Manual 08/2007 Edition

Tool and Mold Making

SIEMENS

SIEMENS

SINUMERIK
Tool and Mold Making

Manual

Valid for:

Control system

SINUMERIK 810D
SINUMERIK 840D
SINUMERIK 840Di
SINUMERIK 802D sl pro

08/2007 Edition

Doc.Ord.No. 6FC5095-0AB20-0BP0

SINUMERIK® Documentation

Printing history, registered trademarks

Brief details of this edition and previous editions are listed below.
The status of each edition is shown by the code in the "Remarks" columns.

Status code in the "Remarks" column:

A .... New documentation.
B .... Unmodified reprint with new order number
C .... Revised version with new edition status

If factual changes have been made on the page since the last edition, this is indicated by the new version code in the header on that page.

Edition	Order No.	Remarks
08.2007	6FC5095-0AB20-0BP0	A

Registered Trademarks

SIMATIC, SIMATIC HMI, SIMATIC NET, SIROTEC, SINUMERIK, SIMODRIVE and SINAMICS are registered trademarks of Siemens AG. Other names in this publication might be trademarks whose use by a third party for his own purposes may violate the rights of the registered holder.

More information is available on the Internet at:
http://www.siemens.com/sinumerik

This document was created using several layout and graphic tools.

Other functions not described in this documentation might be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

We have checked that the contents of this document correspond to the hardware and software described. Nevertheless, we cannot assume responsibility for any deviations that may arise. The information contained in this document is, however, reviewed regularly and any necessary changes will be included in the next edition. Suggestions for improvement are welcome.

Technical data subject to change.

Order No.6FC5095-0AB20-0BP0

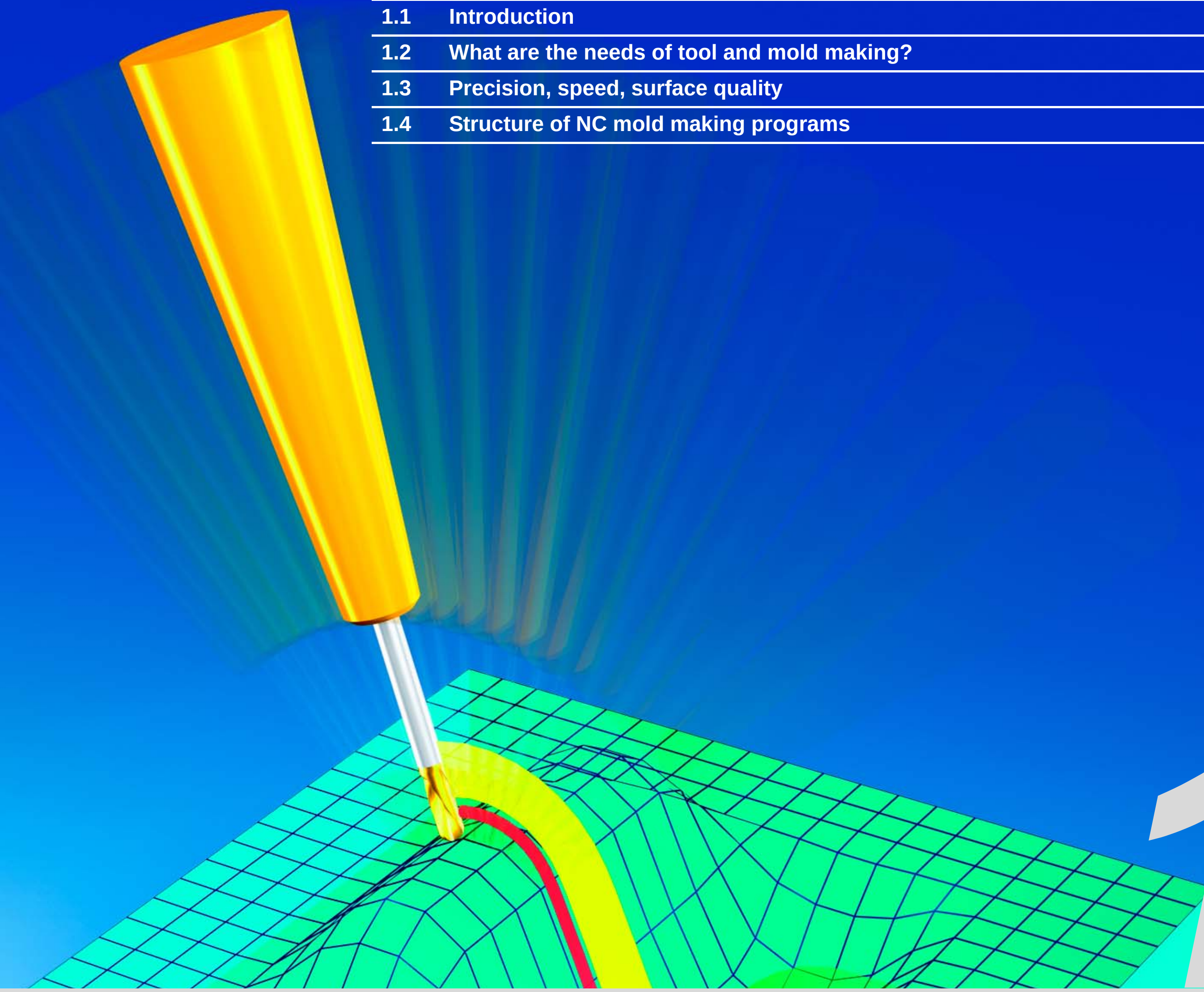
Siemens Aktiengesellschaft

Page

Page

Basic information

Contents



1

1.1 Introduction

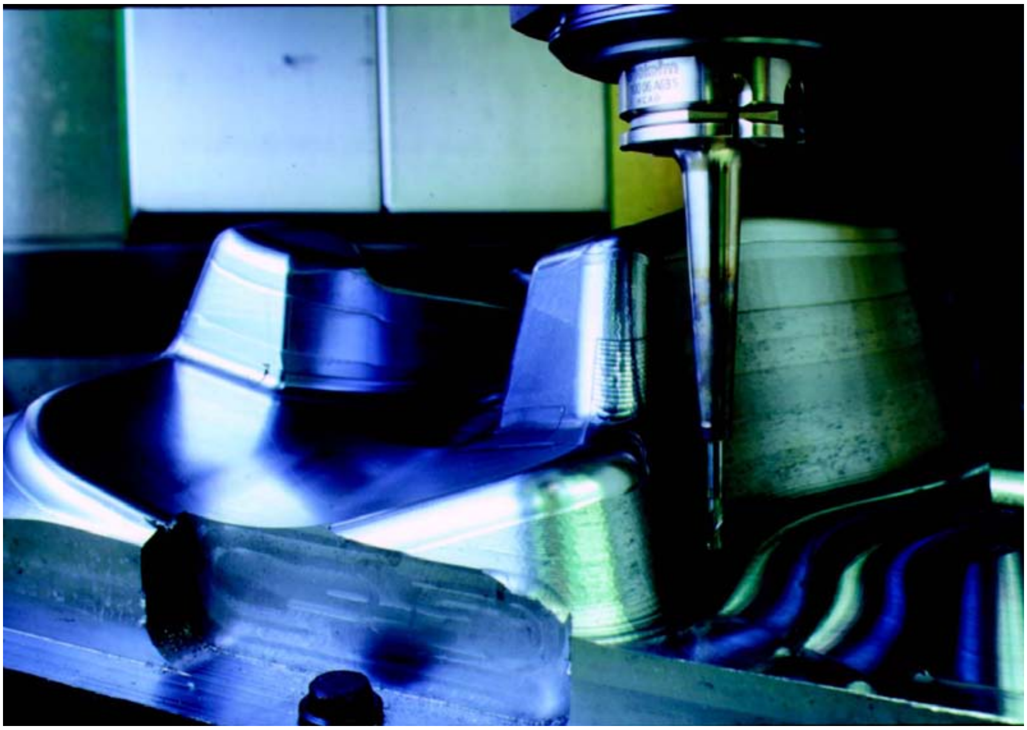
Speed, precision and perfect surface quality without the need for time-consuming re-machining are the main aims in tool and mold making.

SINUMERIK 840D has powerful, highly-developed functions which when intelligently used significantly simplify the complete operation involving 3-axis programming and machining - and at the same time the production result is improved. The SINUMERIK 802D sl is available for standard 3-axis machining.
In a compact form, the brochure offers specialists from industry and research an entry into the most important basics of tool and mold making, especially 3-axis milling. Based on this it provides users and operators at the machine practical information so that they can work efficiently.

This is followed by an explanation of the essential functions of the SINUMERIK control for programmers and the potential for optimizing the overall process is clearly shown using specific examples.

Many aspects will only be able to be briefly discussed in this Manual. You will find additional information in the appropriate Manuals and relevant literature.

Automotive trim



1.2 What are the needs of tool and mold making?

Prototype construction

Design standards in all application areas are becoming increasingly more demanding. More and more - ergonomics, air drag coefficient (CW value) or just simply aesthetics call for rounded forms and shapes. And all of this in shorter times and with a higher degree of precision The design primarily comes from CAD systems, the machining programs for freeform surfaces from CAM stations.

Nevertheless, the skilled machine tool operator still has the technological responsibility for the quality of the mold and the complete tool.

Chess piece

With SINUMERIK 840D and 802D sl Siemens is offering controls that are precisely tailored to the demands of tool and mold making - in the classic 2 ½ D sector, for 3-axis machining and also with 840D, in the 5-axis and high-speed area:

- simple to operate
- user-friendly programming at the machine
- optimum performance in the overall CAD - CAM - CNC process

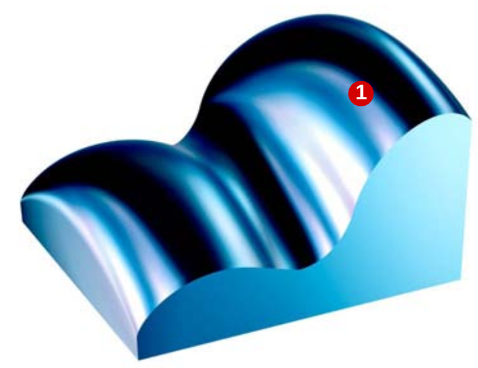
Valve

1.3 Precision, speed, surface quality

Process chain: CAD -> CAM -> CNC

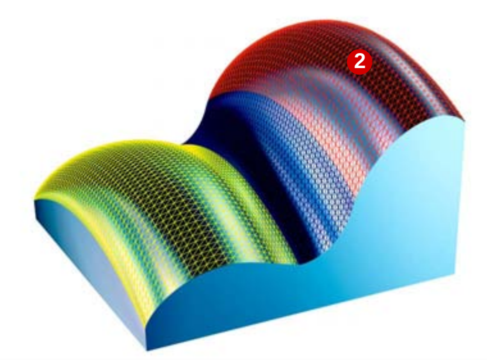
CAD -> CAM CAM system generate NC programs for machining freeforms. The CAM system receives the workpiece geometry from a CAD system.

CAM -> CNC The process chain **CAD -> CAM -> (post processor) -> CNC** must be given considerable thought when it comes to machining freeform surfaces.



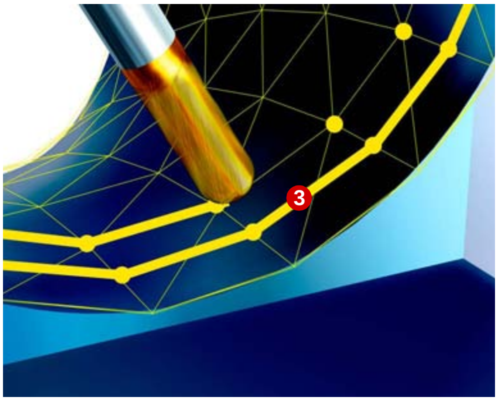
In CAD systems, surfaces 1 with higher orders are constructed (freeform).

For example, in order to be able to mill an entire surface - or for collision checking - the CAM system generally converts the CAD surfaces into a polyhedron.



This means that the smooth design surface is approximated using many tiny planes 2 .

This inevitably produces some minor deviations with respect to the original freeform.

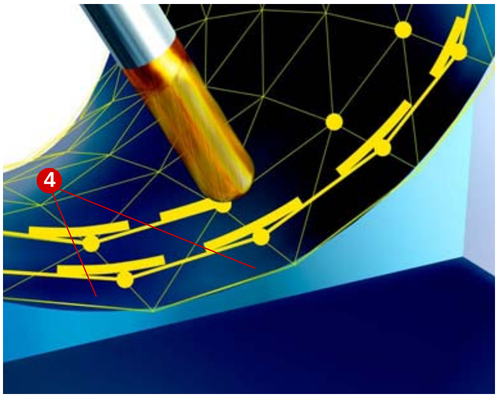


The CAM programmer overlays this polyhedron with tool paths. From these, the post processor generates NC blocks within the specified error tolerances. These usually comprise many straight line elements, G1 X Y Z 3.

This is the reason that the machining result is no longer a freeform surface - but a polyhedron. The small plans of the polyhedron can be visibly mapped on the surface.

This can result in undesirable reworking.

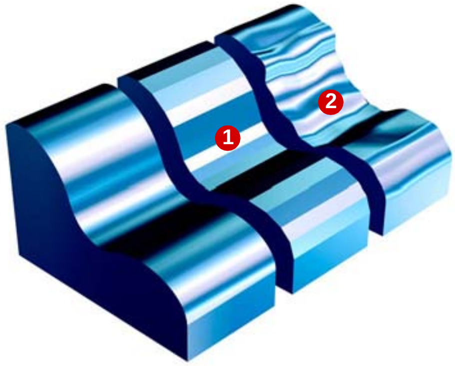
SINUMERIK controls offer various functions to avoid having to rework:



Programmable blending (spline interpolation)

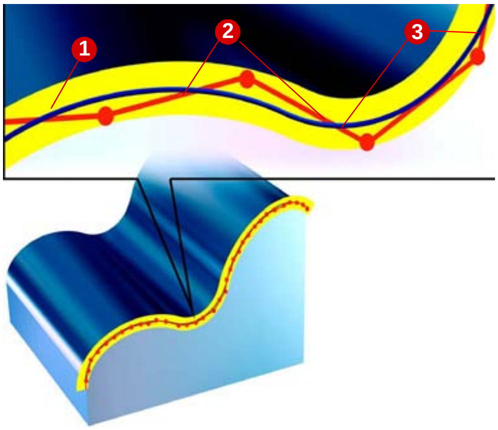
One is defined blending at block boundaries. This involves inserting geometrical elements 4 at the corners (block transitions). The tolerance of these geometrical elements can be adjusted.

Also refer to Chapter 3.5 High Speed Settings.



Compressor function (COMPCAD)

The linear interpolation at the block transitions leads to acceleration jumps in the machine axes, which in turn can cause resonance in the machine elements and can ultimately be detected as a beveled pattern 1 or as vibration 2 on the workpiece surface.



Corresponding to the specified tolerance band 1 the compressor combines a sequence of G1-commands 2 and compresses them to form a spline 3, which is directly executable by the control system.

The surface quality is increased since the machine axes can move more harmoniously and machine resonance is avoided.

As a consequence, this allows more constant traversing velocities that reduce the load on the machine and increase the productivity.

Prerequisites

- The options COMPCAD and spline interpolation must be available and set and the machine must have been set-up for these functions to be used.

NOTE

If the tolerance band for the CAM system is known, this or a slightly higher value should be used for the compressor tolerance.

For COMPCAD this value typically lies between 1.2 ....1.5 of the programmed chord tolerance of the CAM system. If this value is not known, the default setting for CYCLE832 should be used as a starting value. You will find the default settings in Chapter 2.9 High Speed Settings.

With SINUMERIK 840D you can switch-in or switch-out spline compression and COMPCAD using cycle CYCLE832. Please refer to the information about this cycle in Chapter 2.9 and Chapter 3.5.

1.4 Structure of NC mold making programs

A NC program to machine freeform surfaces comprises many NC blocks and is generally not edited by the CNC control.

Structure of an NC mold making program

The most clearly structured NC program is one in which the CAM programmer applies the following program structure.

Example

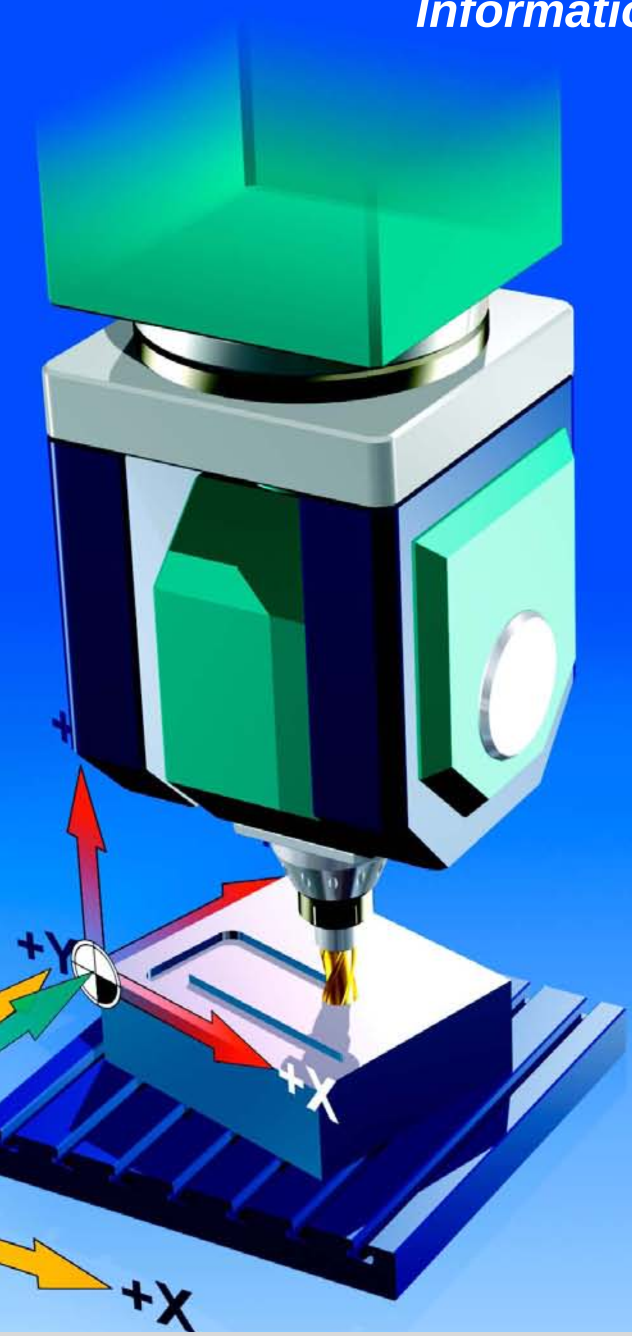
Main program with subroutine call

Tool call	**N10 T1D1** **N15 M6**	
Technology	**N20 M3 M8 S8000 F1000**	; spindle speed, feedrate
Zero point Start position	**N30 G0 G54 X10 Y10 Z5**	; Settable zero offset
High Speed Setting cycle	**N40 CYCLE832(0.05,112003)**	; CYCLE832 sets the ; compressor tolerance and defines ; other path conditions.
Subroutine call	**N50 EXTCALL "Roughing"**	; Calls the "Roughing" subroutine, ; that contains the geometry of the CAM ; program.

The zero offset, all technology values, start point and high-speed settings are defined in the main program. The high-speed setting parameters can be used to influence the quality of the workpiece.
The subroutines contain the typical traversing blocks, which owing to the complexity of the programs should not be changed.

A well-structured NC program can also be resumed from a selected point after a program interruption.

Information for Machine Operators



Contents

2

2.1 Introduction - setting-up/measuring the workpiece and measuring the tool

Setting-up the workpiece and tool in JOG

When setting-up the machine is prepared for machining. This means that the dimensions of the workpiece and the tool - that are still unknown, must be measured.

The reference point for programming a workpiece is always the workpiece zero. When setting-up a clamped workpiece its workpiece zero is determined. The workpiece elements - edge, corner, pocket/hole, lug, plane - can be used when setting-up. When completed, the workpiece zero is defined as the result of the linear and rotary offsets of the coordinate system that have been determined.

Depending on the control, the tool length and diameter can be automatically determined using a mechanical test socket or by scratching at the known workpiece geometry. The values that have been determined are transferred into the tool offset data.

Measuring the workpiece - in-process measurements

The workpiece is measured to determine workpiece tolerances in the production process. Depending on the measuring cycle used, you can select the following options as result of the workpiece measurement:

- Only measurement without offsets (actual value is measured)
- Zero offset correction (setpoint - actual value deviation)
- Tool data offset (setpoint - actual value deviation)

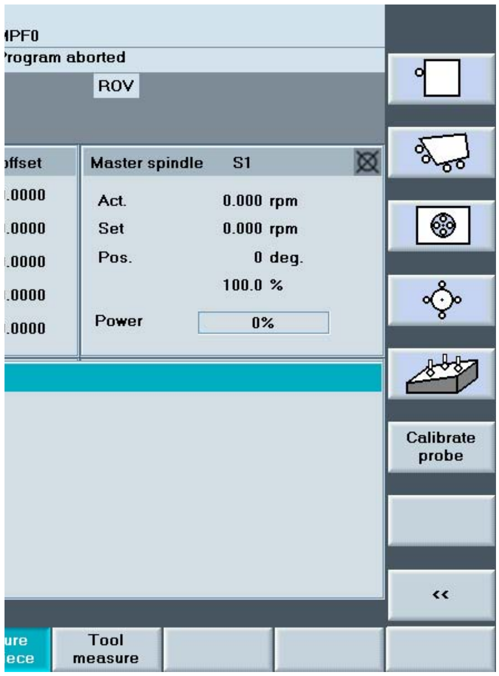
These measurements can be performed using switching or non-switching probes.

You should use switching 3D probes in order to use the full functionality of the measuring cycles.

Measuring the tool - in-process measurement

The tool is measured to monitor specific tool parameters in the production process. The tool parameters are corrected in the result of the tool measurement, i.e. generally the diameter and length of the tool are determined.

Measuring in JOG - manual measurement

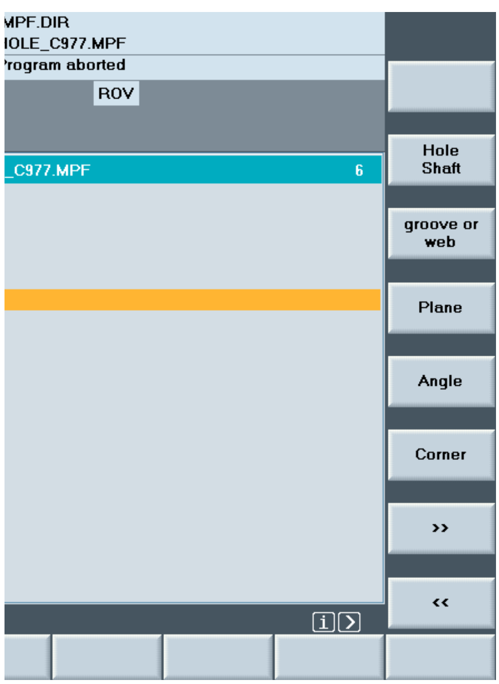


Using the semi-automatic "Measuring in JOG" the required measuring function is selected at the control using the appropriate softkeys. The displayed input screens are used to parameterize the function. You must bring the tool or probe into a permissible starting position for the measurement task, e.g. using the traverse keys or handwheel (manual traverse).

You measure in JOG for the following tasks:

- Manual measurement is used to prepare the machine for machining.
- Manual measurement is used to determine unknown workpiece or tool geometries.
- The measurement is carried-out as interactive operator control with the machine in the manual mode.

Measuring in the automatic mode - in-process measurement



When performing automatic measurements in the automatic mode, NC programs (measuring cycles) are parameterized for the specific measurement task. The input screens of the program editor are used for parameterization. The measuring points to be approached and the measuring task are automatically implemented corresponding to the measuring program.

You perform measurements in the AUTOMATIC mode for the following tasks:

- Automatic measurements are performed to check that the workpiece measurements are to specifications
- Automatic measurements are performed to correct known workpiece and tool geometries.
- The measurement is performed by calling a measuring cycle in the machining program.

2.2 JOG mode - setting-up and measuring workpieces

Setting-up the workpiece

After the machine has been powered-up and the reference point approached the axis positions are referred to the machine coordinate system. The zero offset signals to the control the position of the workpiece in the machine coordinate system.

