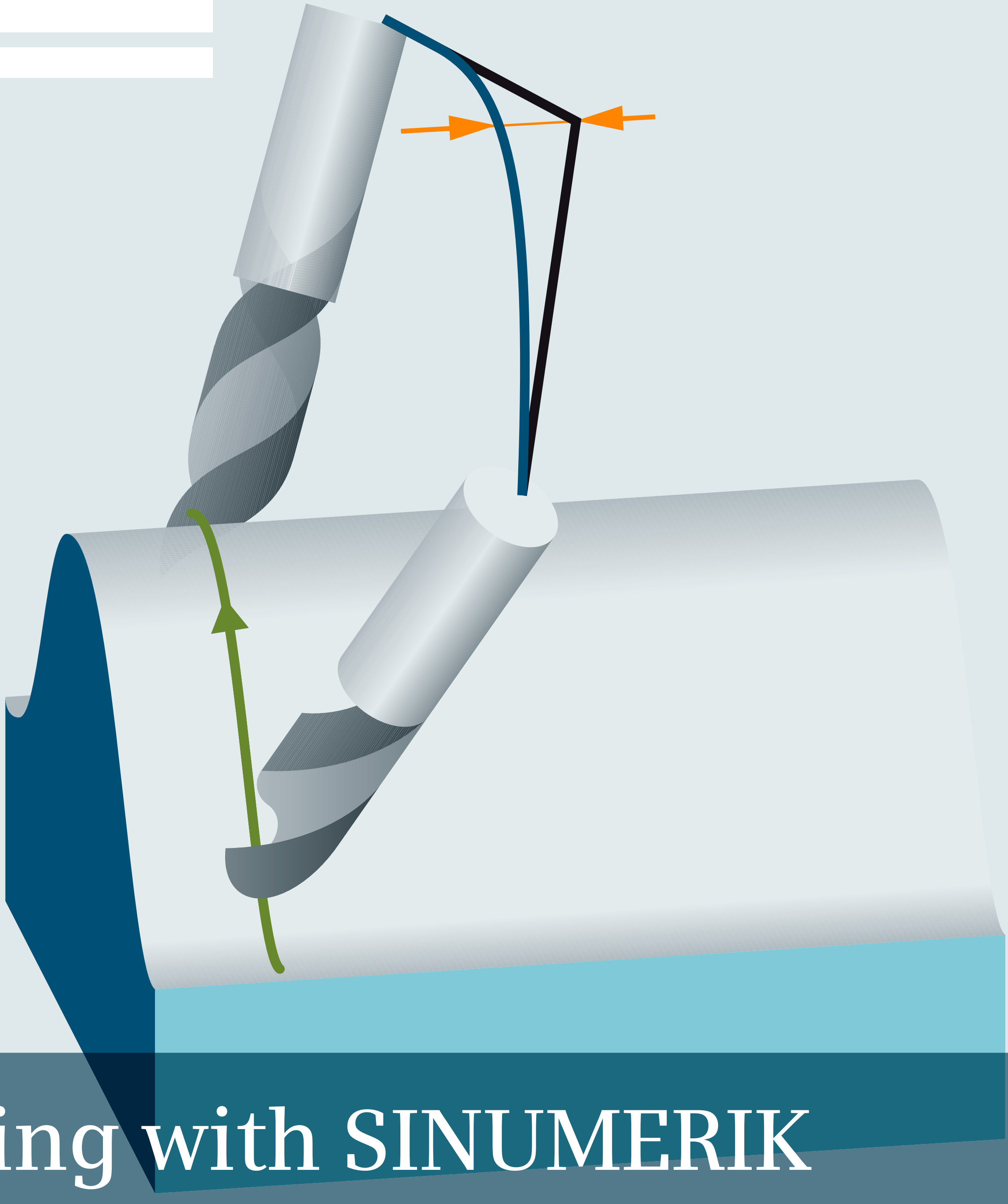
SIEMENS



Milling with SINUMERIK

Mold making with 3 to 5-axis simultaneous milling

Manual

Edition 06/2016

siemens.com

SIEMENS

Milling with SINUMERIK

Mold making with 3 to 5-axis simultaneous milling

Manual

Valid for:

control system

SINUMERIK 828D
SINUMERIK 840D sl

Edition 06/2016

Document Order No.6FC5095-0AB10-0BP4

SINUMERIK® documentation

Printing history, registered trademarks

Brief details of this edition and previous editions are listed below.
The status of each edition that has been published up until now is shown by the code in the "Remarks" columns.

Status code in the "Remarks" column:

A .... New documentation.
B .... Unrevised reprint with new Order No.
C .... Revised edition with new status

Edition	Order No.	Remark
06.2016	6FC5095-0AB10-0BP4	C
09.2013	6FC5095-0AB10-0BP3	C
09.2011	6FC5095-0AB10-0BP2	C
05.2009	6FC5095-0AB10-0BP1	A

Trademarks

All product designations may be trademarks or product names of Siemens AG or supplier companies whose use by third parties for their own purposes could violate the rights of the owners.

You can find additional information 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. Nonetheless, differences might exist and therefore we cannot guarantee that they are completely identical. Nevertheless, the information contained in this document is reviewed regularly and any necessary changes will be included in subsequent editions. Suggestions for improvement are welcome.

Technical data subject to change.

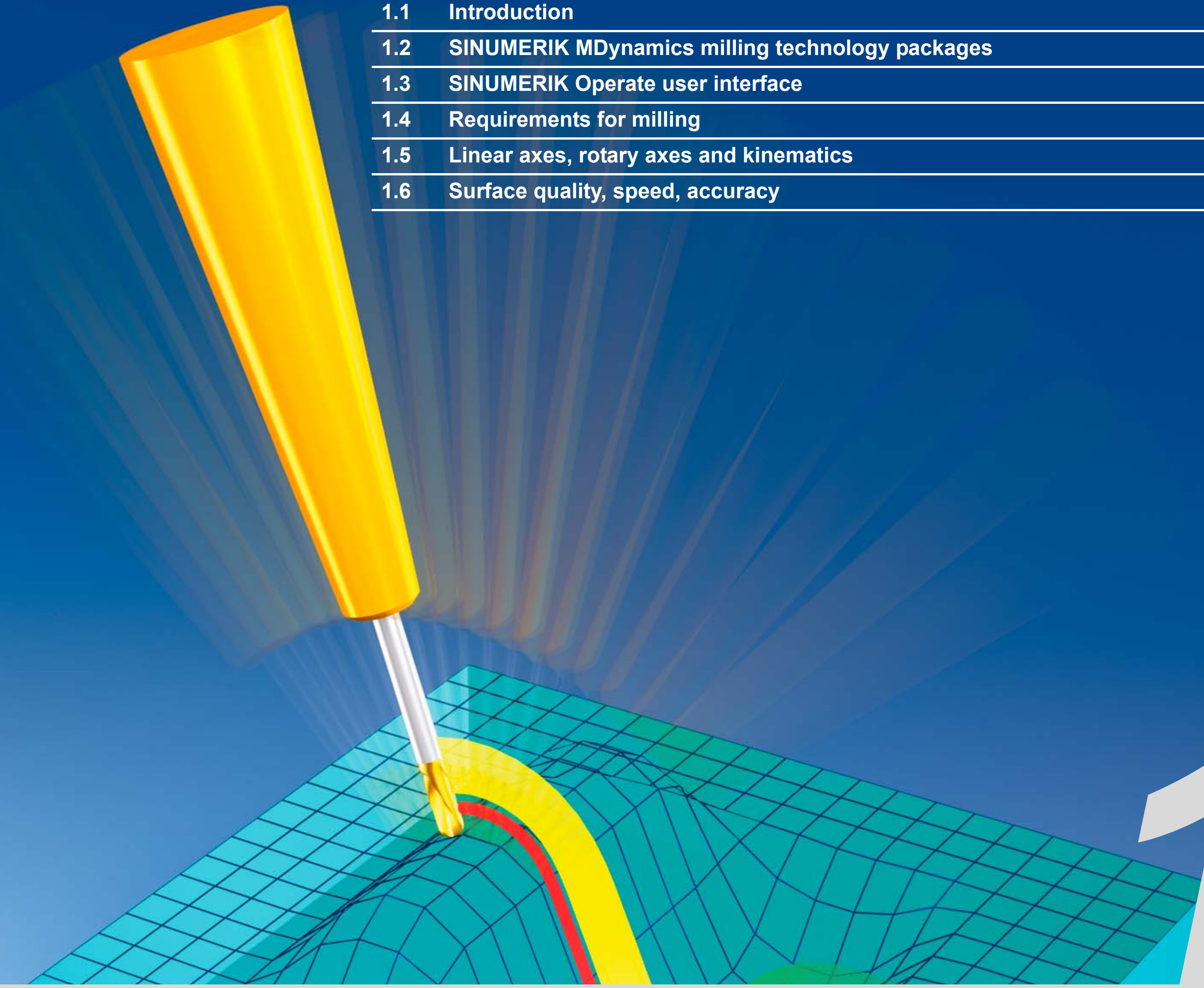
Order No. 6FC5095-0AB10-0BP4
Siemens AG

Page

Page

Basic information

Contents | Page



1

1.1 Introduction

With mold making, the main objectives are to achieve perfect surface quality, precision, and speed without any need for remachining. Within this context, workflow is typically characterized by the CAD-CAM-CNC process chain. From the CAD system right through to the control system, Siemens can offer an integrated solution for these requirements based on its SINUMERIK system.

