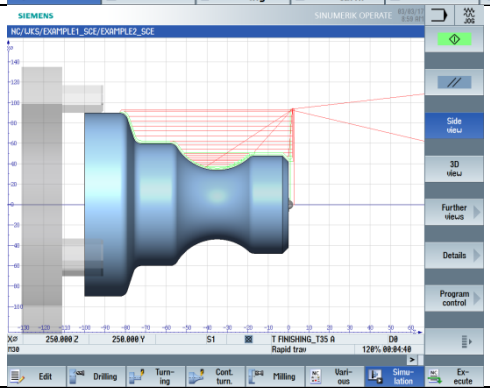
Cont. turn. Contour Contour call Accept	Contour call Contour name CON AK1	Select the contour name variant in the first text box. Enter the name for the contour to be machined.
Cont. turn. Stock removal Accept		Softkey "Cont. turn." Softkey "Stock removal"  Residual material "Yes" because you want to work with out relief cut machining.
Simu-lation		This is now the version after roughing machining without relief cut.

Current program excerpt:

;Longitudinal stock removal against contour WITHOUT relief cut
CYCLE62("AK1",1,,)
CYCLE952("SCHRUPPEN_AK1",,"RESTMAT_AK1",2101311,0.3,0,0,2.5,0.1,0.1,0.5,0.1,0.1,0,1
,0,0,,,,,2,2,,,0,1,,0,12,1110010,1,0,0.1)

WWP

Contour turning with stock removal cycle against the contour (CYCLE952) residual material

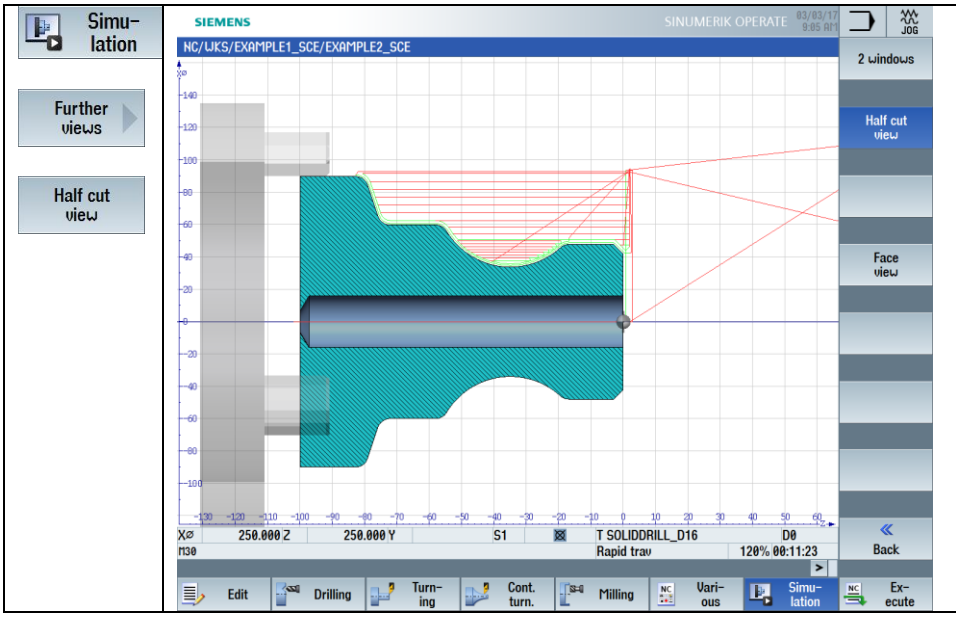
<table>
<tr>
<td>Cont. turn.
Cut resid stock
Accept</td>
<td> 
</td>
<td>Now let us continue with the machining of the residual material.
For this, you first load the finishing tool.
  </td>
</tr>
<tr>
<td></td>
<td>G96 S200 M4</td>
<td>Adaptation of the cutting data for finishing.</td>
</tr>
<tr>
<td>Cont. turn.
Stock removal
Accept</td>
<td> 
</td>
<td>With the finishing, you complete the machining of the outer contour.</td>
</tr>
<tr>
<td>Simu-lation</td>
<td>  </td>
<td>Current version with relief cut and finishing machining including traversing path display</td>
</tr>
</table>