Previously, the workpiece was clamped, manually aligned in parallel to the machine axes, and then the zero offset was determined e.g. by scratching. By looking at two examples that commonly occur in practice, we can see how much easier it is with probes and SINUMERIK cycles. We will show how the control system compensates the basic rotation of the workpiece, eliminating the need for time-consuming, manual alignment.

As an example, setting-up will be shown at two machine configurations.

- Machine without rotary axis in the table (refer to Chapter 2.3)
- Machine with a rotary axis in the table (refer to Chapter 2.4)

Prerequisites

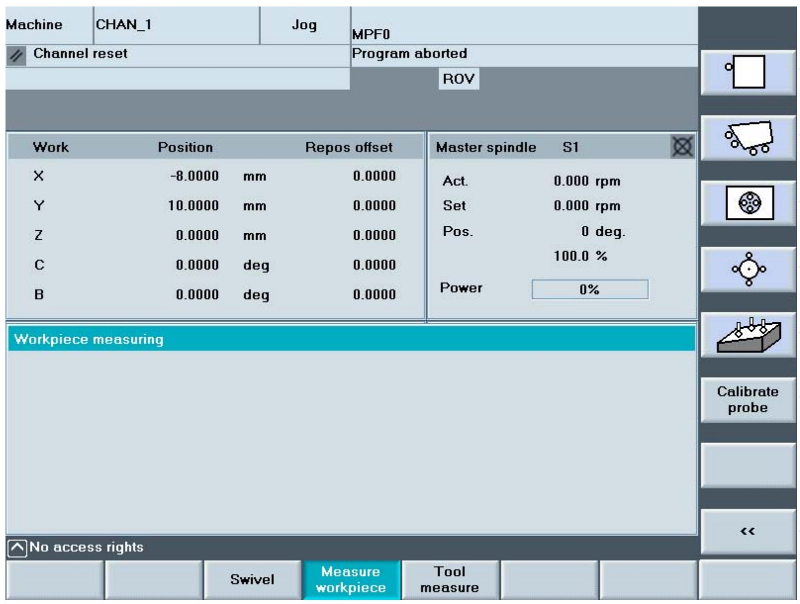
- Measuring cycles have been installed
- Workpiece has been clamped
- Probe has been calibrated, is active and clamped in the spindle; tool offset is activated.

NOTE

If only one single workpiece is to be machined - which is generally the case in tool and mold making - then measurements are performed in the JOG mode (as described below). If several similar parts are to be machined in the same equipment, the measuring cycles are used in the automatic mode (the approximate zero point must be set up).

Selecting measuring cycles for SINUMERIK 840D

Measuring cycles in-line with those required in practice are available for making measurements



→ Measure an edge

→ Measure a corner

→ Measure a pocket/hole

→ Measure a spigot/hole

→ Align a plane

→ Calibrate a probe

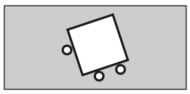
→ Back (exiting measuring in JOG)

2.3 Measuring in JOG - setting-up a workpiece without rotary axis in the table

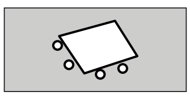
Task

After clamping, there is a non right-angled workpiece rotated in the working area with respect to the machine coordinate system. You must determine the zero offset and the position of the coordinate system - i.e. the basic rotation.

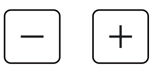
Measuring a corner at the machining plane



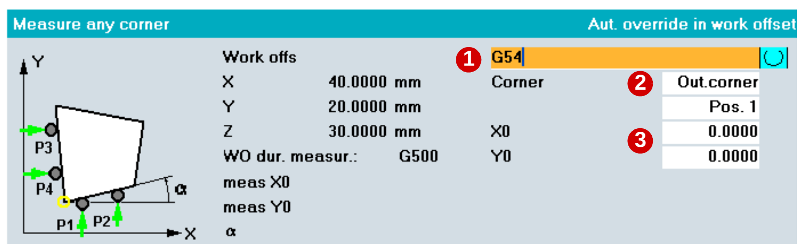
Call "Measure corner". The "Corner" window opens with the new vertical softkeys "Right angled corner" and "Any corner".



Press the "Any corner" softkey, if you want to measure a corner not equal to 90°. 4 measuring points are required: P1, P2, P3 and P4



Traverse the probe to the first measuring point P1 as shown in the help display.



Enter details in the input screen:

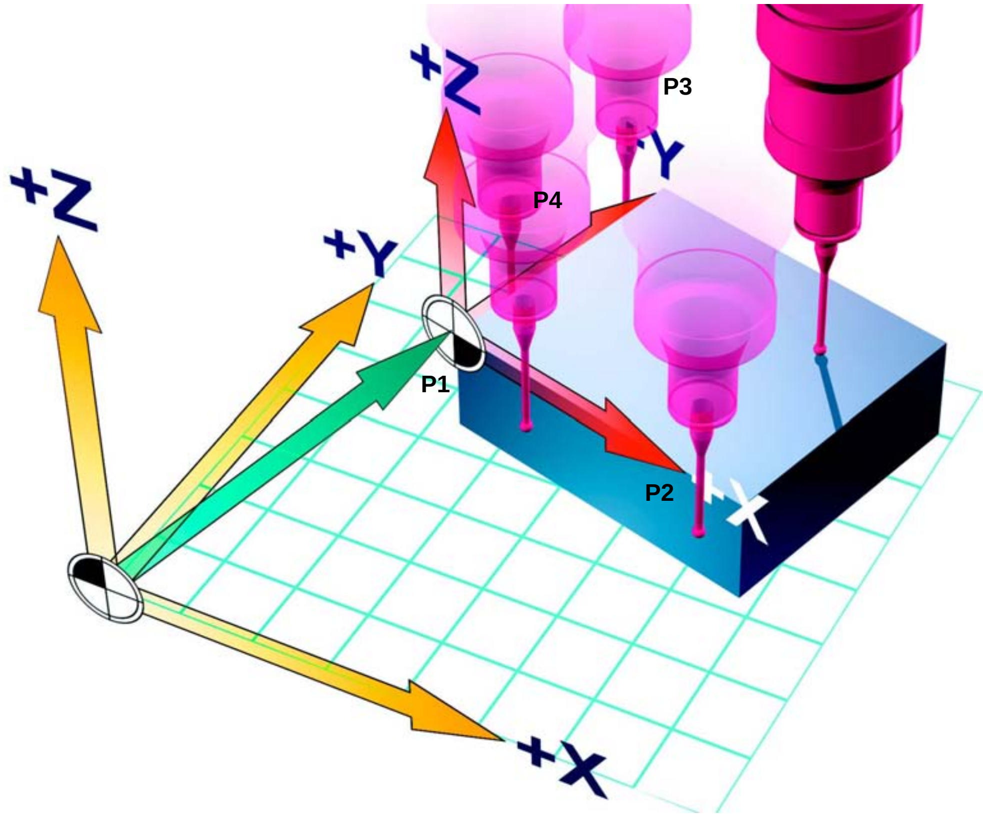
1. Select the zero offset, e.g. G54, G55, G56 or G57. G54 was used in the example.
2. Select either inner or outer corner
   Select the position of the corner
3. Enter the set position required for the reference point (corner) for the selected ZO in both axes.

When measuring a corner in the G17 plane - in the translatory zero offset in X and Y and the rotation around Z are determined in the result. The translatory zero offset in Z should be determined using an additional measurement "Set edge".

With "NC-Start" the particular measuring point (P1 - P4) is automatically approached starting from the preliminary position that was manually selected. This means that the probe approaches the workpiece, is triggered and then returns to the starting position.

NOTE

When performing measurements you can select as to whether the result of the measurement is entered as correction in the zero offset or whether just a measurement is made (refer to 1). When setting-up a correction is made in the zero offset. In order to simply just check the dimension specifications of a corner, you can also use the measuring cycles for "Only measure".



Result

With "NC-Start" and the pre-defined measuring distance, the measurement is performed automatically at P1 with the set measuring feedrate. After the measurement has been successfully performed, the previously inactive softkey "P1 stored" is activated and the coordinates of the 1st measuring point P1 are stored internally.

After manual positioning in front of the 2nd measuring point P2, measuring is performed automatically at this measuring point by pressing "NC start". Proceed in the same way for measuring points P3 and P4. Once all of the measuring points have been successfully completed and all "Px stored" softkeys activated, then a vertical "calculate" softkey is displayed. After this softkey has been actuated, the corner coordinates P0 and the offset are calculated.

Calculating

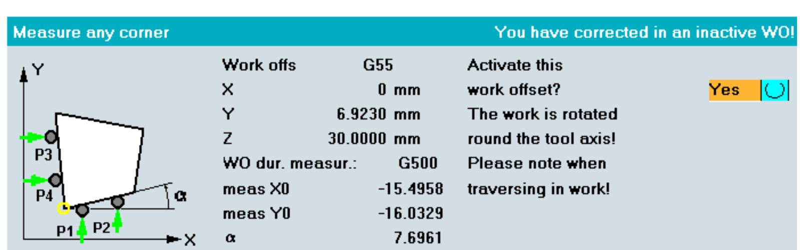
The control calculates

1. the X, Y values of the zero offset from the intersection of the two straight lines,
2. the basic rotation of the workpiece coordinate system around the Z axis.
3. The values are transferred into the zero offset table, zero offset G54.

An offset in the XY plane and a basic rotation around "Z" were determined as result.

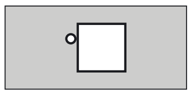
The offset values become immediately effective if the already active ZO was selected as correction goal.

If another ZO is selected the system issues a prompt as to whether this should be activated.

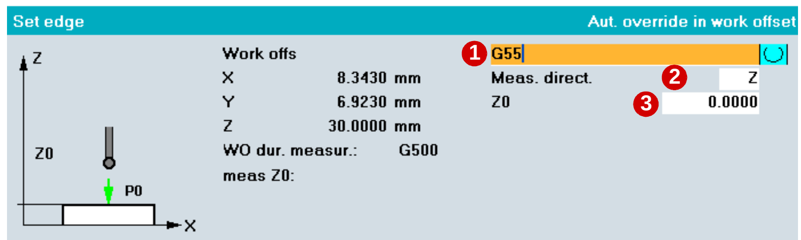


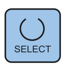
Determining the zero offset in the Z axis

After you have aligned/measured the X, Y planes - you must now measure the zero offset in the Z axis.



Select "Set edge" and move the probe to the first measuring point P1.





Enter details in the input screen:

1. Select a zero offset, e.g. G54.
2. Select the Z axis
3. Enter the set position required for the reference point (edge) for the selected ZO.

With "NC-Start" measuring point P0 is automatically approached starting from the manually selected preliminary position. This means that the probe moves to the workpiece, is triggered and then retracts to the starting position.

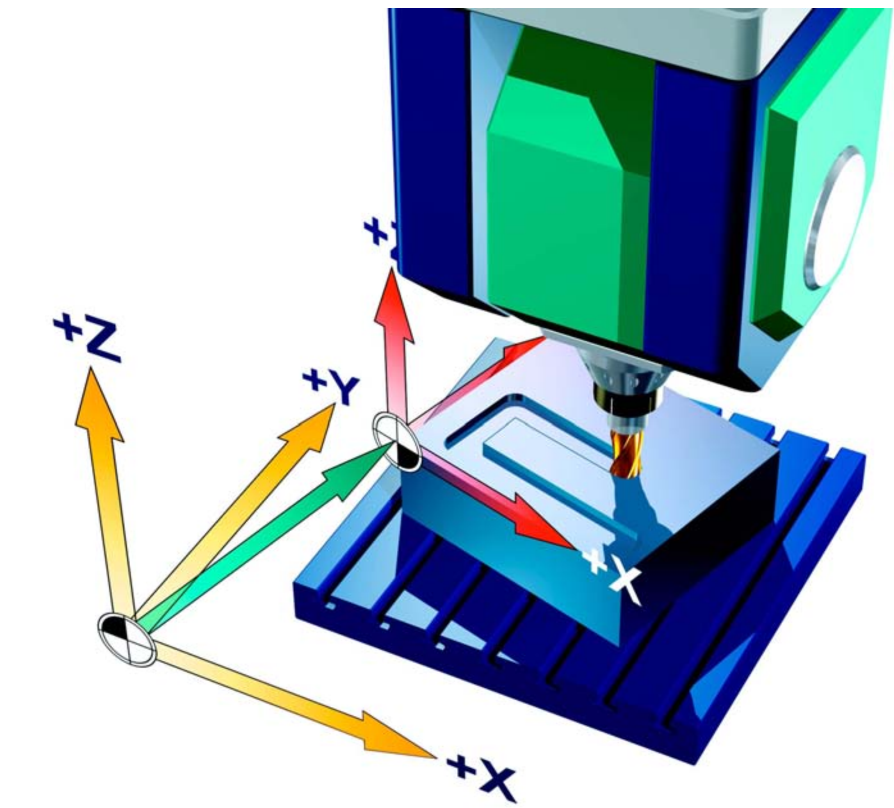
The basic rotation and the zero point in axes X/Y and Z for a 3-axis machine without rotary axis in the table have now been determined.

Summary for machine configurations without rotary axis

The zero point and the basic rotation (coordinate rotation for rotary axes) for the machine have now been determined.

If coordinates are rotated for machines without rotary axes, then the control converts the programmed motion of the X/Y axes - in parallel to the axis - into the corresponding resulting XY motion. This means that tool motion is no longer parallel to the machine axes.

Example 1 **Machine kinematics "without" a C axis in the table**

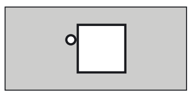


2.4 Measuring in JOG - setting-up a workpiece with a rotary axis in the table

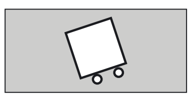
Task description

Example 2 After clamping, a non right-angled workpiece is rotated in the working area with respect to the machine coordinate system. You must determine the zero offset and the position of the coordinate system - i.e. the basic rotation. The machine has one rotary axis

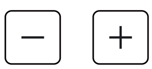
Determining the basic rotation around the Z axis



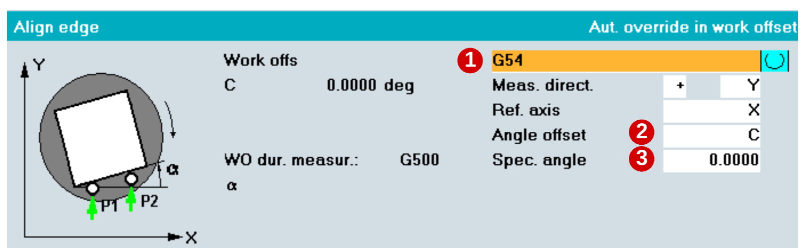
Call "Measure edge". The "Measure edge" window opens the vertical softkeys "Set edge", "Align edge" and "Distance between two edges".



Actuate the "Align edge" softkey. Two measuring points are required!



Traverse the probe to the first measuring point P1 - as shown in the help display.

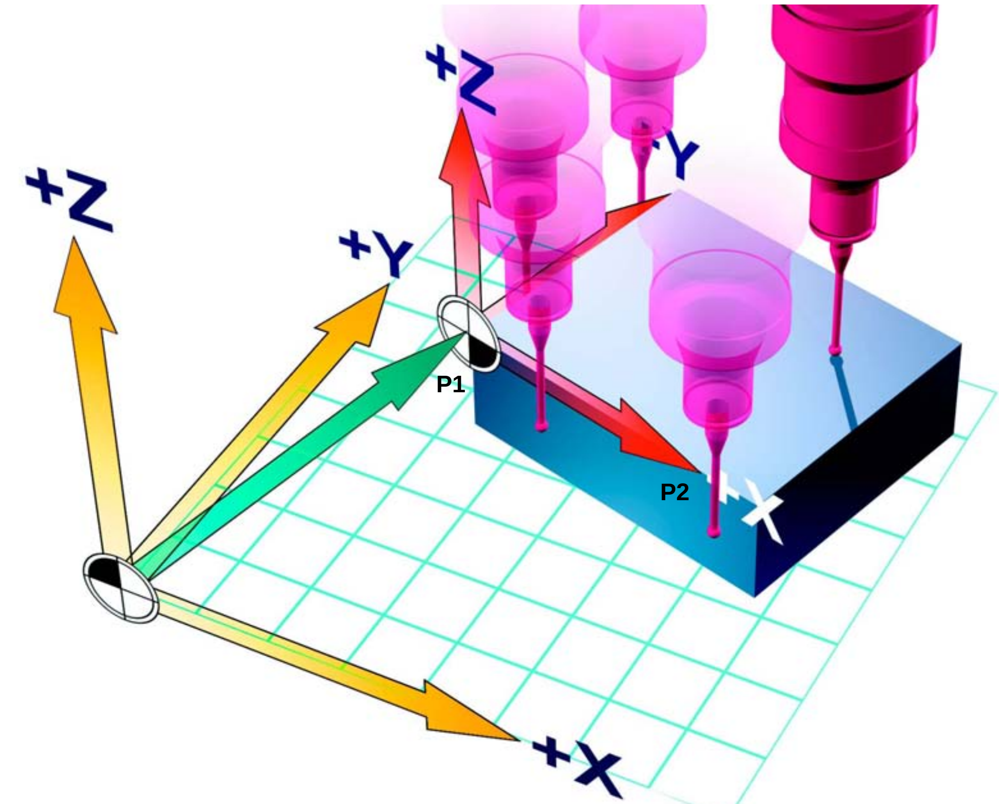


Enter details in the input screen:

1. Select the zero offset, e.g. G54, G55, G56 or G57. G54 was used in the example.
2. Select the type of angular correction, table rotary axis C in the example.
3. Enter a possible setpoint angle.

With "Align edge" the Alpha angle is determined in the G17 plane

With "NC-Start" the measuring points P1 and P2 are automatically approached starting from the manually selected preliminary position. This means that the probe moves to the workpiece, is triggered and then retracts to the starting position.

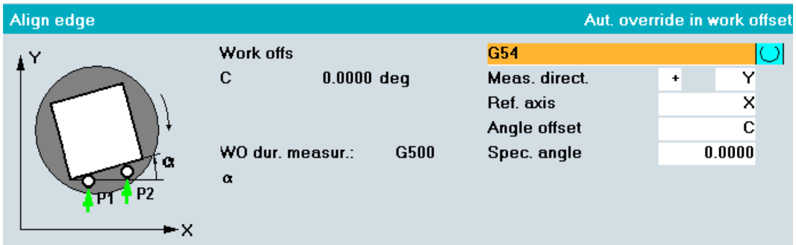


.

Result

Calculating

If the measuring points have been successfully completed and all "Px stored" softkeys activated, then the vertical "Calculate" softkey is displayed. The "Alpha" angle is calculated after this softkey has been actuated.

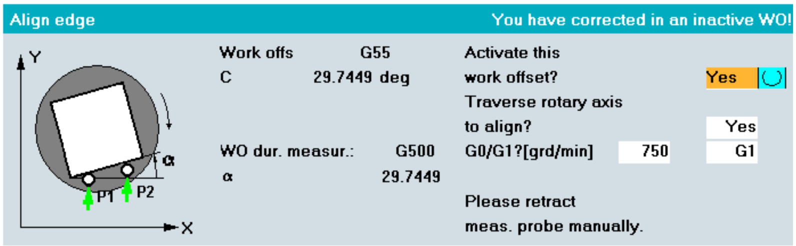


In the result this rotation is entered as zero offset in the rotary axis of the table.
The correction values immediately become effective if the already active ZO is selected as correction goal.

Otherwise a prompt is output as to whether the zero offset should be activated. As an additional step a prompt is issued as to whether the rotary axis - and therefore the workpiece - should be aligned.

NOTE

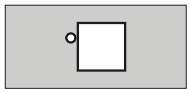
Caution! Before you allow the workpiece to be aligned, you should retract the tool so that no collisions occur on the table if rotary motion is executed.



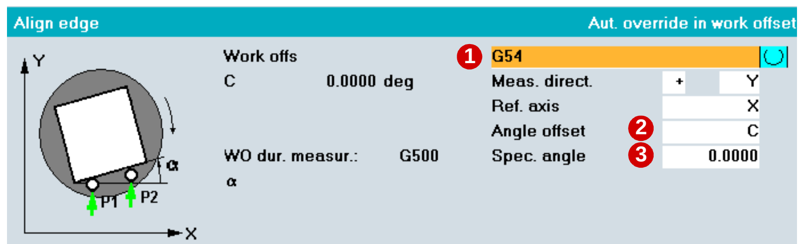
The workpiece is now aligned parallel to the axis.

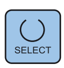
Determining the zero offset in the X/Y axis

After the rotation was set-up you must now determine the ZO in the X/Y plane.



Select "Set edge" and move the probe to the first measuring point P1.





Enter details in the input screen:

1. Select a zero offset, e.g. G54.
2. Select the X or Y axis.
3. Enter the set position required for the reference point (edge) for the selected ZO.

With "NC-Start" the measuring point P1 is automatically approached starting from the manually selected preliminary position. This means that the probe moves to the workpiece, is triggered and then retracts to the starting position.

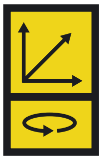
This operation must be executed separately for the X and Y axes. The zero offset in the Z axis should be determined as described in example 1 in Chapter 2.3.

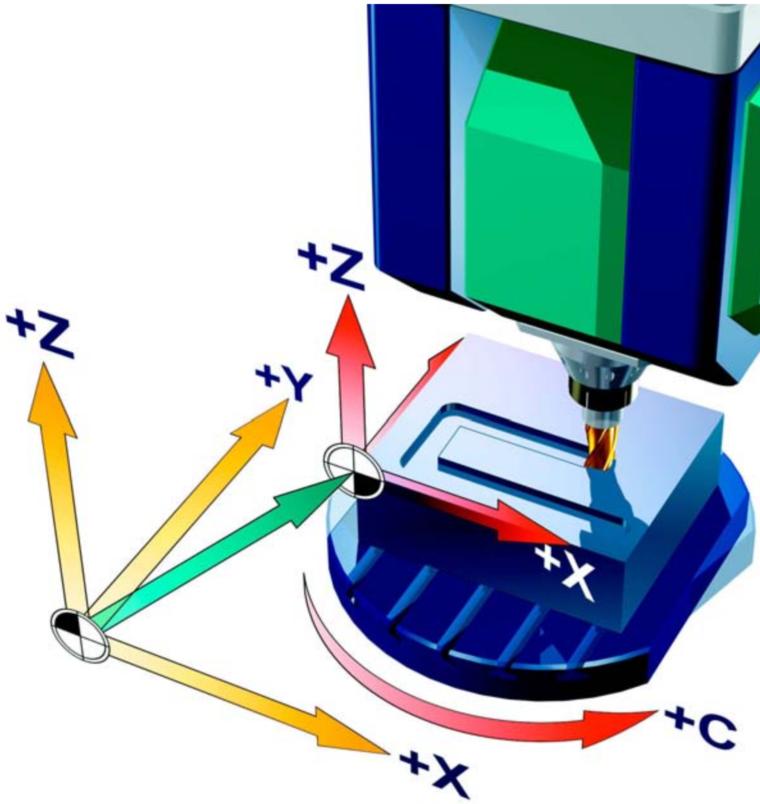
Summary for machine configurations with one rotary axis

Machine with C axis in the table

The table was rotated. Milling paths that run in parallel to the workpiece edges also run in parallel to the axis of the machine coordinate system. When programming the X axis, the machine axis also traverses in X.

Example 2 **Machine kinematics "with" C axis in the table**

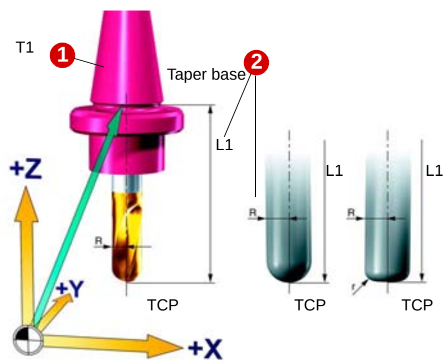




2.5 Measuring tools in JOG

The various tool geometries must be taken into consideration while the program is being executed. These are saved as tool offset data in the tool list.
When calling the tool, the control takes into consideration the tool offset data.
You can determine the tool offset data, i.e. length and radius or diameter, either manually or automatically using a mechanical test socket (cycles for the automatic mode) or semi-automatically in the JOG mode.

Tool reference point



TCP = Tool Center Point

The tool magazine is loaded as usual, tool numbers T1, T2 etc. 1 are entered into the tool table and the tools are assigned a tool offset D 2 consisting of radius "R" and length "L1".

The CAM system already takes into account the tool diameter when the geometry program is being created. The calculated tool path refers to the miller center point (center point path).