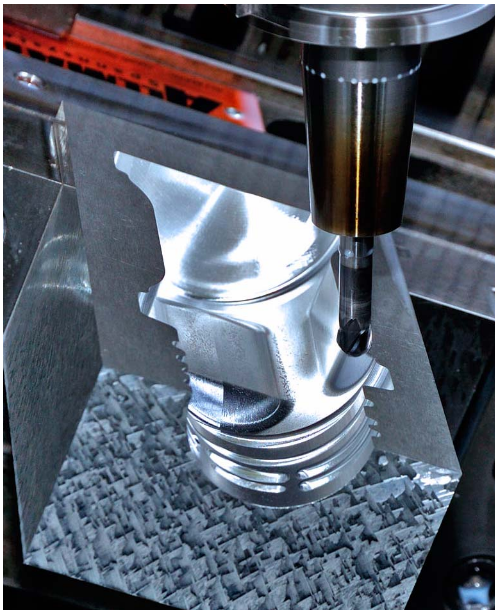
SINUMERIK controls have powerful, highly-developed functions which, if intelligently used, significantly simplify the complete operation involving programming and milling, and at the same time the production result is improved.

This manual is a compact resource aimed at experts working in machining environments who need to get acquainted with the most important basic principles of milling. Based on this, it also provides machine users with practical tips so that they can organize their work efficiently and provides programmers with an insight into the functions of the control and the CAD/CAM system.

As regards the various application areas of machining, the dedicated functions that have been specifically developed for each area are briefly introduced and considered in context.

Although, many of the aspects cannot be covered in depth by this manual, You will find additional information in the appropriate SINUMERIK documentation and relevant literature, See "Further information/documentation" on Page 150.

Range of milling operations



1.2 SINUMERIK MDynamics milling technology packages

SINUMERIK MDynamics bundles SINUMERIK CNC hardware, intelligent CNC functions and our unique CAD/CAM/CNC process chain in technology packages for 3-axis and 5-axis milling. The SINUMERIK MDynamics 3/3+2 or 5-axis milling packages are available for the SINUMERIK 828D and SINUMERIK 840D sl CNC systems.

SINUMERIK 828D

SINUMERIK 828D integrates the milling package for 3 axes. The basic scope can be expanded to include optional functions (see also the 3-axis milling package for SINUMERIK 840D sl)

Basic scope, 3/3+2 axes (excerpt):

- Advanced Surface
- User memory expansion on the user's CF card

Additional options, 3/3+2 axes (excerpt):

- Top Surface
- DXF reader
- EES
- ShopMill machining step programming
- 3D simulation

SINUMERIK 840D sl

For SINUMERIK 840D sl, you can choose between two milling packages for 3-axis and 5-axis machining. The basic scope of the packages includes all the functions needed for the associated machining tasks - and can be expanded.

Basic scope, 3/3+2 axes (excerpt):

- Advanced Surface
- Spline interpolation
- Transmit and peripheral surface transformation
- AUTOMATIC measuring cycles
- 3D simulation
- Residual material detection
- ShopMill machining step programming...

Additional options (excerpt):

- Top Surface
- DXF reader
- EES

The 5-axis package additionally includes:

- 5-axis machining package TRAORI
- 3D tool radius compensation
- Measuring kinematics CYCLE996

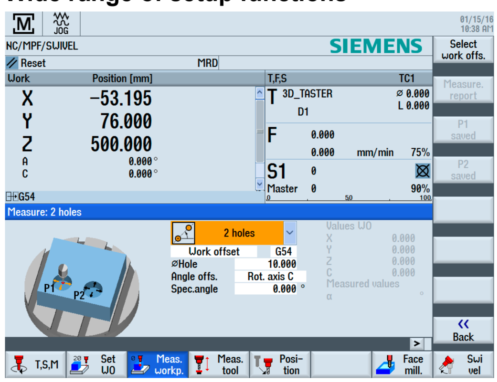
1.3 SINUMERIK Operate user interface

The new SINUMERIK Operate user interface is well laid out and easy to use. It combines the known features of HMI Advanced, ShopMill and ShopTurn in seamlessly integrated, innovative user and programming interface. This allows machining step programming to be combined with high-level language programming, resulting in rational and intuitive NC programming and production planning.

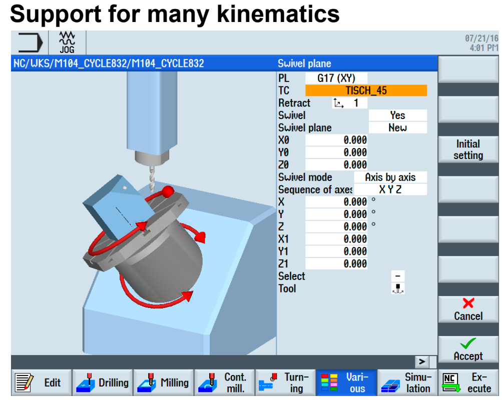
A high degree of user friendliness and expanded setup functions

The machine setup display is clear and manageable thanks to graphical support. Complex workpieces can be quickly and easily machined in one clamping operation. Various kinematics can be easily set up thanks to the cycle support.

Wide range of setup functions



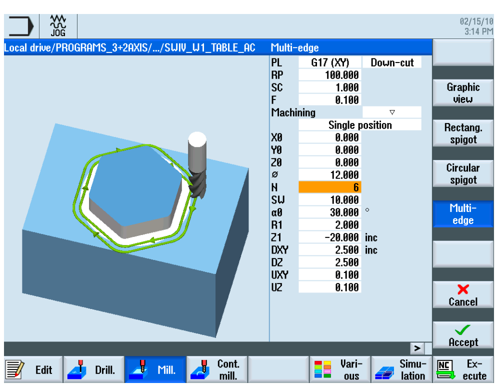
Support for many kinematics



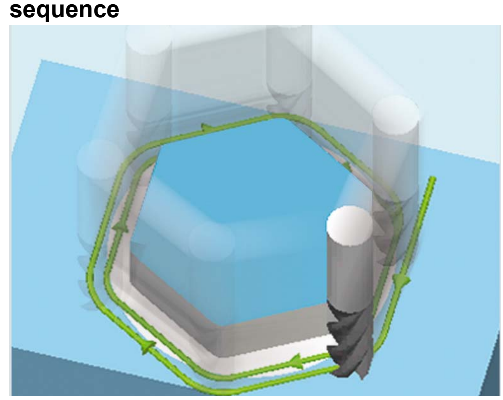
Animated elements clarify the parameters, for example during programming

Using animated elements, the SINUMERIK CNC achieves a unique level of user friendliness for operating and programming - even for technological cycles.

Multi-edge cycle in programGUIDE



Animated elements - moving image sequence



1.4 Requirements for milling

Free-form surfaces
Mold making

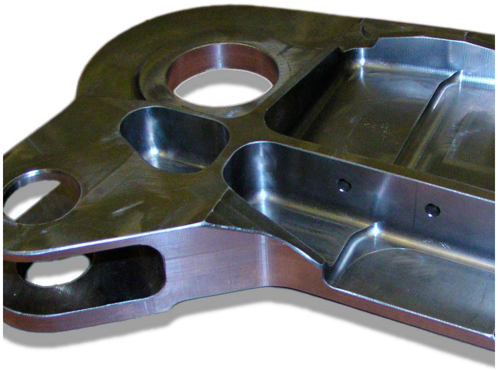
Design standards in all application areas are becoming increasingly more demanding. Economics, the air drag coefficient (cw value) or simply aesthetic appeal are creating a need for more complex surface geometries to be achieved in less time and with the highest precision. The design primarily comes from CAD systems, the machining programs from CAM systems.

Engine and turbine components
for example, impeller

With SINUMERIK, Siemens can provide CNC systems that are perfectly suited to the demands of 3- to 5-axis machining as well as HSC (High-Speed Cutting) and HPC (High-Performance Cutting) applications:

- Simple to operate
- User-friendly programming at the machine
- Optimum performance throughout the CAD - CAM - CNC process chain
- Optimized 5-axis functions

Structural parts
Aerospace industry



Depending on the application, the requirements imposed on the control will vary and a whole range of different functions may be demanded.

Within this context, milling can be broken down into three broad areas:

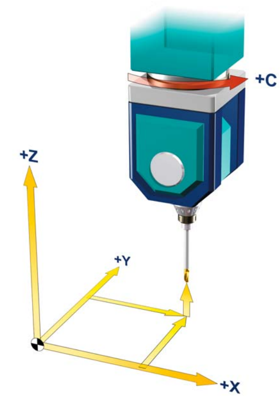
- Free-form surfaces (mold making)
- Turbine and driving gear components (impellers, blisks)
- Structural parts (aviation industry)

SINUMERIK can provide optimum support for each of these areas.

1.5 Linear axes, rotary axes and kinematics

1.5.1 Axes and programming

Tool tip motion



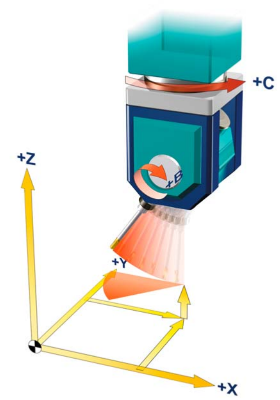
3-axis area

The tool position is approached in space using linear axes X, Y and Z. This enables the tool center point to adopt any position.

With 3-axis machining, you program the three linear axes to achieve the desired machining operations. The contour is milled line by line by moving the three linear axes.

If the tool also has to be set at an angle, you will need rotary axes as well.

Positioning the tool



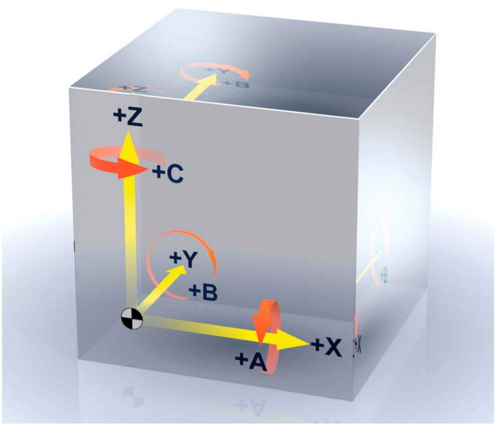
5(3+2)-axis area

The inclination of the tool (i.e. the tool orientation) is changed using two axes of rotation, e.g. B and C.