Current program excerpt:

```
T="FINISHING_T35 A"
G96 S180 F0.2 M4
;Residual material machining (relief cut)
CYCLE952("RESTMAT","RESTMAT_AK1","",1101311,0.15,0.1,0,1,0.1,0.1,0.5,0.1,0.1,0,1,0,,,,,,
2,2,,,0,1,,0,112,1100010,1,0)
G96 S200 F0.2 M4
;Finish outer contour
CYCLE952("SCHLICHTEN_AK1",,"",1101321,0.2,0,0,2.5,0.1,0.1,0.5,0.1,0.1,0,1,0,0,,,,,2,2,,,0,1,
,0,12,1100010,1,0,0.1)
WWP
```

5.2.3 Centric drilling

Centric drilling with solid drill	
T="SOLIDDRILL_D16" ;Centric drilling G95 S1200 M3 M8	After turning, the 16mm through-hole will now be machined with a solid drill. During drilling, constant speed (G95) and feed in mm/revolution are used. Unlike for turning machining, the spindle rotates in the clockwise direction (M3).
G0 X0 Z2	The workpiece is approached in a rapid traverse. Ensure later when executing the program that a collision with the tailstock cannot occur.
G1 Z-102 F0.1	During the feed, the 100 mm long workpiece is drilled through in one pass (with 2 mm allowance).
G0 Z2	Position solid drill back to 2 mm in front of workpiece in rapid traverse.
WWP	Return to the tool change point with the created subprogram.



Current status of the machining steps.

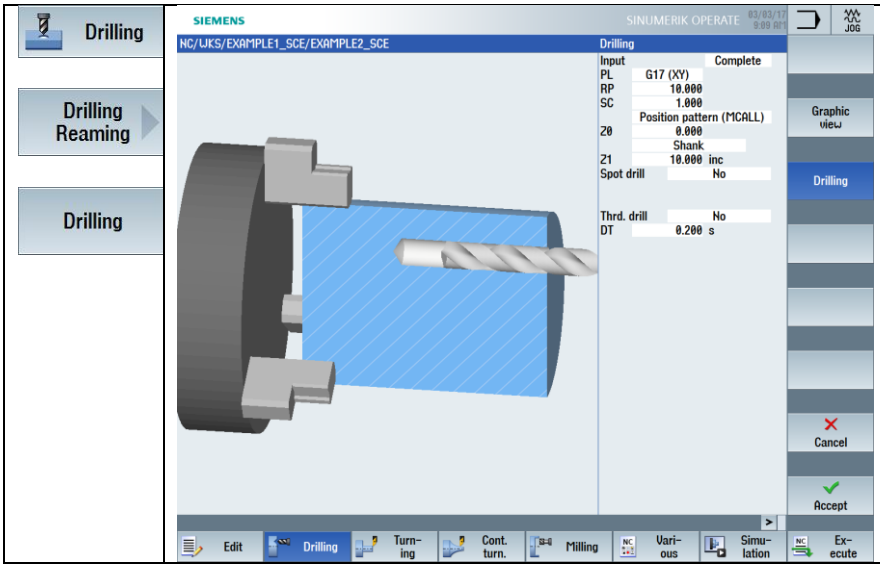
5.2.4 Face machining with TRANSMIT

Increasingly, turning machines also allow execution of milling and drilling machining operations on the front face and peripheral surface with powered tools.

Your SINUMERIK controller supports these machining operations on machines of this type.

The programming for a drilling pattern on the front face is presented here as an example.