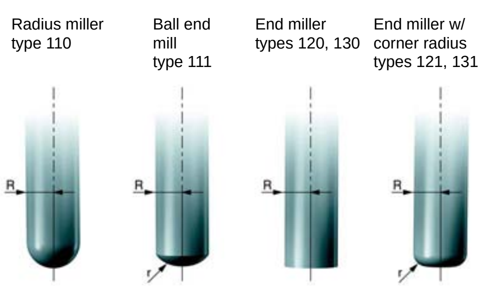
This means that to measure the length of your tool you must use the same reference point (TCP) as the CAM system. For the tool length, always check the reference point the CAM programmer used to measure L1. The TCP can either be located at the tool tip or further upwards in the miller - e.g. for radius milling tools at the center of the radius.

NOTE

CAM systems define the position of the TCP differently depending on the tool shape. For Siemens controls it is assumed that the TCP is at the tool tip. If the CAM system specifies a different TCP position then this difference must be taken into account when specifying the tool length.

NOTE

Coordinate the following with the CAM programmer: To avoid significant tool deflection the CAM programmer should keep the tool length as short as possible.



You can specify additional tool data for face milling depending on the tool type.

In a CNC program the control system uses this data and path corrections G41, G42 -defined in the program - to execute the necessary path and length corrections.

Manually entering tool offset data

General information

Tool offset data comprises data that describe the geometry, the wear, cutting edge number (D) and the tool type. The units used for the tool dimensions are displayed.

Task description

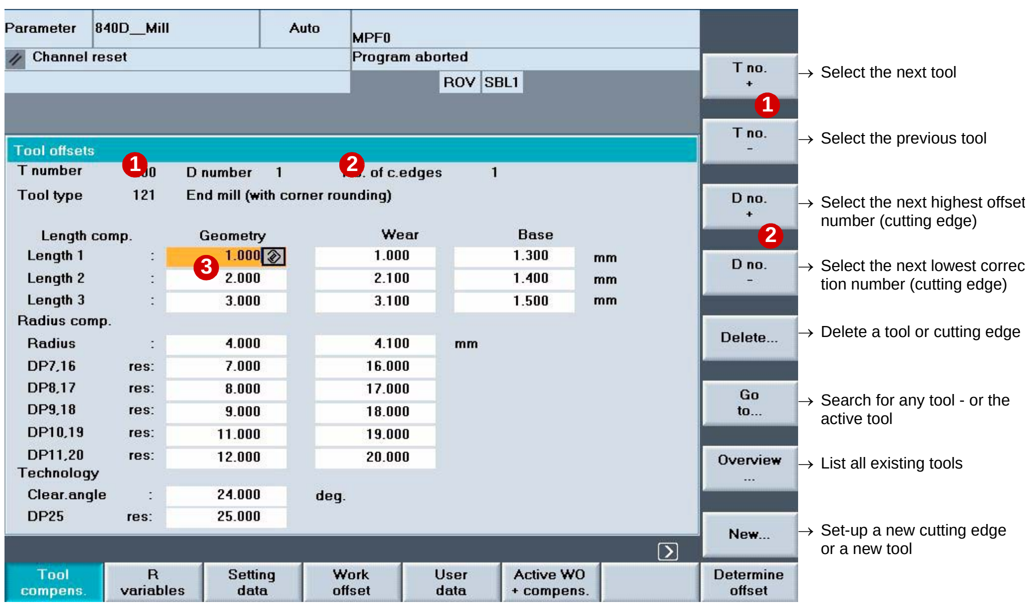
The tool offset data "length" and "radius" were externally determined using a tool pre-setting device and the tool was inserted in the tool magazine. The tool offset data are then entered.

Parameters

Select the "Parameters" operating area.

Tool offset

Select "Tool offset". The following window is displayed. The input field is marked.

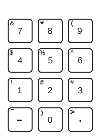


T No. +

Select a tool with "T No. + " or "T No. - ", or 1

D No. +

Select the offset data with "D No. + " or "D No. - ". 2



Enter new values. 3

Measuring a tool in JOG

Function

The "Tool measuring" function permits the following functions:

- Calibrating a mechanical test socket (calibrate)
- Determining the tool length or the radius of millers or the tool length of drills and entering these into the tool offset memory.

Prerequisites

- The measuring cycles have been installed
- The mechanical test socket has been calibrated and the tool has been clamped

Machine

Select the "Machine" operating area.

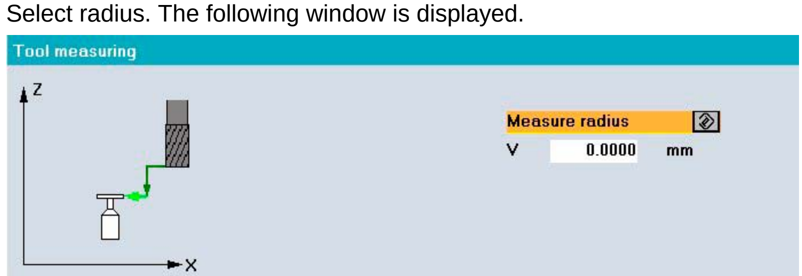
Select the "JOG" mode at the machine control panel.

Measuring tool

Call "Measure tool". The following selection is included on the vertical soft-key bar: "Length", "Radius", Calibrate probe".

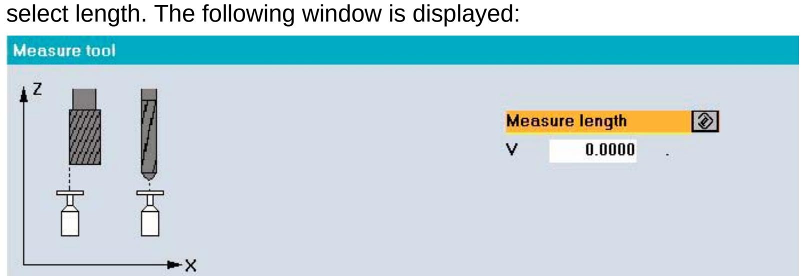
Radius

Select radius. The following window is displayed.



Length

or
select length. The following window is displayed:



Enter details into the input screen - if required, enter the offset (V, positive value).

The measurement automatically starts after "NC-Start".
The tool offsets "radius" or "length 1" are calculated and are entered into the active tool offset data.

2.6 Setting-up a workpiece and measuring a tool with 802D sl - measuring in JOG

SINUMERIK 802D sl supports you when setting-up the machine, i.e. the workpiece and the tool. SINUMERIK 802D sl provides you with manual measuring functions - and for tool measurements - also automatic measuring functions.

Setting-up the workpiece

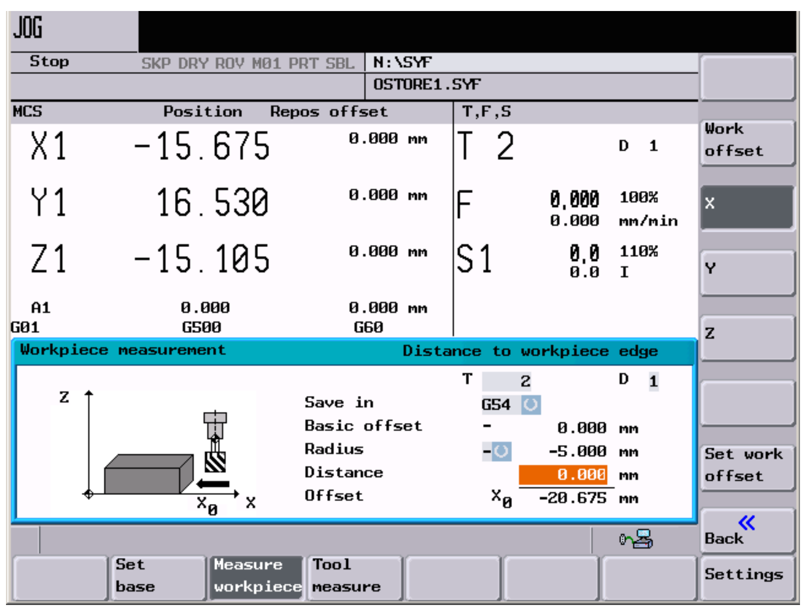
Setting-up is realized by scratching or probing the workpiece using a manual measuring or milling tool with known geometry in the JOG mode.

Measuring workpiece

Select "Measure workpiece".

x

Select the axis of the workpiece edge that is to be set-up (X, Y, Z).



Activate the zero offset (G54) in which the measured offset should be entered.

Traverse the tool or the measuring probe to the edge of workpiece.

Set zero offset

Press "Set zero point" in order to accept the actual position in the zero offset. The zero offset is now determined for each axis.

Measuring a tool

In the setting-up mode you can determine the offset values for the tool directly at the machine. The following two options are available:

- Manual measurement where the tool is scratched at a known tool geometry
- Semi-automatic measurement using a mechanical test socket

Measuring a tool using a mechanical test socket

Prerequisites

- The tool has been clamped
- The mechanical test socket has been calibrated

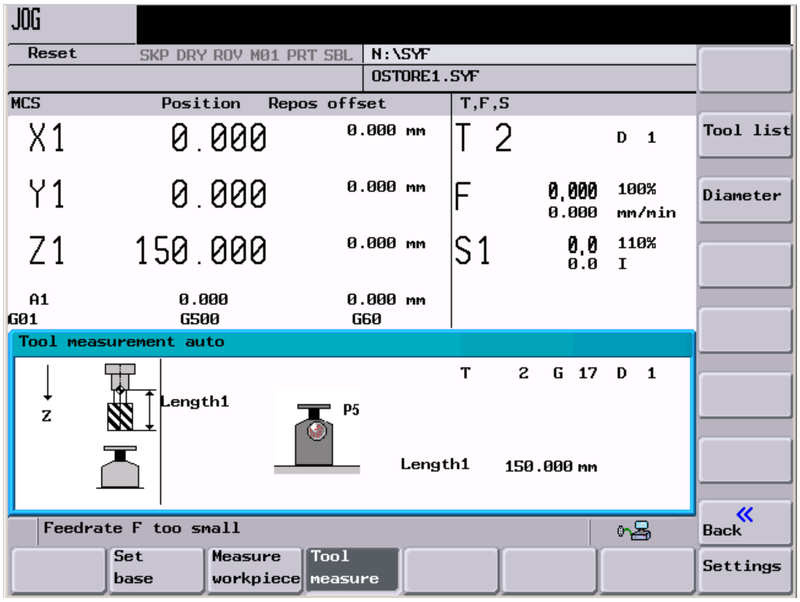
Measuring tool

Select "Measure tool".

Measuring auto

Select "Auto measurement" for automatic measurement using the mechanical test socket.

Traverse the tool to the mechanical test socket using the handwheel. When the mechanical test socket triggers this is displayed and the tool offset data are determined.



Diameter

Length 1

You can now measure the diameter and the length of the tool. Press the appropriate softkey.

2.7 AUTOMATIC mode - in-process measurement

With in-process measurements in the automatic mode, NC programs are specifically parameterized for the measuring task (measuring cycles). For measuring tasks that are repeated, e.g. setting-up for multi-part machining, you can simply run the NC programs and the workpieces are automatically measured, aligned and the tools measured.

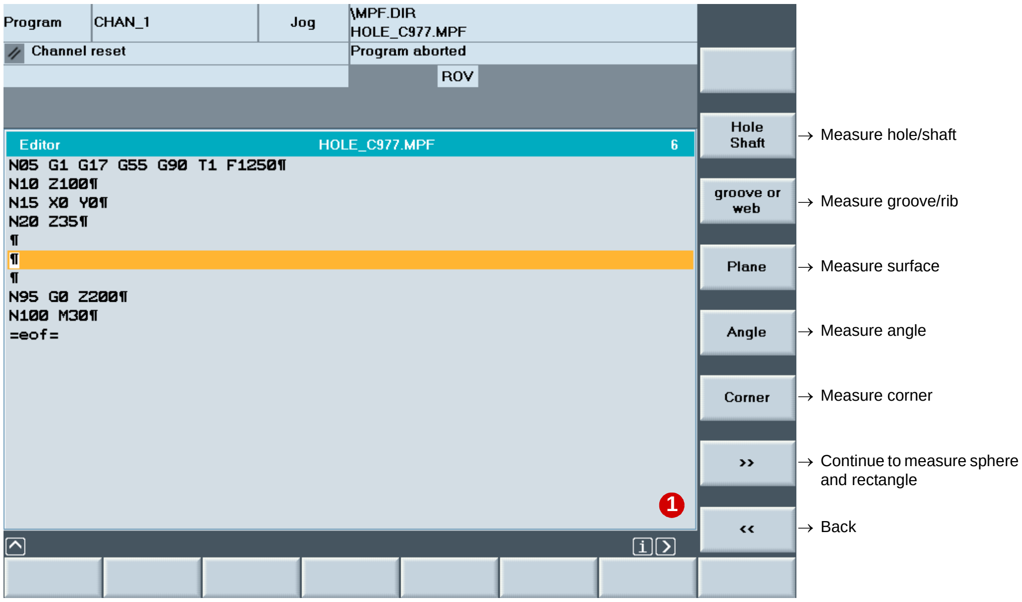
Measuring cycles for in-process workpiece measurements

You can use measuring cycles in-line with those required in practice for in-process measurements

▶ You can select the measuring cycles within the NC program using the softkeys **Measuring Milling > Measure** workpiece.

NOTE

The softkeys for in-process measurement are on the expanded softkey bar.
Go to this softkey bar by pressing the expand arrow > (1).



Measuring the workpiece in the automatic mode

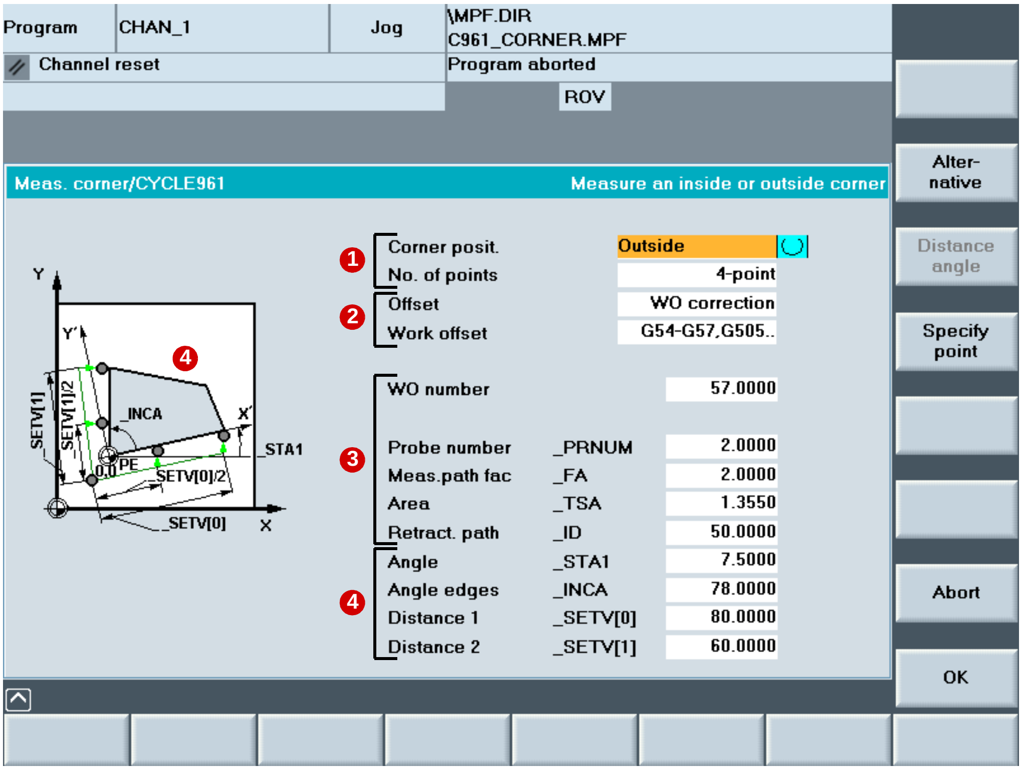
The sequence is described using as an example setting-up the workpiece using the measuring cycles set corner (CYCLE961) and measure surface – 1-point measurement (CYCLE978).

Prerequisites

- The measuring cycles have been installed
- The workpiece has been clamped
- The probe has been calibrated, is active and has been clamped in the spindle; tool offset is active

Setting/measuring a corner for X/Y axes:

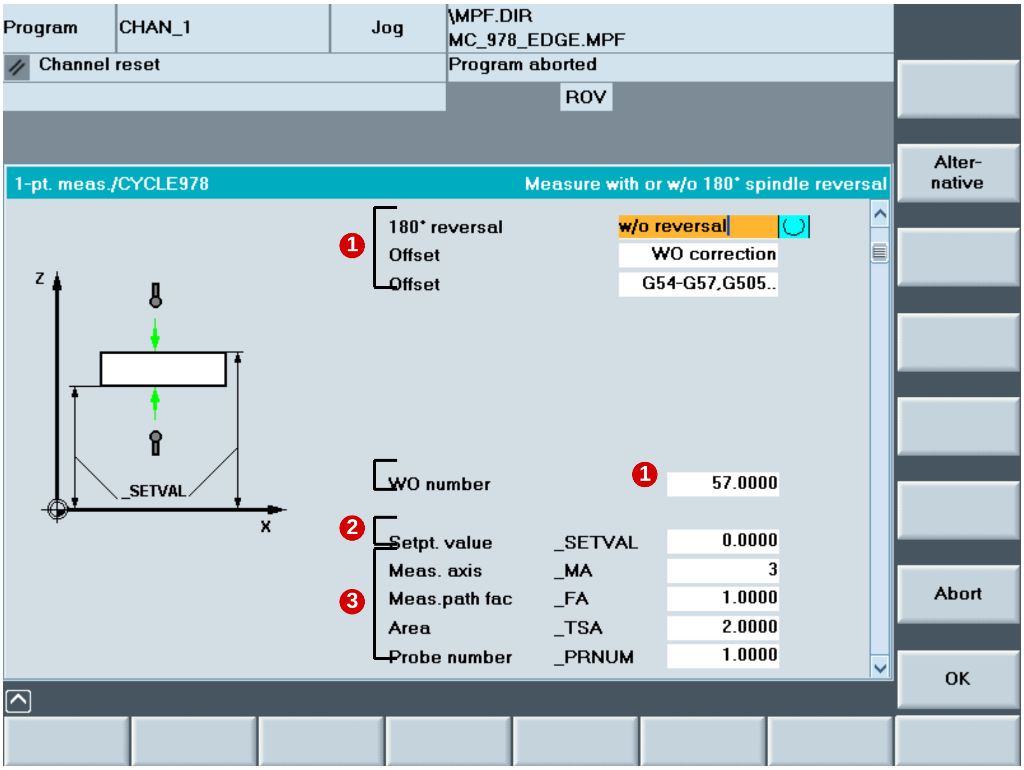
- Generate a new program to set-up the workpiece.
- Select the **corner measuring** cycle.
- Select the position of the corner and the number of measuring points (1).
- You can define whether the result of the measurement is a correction or only a measurement (2).
  - Correction in the zero offset, specifying the ZO
  - Correction in the tool offset data
  - Only measurement
- Parameterize the measuring operation and the probe (3).
- Enter the approximate dimensions of the corner to be measured. The help display supports you when making the entry (4).



Measuring a point for the Z axis:

- Select the **Surface** softkey.
- You can define whether the result of the measurement is a correction or only a measuring operation (1).
  - Correction in the zero offset, specifying the ZO
  - Correction in the tool offset data
  - Only measurement

  As you are setting-up the workpiece here, the correction is made in the ZO.
- Enter the approximate dimensions of the point (2).
- Parameterize the measuring operation and the probe (3).



Measuring cycles for in-process tool measurements

You can use a practical measuring cycle for in-process tool measurements. The cycle determines the length and the diameter of the tool using the calibrated mechanical test socket.

▶ You can select the measuring cycles in the NC program in the expanded softkeys **Measuring Milling > Measure tool**.

Prerequisites

- The measuring cycles have been installed
- The mechanical test socket has been calibrated
- The tool has been clamped

Measuring a tool in the automatic mode

IIn the automatic mode you can automatically measure the tool data or enter as tool offset. In the following example you will generate a program that determines the tool length and the radius and enters these into the tool correction.

Determining the tool length:

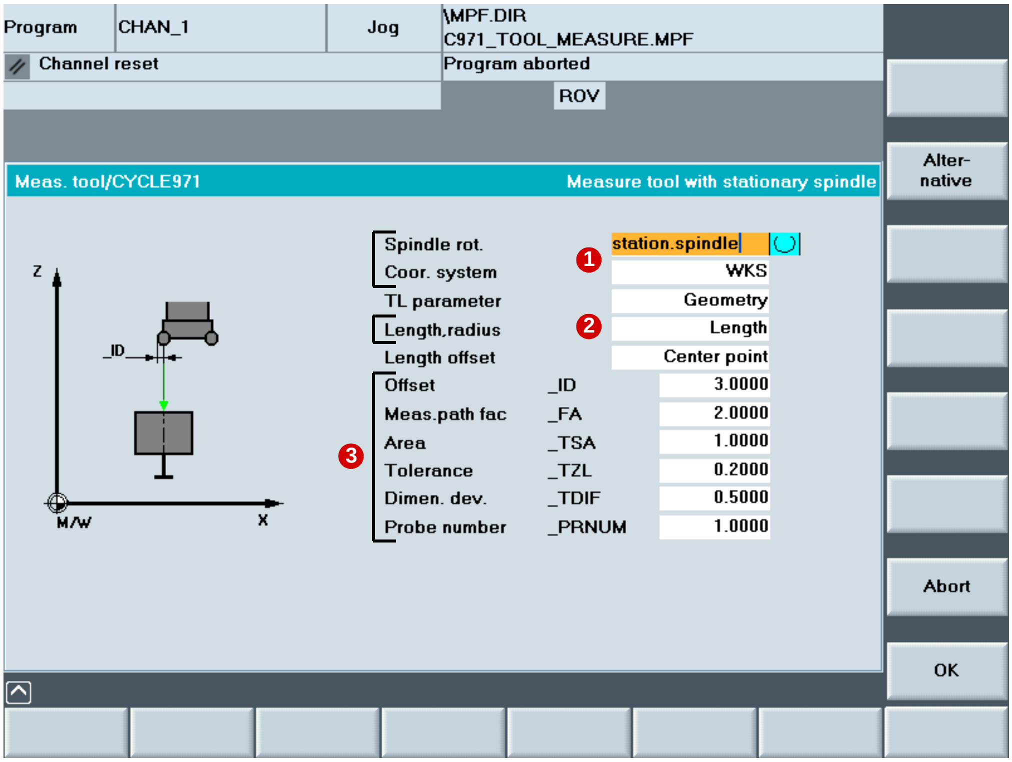
▶ Generate a new program to measure the tool.

▶ Select the measuring cycle - **measure tool**.

▶ The measurement is performed with the spindle stationary and the measured values are entered into the tool geometry component (1).

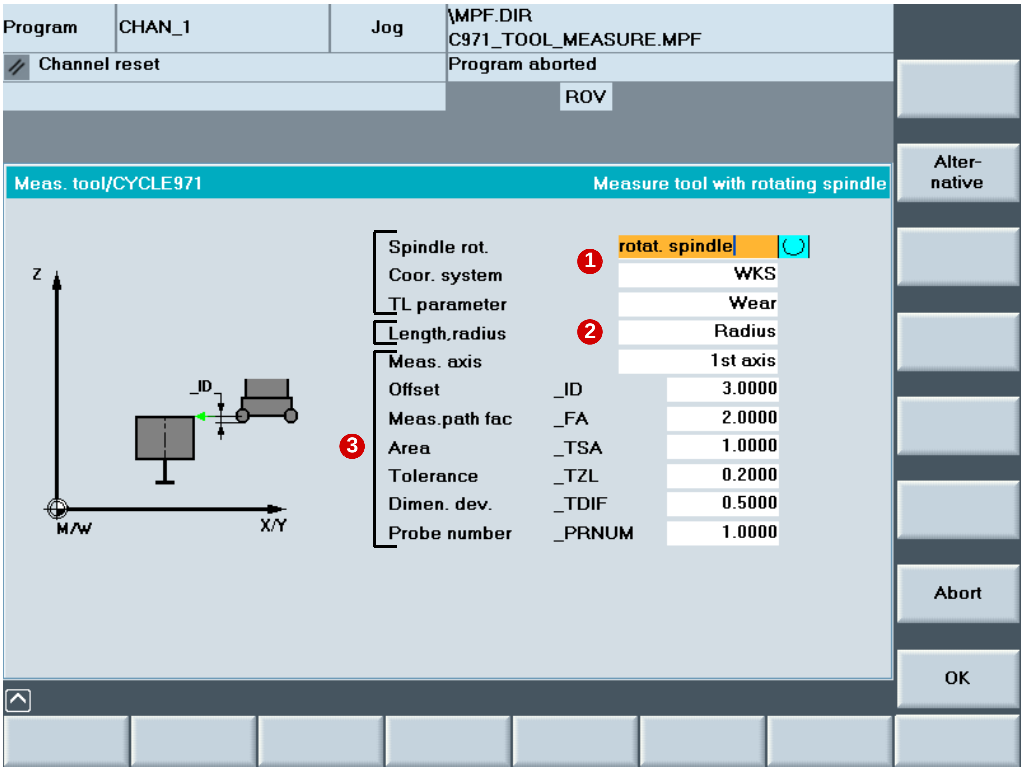
▶ Select the length as measured value (2).