This is necessary, for example, if the tool needs to be angled in relation to the machining surface or if you want to mill a pocket with inclined walls within the context of circumferential milling.

Using three linear axes and two rotary axes, theoretically any point in space can be approached with any tool orientation. This is the basis of 5-axis machining.

Linear and rotary axes

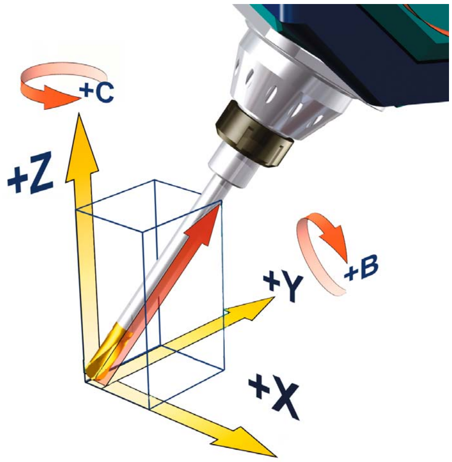


5-axis simultaneous

To accommodate machining scenarios involving tools set at an angle or in order to mill undercuts or geometries located anywhere in space, the three linear axes X, Y and Z are required along with two of the rotary axes A, B or C. It must be possible to control the axes simultaneously.

CNC programming options in the 5-axis area

Programming rotary axes



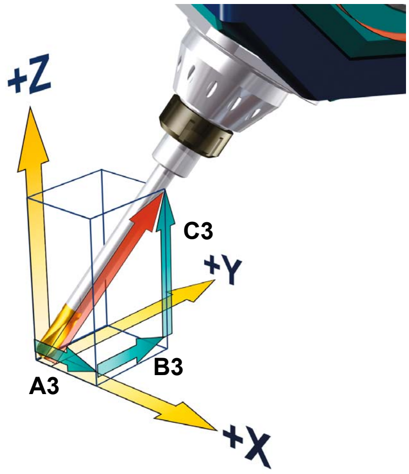
For the purpose of 5-axis machining, the orientation of the tool needs to be defined in addition to the position setpoint of the machining point. The position setpoint is defined in the CNC program by means of the coordinate axes X, Y, and Z.

The tool orientation can be specified on the basis of the rotary axis positions. This enables the position setpoint and tool orientation to be uniquely defined.

The example below shows the tool tip in the position (0,0,0), and the tool shank is the diagonal of a cube (35.26° relative to the X-Y plane).

N100 G1 X0 Y0 Z0 B=54.73561 C=45

Programming direction vector



When describing the tool orientation, it also makes sense to specify the direction vectors A3, B3, and C3 so that the orientation can be programmed independently of the machine kinematics.

The programming of the example above would look like the following after specifying the rotary axis positions:

N100 G1 X0 Y0 Z0 A3=1 B3=1 C3=1

TIP

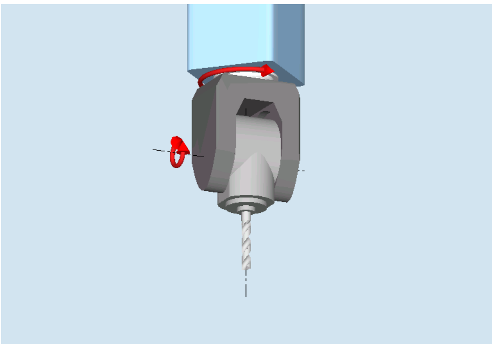
In addition to programming based on the direction vector and rotary axis positions, there are also other forms of angle programming. These include, for example, Euler or RPY angles. Further information regarding this topic can be found in Section "Programming the tool orientation" on page 72.

1.5.2 Kinematics of 5-axis machining centers

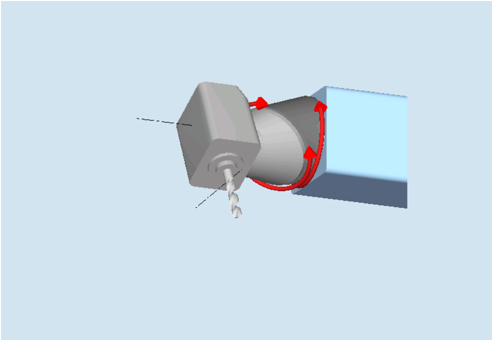
A 5-axis machine can control tool motion in five axes. These are the three linear axes (with which you will already be familiar) and two additional rotary axes. There are different kinematic solutions for the two rotary axes. We will present the most common of these schematically. With SINUMERIK controls, even special kinematics can be controlled on the basis of the integrated, kinematic transformation feature. Special cases such as hexapods, etc. will not be explored in further detail here.

Two rotary axes in the head

Fork head

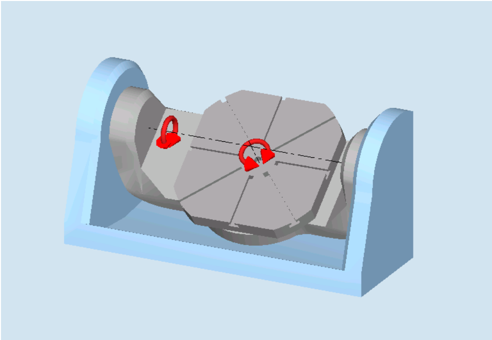


Head kinematics nutated *

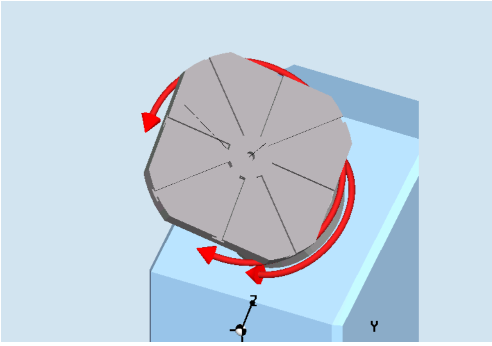


Two rotary axes in the table

Rotary/swiveling table

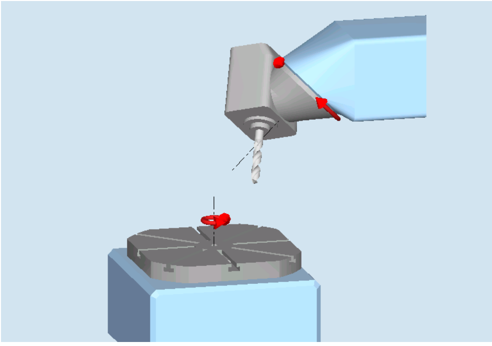


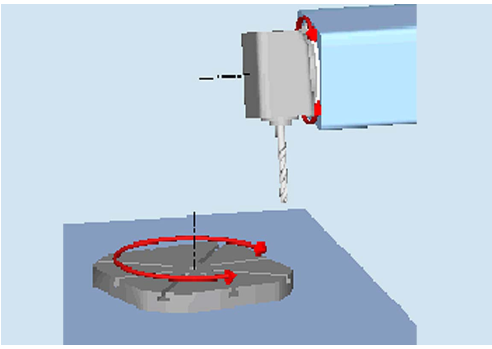
Rotary/swiveling table nutated *



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

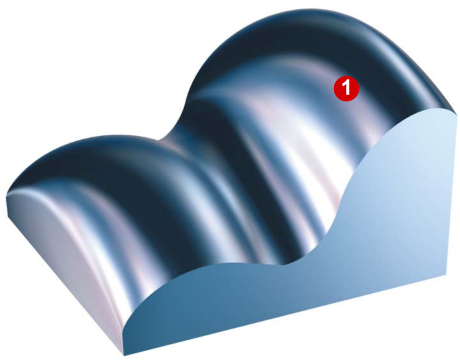
Mixed kinematics





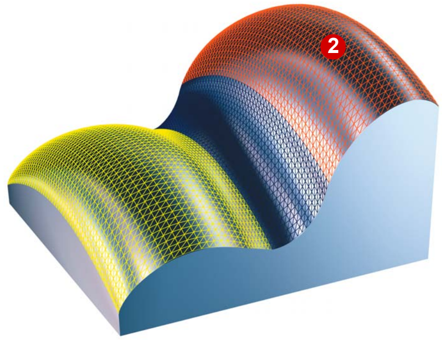
1.6 Surface quality, speed, accuracy

Special attention must be paid to the **CAD -> CAM -> (post processor) -> CNC** process chain when machining three-dimensional geometries, e.g. free-form surfaces. CAM systems generate NC programs for free-form surface machining. The CAM system receives the workpiece geometry from a CAD system. The CNC machine has to process the NC data generated and convert it into axis motion.



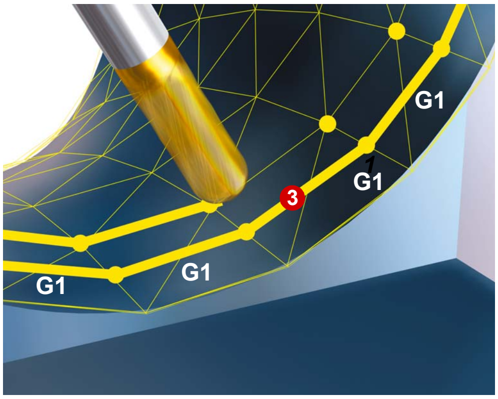
Higher-order surfaces are designed (1) in CAD systems (free-form).

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



This means that the smooth design surface is approximated using many small (2) planes.

This produces deviations from the original free-form surface.

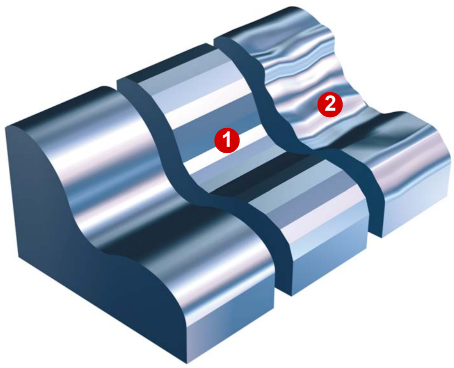


The CAM programmer overlays this polyhedron with tool paths. From these, the postprocessor generates NC blocks within the specified error tolerances. These usually comprise many short straight line elements, G1 X Y Z. (3)

Thus, the machining result is no longer a free-form surface, but a polyhedron. The small planes of the polyhedron can be visibly mapped on the surface.