Drilling on subcircle D30	
T="DRILL_5"	Tool call of powered drill; The tool call is the same as for a static tool.
SETMS(3)	The spindle that is now to be turned is located on the turret and configured as spindle 3 in SinuTrain for standard SIEMENS machines. For this reason, this spindle must be activated with the extended language word SETMS(X) (X = spindle number). This means you may have to make an adaptation if you want to apply the example to your machining, e.g. SETMS(1).
G95 S2200 F0.1 M3 M8	This line triggers a constant speed of 2200 rpm and a feed of 0.1 mm/revolution and clockwise spindle rotation.
;Drilling pattern face-end hole circle D30 4xD5	Comment line for better readability of the program.
TRANSMIT	This function (**Trans**form **M**illing **I**nto **T**urning) is used to transform the axes for milling and drilling on the front face. The following traversing motions can occur in the usual Cartesian coordinate system of milling (X, Y). The controller converts these program blocks for the real axes (X, C). The Z axis remains unchanged. (For peripheral surface machining, the corresponding function is called TRACYL).



Inputs in the drilling cycle
PL ➔ Machining plane
RP ➔ Retraction plane
SC ➔ Safety clearance
Cycle machining in combination with one or more hole patterns
Z0 ➔Start position in the Z direction
Dimension Z1 refers to the cylindrical part of the drill hole
Z1 ➔ 10 mm incremental from Z0
DT ➔ Dwell time in seconds

Now let us continue with the positioning of the previously defined "Drilling" machining.

The various input options of positions can be found under the "Drilling" softkey, "Positions" softkey.

Positioning options are:

Cartesian and polar single positions

Linear hole patterns - LINE

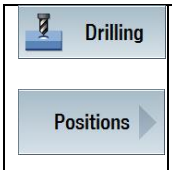
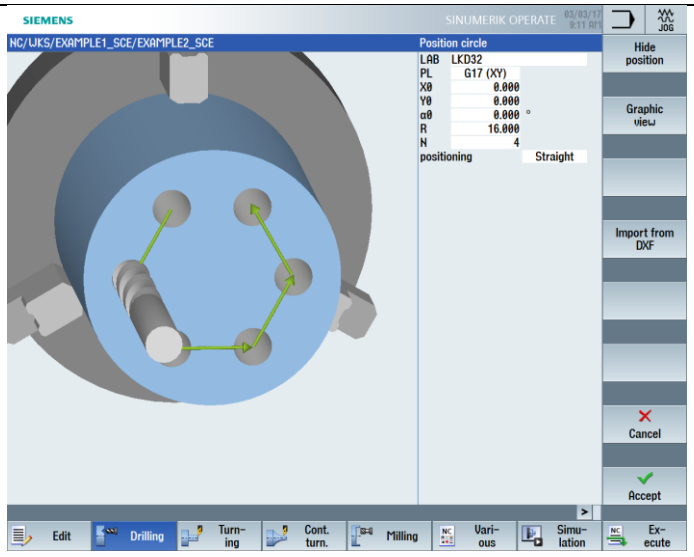
Linear hole patterns – GRID

Linear hole patterns – POSITION FRAME

Circular hole patterns – FULL CIRCLE

Circular hole patterns – SEMICIRCLE

You can also HIDE positions of hole pattern arrangements.