▶ Parameterize the measuring operation(3).



Determining the tool radius:

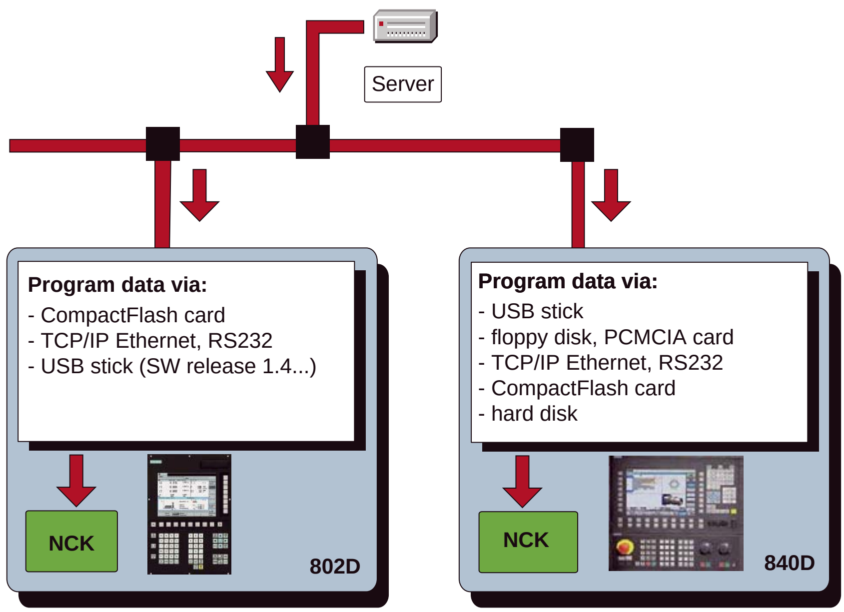
- The measurement is performed with the spindle stationary and the setpoint/actual value difference is entered into the radius wear (1).
- Select the radius as measured value (2).
- Parameterize the measuring operation (3).



2.8 Program data transfer/managing programs

NC programs are stored in the control, if required, downloaded into the NCK working memory (RAM) and executed on the machine.

For mold making programs - that generally comprise technology and geometry programs - the geometry program with up to 100 MB is often so large that it can no longer be saved in the NCK working memory (RAM) and/or cannot be processed. This is the reason that mold making programs must be saved on an external memory so that they can be successively processed.



External program memory (hardware configuration)

Depending on the system, the existing operator interface (HMI) and the options that have been purchased, you can use external program memories with the following characteristics:

- TCP/IP Ethernet (network drivers), serial interface RS232/V.24 (low date rate)
- Hard disk (PCU 50)
- Compact Flash card (TCU, 802D)
- USB interface (USB stick)
- PCMCIA card (PCU 20)
- Floppy disk

Calling the program data in the external memory using EXTCALL

An EXTCALL command is programmed in the main program that in turn calls the geometry program that has been externally stored corresponding to the network path on the server, the USB interface, hard disk etc.
Using EXTCALL, you can subsequently download a program from the HMI in the execute from external mode.
In this case, all programs that can be reached through the directory structure of the HMI, can be subsequently downloaded and executed.

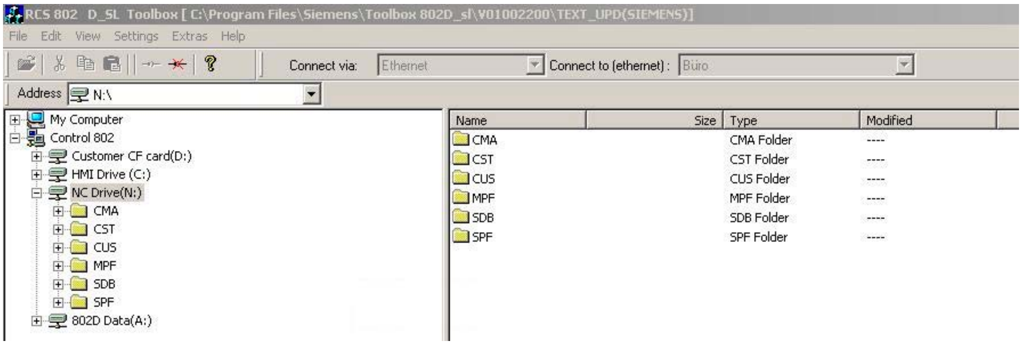
Procedure when calling the geometry program using EXTCALL

- In machine data SD 42700: EXT_PROG_PATH, define the source directory to the geometry program, e.g. to a server "\\R4711\workpieces\subprograms". The default setting is optional. The directory can first be specified when making the call with EXTCALL.
- Program the geometry program call, e.g. SAMPLE in the main program. The call differs depending on the control and where the data is saved.
  - PCU 50, subroutine is on the hard disk
    EXTCALL "sample"
  - PCU 20, 802D, subroutine is directly on the CompactFlash card
    EXTCALL "C:\sample.spf"
  - PCU 20, 802D, subroutine is in the directory on the CompactFlash card
    EXTCALL "C:\programms\sample.spf"
  - Network connected to Ethernet and path in the machine data SD 42700
    EXTCALL "sample.spf"
  - Network connected to Ethernet and no path in the machine data SD 42700
    EXTCALL "\\myserver\programms\workpieces\sample.spf"

Managing large programs using the 802D sl – RCS tool

The RCS tool (Remote Control System) is an Explorer-type tool for your PC/PG to assist you in your daily work with SINUMERIK 802D sl.
The connection between the control system and the PC/PG can be established either via an RS232 cable, peer-to-peer cable, or a local network cable (optional).

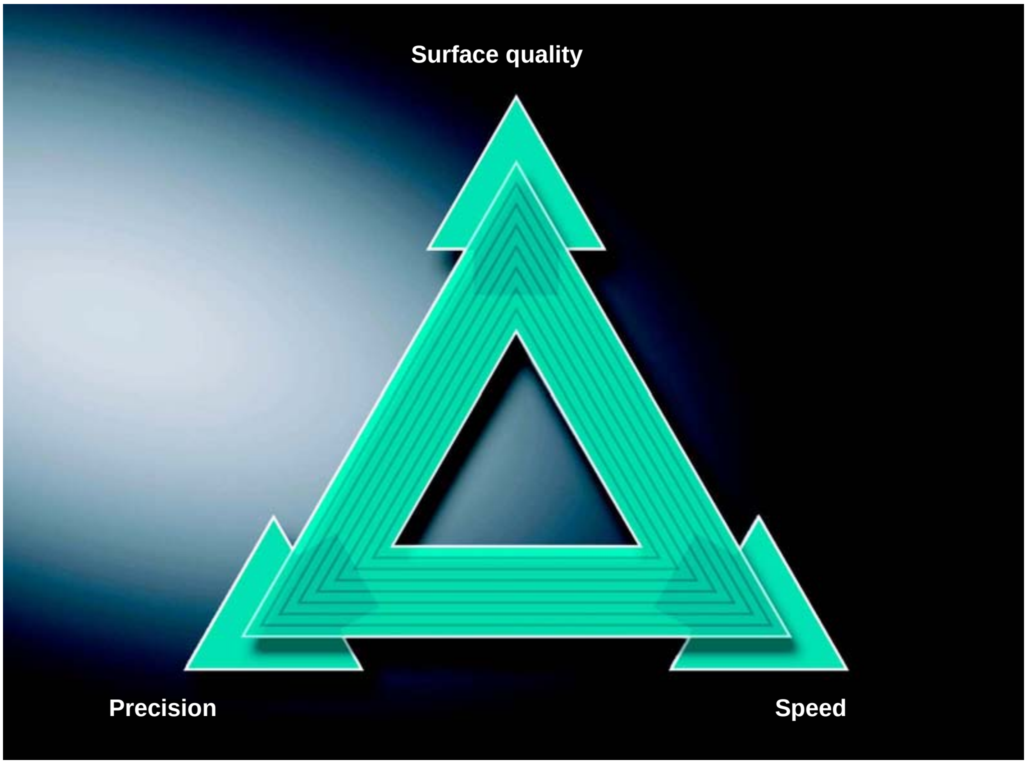
When executing from external with 802D sl please note that programs on the CF card cannot be edited. If these programs are larger than the NCK memory then you must externally edit the programs (e.g. PC). The RCS tool is provided for precisely this task. In a transparent Explorer-type display you can copy, shift and delete programs or other data.



2.9 High Speed Settings – CYCLE832

Application

You can influence the sequence of CAM programs using CYCLE832 of the SINUMERIK 840D. It is used to provide technological support when machining freeform contours in the 3-axis high-speed machining sector (High Speed Cutting - HSC).
CYCLE832 combines the essential programming commands and G codes that are required for HSC.



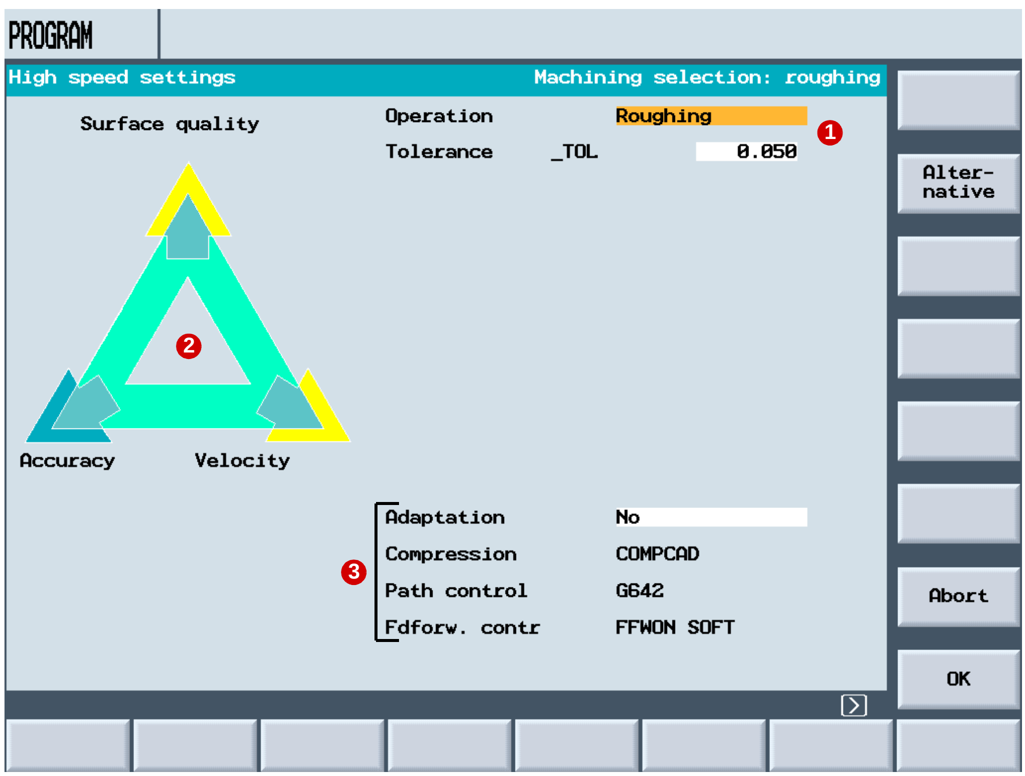
When executing CAM programs in the HSC area, the control has to process high feedrates with the shortest NC blocks. A good surface quality with high precision in the µm range at high machining feedrates >10 m/min is expected. By applying different machining strategies you can use CYCLE832 to fine tune the program.

- When **roughing**, the emphasis is on speed due to the blending of the contour.
- When **finishing,** the emphasis is on precision and surface quality.

In both cases, specifying a tolerance ensures that the machining contour is observed in order to achieve the desired surface quality and precision. Generally, when roughing, the tolerance is selected higher than when finishing.

Calling CYCLE 832 in HMI menu tree

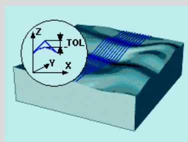
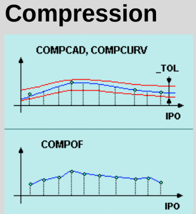
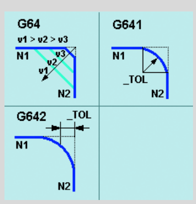
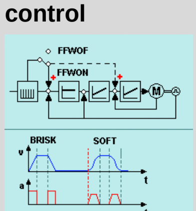
Programs	Opens the "Programs" operating area.
Milling	Press the "Milling" softkey.
>>	Display additional softkeys.
High Speed Settings	Press "High Speed Settings". The cycle is called.



Corresponding to the parameter selection ❶ the yellow arrows ❷ either point towards "Speed", "Surface quality" or in the direction "Precision".
The other options ❸ are released by the machinery OEM and are generally password protected.

Parameters for the High Speed Setting cycle

In the **Machining** field the user only has to select between finishing, pre-finishing and roughing and enter a value in the **Tolerance** field. The values in all of the other fields have already been entered by the machinery OEM. The machinery OEM can enable the other fields using the **Adaptation** field (password-protected).

Machining	■ Finishing (default) ■ Pre-finishing ■ Roughing ■ Deselection	By calling "De-selection" the modified machine/ setting data are reset to the value generated by the machinery OEM
Tolerance_tol. 	■ Chord tolerance (chord tolerance should be taken from the CAM system or weighted with a factor of 1.2 ... 1.5)	Tolerance of linear/rotary axes, default settings: → 0.01 mm/ 0.08° (finishing) → 0.05 mm/ 0.4° (pre-finishing) → 0.1 mm/ 0.8° (roughing) → 0.1 mm/ 0.1° (deselection)
Adaptation	■ Yes ■ No	→ The following fields can be modified → The following fields are invisible - and are released by the machinery OEM.
Compression 	■ COMPOF (default) ■ COMPCAD ■ ■ B SPLINE	→ Compressor off → Compressor on, continuous acceleration rate for mold making applications → Jerk-free for circumferential milling → Spline interpolation
Continuous path control 	■ G642 (default) ■ G641 ■ G64	→ Blending with single-axis tolerances → Programmable blending clearance → Continuous path mode With an NC block compressor with COMPCAD, COMPCURV, G642 is always permanently selected.
Feedforward control 	■ FFWOF SOFT ■ FFWON-SOFT ■ FFWOF-BRISK	→ Without feedforward control, with jerk limiting → With feedforward control, with jerk limiting → Without feedforward control, without jerk limiting The selection of feedforward control (FFWON) and jerk limitation (SOFT) requires that the machine manufacturer has optimized the con-trol and the machining axes.

Notes

- CYCLE832 is based on the use of G1 blocks. The tolerance is not important when using G2/G3 and CIP programs.
- When making changes you should align to the tolerance value to that specified in the CAM program. Tolerances that are lower than specified there are not practical.
- Please note that there are dependencies between the fields: For instance, if **compression** is switched-out then various grinding types can be selected under continuous **path control**.

NOTE

Please refer to Chapter 3.5 for additional information; individual parameters are described in detail here.

Programming

Ideally, you program CYCLE832 in the higher-level NC master program that then calls the geometry program. This means that you can apply the cycle to the complete geometry or - depending on the transparency of the CAM program - to individual program sections or freeform surfaces.

NOTE

For information regarding an optimum program structure please refer to Chapter 1.4, specifically for CYCLE832, please observe the information in Chapter 2.10.

2.10 Program structure for mold making

Recommendation for a practical program structure with CYCLE 832

For machining a main program is generated 1 that includes all technology data. The main program calls one or several subroutines 2 , 3 that contain geometry data of the workpiece. The tool change defines the sub-division into subroutines.

Example

Main program

```
Call.MPF 1
N1      T1 D1
N2      M6                                  ; Tool change
N3      M3 S15000
N4      CYCLE832 (0.05,112003)      4
N5      EXTCALL "CAM_Rough"         5       ; all programs should
                                            ; be located in one
                                            ; directory. If this is not
                                            ; the case, then the
                                            ; paths must be
                                            ; specified.
N6      T2 D1
N7      M6                                  ; tool change
N8      M3 S20000
N9      CYCLE832 (0.005,112001)     4
N15     EXTCALL "CAM_Finish"        5
N16     M30
```

Subroutine

```
CAM_Rough.SPF 2
N1      G90
N2      G0 X0 Y0 Z10

N3      G1 Z0 F500
N4      G1 X-1.453 Y0.678 F10000
N17     G1 X-1.814 Y0.842           6
N18     G1 X-1.879 Y0.684 Z-0.001
...
N5046   G1 X-4.118 Y-11.442
N5047   G0 Z10
...
N5051   G1 Z-2.132 F5000
...
N6582   G1 X7.609 Y3.555
N6583   G0 Z50
N6584   M17
```

Subroutine

```
CAM_Finish.SPF 3
N1      G90
N2      G0 X0 Y0 Z10
....    .......
N7854    M17
```

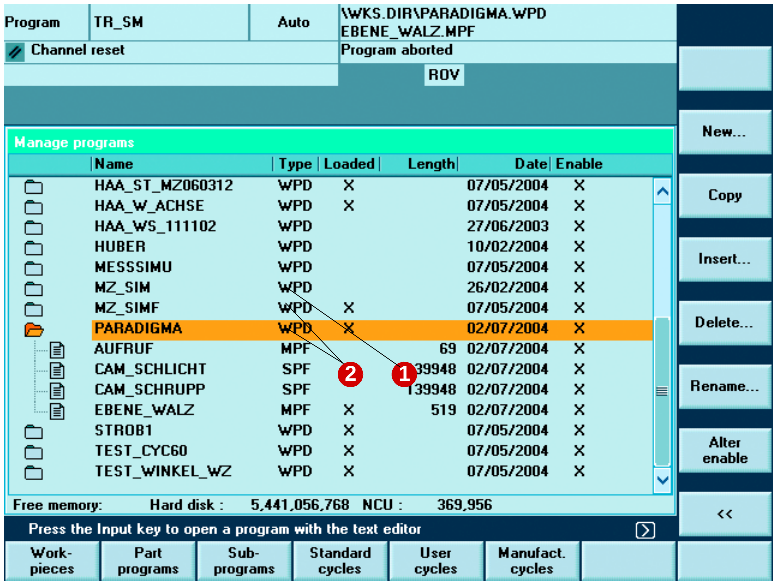
Main program: The main program includes two important functions for milling, CYCLE832 4 and EXTCALL 5 .

CYCLE832 4 : CYCLE832 has been specifically developed for the program structure shown where technology and geometry data are separated. The machining technology for milling is defined in CYCLE832. For the roughing program "CAM_Rough" using T1, the parameters in CYCLE832 were set towards achieving a high speed. For the finishing program "CAM_Finish" the parameters were set towards achieving high precision and surface quality.

EXTCALL 5 : CAM programs are generally extremely large which is why they are stored an external memory. EXTCALL calls the subroutines from the external memory.

Subroutine: In the subroutine - G90 for absolute programming is immediately followed by the geometry sets. In our particular example these are the blocks for 3-axis milling 6 .

2.11 Selecting/starting/stopping/interrupting/continuing a program



Machine

Select the "Machine" operating area.

AUTO

Select the "Automatic" mode.

Program overview

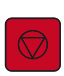
Workpieces

Press "High Speed Settings". The cycle is called.

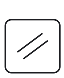
Selection

In the workpiece directory highlight the part program ❶ - in this case the program "Call.MPF" ("Aufruf.MPF") and press "Select" ("Anwahl").

Press "NC-Start" to start the part program. This calls the geometry programs "Roughing.SPF" ("Schrupp.SPF") ❷ and "Finishing.SPF" ("Schlicht .SPF"), which are loaded block-by-block into the control system from the external memory during machining.



Press "NC-Stop" to stop the part program.



Press "Reset" to interrupt the part program.

Note A part program interrupted with "NC-Stop" can be continued by pressing "NC-Start". A part program interrupted with "Reset" is executed from the beginning if "NC-Start" is pressed - or, with a block search jumps to the point of interruption where it continues the program.

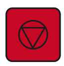
2.12 Interrupting a program

REPOS – repositioning after an interruption

Function

When a program is interrupted - or after an NC-Stop - the tool can be moved away from the contour in the JOG mode, e.g. to check the cutting edge of a tool. The control saves the interruption point coordinates. The differential travel of the axes is displayed.

Operation



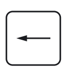
Initial situation: Program interrupted with "NC-Stop".

Machine

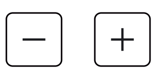
Select the "Machine" operating area.

Select the "JOG" mode.

Reposition after a program interrupt.



Select axes.



Move the axes to the point of interruption according to the differential travel displayed. It is not possible to pass the point of interruption.

AUTO

Changeover from the "JOG" into the "Automatic" mode.

Continue machining

Accelerated external block search without calculation

Function

This SINUMERIK 840D function was specifically developed for programs that are called with EXTCALL. This means that it is admirably suited for large programs that are received from CAM systems.

After machining has been interrupted with "Reset" using the function "Accelerated external block search without calculation" any location in the part program can be selected at which machining is to be started or continued.

Operation

Initial situation: Program was interrupted with "Reset".

Example

Call.MPF 1

N1	G54
N2	T1 D1
N3	M3 S15000
N4	CYCLE832 (0.05,112003)
N5	EXTCALL "CAM_Rough"
N6	T2 D1
N7	M3 S20000
N8	CYCLE832 (0.005,112001)
N16	EXTCALL "CAM_Finish" 3
N10	M30

CAM_Roughing.SPF 1

N1	G90
N2	G0 X0 Y0 Z10
N3	G1 Z0 F500
N4	G1 X-1.453 Y0.678 F10000
N17	G1 X-1.814 Y0.842
N18	G1 X-1.879 Y0.684 Z-0.001

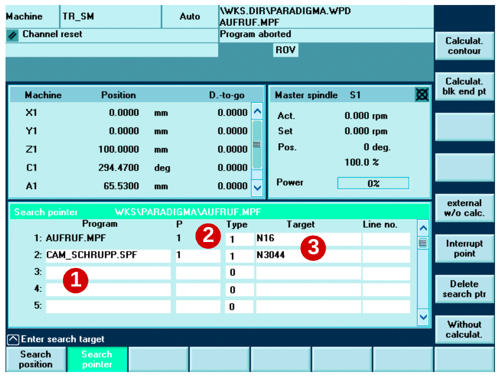
CAM_Finishing.SPF

N1	G90

Block search — Press the "Block search" softkey.

Search pointer — Press the "Search pointer" softkey.

Break point — Press the "Breakpoint" softkey.

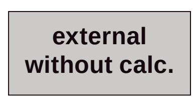


Pressing the "Breakpoint" softkey inserts the screen with the entire program sequence ❶ up to the breakpoint:

In this example, the main program "Call.MPF" ("Aufruf.MPF") calls the subroutine "CAM_Roughing.SPF" ("CAM_Schrupp.SPF"). The EXTCALL for the subroutine is located in block N16 ❸. Block 3044 in "CAM_Roughing.SPF" ("CAM_Schrupp.SPF") is where the program was interrupted.

There are now two possibilities:

1. Jump directly to the breakpoint in the subroutine: Press the "External without calc." softkey. The program jumps immediately to block 3044.
2. To do this you must select a (search) type ❷ - when searching in external programs always select type 3 for a string search. Then enter the type number and next to it the required search text - e.g. block or line number.

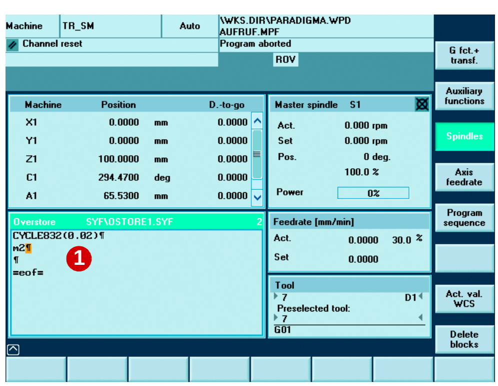


Press the "External without calc." softkey.