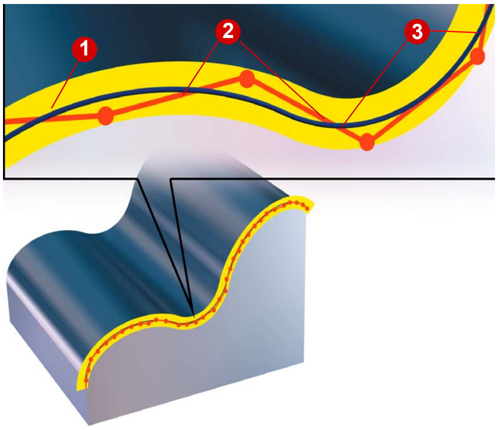
This can result in undesirable remachining.

SINUMERIK controls offer various functions so that remachining can be avoided:



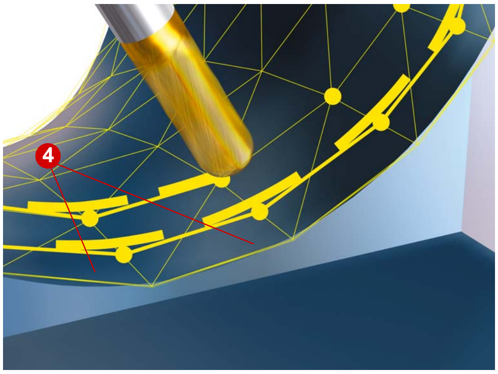
Compressor function (COMPCAD/COMPSURF)

At the block transitions, the interpolation leads to speed step changes 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 make itself apparent as effects of the vibration 2 on the workpiece surface.



The compressor combines a sequence of NC commands in accordance 1 with the set tolerance range 2 and compresses them into a spline 3 that can be executed by the control system directly.

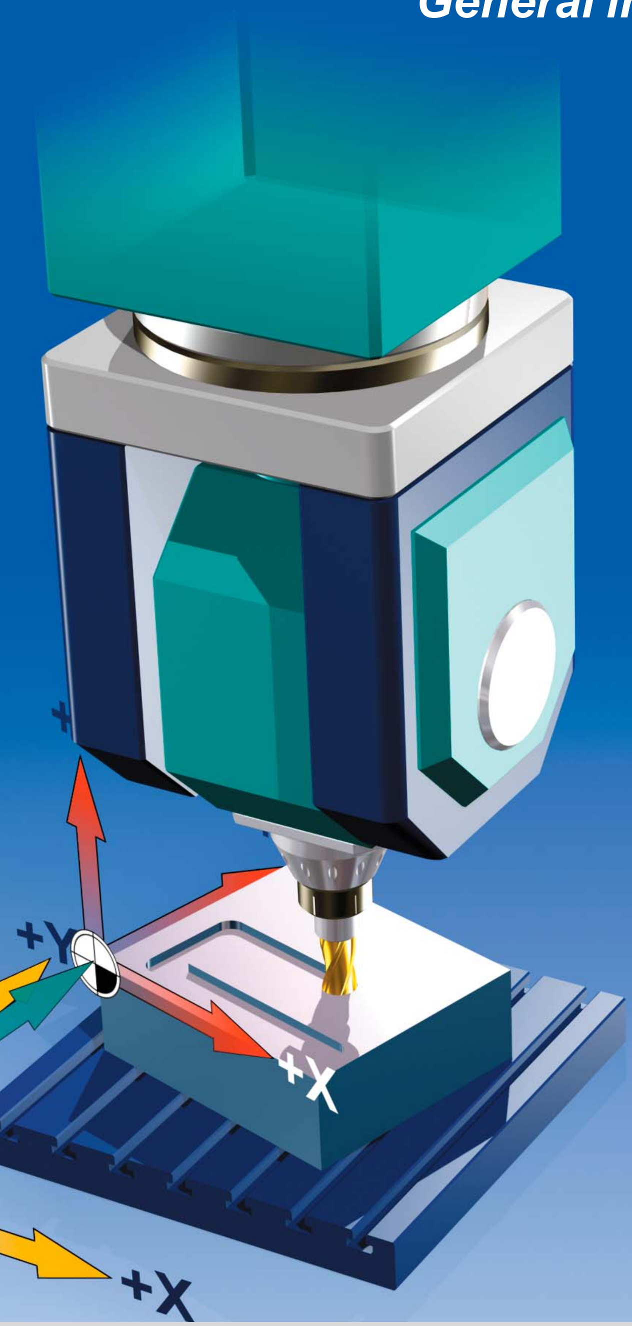
The compressor generates smooth paths and paths with constant curvature. The constant curvature results in a steady velocity and acceleration characteristic. The surface quality is improved and the machine can run at higher speeds, thereby increasing productivity.



Programmable blending (G645)

Discontinuous block transitions can be blended using the blending function to create steady characteristics. This involves inserting geometrical elements at the corners 4 (block transitions).
The tolerance of these geometrical elements can be adjusted (see also 3.6).

General information on workpiece machining



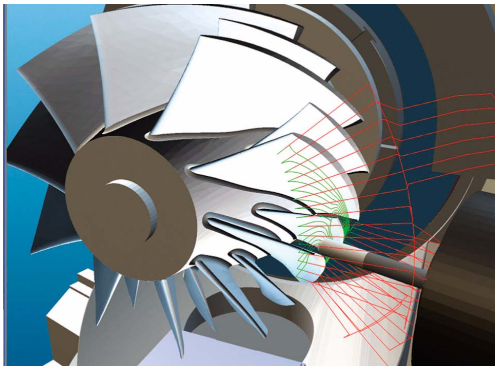
Contents | Page

2

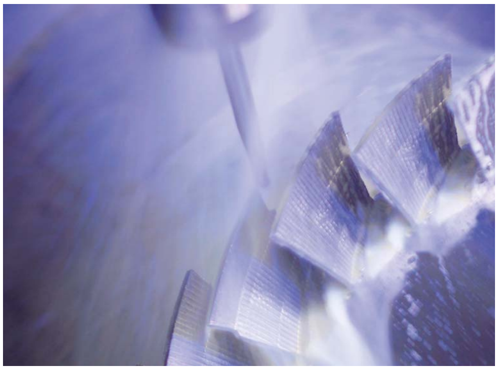
2.1 Process chain for producing 3 to 5-axis workpieces

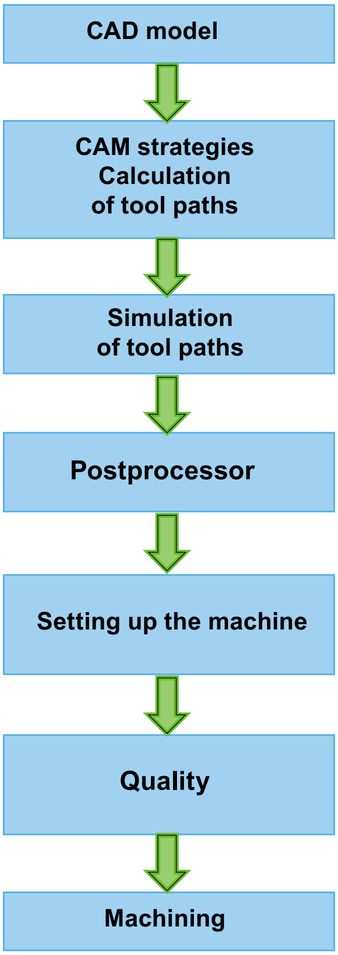
The production process chain generally starts with workpiece design. The data generated at this stage provides the basis for further processing and, ultimately, for production.

CAD/CAM



Production





Creating a CAD model

Use of the CAM-internal strategies and methods to generate the individual machining steps and calculate the paths.

Simulation of the calculated tool paths with collision checking.

Generation of the NC code in the postprocessor. Generally, the path at the tool tip (TTP) is output in NC code.

Data transfer of the NC programs and setting up of the tool and workpiece. If required, creation of a main program that calls the generated geometry programs.

Specification of the high-speed settings, e.g. in CYCLE832, if this is not already taken into account by the postprocessor. These are tolerance, compressor, path control, smoothing, jerk and velocity.

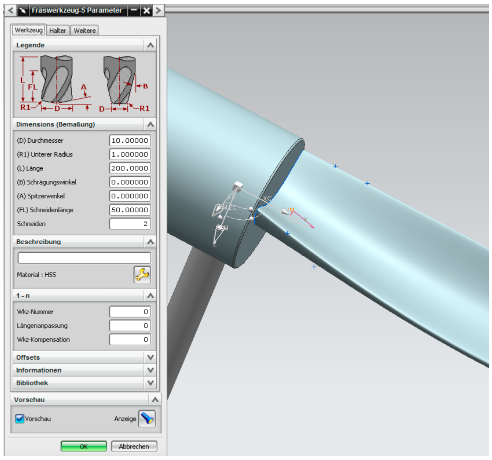
Manufacture of the workpiece on the machine.

2.2 CAM systems

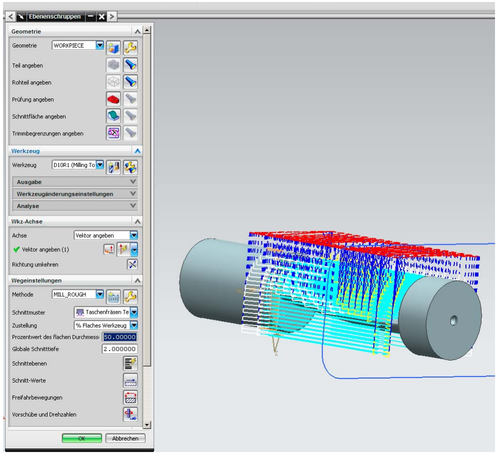
Within the context of the process chain, it is the CAM system that is responsible for the key task of generating the NC programs. The quality of this data plays a decisive role in determining the results of workpiece production.