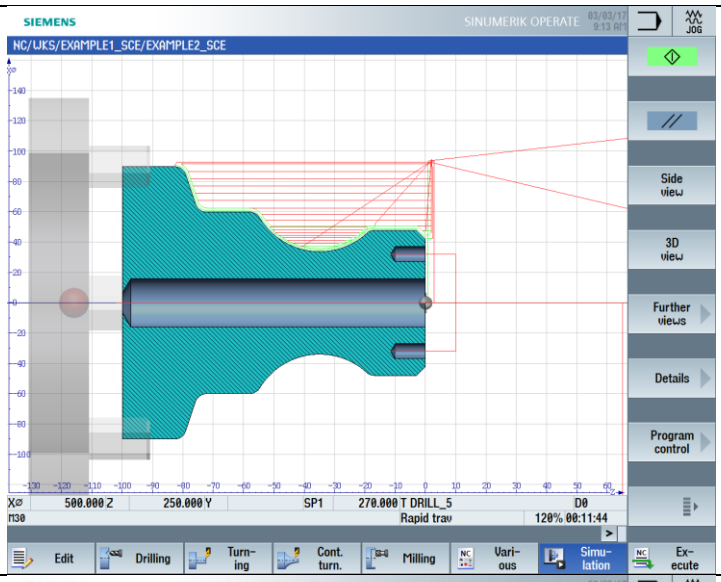
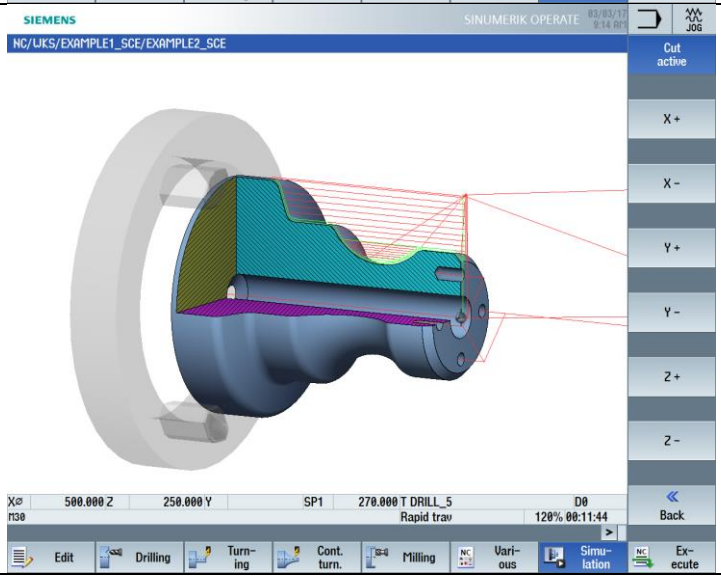
		Select horizontal softkey for opening the "Drilling" main menu. We now continue with the Full circle position input for the "Circular hole pattern" in the "Positions" submenu command on the vertical softkey bar. **LAB** ➔ Label name **PL** ➔ Machining plane Variant selection **X0** ➔ X position **Y0** ➔ Y position **α0** ➔ Start angle **R** ➔ Subcircle radius **N** ➔ Number of drill holes equal to division factor for angle increment calculation **Positioning** shape between the drill holes

MCALL	Deselection of the modal cycle call from the machining cycle
TRAFOOF	The TRANSMIT transformation function is deactivated again.
SETMS(1)	Once you have activated the spindle on the turret, the main spindle (turning spindle) is to be switched back to. This spindle is configured as spindle 1 in SinuTrain for standard SIEMENS machines. For this reason, this spindle must be activated with the extended language word SETMS(X) (X = spindle number). This means you may have to make an adaptation if you want to apply the example to your machining, e.g. SETMS(4).

Current program excerpt:

```
T="DRILL_5"
SETMS(3)
G95 S2200  F0.1 M3
TRANSMIT
MCALL CYCLE82(10,0,1,,10,0.2,10,1,11)
LKD32: HOLES2(0,0,16,0,30,4,1000,0,,,1)
MCALL
TRAFOOF
SETMS(1)
WWP
M30
```

5.2.5 Simulating the turning program

Simu-lation Further views Half cut view		Simulation of the machining operations in the side view with traversing paths. Red ➔ Rapid traverse motions Green ➔ Feed motions.
Simu-lation 3D view Details Cut		3D view in the Cut display.

5.2.6 "Complete" solution program

```
G18 G40 G71 G90
G54
LIMS=3500
WORKPIECE(,,,"CYLINDER",0,1,-101,-92,90)
;=================================================
;Example by Turning made easy with G-Code programGUIDE
;Example 2 : Guide shaft
;created with SinuTrain OPERATE V4.7
;=================================================
R1=250 ;WWP-Position X
R2=250 ;WWP-Position Z
;=================================================
WWP
T="ROUGHING_T80 A"
```

```
G96 S180 F0.2 M4
;Face turning
G0 X92 Z0
G1 X-1.6
G0 Z2
G0 X92
;Longitudinal stock removal against contour WITHOUT relief cut
CYCLE62("AK1",1,,)
CYCLE952("SCHRUPPEN_AK1",,"RESTMAT_AK1",2101311,0.3,0,0,2.5,0.1,0.1,0.5,0.1,0.1,0,1
,0,0,,,,,2,2,,,0,1,,0,12,1110010,1,0,0.1)
WWP
T="FINISHING_T35 A"
```