Continue machining at the destination block.

Corrections

While making an entry for CYCLE Stop the "Overstore" function can be selected that so that you can correct the destination block before starting the program.



A typical case is shown here, where the compressor tolerance needs to be subsequently changed. CYCLE832 was called to do this and the compressor tolerance was manually changed to 20 µm ❶. This was possible by just entering one single parameter (tolerance = 0.02). CYCLE832 is now executed before the main program is started.

The tolerance is activated by pressing NC-Start.

2.13 Program overview/status of external programs

Function

When executing programs from external you can display the current status and the program runtime.

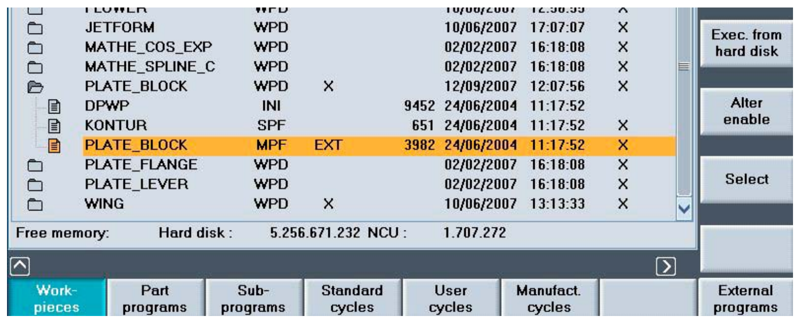
Displaying the status for 840D standard HMI

Program overview

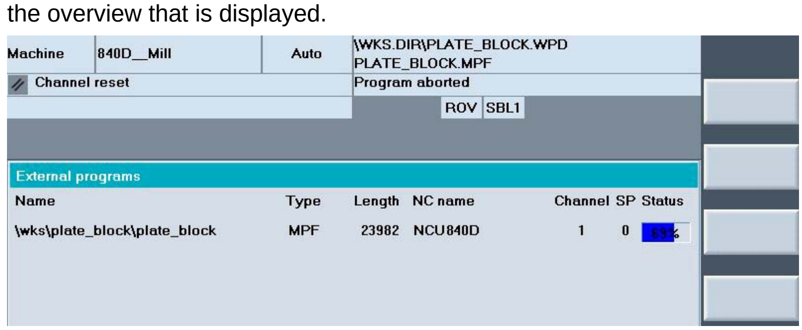
In the "Automatic" mode, select the softkey **Program Overview**. The program overview is displayed.

External programs

Select the softkey **External Programs**.



The current status of the external program is displayed as a percentage in the overview that is displayed.



2.14 Simulating the part program

Function

Using the simulation function you can obtain an overview of the individual machining steps and you can check the workpiece programming.

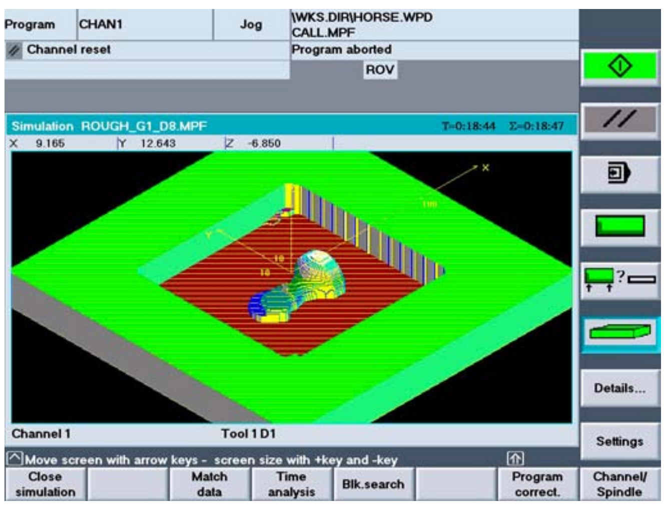
Selecting the simulation function:

- Select the Simulation softkey in the Program Editor.

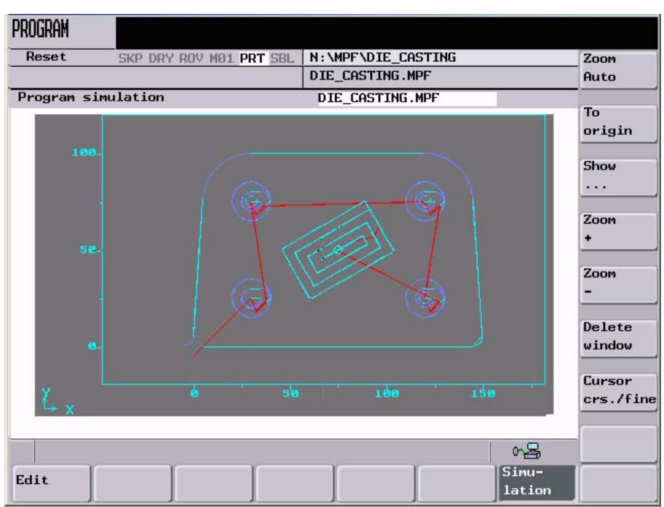
Simulation functions depending on the control:

- Simulation is displayed in the 3 planes (840D) or the machining plane (802D sl)
- Turning, scaling and zooming in/out
- 3-D volume model in any view and any sections (only 840D)

Example, simulation 840 D



Example, simulation 802D sl

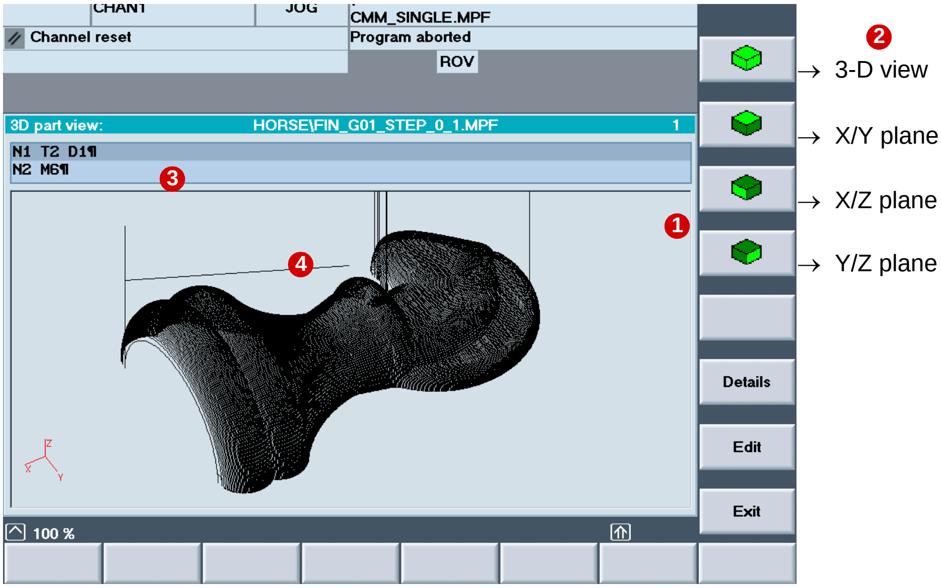


2.15 Quick View / fast display

Function

Quick View of SINUMERIK 840D with PCU50 allows mold making part programs that contain G01 blocks to be visualized. Program loops, polynomials, transformations and G02/03 blocks are not supported. Four views ❶ are available: 3-D view ❷ , X/Y plane, X/Z plane, Y/Z plane.

The two editor lines ❸ display the block currently highlighted in the graphic. Scrolling through the editor window automatically highlights the position ❹ in the graphic.



The following functions are also available

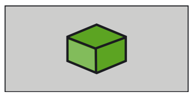
- Search for a specific block
- "Zoom in/out"
- Shift, rotate
- Measure the distance between two points
- Edit the NC part program displayed

Notes You can use Quick View (fast display) for the 840D with standard interface and for ShopMill. For 840D Standard Quick View is in the **Program Manager**, for ShopMill, you can open Quick View in the **Program Editor**.

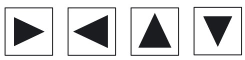
Operation

Quick View

Call the "Quick View" function.

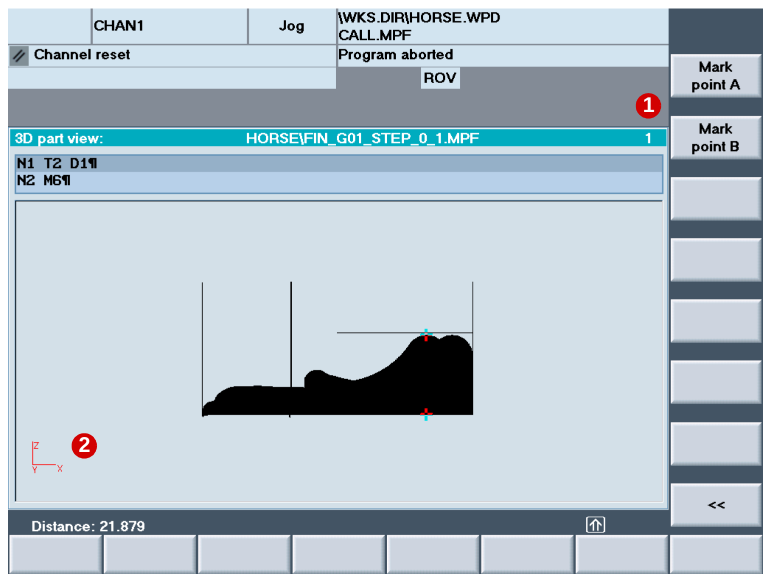


Choose the view you require - in this case the X/Z plane.



Use the cursor to select a point in the graphic. The associated block is displayed in the editor line.

Call the block, e.g. to change it in the program.



Measuring distances in Quick View

In Quick View you can also measure distances between two points. This function is extremely helpful, e.g. if you want to know the dimensions of a workpiece - because it is only very difficult to assess the size of a workpiece from the G1 blocks of a program.

By highlighting two points 1 you are shown the distance 2 between the points in the footer area.

2.16 ShopMill

For SINUMERIK 840D the user-friendly ShopMill interface provides a real alternative to the universal SINUMERIK 840D standard DIN/ISO user interface.

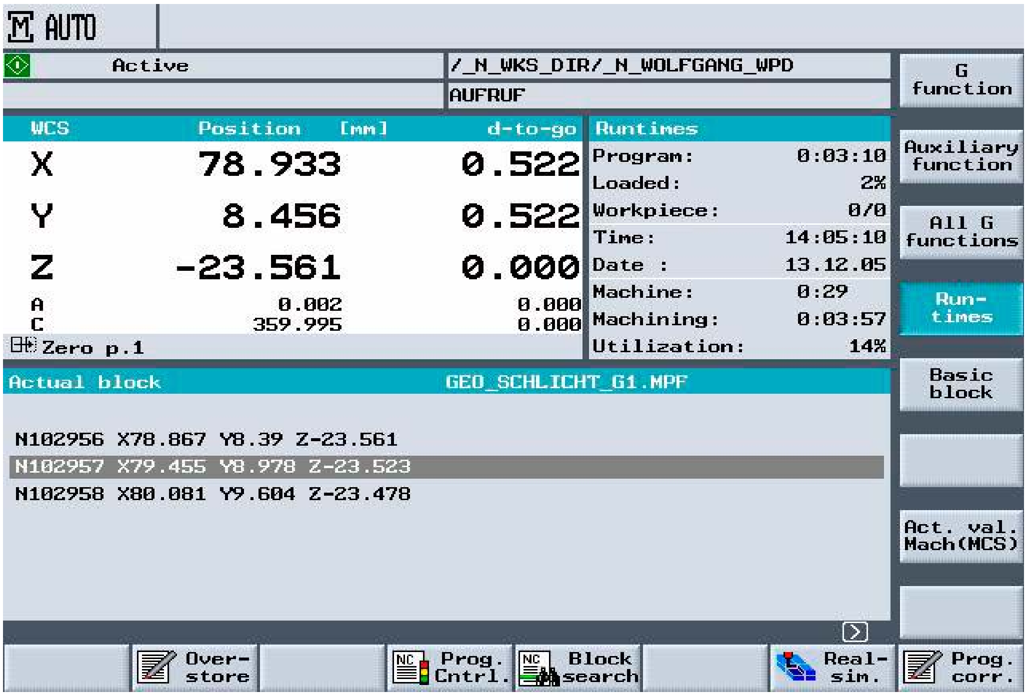
ShopMill has been supplemented by many mold making functions, greatly simplifying its use for mold makers.

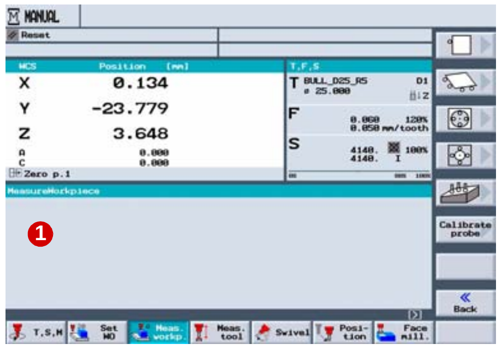
As a consequence, ShopMill is no longer restricted to sequencer programming using partial machining steps - and in fact it even supports demanding 3+2-axis and 5-axis applications.

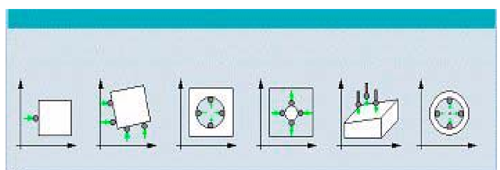
The complete description of ShopMill functions can be found in "SINUMERIK 810D/840D Using and Programming ShopMill".

ShopMill user interface

Simple operation and programming in the workshop







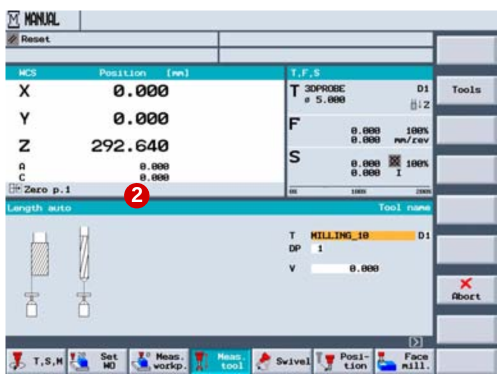
ShopMill functions

Setting-up

Powerful setting-up functions in ShopMill ensure rapid and accurate detection of the workpiece position. Special measuring cycles simplify measuring tools and the workpiece. Any offsets are automatically compensated by the control system.

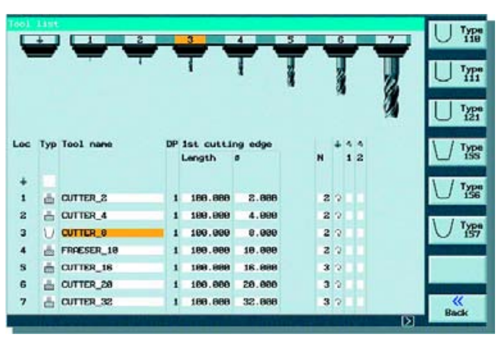
1. Measuring a workpiece
   Edge, corner, pocket/hole, spigot/ rectangle, plane))

The measuring functions are available when measuring in JOG. Measuring cycles support you when measuring in the AUTOMATIC mode.



2. Measuring a tool

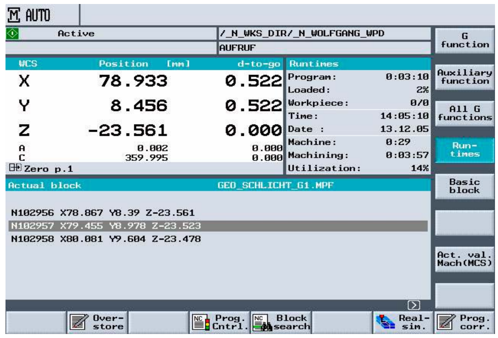
The measuring functions are available when measuring a tool in JOG and when measuring in the AUTOMATIC mode.



Tool management

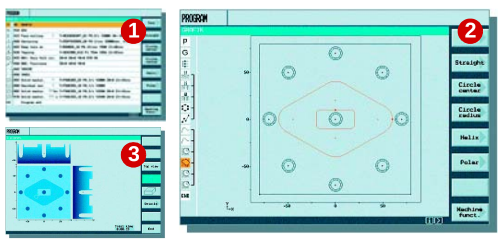
The ShopMill tool management is clearly structured and supports various tool types, tool names in plain text, daughter tools and the tool geometry with lengths, radii and number of cutting edges.

By entering the unit quantity, lifetime or wear parameters you can automatically monitor the time that tools are used - therefore ensuring a uniform machining quality.



AUTOMATIC

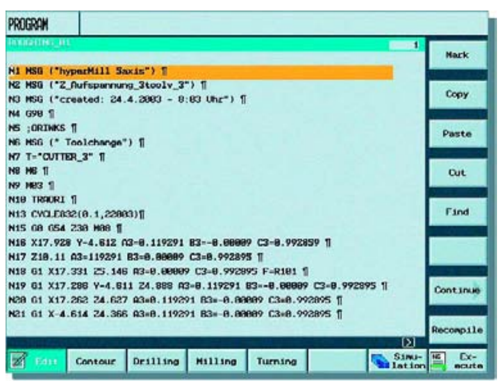
Displaying runtimes in the basic automatic screen.



Sequencers programming

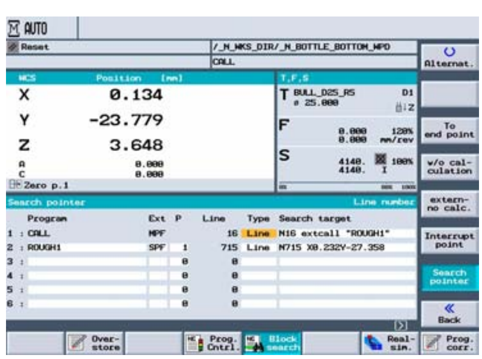
The ShopMill sequencer programming permits simple programming of basic 2 1/2D machining tasks directly at the machine. This is an ideal add-on for mould makers.

1. Program
2. 2D display
3. 3D display



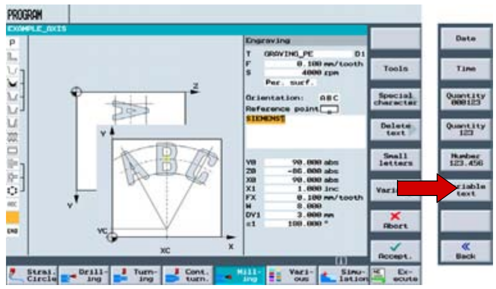
G code editor

ShopMill comes with an integrated, powerful G code editor, which supports mold making programs in a user-friendly fashion.



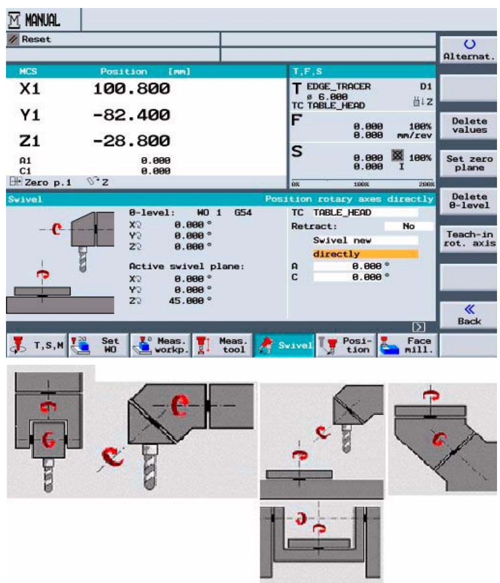
Block search

Fast block search in an external program (with and without calculation).



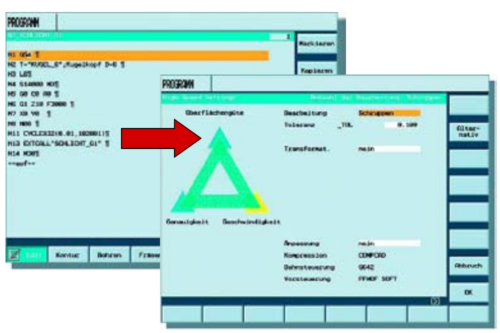
ShopMill cycle for engraving

- Text with special characters
- Date, time, workpiece counter, variable



Swiveling in JOG

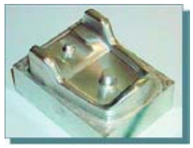
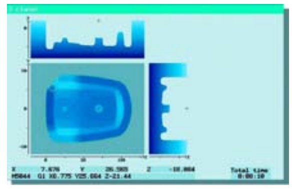
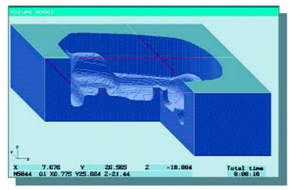
- Swivel cycle for all swivel tasks in the setting-up mode using softkeys
- Input is either directly or axis by axis
- All machine kinematics are supported



"High Speed Setting" cycle

The "High Speed Setting" cycle is now an integral component of the ShopMill user interface in the G code editor.

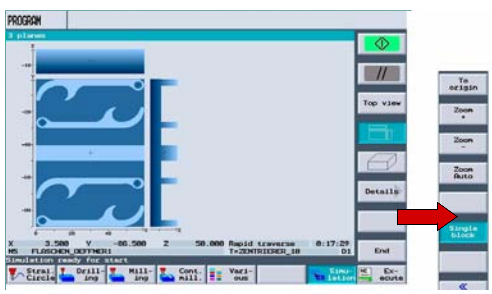
- Program editor
- CYCLE832, High Speed Settings

Simulation

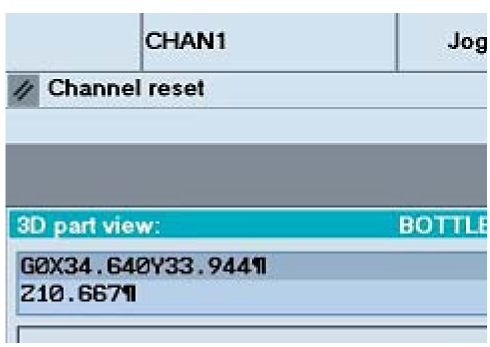
ShopMill provides various extensive and detailed simulation functions for displaying machining paths. To simulate the machining process, the control system completely calculates the currently selected program and displays the result in graphical form.
You can select one of the following representation types for the simulation:

- Top view
- 3-plane view
- Volume model



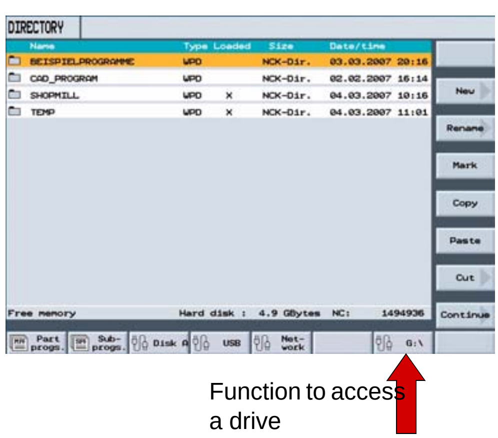
Program - simulation

- Start / stop / single block / reset of the simulation using softkeys
- Velocity can be controlled using the override function



Fast display / Quick Viewer for mold making

Quick display of traverse paths is possible for large part programs. With this dotted-line quick view display all programmed positions from G1 are displayed as resulting axis paths.