This section outlines the procedure for generating the NC programs. Given the diverse range of systems available on the market, only a brief summary can be provided.

Tool definition



Plane roughing strategy



Procedure when working with a CAM system:

- **CAD data**
  Reading in CAD data into the CAM system. When reading the data in, a check should be performed to ensure that the surface geometries are free from defects, i.e. that there are no steps or jumps. Flaws in the data such as these will manifest themselves on the surface of the finished workpiece.
- **Clamping situation:**
  Define the clamping situation and the geometry that can be freely machined in one clamping operation. Define geometries such as a zero point.
- **Defining tools**
  Define the necessary tools on the basis of the machining task at hand and enter the technology data. As a general rule, CAM systems are able to read the data in from tool databases. Amongst other things, the tools determine which subsequent machining strategies can be used, e.g. whether plunge cutting is possible with the tool.
- **Defining machining strategies**
  Define the machining process for the various workpiece geometries using the appropriate strategies. First of all, roughing strategies are applied, e.g. roughing in the Z plane or equidistant from the surface geometry. CAM systems offer various options from 2 1/2-axis to 5-axis machining.
  The tool paths can be generated automatically or defined manually; these include, for example, approach and retract strategies or special milling strategies such as trochoidal milling.
  The machining strategies are gradually introduced in stages as part of this procedure and are supported by automatic residual material detection, for example.

- **Calculation and simulation**
  When simulating the calculated tool paths / machine movements, different levels of quality can be used. from straightforward simulation of the tool paths through to complete simulation of the G and M codes that takes account of all machine-specific and control-specific data. Here, potential collisions can be detected and avoided, for example, and the machine's maximum axis traversing ranges can be taken into account.
- **Output of the NC code using the post processor**
  The postprocessor converts the machining steps in the NC programs taking into account control-specific syntax and special control system functions. For this purpose, CAM systems make use of universal postprocessors or special postprocessors that have been optimized for the SINUMERIK system. Manufacturer-specific functions such as separate coolant strategies must be implemented in the postprocessor in consultation with the machine manufacturer.

Important parameters

When working with CAD/CAM systems, certain tolerances and levels of accuracy that will have an impact on subsequent machining must be observed.

Tolerance The CAM system uses the CAD surface (spline) to generate a contour consisting of linear traversing blocks (straight line elements). The extent to which the linear contour deviates from the real contour from the CAD system is known as the chord error or chord tolerance. This tolerance depends on the strategy used and is greater in the case of roughing strategies than with finishing strategies. When the NC programs are executed on the machine, the tolerance is specified by the CAM system in CYCLE832 so that optimum results can be achieved in terms of surface quality and contour accuracy.

Accuracy When outputting the NC blocks from the CAM system, you can specify the number of decimal places. The required level of accuracy is dependent on the type of interpolation. In the case of linear axes (X, Y, Z), at least three decimal places should be used for 3-axis programs.

If the blocks are to be output as rotary axis positions, with 5-axis programs, five decimal places should be used in the linear and rotary axes for optimum surface quality. If they are to be output in the form of a direction vector, we recommend 5 decimal places in the linear axes and at least 6 for the direction vectors.

2.3 Program structure for milling

CYCLE832 can be programmed in the **main program** or in the**subprogram** for the geometry data. Both versions are briefly described.

CYCLE832 in the main program

For machining, a main program is written (1), which contains all technological data. The main program calls one or several subprograms (2), (3) which contain only the pure geometric data of the workpiece. The tool change defines how the content is divided into subprograms.

Main program

HAUPTPROGRAMM.MPF (1)

```
N10     T1 D1                                   ; tool change
N20     M6
N30     S10000 M3                               ; high-speed settings
N40     G54                                     ; subprogram call
N50     CYCLE832 (0.1,_ROUGH,1) (4)             ; deselect CYCLE832
N60     EXTCALL "CAM_schrupp" (5)
N65     CYCLE832(0,0,1)(4)
N70     T2 D1                                   ; tool change
N80     M6
N90     S15000 M3
N100    G54
N105    CYCLE832(0.005,_ORI_FINISH,0.5) (4) ; High-Speed settings (6)
N110    EXTCALL "CAM_schlicht" (5)              ; subprogram call
N120    CYCLE832(0,0,1)(4)                      ; deselect CYCLE832
N130    M30                                     ; end of program
```

Subprogram

Subprogram, CAM_schrupp.SPF (2)

```
N15     G54
N20     G0 X0 Y0 Z10
N30     G1 Z0 F500
N40     G1 X-1.45345 Y0.67878 F10000
...     ...
N5046   G1 Z-2.13247 Y=0.34202 F800 (7)
N5047   ...
N6580   M17
```

Subprogram

Subprogram, CAM_schlicht.SPF (3)

```
N10     TRAORI                     ; only for 5-axis
N15     ORIWKS ORIAXES             ; orientation reference and orientation interpolation
                                   ; optional, already switched in CYCLE832
N20     G54                        ; Nullpunktverschiebung
N20     ORISON (6)                 ; activation orientation smoothing
                                   ; optional, already switched in CYCLE832
N25     G0 X0 Y0 Z10 A3=0 B3=0 C3=0
N23     G1 Z0 F500
...     ...
N5050   G1 X7.60978 Y3.55541 A3=0.34202 B3=0 C3=-0.9396
N5060   G0 Z50 A3=0.34202 B3=0 C3=0.93969
N6570   TRAFOOF
N6580   M17
```

Main program: The main program includes the two important functions for milling, CYCLE832 ❹ and the subprogram call (e.g. using EXTCALL) ❺ .

CYCLE832 ❹ : CYCLE832 was specifically adapted for the program structure shown, where technology data and geometry data are separated. It brings together all the key commands and activates control functions. 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 velocity. For the finishing program "CAM_Finish", the parameters were set towards achieving high surface quality and precision. CYCLE832 is deselected at the end of the geometry blocks. More information is available on the individual parameters in Section CYCLE832 (See "High Speed settings - CYCLE832" on Page 87.)

TiP

The call syntax changes if the Top Surface option is active.

EXTCALL ❺ : CAM programs are generally extremely large, which is why they are stored in an external memory. The EXTCALL command is used to call the subprograms from various locations, including external memories. All programs should be located in the same directory. If this is not the case, the paths must also be specified during the call.

TiP

If option EES is active, then the subprogram call is made without EXTCALL. (See "EES - Execution from External Storage (NC Extend)" on Page 26.)

ORISON/OTOL ❻ : The NC command ORISON is a vector smoothing function that has been specially developed for the 5-axis area. This function can be used to smooth fluctuating orientation across several blocks. The aim is to achieve a smooth characteristic for the orientation and a more harmonious movement of the axes. As of SW 4.5, this command can be part of CYCLE832 (_ORI_ROUGH, _ORI_SEMIFIN, _ORI_FINISH). With older software versions, it is recommended that ORISON is programmed after CYCLE832.

The OTOL command can be used to define the orientation tolerance for vector smoothing with ORISON. The value is entered in degrees. The set value depends on the component.

Subprogram: In the subprogram, ORISON programming is immediately followed by the geometry blocks. In our example, these initially take the form of blocks for 3-axis milling, which are then followed by the blocks for 5-axis simultaneous milling ❼ These are designated A3, B3, and C3.

CYCLE832 in the subprogram

For machining, a main program is written ❶, which only serves as program to call the individual subprograms. The main program calls one or several subprograms ❷, ❸ which contain all the technological and geometric data for the production of the workpiece.

This program structure is suitable for the use of CAM systems whose postprocessors support the SINUMERIK cycles, output these completely with the cycle call and deselect them during the program generation. The descriptions for ❹ to ❼ are identical (see Page 22).

Main program

```
HAUPTPROGRAMM.MPF ❶
N10    CYCLE800(4,"TABLE",200000,57,0,0,0,0,0,0,0,0,0,1,,1)  ; swivel basic position
N20    CYCLE800()                                             ; deselect swiveling
N30    WORKPIECE(,,,"CYLINDER",0,0,-32,-80,46)                ; define blank
N40    EXTCALL "CF_CARD:/PROG/CAM_SCHRUPP.SPF" ❺              ; call subprogram
N50    STOPRE                                                 ; stop preprocessing
N60    EXTCALL "CF_CARD:/PROG/CAM_SCHLICHT.SPF" ❺             ; subprogram call
N65    STOPRE                                                 ; deselect CYCLE832
N70    ...                                                    ; additional subprograms
N200   M30                                                    ; end of program
```

Subprogram

```
Subprogram, CAM_SCHRUPP.SPF ❷
N10    T1 D1                              ; tool call
N20    M6                                 ; tool change
N30    S10000 M3                          ; technological data
N40    G54                                ; zero offset
N50    CYCLE832 (0.1,_ROUGH,1)❹           ; CYCLE832 roughing
N60    G0 X0 Y0 Z10                       ; 3 axes
N70    G1 Z0 F500
N80    G1 X-1.45345 Y0.67878 F10000       ; deselect CYCLE832
N90    ...                                ; end of subprogram
N6000  CYCLE832(0,0,1)
N6010  M17
```