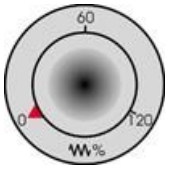
```
G96 S180 F0.2 M4
;Residual material machining (relief cut)
CYCLE952("RESTMAT","RESTMAT_AK1","",1101311,0.15,0.1,0,1,0.1,0.1,0.5,0.1,0.1,0,1,0,,,,,,
2,2,,,0,1,,0,112,1100010,1,0)
G96 S200 F0.2 M4
;Finish outer contour
CYCLE952("SCHLICHTEN_AK1",,"",1101321,0.2,0,0,2.5,0.1,0.1,0.5,0.1,0.1,0,1,0,0,,,,,2,2,,,0,1,
,0,12,1100010,1,0,0.1)
WWP
T="SOLIDDRILL_D16"
```

```
G95 S1200 M3 M8
G0 X0 Z2
G1 Z-102 F0.1
G0 Z2
WWP
```

```
T="DRILL_5"
SETMS(3)
G95 S2200  F0.1 M3
;Drilling pattern face-end hole circle D30 4xD5
TRANSMIT
MCALL CYCLE82(10,0,1,,10,0.2,10,1,11)
LKD32: HOLES2(0,0,16,0,30,4,1000,0,,,1)
MCALL
TRAFOOF
SETMS(1)
G97 S500
WWP
M30
;---------------------------------------------
;Local subprograms
;Outer contour
E_LAB_A_AK1: ;#SM Z:3
;#7__DlgK contour definition begin - Don't change!;*GP*;*RO*;*HD*
G18 G90 DIAM90;*GP*
G0 Z0 X42 ;*GP*
G1 X48 CHR=3 ;*GP*
Z-18.477 RND=4 ;*GP*
G2 Z-55.712 X60 K=AC(-35) I=AC(80) RND=4 ;*GP*
G1 Z-75 RND=4 ;*GP*
Z-80 X90 RND=4 ;*GP*
Z-82.883 ;*GP*
;CON,V64,2,0.0000,5,5,MST:3,2,AX:Z,X,K,I,TRANS:0;*GP*;*RO*;*HD*
;S,EX:0,EY:48,ASE:90;*GP*;*RO*;*HD*
;F,LFASE:3;*GP*;*RO*;*HD*
;LL,DIA:225/0,ASE:180;*GP*;*RO*;*HD*
;R,RROUND:4;*GP*;*RO*;*HD*
;ACW,DIA:210/0,EY:60,CX:-35,CY:80,RAD:23;*GP*;*RO*;*HD*
;R,RROUND:4;*GP*;*RO*;*HD*
;LL,EX:-75;*GP*;*RO*;*HD*
;R,RROUND:4;*GP*;*RO*;*HD*
;LA,EX:-80,EY:90;*GP*;*RO*;*HD*
;R,RROUND:4,ASE:180;*GP*;*RO*;*HD*
;#End contour definition end - Don't change!;*GP*;*RO*;*HD*
E_LAB_E_AK1:
```

6. Tips and key combinations

6.1 Tips for execution

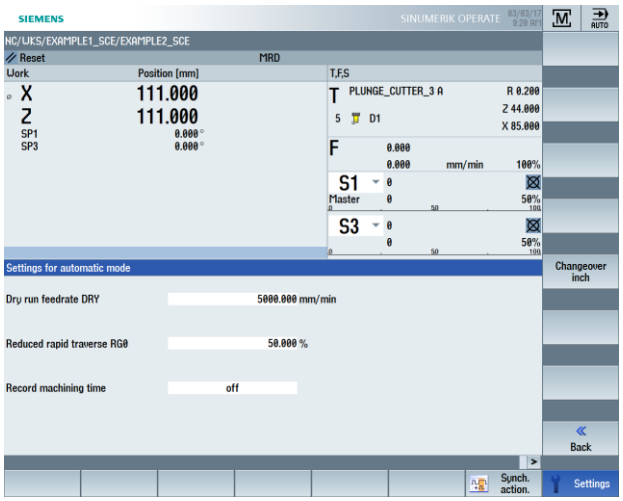


Due to the fact that the machining plan has not yet been executed with control, turn the feedrate potentiometer to zero position to ensure that you keep everything under control from the beginning.