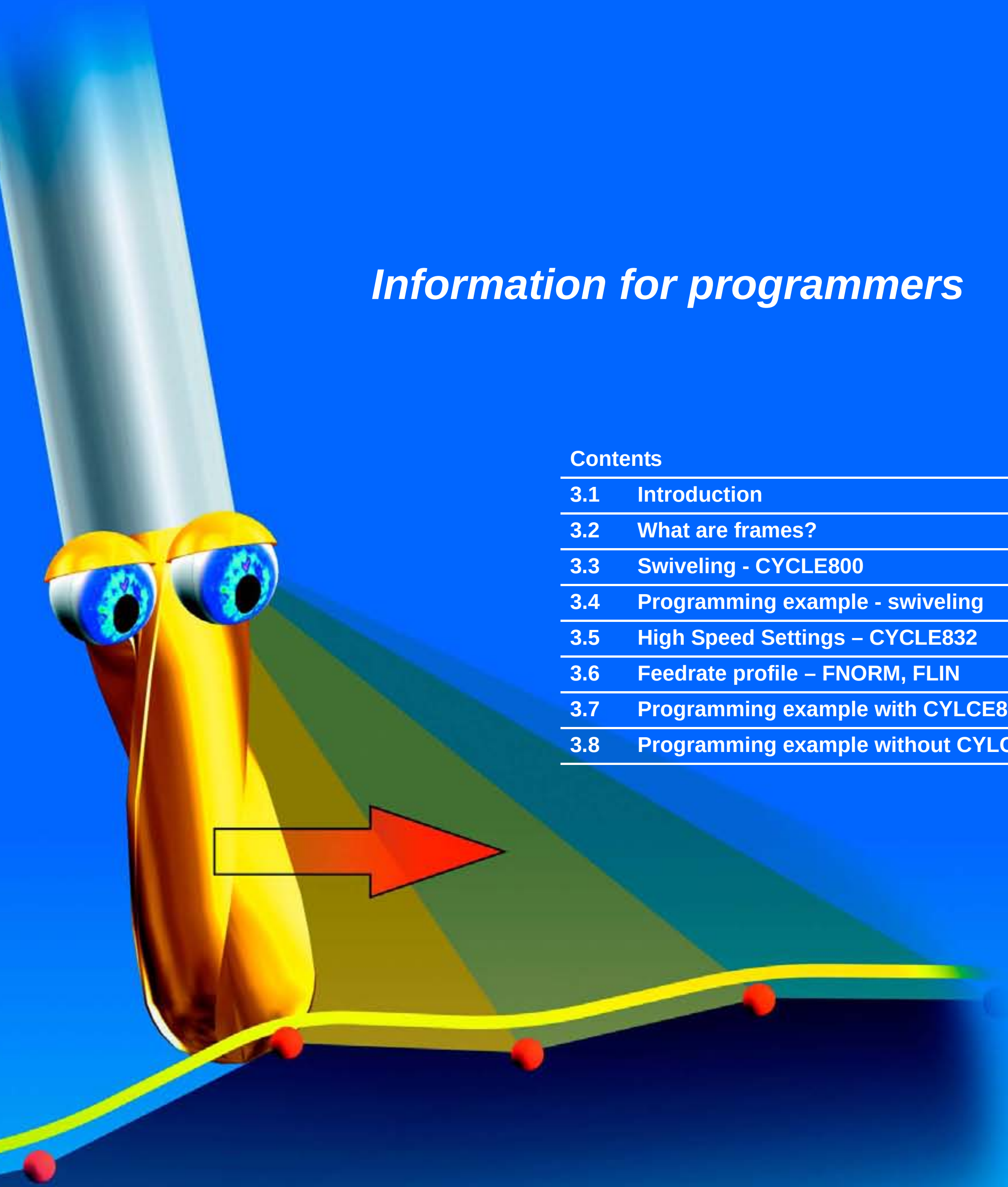
Function to access a drive

External drives

The ShopMill Program Manager allows direct access to external drives via Ethernet. Extensive mold making programs can be saved there.

- HMI hard disk (PCU 50)
- Flash card (PCU 20)
- Network drives
- USB stick

Information for programmers



Contents

3

3.1 Introduction

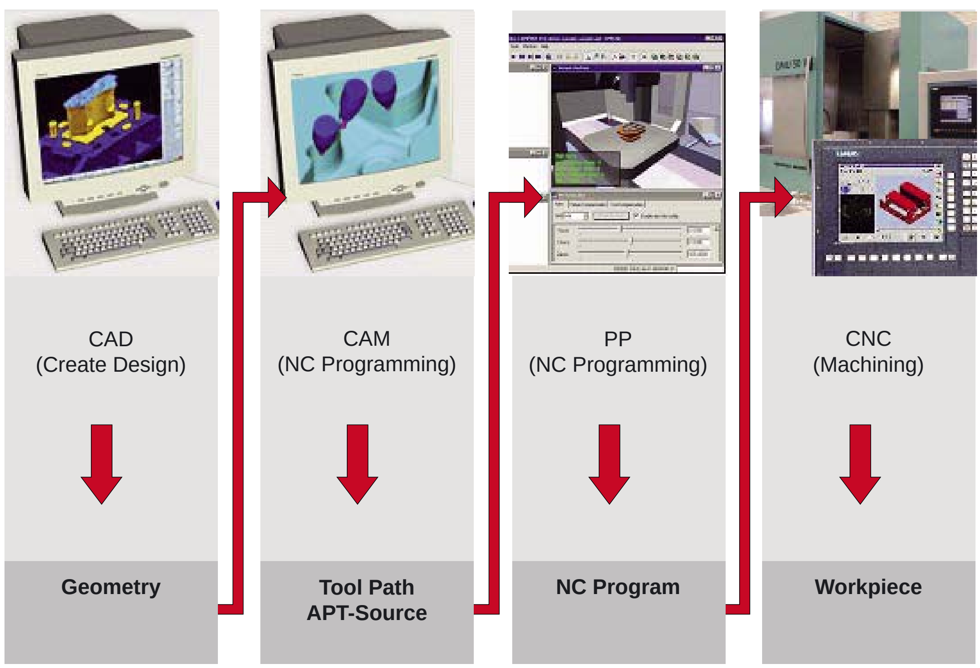
When programming freeform surfaces the entire CAD/CAM/CNC process chain is of vital importance.

The CAD system generates the geometry of the desired workpiece. Based on this geometry file the CAM system generates the corresponding machining strategy with the associated technology information.

The data format output from the CAM system is generally an APT or CL data file. This is converted in the post processor into an executable CNC code.

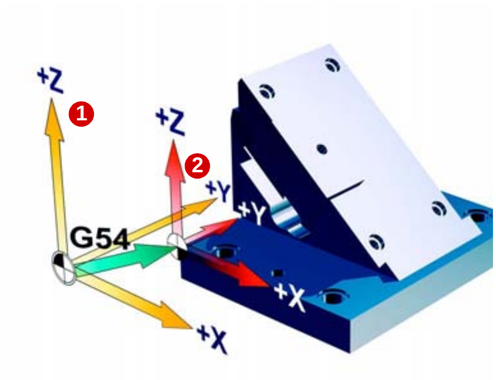
The upstream post processor is of the utmost importance in being able to fully utilize the capabilities and performance of SINUMERIK controls to the maximum.

The post processor must ensure that the higher-order functions of SINUMERIK controls - described in this Chapter - are activated in the best possible way. An overview of all of the higher-order SINUMERIK 840D functions can be found in Chapter .



3.2 What are frames?

Coordinate systems



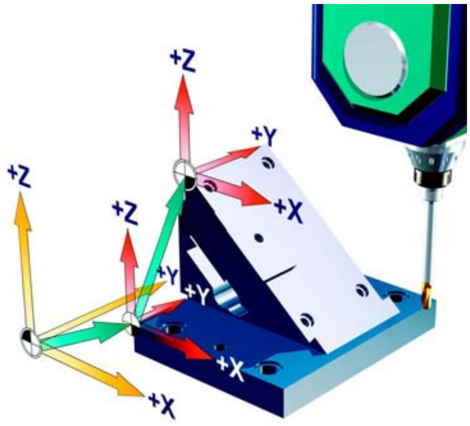
Machine coordinate system 1 with reference point and zero offset (G54, G55, ...) are known terms.

Using frames, coordinate systems can be shifted, rotated, mirrored and scaled so that they are aligned to the workpiece surface. This allows the programming time & costs to be reduced to a minimum.

With frames, starting from the actual workpiece coordinate system 2 the position of a target coordinate system is defined by specifying coordinates and angles. Possible frames include

- Basic frame (basic offset) (G500)
- Settable frames (G54, G55...)
- Programmable frames (TRANS, ROT...)

Coordinate systems and traversing motion

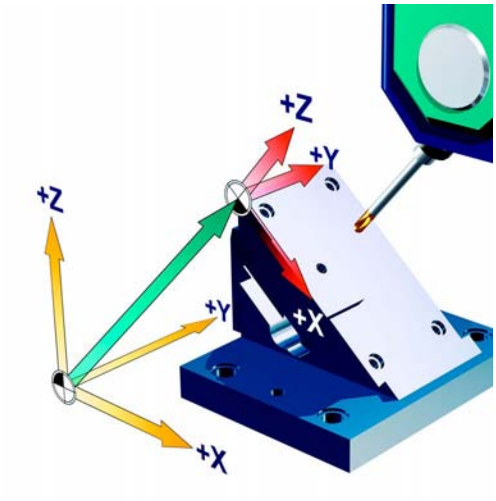


Using a 3+2-axis machine it is possible to machine surfaces that can be shifted and rotated in space as required.

The workpiece coordinate system only has to be shifted using frames and then rotated into an inclined plane.

This is precisely why we need **FRAMES**. All of the subsequent traversing commands now relate to the new workpiece coordinate system shifted using frames.

Using frames



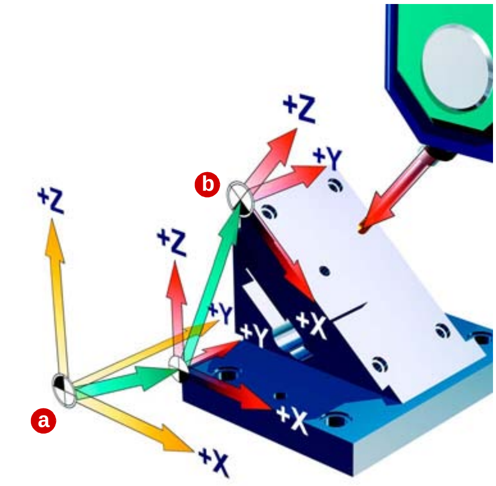
When the settable zero offset has been activated (G54, G55), the workpiece coordinate system is at the workpiece zero.

With the exception of special kinematics, the axes are now aligned parallel to the machine axes.

Using a **FRAME** this coordinate system can now be shifted and rotated anywhere in space.

CYCLE800 must have been installed on the control (only 840D) in order to align the machine axes to the workpiece axes using graphic dialog.

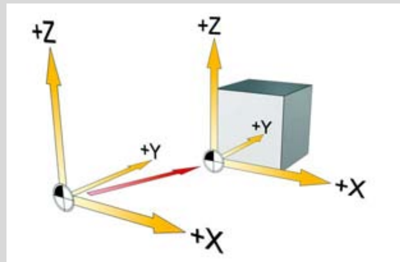
Example **Machining on an inclined plane**



In the example, using frames the coordinate system is first shifted from a to b in two steps and is then rotated to the inclined surface.

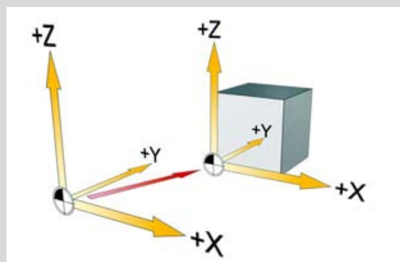
You now no longer have to take into account the inclination when programming. You program as usual, vertical to the workpiece surface and using the machining cycles - e.g. the drilling cycle.

Frames - programming components

<table>
<tr>
<td>Offset (coarse)</td>
<td>
</td>
<td>programmable with

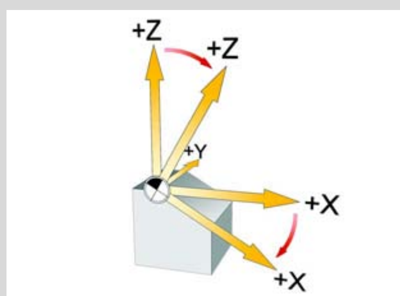
TRANS
ATRANS (additive translation com-ponent)
CTRANS (zero offset for multiple axes)

and

G58 (axial zero offset)
</td>
</tr>
<tr>
<td>Offset (fine)</td>
<td>
</td>
<td>programmable with

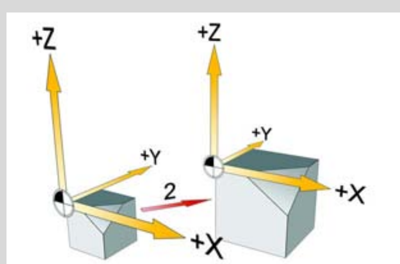
C-FINE

and

G59 (axial zero offset)
</td>
</tr>
<tr>
<td>Rotation</td>
<td>
</td>
<td>programmable with

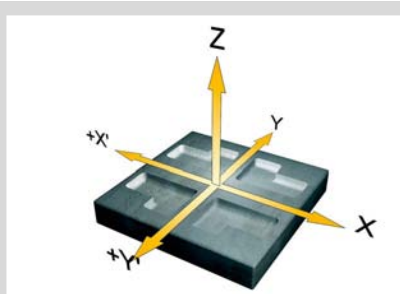
ROT / ROTS
AROT / AROTS

and

CROTS
</td>
</tr>
<tr>
<td>Scaling</td>
<td>
</td>
<td>programmable with

SCALE

and

ASCALE
</td>
</tr>
<tr>
<td>Mirroring</td>
<td>
</td>
<td>programmable with

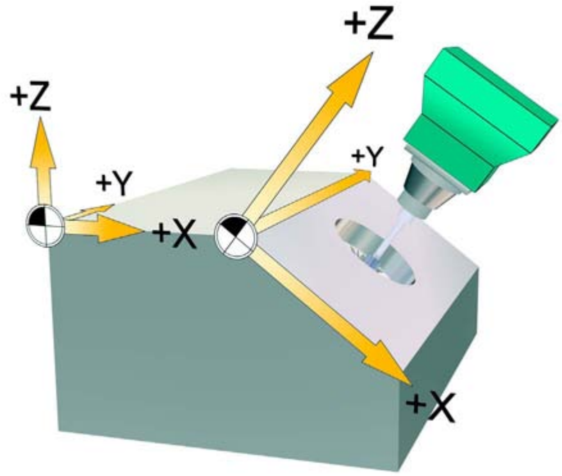
MIRROR

and

AMIRROR
</td>
</tr>
</table>

3.3 Swiveling - CYCLE800

Function

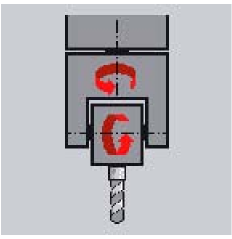
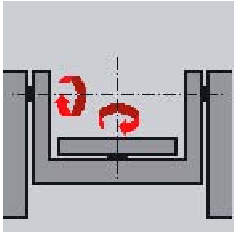
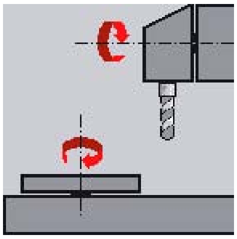


You can use swivel heads or swivel tables to set-up and machine inclined planes. Swiveling is possible in the JOG and AUTOMATIC modes. When parameterizing and/or programming swivel motion you are supported by transparent graphic displays.

You can program the swiveling axes of the machine (A,B,C) - or can simply specify the rotation around the geometry axes (X,Y,Z) of the workpiece coordinate system as described in the relevant workpiece drawing. The rotation of the workpiece coordinate system in the program is then automatically converted to a rotation of the relevant swiveling axes of the machine during machining.

The swivel axes are always rotated to place the machining plane perpendicular to the tool axis for machining. The machining plane remains fixed during machining.
When the coordinate system is swiveled the previously set zero offset is automatically converted for the swiveled state.

Machine kinematics

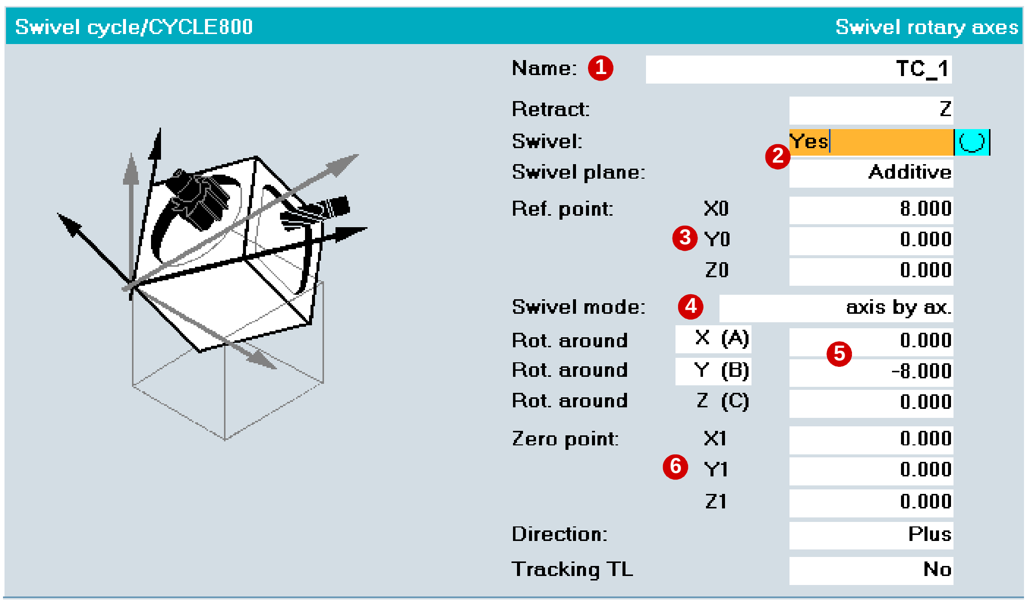
Swivel head (type T)	Swivel table (type P)	Swivel head + swivel table (type M)
Tool carrier that can be swiveled	Tool carrier that can be swiveled	Mixed kinematics
		

Procedure when programming swivel motion and subsequent machining:

- Swivel the coordinate system into the plane to be machined.
- Program machining as usual in the X/Y plane.
- Swivel the coordinate system back to its original position.

Basic procedure when generating a swivel data set

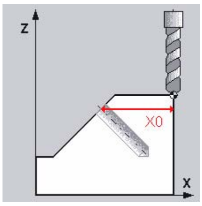
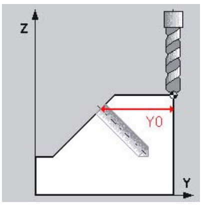
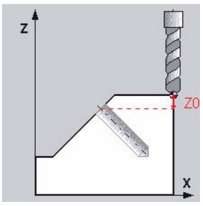
- Call the swivel function in the program.
- Select the name of the swivel data set 1 .
- Select yes for swivel if you wish to make a swivel movement. Select new as swivel movement if you wish to make a new swivel movement, or additive if you wish to base the movement on a previous swivel movement 2 .
- Specify the reference point before rotation (X0, Y0, Z0) 3 .
- Select the axis by axis swivel mode, directly, using the projection angle or the space angle 4 .
- Enter the angle through which the axis should swivel. In the axis-by-axis mode you can enter the angle for each axis 5 .
- Enter the zero point after the rotation 6 .

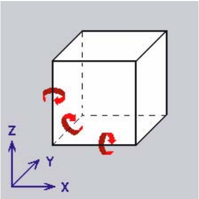
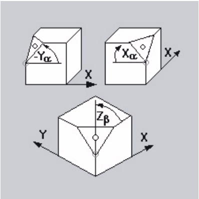
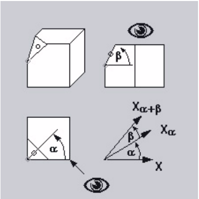
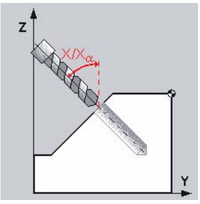
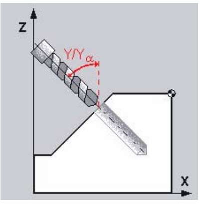
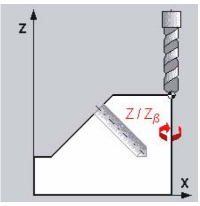
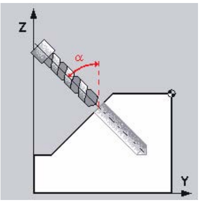
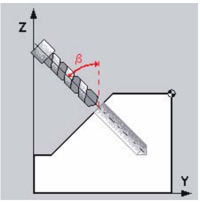
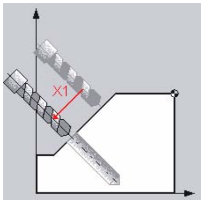
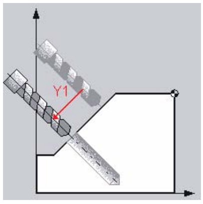
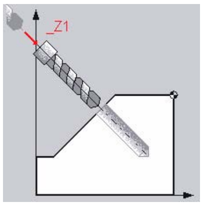


NOTE

You can program several swivel movements one after the other. A subsequent swivel motion can be based on a previous one (additive). This means that you can transparently represent swivel motion in the program code.

Parameters of the input screen

Name of swivel data record _TC	The swivel data records that have been set-up can be selected (toggle). Every swivel data record is assigned a name. A name does not have to be declared if there is only one swivel data record. "0" → De-select swivel data record.
Retraction _FR (prior to swiveling the rotary axis)	■ Do not retract ■ Retract Z axis ■ Retract axis Z, XY ■ Maximum retraction in tool direction (with Cycles SW 6.5 and higher) ■ Incremental retraction in tool direction (with Cycles SW 6.5 and higher) The incremental value for the traversing path in the tool direction must be entered in the input field. The retraction positions can be entered in the CYCLE800 startup menu.
Swivel, direction _DIR	■ **Swivel, yes** → Rotary axes are positioned or are manually rotated into position ■ **Swiveling, no** (only calculated) → Rotary axes are not traversed, e.g. auxiliary swivel plane according to the workpiece drawing ■ **Minus/plus direction** When selecting the traversing direction the reference is rotary axis 1 or 2. The NCU can calculate two possible solutions as a result of the angular range of the rotary axes of the machine kinematics. Usually, one of these solutions is technologically suitable. The rotary axis (1st or 2nd rotary axis) that is to be used as reference for the two solutions is selected in the CYCLE800 startup menu. Selecting the "minus" or "plus" direction determines which of the two possible solutions is to be applied. Observe the machine manufacturer's instructions!
Swivel plane _ST	■ **New** Previous frames are deleted -> defined values -> swivel frame. ■ **Additive** The swivel frame is based additively on the swivel frame, programmed active frames (e.g. AROT ATRANS) and the currently effective rotation in the ZO are taken into consideration.
Reference points before rotation X0, Y0, Z0	Help displays refer to machining plane G17 (tool axis Z).   

Swivel mode_MODE	   **Axis by axis** rotation around the individual axes of the coordinate system, **Projection angle** the angle of the swiveled surface is projected onto the first two axes of the coordinate system. **Angle in space** rotation first around the Z axis and then around the Y axis The axis sequence can be freely selected.
Rotation around A, B, C	   Rotation (axis by axis, projection angle)   Rotation (angle in space)
Zero point after rotation X1, Y1, Z 1	  
Correct tool	■ **Yes** When swiveling to a machining plane, the linear axes can be corrected to prevent the risk of collision. (Beforehand.:TRAORI and TOOLCARR.SPF have been adapted) ■ **No** Linear axes are not corrected when swiveling.

Observe the machine manufacturer’s instructions. You can set the available parameters in the CYCLE800 startup menu.

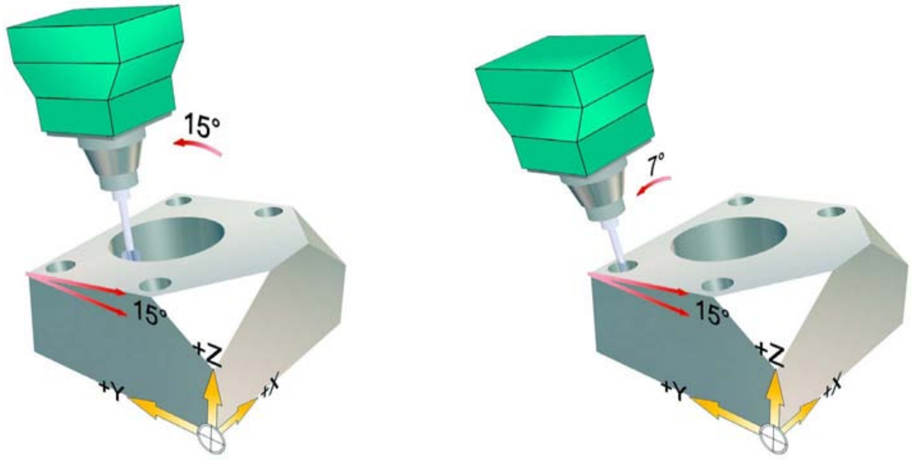
3.4 Programming example - swiveling

Standard milling and drilling cycles are applied to swiveled machining surfaces in the following example.