Subprogram

```
Subprogram, CAM_SCHLICHT.SPF ❸
N10    T2 D1                                          ; tool call
N20    M6                                             ; tool change
N30    S15000 M3                                      ; technological data
N40    CYCLE832(0.005,_ORI_FINISH,0.5)❹               ; CYCLE832 finishing
N45                                                   ; 5 axes
N50    TRAORI                                         ; TRAORI for 5 axis on
N60    ORIWKS          ; orientation reference
N70    ORIAXES         ; orientation interpolation, optionally already activated in CYCLE832
N80    G54             ; work offset
N90    ORISON ❻        ; activation orientation smoothing
                       ; optional, already switched in CYCLE832
N100   G0 X0 Y0 Z10 A3=0 B3=0 C3=0                    ; TRAORI off
N110   G1 Z0 F500                                     ; deselect CYCLE832
N120   G1 X7.60978 Y3.55541 A3=0.34202 B3=0 C3=-0.9396 ; end of subprogram
N130   G0 Z50 A3=0.34202 B3=0 C3=0.93969 ❼
N140   ...
N6000  TRAFOOF
N6010  CYCLE832(0,0,1)
N6020  M17
```

2.4 Programs, data transfer and editor

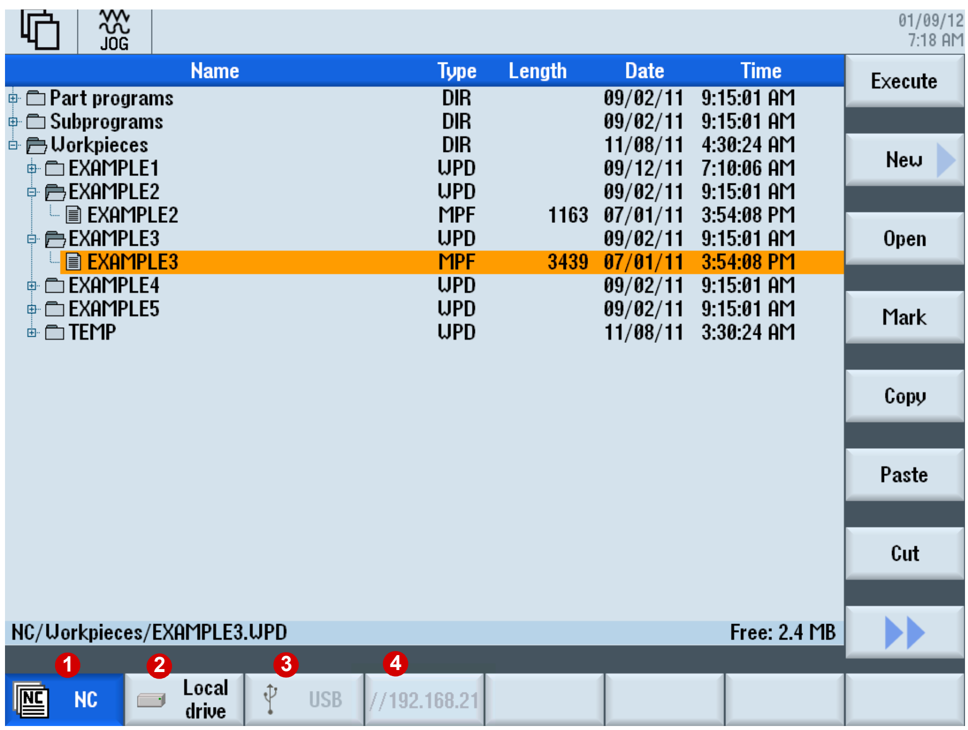
2.4.1 Program manager

The program manager offers you an optimum overview of the directories and programs, and very easy-to-use file handling. It supports plain names of up to 24 characters for directories and files. Subdirectories can also be managed on external storage media such as CF cards, USB sticks and on the NC.

All storage media including the network drives are displayed in the program manager. The part programs can be edited in all media. You can create, paste, copy, delete and cut programs via the horizontal softkey bar.

TIP

In the program manager, you can use standard Windows short-cuts such as CTRL+C, CTRL+X, and CTRL+V.



Possible storage locations for programs are:

1. NC
2. Local drive (CompactFlash card or hard drive)
3. USB drives
4. Network drives

2.4.2 External storage media - data transfer

NC programs are stored in the control, if required, downloaded to the working memory, and executed on the machine.

Mold-making programs are often too large for the NC memory or they cannot be processed. Therefore these are swapped to an external memory and processed successively. In the main program, an EXTCALL command is programmed, which calls up the program according to the network path on the server, the USB port, hard drive, etc.

Procedure when calling the geometry program using EXTCALL/CALL

▶ Program the geometry program call, e.g. SAMPLE in the main program. The call differs depending on the control and where the data is stored.

- Subprogram is resident in the HMI user memory (NC)
  CALL "SAMPLE"
- The subprogram is located in the directory on the CompactFlash card
  EXTCALL "CF_CARD:/PROGRAMS/SAMPLE.SPF"
- The subprogram is located on a local hard drive
  EXTCALL "LOCAL_DRIVE:/PROGRAMS/SAMPLE.SPF"
- The subprogram is located on a USB flash drive
  EXTCALL "USB:/MOLD_DIE/CAM_SCHRUPP.SPF"
- Network connected with Ethernet and path in the machine data SD 42700, e.g. on a server "//R4711/workpieces/subprograms". The default setting is optional. The directory can first be specified when making the call with EXTCALL.
  EXTCALL "SAMPLE.SPF"

Executing from USB flash drive / CF card

On the control, there is a USB port on the front, and on the SINUMERIK 828D, there is a USB port on the front and on the rear. The slot for a CompactFlash card on the SINUMERIK 828D is on the front side.

- Storage media can be inserted or removed during operation, i.e. the machine does not have to be restarted in order for the storage medium to be recognized
- Loading, editing and executing of part programs from the storage medium.
- When executing part programs from a storage medium there is no loss of speed (DNC operation), in which case executing from a CF card is recommended.
- No special software is necessary for reading from or writing to the storage medium on the PC.

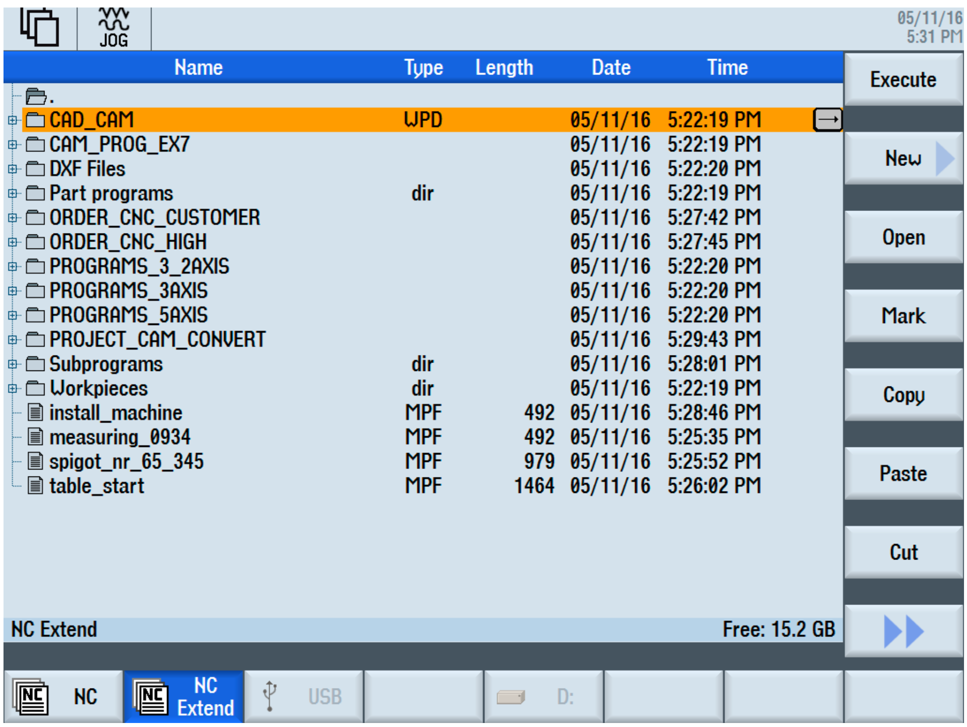
TIP

Direct execution from a USB FlashDrive is not recommended. Interrupting the connection while machining results in a stop, and under certain circumstances, will damage the workpiece.

2.4.3 EES - Execution from External Storage (NC Extend)

Using function **EES - Execution from External Storage** (from SW 4.7 SP2) you can call and execute part programs by the NC directly from all connected data storage devices (e.g. local drive, PCU50, USB global and at the TCU, network). The behavior is the same as that for processing from the NC part program memory without the restrictions that apply to "processing external".

With this function, there is practically no longer a restriction regarding the part program size. This is only limited by the capacity of the external data storage. There is a standard syntax for all subprogram calls, independent of where the subprogram is archived. Program correction for NC stop is possible



The EES function makes EXTCALL calls superfluous. If required, existing EXTCALL calls can be converted. EXTCALL calls without conversion are also executed for an active EES, although still as EXTCALL.

Syntax example for a call with/without option EES

Without EES	With EES
Call subprogram in another folder in the local drive	
`EXTCALL "//LOCAL_DRIVE: /CAM_PRG/ROUGH_01.SPF"`	`CALL "//LOCAL_DRIVE: /CAM_PRG/ROUGH_01.SPF"`
Subprogram call from the main program in the local drive (main and subprogram in the same folder)	
EXTCALL "ROUGH_01.SPF"	CALL "ROUGH_01.SPF"
EXTCALL "ROUGH_01.SPF"	ROUGH_01.SPF

2.4.4 Graphic and file formats in the program manager

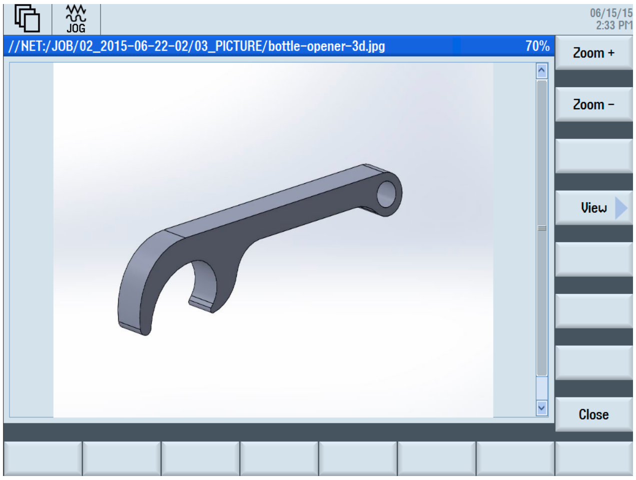
In addition to the standard NC formats (*.mpf, *.spf), the program manager also directly displays additional graphic and file formats at the control system. This means that you can archive and display, for example, production documentation as PDFs, photographs of the clamping situation or also CAD drawings (DXF). This means that you have all information in the production environment directly available at the control system - a paperless production environment.