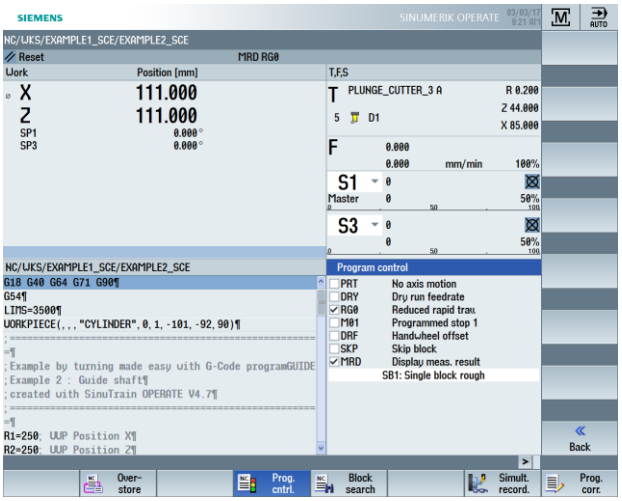
If you also want to see a simulation during the machining, select the "Simult. record." softkey before starting. Then all traversing paths and their effects are also displayed.

Start the machining and control the speed of the tool motions using the feedrate potentiometer.

To move in at reduced rapid traverse RG0, the percentage of the maximum rapid traverse of the machine, in the second level of the horizontal softkey bar, must be set under Settings,
e.g. 50%.



Once the percentage is set, the reduced rapid traverse must also be selected under "Program control" with the toggle key.



Keyboard shortcuts

OPERATE keyboard shortcuts

Control keys:	
CTRL + P	For screenshots – Storage location: Commissioning (password) – System data – HMI data – Logs – Screenshots
CTRL + L	Language switchover
CTRL + C	Copy
CTRL + X	Cut
CTRL + V	Paste
CTRL + Y	Redo (editor functionality)
CTRL + Z	Undo - max. 5 lines in the editor (editor functionality)
CTRL + A	Select All (editor function)
CTRL + NEXT WINDOW	Go to start of program
CTRL + END	Go to end of program
CTRL + ALT + S	Save complete archive – NCK/PLC/Drives/HMI
CTRL + ALT + D	Backup log files on USB or CF card
CTRL + E	Control energy

Keyboard shortcuts, continued

CTRL + M	Maximum simulation speed
CTRL + F	Search in all screen forms. Wildcards "?" and "*" can be used in search screen forms. "?" stands for any character, "*" for any number of any characters.
Miscellaneous:	
Shift + INSERT	Comment out cycles and direct editing of programGuide cycles
Shift + END	Select up to end of block
Shift + NEXT WINDOW	Select up to start of line
ALT + NEXT WINDOW	Jump to start of line
ALT + S	Enter Asian characters
=	Calculator function
HELP	Help function
END	Jump to end of line

Keyboard shortcuts, continued

Simulation/simultaneous recording:	
	Move
	Rotate in 3D display
	Move section
	Override +/- (simulation)
	Single block on/off (simulation)
Insert key:	
	It brings you into Edit mode for text boxes or into Selection mode of combo boxes and toggle fields. You can exit this without making any changes by pressing "Insert" again.
	Undo function, as long as no Input key is pressed or no data has already been applied in the fields.
Toggle key:	
	You can directly switch between toggle fields using the Toggle key (Select) without having to open them. With Shift-Toggle you can switch through these in the reverse direction.
Cursor key:	
	Open/close directory Open/close program Open/close cycle