Workpiece

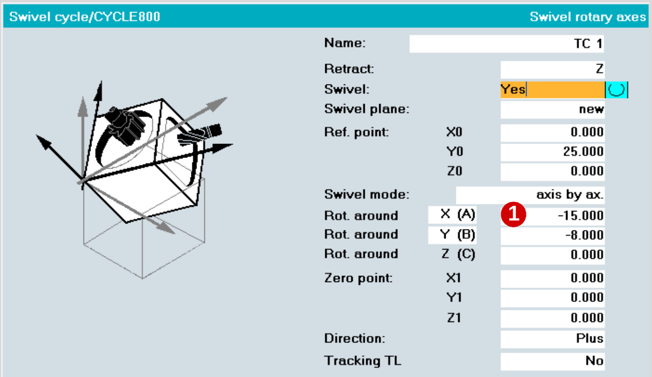
Task description

Face milling the workpiece.
Swivel the machining plane through X=-15 Degrees and mill a circular pocket.
Swivel through Y=-8 Degrees and machine four holes under this angle.

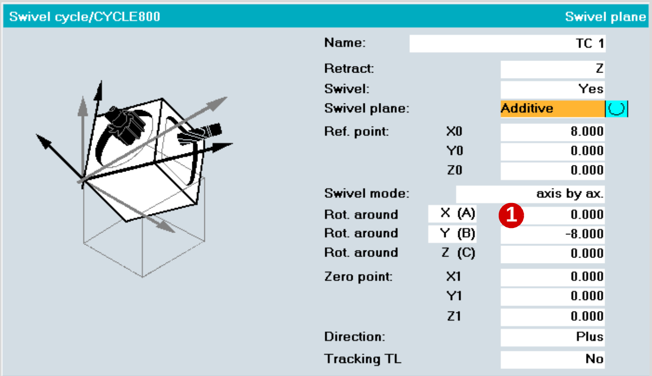


SWIVEL_ALU.spf (SCHWENK_ALU.spf)

```
N1 T10 D1 ;miller D=10mm
; define ZO
N3 G54
; Swivel to the normal position
N4 CYCLE800(1,"",0,57,0,0,0,0,0,0,0,0,0,-1)
N5 M3 S8000 M8
;Face mill the workpiece at the normal position
N6 CYCLE71(50,2,1,0,0,0,70,30,0,2,8,2,0,1000,31,2)
; Swivel the machining plane through X=-15 Degrees  1
N7 CYCLE800(1,"",0,57,0,25,0,-15,0,0,0,0,0,-1)
```




```
; Face milling the swiveled plane
N8 CYCLE71(50,10,1,0,0,0,75,54,0,3,8,2,0,1200,31,2)
N10 ;T="MILL_10mm"
N11 ;M6
N12 ;M3 S8000
; mill a circular pocket at the swiveled plane using the circular pocket cycle
N13 POCKET4(50,0,1,-10,16,0,0,4,0.5,0.5,1000,400,0,11,,,,,)
N14 POCKET4(50,0,1,-10,16,0,0,20,0,0,1000,600,0,12,,,,,)
; Swivel the machining plane through Y=-8 Degrees  1
N15 CYCLE800(1,"",1,57,8,0,0,0,-8,0,0,0,0,-1)
```




```
;Drill the holes inclined by -8 Degrees
N16 MCALL CYCLE82(50,2,1,,6,0)
N17 HOLES1(0,0,90,8,30,2)
N18 MCALL
…
```

3.5 High Speed Settings – CYCLE832

To simplify programming and ensure that the program is transparent SINUMERIK 840D offers the CYCLE832. This includes the most important functions when milling freeform surfaces. Further, using CYCLE832 the machine operator can more simply influence the program.

Programming

CYCLE832(_TOL,_TOLM)	Programming the cycle
CYCLE832()	Abbreviated program call. Corresponds to selecting the "Machining" "Deselection" input screen.
CYCLE832(0.01)	Abbreviated program call. Enter the tolerance value. The active G commands are not changed in the cycle.

Explanation of the parameters

Parame-ters	Data type	Meaning
_TOL	real	Tolerance, machining axes → units: mm/inch;Degrees
_TOLM	integer	Tolerance mode 7 6 5 4 3 2 1 0 Decimal places Bit 0: 0: De-select 1: Finishing (default)1) 2: Pre-finishing 3: Roughing -- Bit 1: 0: 1: -- Bit 2: only 5-axis transformation -- Bit 3: 0: G64 1: G641 2: G642 (default) [a] -- Bit 4: 0: FFWOF SOFT (default) [a] 1: FFWON SOFT 2: FFWOF BRISK -- Bit 5: 0: COMPOF (default) [a] 1: COMPCAD 3: B spline -- Bit 6: reserved -- Bit 7: reserved --

a. The machinery construction OEM can change the setting.

Decimal point				
	0	**Tolerance_tol.** This refers to the tolerance of axes involved in machining. The tolerance value is effective for G642 and for COMPCAD. If the machining axis is a rotary axis, the tolerance value is written with a factor (default factor = 8) to MD 33100: COMPRESS_POS:_TOL (AX) of the rotary axis. For G641, the tolerance value corresponds to the ADIS value. When first entered, the tolerance is pre-set to the following values:		
		0: Deselection	0.1 (linear axes)	0.1 Degrees (rotary axes)
		1: Finishing	0.01 (linear axes)	0.08 Degrees (rotary axes)
		2: Pre-finishing	0.05 (linear axes)	0.4 Degrees (rotary axes)
		3: Roughing	0.1 (linear axes	0.8 Degrees (rotary axes)
			5-axis transformation must be set-up by the machinery construction OEM if the tolerance value should also be effective on the rotary axes.	
	1	**No function**		
	2	**Only 5-axis transformation**		
	3	**Continuous path control (_TOLM)**		
		0: G64	default)	
		1: G641	Blending with ADIS, ADISPOS	
		2: G642	Blending with single-axis tolerance With an NC block compressor with COMPCAD, G642 is permanently selected.	
	4	**Compression, NC block compressor (_TOLM)**		
		0: FFWON SOFT	With feedforward control, with jerk limiting	
		1: FFWOF SOFT	Without feedforward control, with jerk limiting	
		2: FFWOF BRISK	Without feedforward control, without jerk limiting	
	5	**Compression, NC block compressor (_TOLM)**		
		0: none (COMPOF)	(default)	
		1: COMPCAD		
		3: B SPLINE		
		The selection of feedforward control (FFWON) and jerk limitation (SOFT) requires that the machine manufacturer has optimized the control or the machining axes.		

NOTE

To use the functions listed here requires that the machinery construction OEM has correctly optimized the CNC-machine.

Compressor - COMPCAD

The compressor is ideally called in CYCLE832. If it is to be separately programmed then proceed as described below.

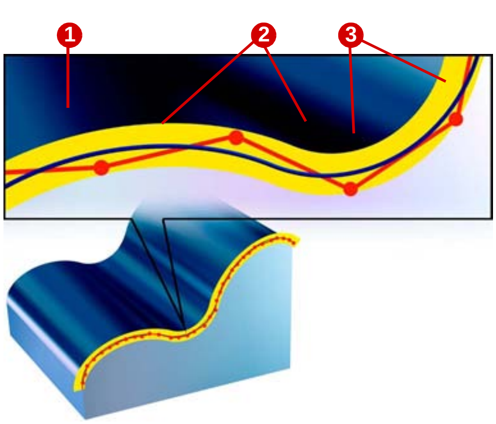
Programming

COMPOF
COMPCAD

Explanation of the commands

COMPOF	Compressor off
COMPCAD	Compressor on - surface quality and speed are further optimized. COMPCAD smoothes the points along the characteristic before approximation (B spline) and offers, at a high path velocity, the highest degree of precision with transitions that have a constant acceleration rate (the compression rate is unlimited, however the max. path length is 5 mm) Preferably used to mill **freeform surfaces (recommended)**

Mode of operation of the spline compressor



According to the specified tolerance band ❶ the compressor combines a sequence of G1 commands ❷ and compresses them into a spline ❸, which can be directly executed by the control. A new contour is created whose characteristic lies within the specified tolerance envelope.

This makes the surface much smoother since the machine axes can move more harmoniously and machine resonance is avoided.

In turn higher traversing speeds are possible and the stress on the machine is reduced.

NOTE

If the High Speed Setting CYCLE832 is not available you can still use the compressor functionality. You only have to formulate the necessary machine data and G codes in the program. Refer to the following example and the example in Chapter 3.8.

Program example for COMPCAD using a subroutine

COMPCAD can be elegantly called using a sub-routine. Normally, a specific tolerance must be specified for all axes for each machining operation (roughing, finishing). In the subroutine the tolerance value is defined as variable and the actual tolerance value is transferred when called.

Call the subroutine in the main program with tolerance transfer

```
;*****************************************************************
;Program call in the main program
;*****************************************************************
...
N40....
N45 TOL(0.015)
N50...
```

Subroutine that sets the tolerance value for the axles

```
;*****************************************************************
;Technology program for 3-axis HSC-MILLING
;*****************************************************************
PROC TOL(real tolerance)

N20 SOFT
N30 COMPCAD
N30 G642
N40 $MA_COMPRESS_POS_TOL[X]=tolerance
N50 $MA_COMPRESS_POS_TOL[Y]=tolerance
N60 $MA_COMPRESS_POS_TOL[Z]=tolerance
N70 NEWCONF
M17
```

Continuous-path mode, Look ahead – G64, G642

If the continuous-path mode is called in CYCLE832 - for G641 the ADIS value corresponds to tolerance value TOL_. If you are programming without CYCLE832, then also specify the ADIS value.

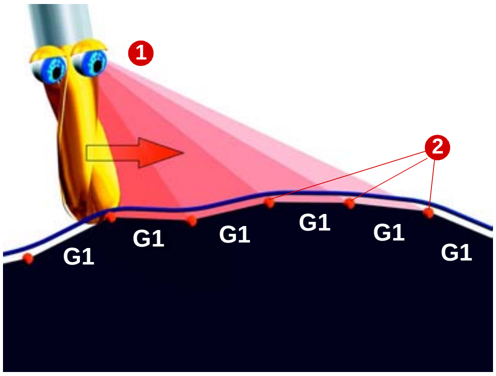
Programming the retraction clearance using ADIS

G64
G642 ADIS=… or ADISPOS=…

Explanation of the commands

G64	Continuous-path mode – **Look ahead** where the axis only brakes at corners
G642	Blending with axial tolerance **(recommended)** **Look ahead** with additional **grinding at the corners** corresponding to MD 33100 (machine data) The following applies to G642: There are 2 possibilities to specify the tolerance 1. Individual axes are specified – refer to the programming example on the previous Page or 2. Program the retraction clearance using ADIS Preferably when milling **freeform surfaces**
ADIS=	Blending clearance for path functions G1, G2, G3
ADISPOS=	Blending clearance for rapid traverse G0 (not suitable for freeform surfaces)

Using G64, G642



The objective of the continuous-path mode is to increase the speed and harmonize the traversing behavior. For the continuous-path function G64 etc. this is implemented using two functions.

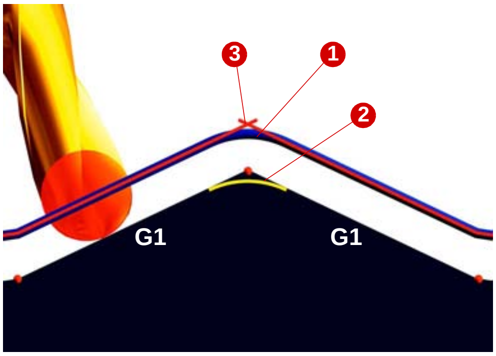
Look ahead - look-ahead speed control 1

The control calculates several NC blocks ahead and determines a modal speed profile. The way in which this speed control is calculated can be set using the functions G64 etc.

Corner rounding 2

The look ahead function also means that the control system is able to round the corners that it detects. The programmed corner points are therefore not approached exactly. Sharp corners are rounded.

These two functions mean that the contour is created with a uniform path velocity profile. This results in improved cutting conditions, increases the surface quality and reduces the machining time.



To round sharp corners 3 the continuous-path command **G642** forms transition elements 1, 2 at the block boundaries. The continuous-path commands differ in the way that they form these transition elements.

With G642 you can define the degree of rounding 2 using the ADIS value.

G642 inserts transition polynomials with constant curvature. These avoid acceleration jumps at the block boundaries. We recommend G642 for mold making applications.

Feedforward control and jerk limiting – FFWON, SOFT, ...

Feedforward control and jerk limiting can only be called in CYCLE832 as a combination of the two functions. This is because this combination offers ideal conditions for freeform surface milling. Of course, both of these functions can also be separately programmed.

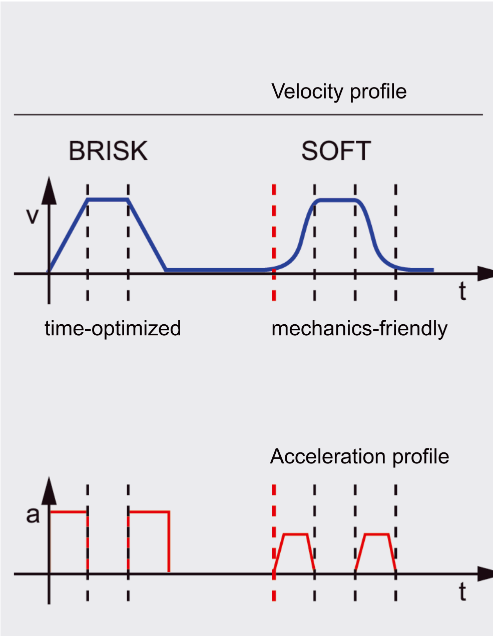
Programming

FFWON/
FFWOF
BRISK
SOFT

Explanation of the commands

FFWON	Feedforward control "on"
FFWOF	Feedforward control "off"
BRISK	**Without jerk limiting** Path axes accelerate abruptly
SOFT	**With jerk limiting** Jerk limited acceleration of the path axes Axial jerk limiting (maximum jerk in the machine data JOG_AND_PS_MAX_JERK (jog and positioning) or MAX_AX_JERK (continuous-path mode)

Jerk limiting function



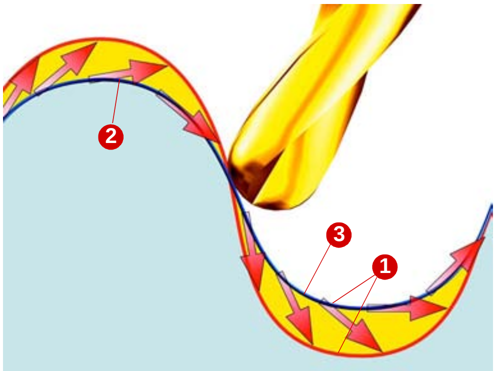
To make acceleration as gentle on the machine as possible the acceleration profile of the axes can be influenced using the commands **Soft**, **Brisk**. If **Soft** is activated, the acceleration behavior does not change abruptly but in the form of a linear characteristic. This reduces the stress on the machine. It also has a positive impact on the surface quality of workpieces, since machine resonance effects are excited far less frequently.

BRISK:
acceleration behavior: The path axes accelerate abruptly according to the machine data that has been set

The axis slides travel with maximum acceleration until the feedrate is reached. BRISK enables time-optimized machining, but with jumps in the acceleration curve.

SOFT:
Acceleration behavior: Jerk-limited (soft) acceleration of the path axes

The axis slides travel with constant acceleration until the feedrate is reached. As a result of the jerk-free acceleration characteristic **SOFT** permits a higher path accuracy and less stress on the machine.



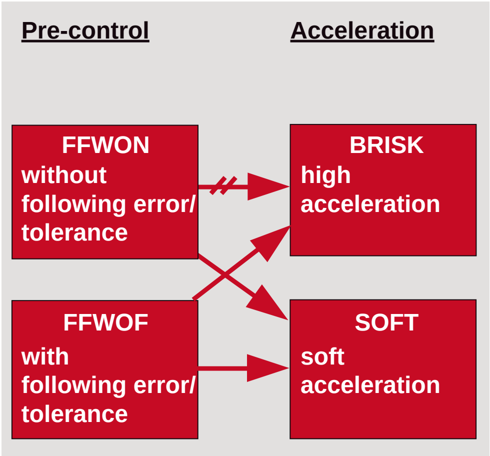
Feedforward control

Following errors cause contour violation 1. The inertia in the system means that the tool tends to leave the setpoint contour 2 tangentially, i.e. the actual contour 3 that is produced deviates from the setpoint contour. A following error is a combination of the system (position control) and the speed.

Feedforward control **FFWON** reduces the speed-dependent following error when contouring towards zero. Traversing with feedforward control permits higher path accuracy and thus improved machining results.

Recommendations

CYCLE832 includes the following combinations:



FFWON SOFT
The emphasis is on a high path accuracy. This is achieved using a soft speed control that is essentially free from following errors.

FFWOF SOFT
The priority is not on achieving a high path accuracy.
Additional rounding is achieved by means of following errors. For use with older part programs/machines.

FFWON BRISK
not recommended

FFWOF BRISK
For use when roughing and when maximum speed is required.

3.6 Feedrate profile – FNORM, FLIN

Programming

F… FNORM
F… FLIN

Explanation of the commands

FNORM	Basic setting. The feedrate value is specified as a function of the traversing path of the block and is then valid as a modal value.
FLIN	Linear path velocity profile: The feedrate is traversed linearly from the actual value at the beginning of the block to the end of the block and is subsequently valid as modal value.

Function

What is a feedrate profile?

To allow the feedrate profile to be more flexibly programmed, linear and cubic characteristics have been added to the feedrate programming in accordance with DIN 66025. The cubic characteristics can be programmed either directly or as interpolating splines. These additional characteristics make it possible to program continuously smooth speed characteristics depending on the curvature of the workpiece to be machined.

These speed characteristics make it possible to program limiting acceleration changes therefore producing uniform workpiece surfaces.

3.7 Programming example with CYLCE832

Knight chess piece

Introduction

Important G codes and machine data are automatically set and harmonized with one another for high-speed machining using CYCLE832. You only have to call CYCLE832 in the program for the appropriate machining (roughing, finishing) - therefore allowing you to work under the optimum process conditions.

Workpiece

A knight chess piece is to be milled. The machining sequence comprises a roughing operation followed by a finishing operation. A separate program is used for each machining operation.

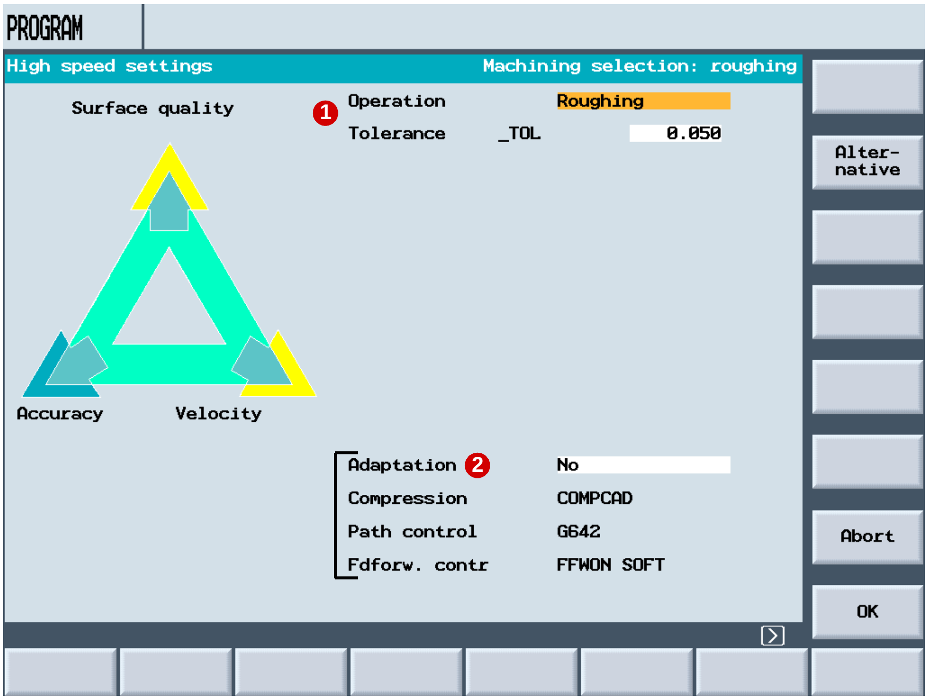
Program for roughing

```
N1 T1 D1
N2 M6
N3 G54 D1
N4 S4500 M3
N5 MSG( "Roughing end mill no arcs, Chord 0.05" )
N6 MSG( "End Mill 8mm" )
N7 CYCLE832(0.05,112003)        ; Roughing with tolerance 0.05, refer to another Chapter
N8 G0 X-51.027 Y-60.935
N9 G0 X-51.027 Y-60.935 Z8.1
N10 G0 Z3.15
N11 G1 Z-1.85 F1000             ; G1 blocks of the geometry
N12 G1 X-50.131 Y-52.985 F2500
N13…
```

CYCLE832 for roughing

You can use the High Speed Settings dialog box to make the settings for CYCLE832.

- To do this, select the "High Speed Settings" softkey in the Program Editor.
- To start, select roughing and the tolerance - 0.050 in the example 1.
- Under Adaptation select whether you wish to accept the standard setting for additional parameters, or you wish to explicitly set other values, e.g. for the compressor. Generally, another adaptation is not required 2
  You can only change the parameters below if the machinery construction OEM has released this function.
- Acknowledge the dialog box with OK. The settings for CYCLE832 are accepted.



Program for finishing

```
N1 T2 D1
N2 M6
N3 G54 D1
N4 S10500 M3
N5 MSG( "Finishing Chord 0.005" )
N6 MSG( "Ball Mill D3" )
N7 CYCLE832(0.005,112001)      ; Finishing with tolerance 0.005, refer to another Chapter
N8 G0 X26.499 Y-12.096
N9 G0 X26.499 Y-12.096 Z10.
N10 G0 Z-11.
N11 G1 Z-16. F500              ; G1 blocks of the geometry
N12 G1 Y-12.079 Z-15.666 F2000
N13…
```

CYCLE832 for finishing

You can also use the High Speed Settings dialog box for the finishing settings.

- ▶ To do this, select the "High Speed Settings" softkey in the Program Editor.
- ▶ Select finishing and the tolerance, this is 0.0050 in the example 1 .
- ▶ You can make additional settings under Adaptation 2 .
- ▶ Acknowledge the dialog box with OK. The settings for CYCLE832 are accepted.

3.8 Programming example without CYLCE832

Knight chess piece

Introduction

Important G codes and machine data are automatically set and harmonized with one another for high-speed machining using CYCLE832. You can also use the advantage of high-speed machining even on controls without CYCLE832 support (e.g. 802D sl) - without having to use CYCLE832. You must then formulate the necessary machine data and program the appropriate commands in the program.
The procedure will be shown using the machining of a knight chess piece as example
.

Workpiece

A knight chess piece is to be milled. The machining sequence comprises a roughing operation followed by a finishing operation. One program is used for each machining operation.