TIP

An existing DXF file can be read into the contour editor as editor.

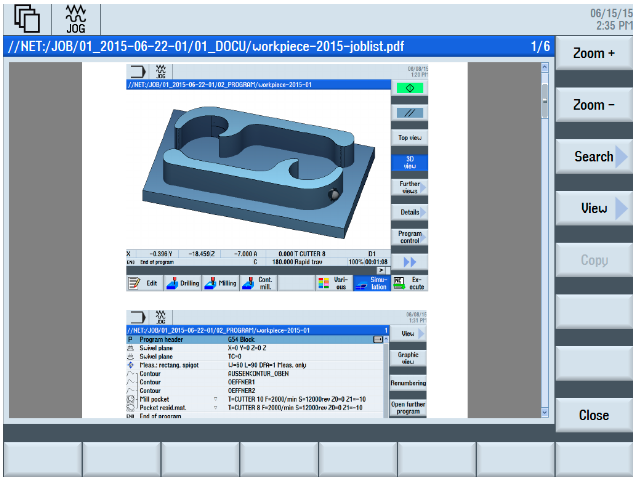
Pixel files

BMP, PNG, GIF, JPG



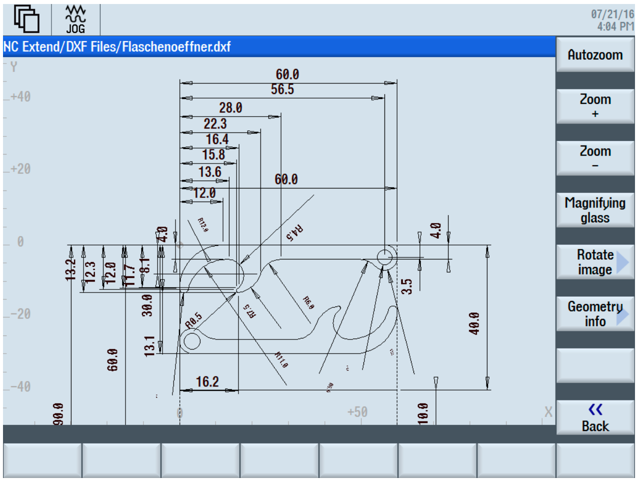
Syntax documents

TXT, PDF



Vector graphic

DXF



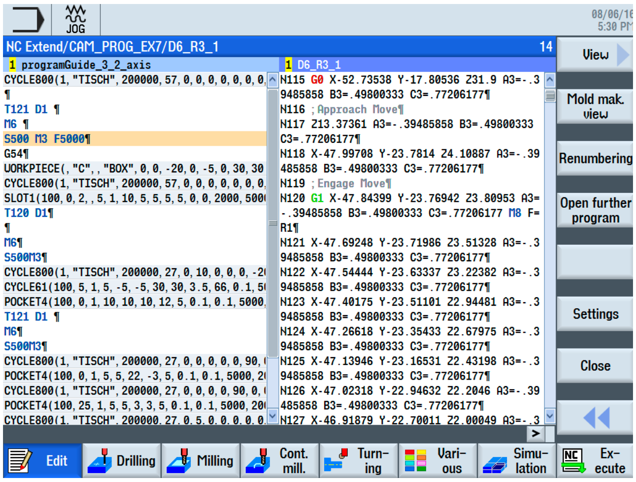
2.4.5 Program editor - useful functions

In the program editor, you can conveniently open larger mold making programs yourself; you can also edit these and this, independent of the archiving location (NC, CF card, USB stick...). In addition to standard functions, it offers additional helpful functions such as cutting out, copying, inserting and renumbering for effectively programming and displaying large programs.

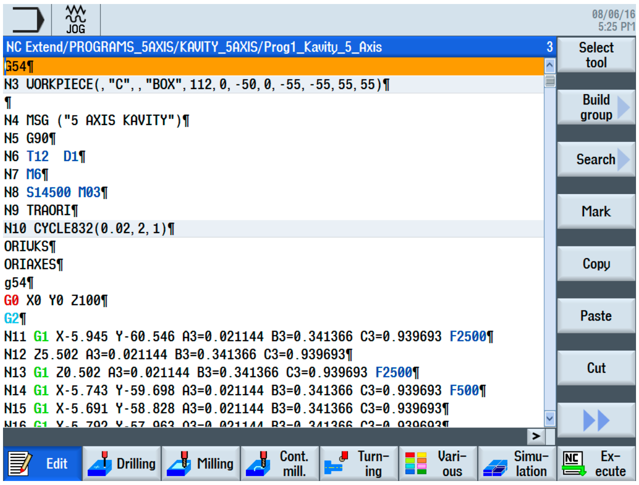
Supplementary programs program editor

- Dual editor - two programs can be simultaneously opened next to one another
- Structuring large programs by creating blocks (blocks are opened and closed)
- Scalable display, changing the font size
- Syntax highlighting, color coding NC functions
- Determine machining times through simulation and display block by block/non-modally
- Subprogram calls including path information directly using softkey **Various**

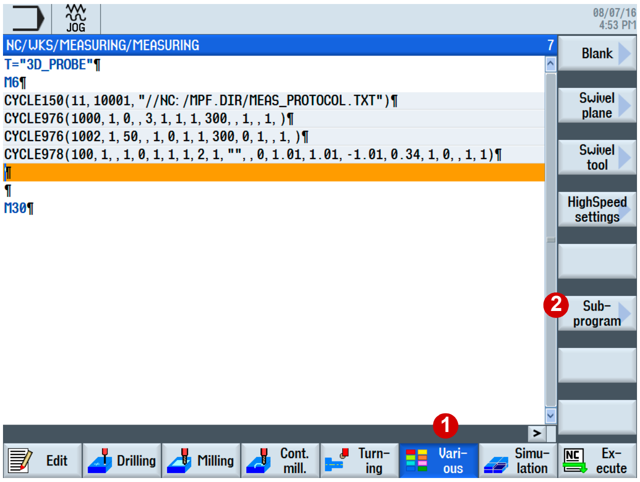
Dual editor



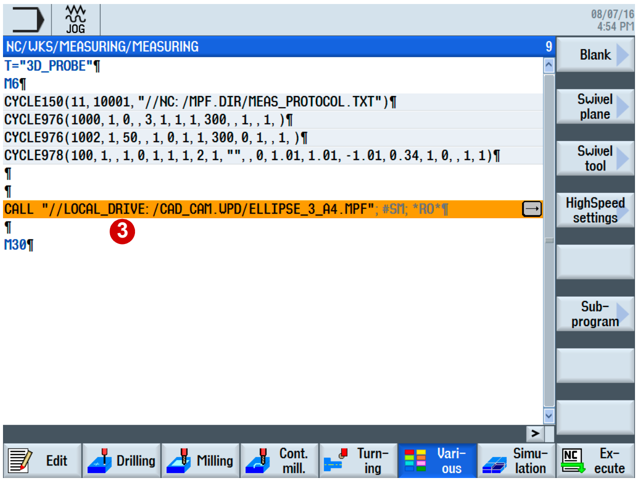
Syntax highlighting



Subprogram call



The complete path is transferred



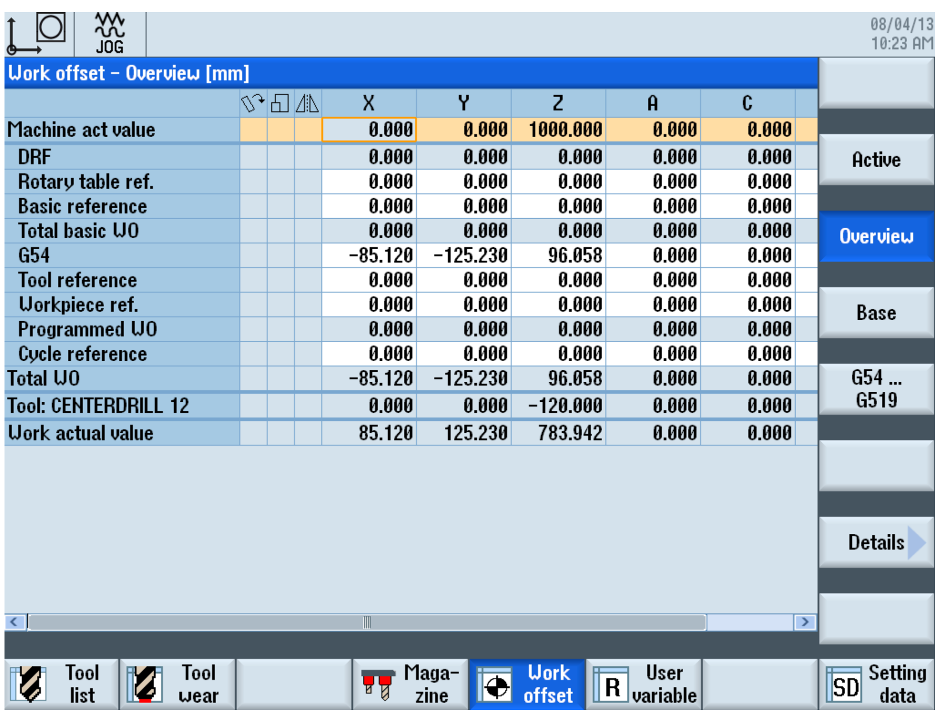
2.5 JOG mode, zero points, tools

2.5.1 Work offsets

After reference point approach, the actual value display for the axis coordinates is based on the machine zero (M) of the **machine coordinate system** (**MCS**). The program for machining the workpiece, however, is based on the workpiece zero (W) of the **workpiece coordinate system** (**WCS**). The machine zero and workpiece zero are not necessarily identical. The distance between the machine zero and the workpiece zero depends on the workpiece type and how it is clamped. This zero offset is taken into account during execution of the program and can be a combination of different offsets.