Program for roughing

```
N1 T1 D1
N2 M6
N3 G54 D1
N4 S4500 M3
N5 MSG("Roughing with compressor tolerance 0.05")
N8 SOFT                 ; Switch-in the jerk-limited path acceleration, therefore
                        ; improved surface
N9 COMPCAD              ; Switch-in the compressor for surface optimization
N10 G642                ; Blending of the contour transitions
N11 $MA_COMPRESS_POS_TOL[X]= 0.05   ; Set the compressor tolerance Axes x, y, z
N12 $MA_COMPRESS_POS_TOL[Y]= 0.05   ; The tolerance value should be approx. 10% to
N13 $MA_COMPRESS_POS_TOL[Z]= 0.05   ; 20% above the Calculation tolerance of the
                                    ; CAM system
N14 G0 X50.899 Y-57.933             ; positioning in rapid traverse and programming
                                    ; Contour for finishing
N15 G0 X50.899 Y-57.933 Z10.15
N16 G0 Z5.15
N17 G1 Z0.15. F2500
N18 G1 X49.986 Y-51
N19 …
```

Program for finishing

```
N1 T2 D1
N2 M6
N3 G54 D1
N4 S+4500 M3
N5 MSG( "Finishing with compressor tolerance 0.005" )
N8 SOFT                                  ; Comments - refer to roughing
N9 COMPCAD
N10 G642
N11 $MA_COMPRESS_POS_TOL[X]= 0.005       ; Sets the compressor data for  axes x, y, z
N12 $MA_COMPRESS_POS_TOL[Y]= 0.006       ; The tolerance value should be approx. 10%
N13 $MA_COMPRESS_POS_TOL[Z]= 0.006       ; to 20% above the Calculation tolerance of
                                         ; the CAM system
N14 G0 X26.499 Y-12.096                  ; Positioning in rapid traverse and
                                         ; Programming Contour for finishing
N15 G0 X26.499 Y-12.096 Z10.
N16 G0 Z-11.
N17 G1 Z-16. F500
N18 G1 Y-12.079 Z-15.666 F2000
N19 …
```

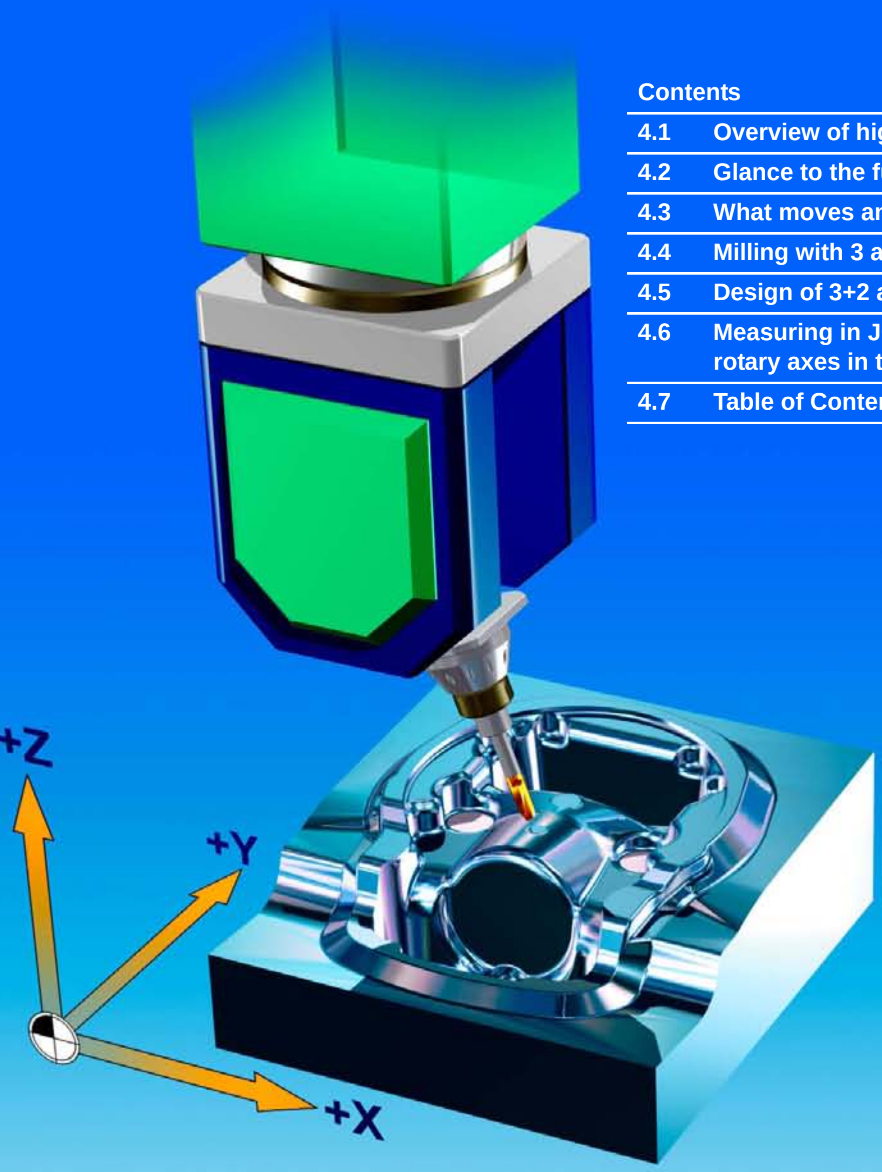
NOTE

If you have more than two machining operations, generally you only have to adapt the compressor tolerance in order to achieve a better surface quality. The other settings can be accepted.

NOTE

You can also define the setting for the compressor tolerance (COMPCAD) using a separate subroutine. The tolerance value is then transferred as variable. Also refer to the example in Chapter 3.5 on COMPCAD.

Reference Information



4

4.1 Overview of higher-order functions

Higher-order functions of the 840D are summarized on the following pages. This provides you with an overview of the commands that go beyond the requirements laid down in DIN 66025 and that facilitate significant improvements in the area of 3-axis mold making.

Traversing commands

Language elements with circular interpolation programming

TURN	Number of full circles to be traversed G3 X… Y… I… J… TURN = Additional parameters:
CR=	Circle radius
I1, J1, K1	Intermediate points in Cartesian coordinates (in X, Y, Z direction)

Compressor

COMPCAD	Surface-optimized compressor (constant acceleration) With the corresponding single-axis tolerances: **$MA_COMPRESS_POS_TOL[X] = ...** also refer to CYCLE832

Technology G groups

DYNNORM	Standard dynamics, as previously
DYNPOS	Positioning mode, tapping
DYNROUGH	Roughing
DYNSEMIFIN	Finishing
DYNFINISH	Smooth-finishing

Dynamic response

Look Ahead

G60	**Exact stop** at block end
G601	Block change on reaching the fine in-position positioning window
G602	Block change on reaching the coarse in-position window
G603	Block change at the end of interpolation
G64	**Overrun of block end (LOOK AHEAD)**
	Blending
G641	ADIS = … blending clearance ADISPOS =… blending clearance for G0, constant speed
G642	Blending with single-axis tolerances ($MA_COMPRESS_POS_TOL[X] = …) or ADIS, ADISPOS with intermediate blocks, constant-acceleration **(recommended)**
G60, G64, G641, G642	G code group 10
G601 – G603	Internal G code group (group 12)

Speed programming

	Conventional blockwise (non-modal) speed programming in
G94	inches/min or mm/min
G93	inverse time
G95	inches, mm per spindle revolution
G96	Constant cutting rate

Jerk

SOFT	Jerk limiting
BRISK	Acceleration limiting

Feedforward control

FFWON	Feedforward control on
FWOF	Feedforward control off

Tool radius compensation

G40	Deactivation of all variants
G41	Activation for circumferential milling, compensation direction left
G42	Activation for circumferential milling, compensation direction right
G450	Circles at external corners (all compensation types)
G451	Intersection traversing at external corners (all compensation types)

2½-D

CUT2D	2 1/2-D COMPENSATION with compensation plane determined using G17 - G19
CUT2DF	2 1/2-D COMPENSATION with compensation plane determined using a frame

FRAMES

Programmable frames

TRANS X… Y… Z…	Absolute offset
ATRANS X… Y… Z…	Incremental offset, relative to the currently active frame
ROT X… Y… Z…	Absolute rotation
AROT X… Y… Z…	Incremental rotation, relative to the currently active frame
ROTS X… Y…	Absolute rotation, defined by two angles. The angles are the angles of the lines of intersection of the inclined plane with the main planes with respect to the axes.
AROTS X… Y…	Incremental rotation, relative to the currently active frame, angles as for ROTS
RPL=…	Rotation in the plane
MIRROR X… Y… Z…	Absolute mirroring
AMIRROR X… Y… Z…	Incremental mirroring, relative to the currently active frame
SCALE X… Y… Z…	Absolute scaling
ASCALE X… Y… Z…	Incremental scaling, relative to the currently active frame

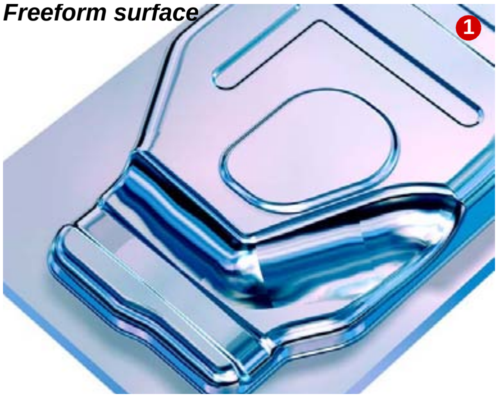
Frame operators

	Frame operators can be used to define frame variables as a chain of individual frame types:
CTRANS (X… Y… Z…)	Absolute offset
CROT (X… Y… Z…)	Absolute rotation
CROTS (X… Y… Z…)	Absolute rotation
CMIRROR (X… Y… Z…)	Absolute mirroring
CSCALE (X… Y… Z…)	Absolute scaling

FRAME = CTRANS(…) : CROT (X… Y… Z…) : CMIRROR (X… Y… Z…)

4.2 Glance to the future, 5-axis machining

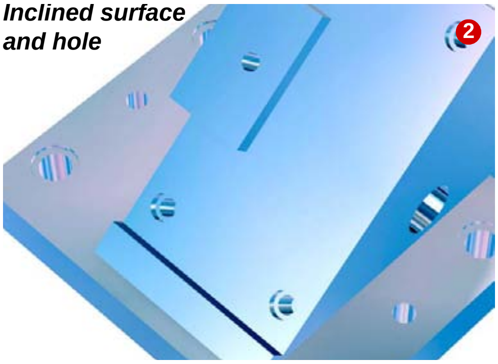
Freeform surface



The requirements placed on shapes, surface quality and machining speed when cutting – especially in tool and mold making – are rapidly increasing.

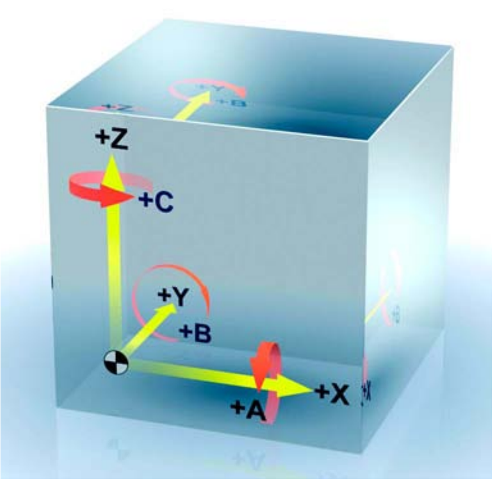
In order to achieve optimum cutting conditions when machining curved surfaces 1 in space, ...

Inclined surface and hole



... and to machine any geometries in space 2 (here, the angle of inclination of the tool axis must be able to be changed) ...

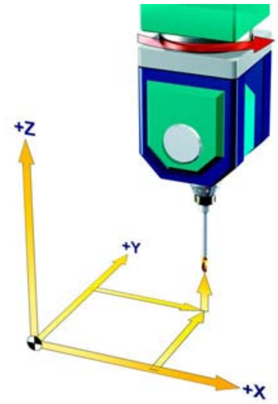
3 + 2 axes



To do this, in addition to the 3 linear axes X, Y and Z, 2 rotary axes A, B or C are required.

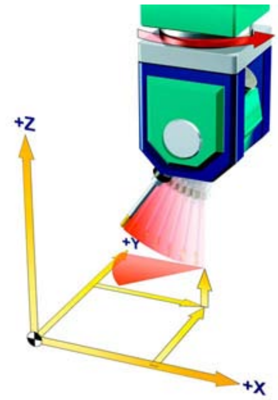
4.3 What moves and how?

For certain tasks, it is sufficient to work with a fixed orientation, e.g. position the tool on an inclined plane. State-of-the-art 5-axis control systems such as the SINUMERIK 840D allow you to program elements such as inclined holes and pockets with a fixed tool orientation directly at the machine – they also allow you to influence the essential machining parameters for programs from CAM systems.



Machine motion

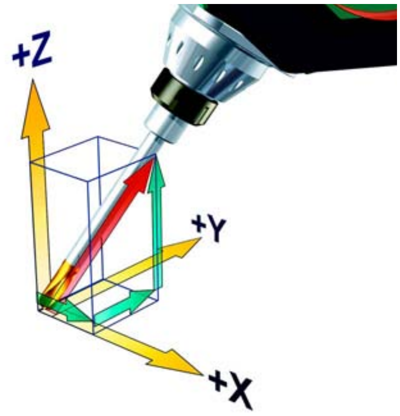
A tool position is approached in the working area using linear axes X, Y and Z.



The inclination of the tool and the tool orientation are changed using 2 rotary axes – e.g. B and C.

Using 3 linear axes and 2 rotary axes, theoretically any point in space can be approached with the required tool orientation.

Direction vector of the tool orientation

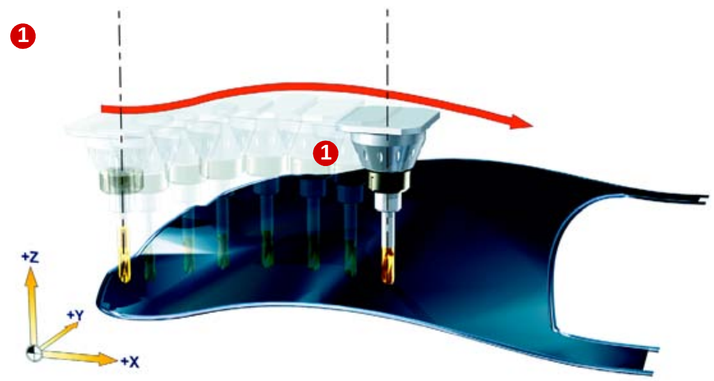


CNC programming

Coordinate axes X, Y and Z are used to describe a set position in the NC program. To describe the tool orientation, we recommend that direction vectors A3, B3, C3 are preferably used in order to program the orientation independently of the kinematics.

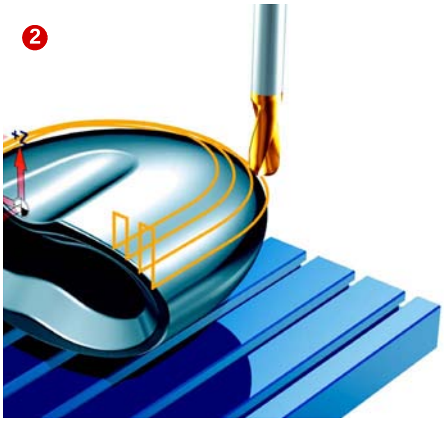
4.4 Milling with 3 axes or 3 + 2 axes?

It is especially uniform, convex curved freeform surfaces that are usually machined using 3 controlled axes. However, 5 controlled axes are required for deep cavities or frequent curvature changes.



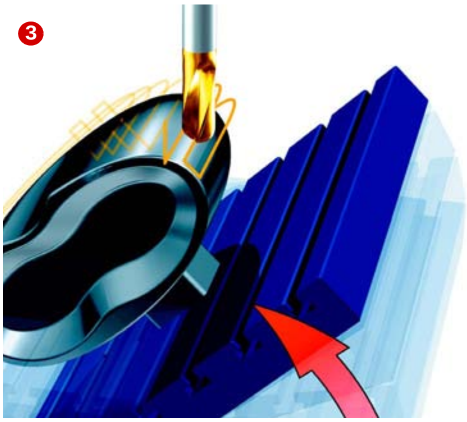
3 axes
Controlled path axes X, Y, Z

The miller orientation does not change along the complete milling path. The cutting conditions at the tip of the tool are never optimum.



3 + 2 axes
Controlled path axes X, Y, Z
Fixed rotary axes, e.g. A, C (table)

For these machine tools, the orientation of the tool or the position of the table can be modified, e.g. by **retrofitting**.
In this case, the milling tool cuts under the optimum conditions. The cutting conditions deteriorate the further that the tool moves to the top or to the side on the workpiece.



3 + 2 axes
Controlled path axes X, Y, Z
Fixed rotary axes, e.g. A, C (table) swiveled

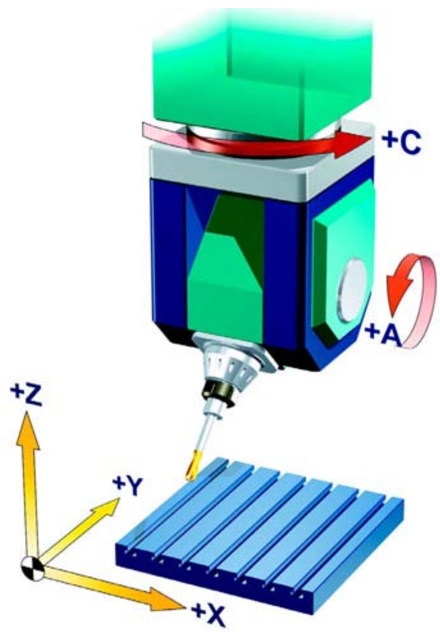
To also obtain the optimum cutting conditions here, the table is swiveled. In order to completely machine a freeform surface, it is often necessary to swivel a multiple number of times in various directions.

4.5 Design of 3+2 axis milling machines

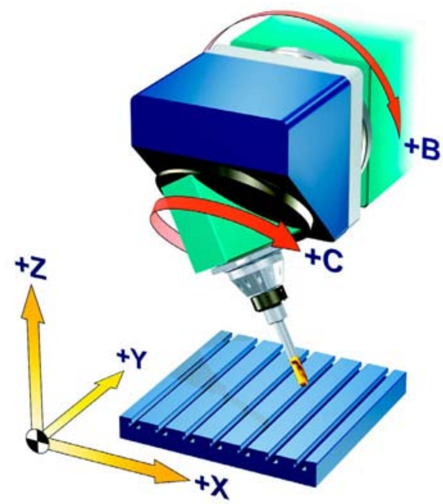
A 5-axis machine can control tool motion in 5 axes. These are the 3 known linear axes and an additional 2 rotary axes. There are different kinematic solutions for the two rotary axes. We will now schematically show the most usual solutions.

2 rotary axes in the head

1 **Fork**



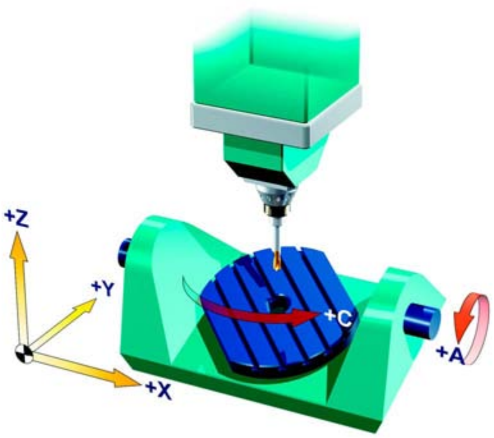
2 **nutated ***



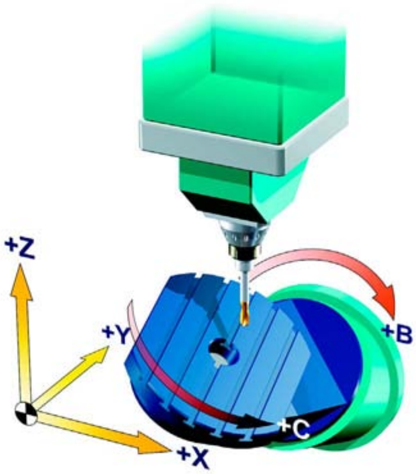
* Term:
If the rotary axis is not perpendicular to a linear axis, then this is known as a "nutated" axis.

2 rotary axes in the table

❸ **Rotate/swivel**



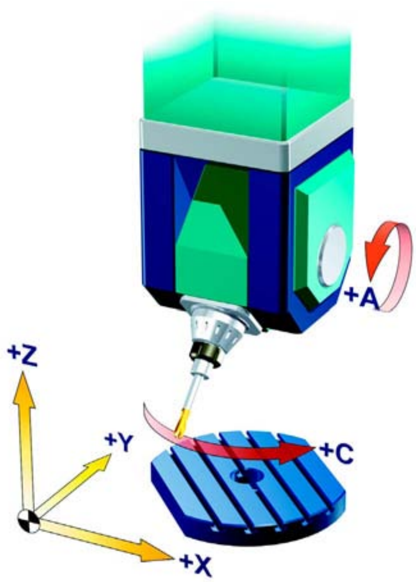
❹ **nutated ***



* Term:
If the rotary axis is not perpendicular to a linear axis, then this is known as a "nutated" axis.

2 rotary axis in the head / 1 rotary axis in the table

❺



4.6 Measuring in JOG - setting-up a workpiece with two rotary axes in the table

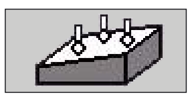
Task description

Example For machines that in addition to the table rotary axis (C axis) also have an A or a B axis and are therefore in a position to 3-dimensionally align the tool - then it is possible to correct the work-piece surface (machining surface) so that it assumes a horizontal position.

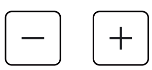
This can be implemented using the combination of two functions – "align plane" and "swivel" in the JOG mode.

Aligning the plane

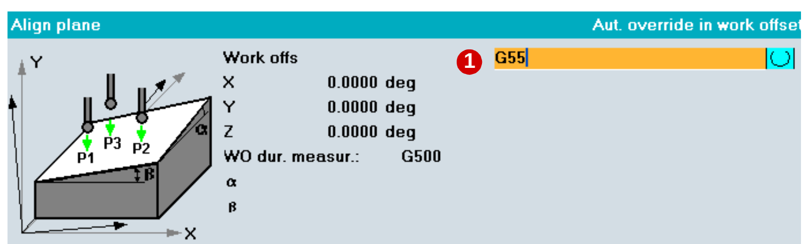
To start, align the plane.



Call "Align plane".



Traverse the probe to measuring point P1.



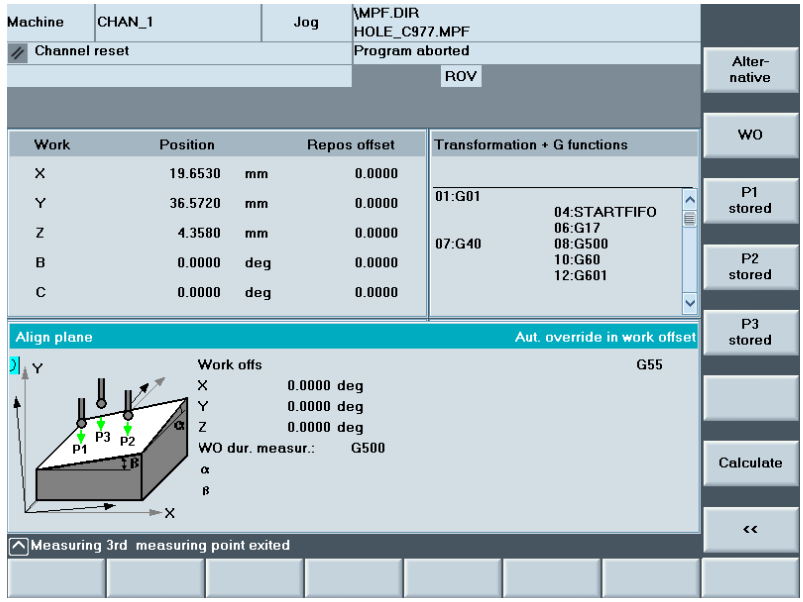
Enter details in the input screen:

1 Select the zero offset e.g. G54, G55, G56 or G57. G54 was used in the example.

With "NC-Start" measuring points P1, P2 and P3 are automatically approached starting from the manually selected preliminary position. This means that the probe approaches the workpiece, is triggered and then retracts to the starting position.

Press "Calculate" after all of the points have been approached.

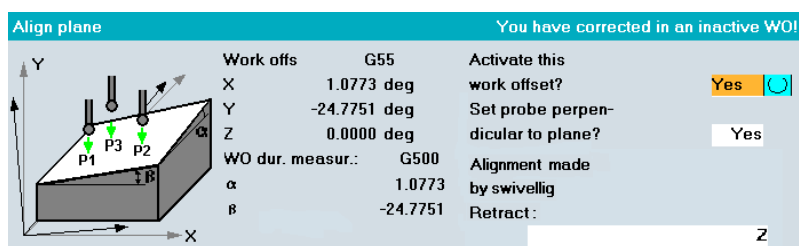
Calculating



Result

The plane is horizontally aligned. If swivel has been set-up on the machine, you can now immediately align the workpiece to the axes. The table or the head is aligned corresponding to the correction values.

If swivel has not been set-up, you can align the probe so that it is perpendicular to the measured plane. The correction is then only made in the coordinate axes without any visible swivel motion of the table or head.



After the plane has been aligned, you only have to determine the ZO for for X, Y and Z. Proceed as described in Chapter 2.3 and 2.4.

4.7 Index

T

U

W

Z

Siemens AG
Automation and Drive Technology
Motion Control Systems
Postfach 3180
91050 Erlangen
GERMANY

www.siemens.com/automation/mc

Order No.: 6FC5095-0AB20-0BP0