When setting up the workpiece, set the workpiece zero as work offset, e.g. G54. Using softkey **Work offsets,** you can open the list and select various views using the vertical softkeys.

- Active
  Work offsets, for which active offsets are included or for which values, settable work offsets and total work offset are entered
- Overview
  The active offsets or system offsets are displayed for all axes that have been set up. In addition to the offset (course and fine), the rotation, scaling and mirroring defined using this are also displayed.
- Basis
  The defined channel-specific and global base offsets, divided into coarse and fine offsets, are displayed for all axes that have been set up.
- G54..G57
  All of the settable offsets, divided into coarse and fine offsets, are displayed along with turns, scaling and mirroring.



Work offset – Overview [mm]

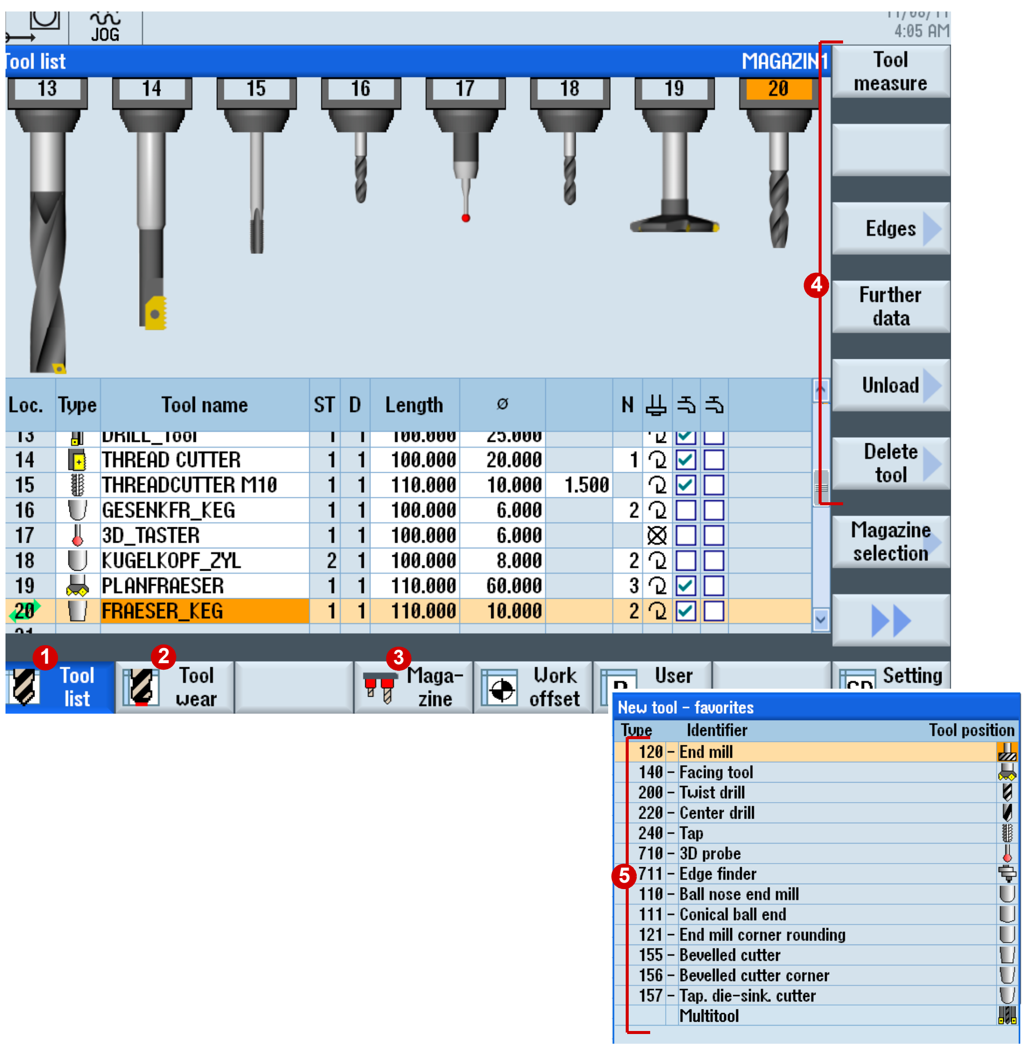
	X	Y	Z	A	C
Machine act value	0.000	0.000	1000.000	0.000	0.000
DRF	0.000	0.000	0.000	0.000	0.000
Rotary table ref.	0.000	0.000	0.000	0.000	0.000
Basic reference	0.000	0.000	0.000	0.000	0.000
Total basic WO	0.000	0.000	0.000	0.000	0.000
G54	-85.120	-125.230	96.058	0.000	0.000
Tool reference	0.000	0.000	0.000	0.000	0.000
Workpiece ref.	0.000	0.000	0.000	0.000	0.000
Programmed WO	0.000	0.000	0.000	0.000	0.000
Cycle reference	0.000	0.000	0.000	0.000	0.000
Total WO	-85.120	-125.230	96.058	0.000	0.000
Tool: CENTERDRILL 12	0.000	0.000	-120.000	0.000	0.000
Work actual value	85.120	125.230	783.942	0.000	0.000

2.5.2 Tool management

For tool management, SINUMERIK Operate provides you with the tool list (1), the tool wear list (2) and the magazine (3). All of the relevant tool data is displayed in the tool list. You can create, delete, load or unload tools - and go directly into the program - using softkey (4). Using the tool catalog (5), which is displayed when new tools are created, you can quickly create tools based on predefined tool types.

Schematic diagrams of the tools are displayed above the tool list. The display can be configured. You can activate or deactivate this using the expanded softkey bar **>>** > **Settings**.

For automatic tool monitoring, you can define the wear limits and service life. Providing a new tool or the disabling of a worn tool is done automatically, based on the specified wear values. Magazine-related data is displayed in the magazine. Here, you can disable locations or assign tools to locations.

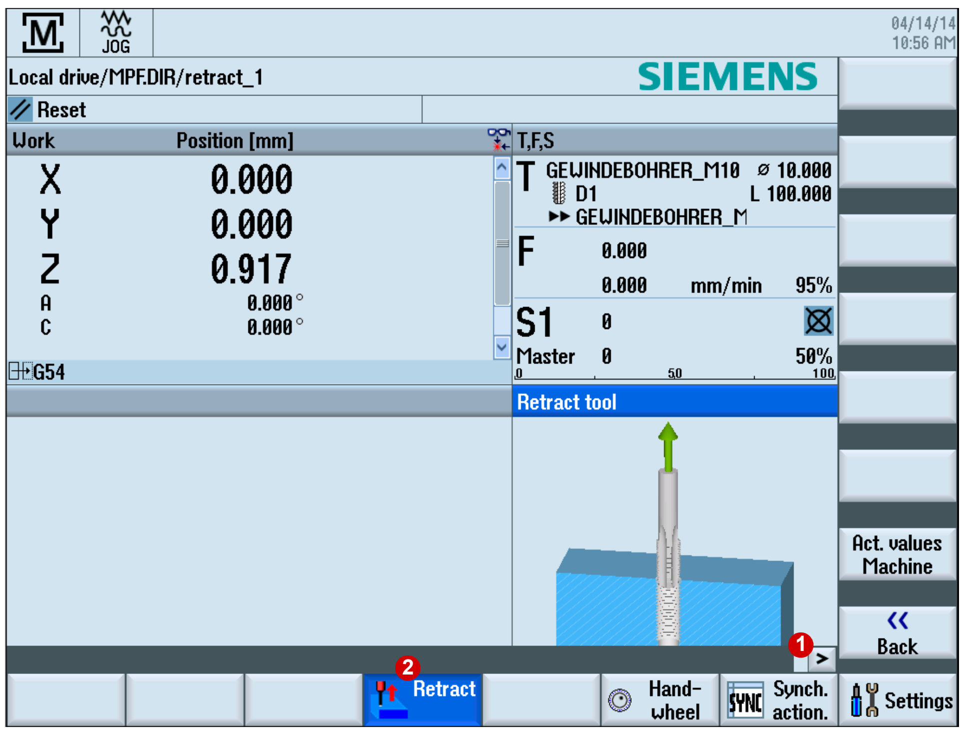


2.5.3 Manual retraction - retract

After an interruption of a tapping operation (G33/G331/G332) or a general drilling operation (tools 200 to 299) due to power loss or a RESET at the machine control panel, you have the possibility to retract the tool in the JOG mode in the tool direction without damaging the tool or the workpiece.

The retraction function is especially useful when the coordinate system is swiveled, i.e. the infeed axis is not in the vertical position.

An active CYCLE800 and TRAORI are taken into account, and for tapping, the spindle is interpolated in the Z axis for retract motion.



Procedure

- After a power failure, switch on the machine again and select the JOG operating MODE.
- Press the menu forward key 1 followed by **Retract** 2.
- Select the coordinate system WCS at the machine control panel and traverse the tool corresponding to the retraction axis displayed in window **Retract tool** (e.g. Z+).

2.6 Fundamentals, measuring in JOG and AUTOMATIC

Measuring in JOG

When **measuring in JOG** (setting up), the machine is prepared for actual machining. This involves determining the dimensions of the workpiece and the tool which are still unknown.

- Manual measurement is used to prepare the machine for machining.
- Manual measurement is used to determine unknown workpiece or tool geometries.
- The operator interacts with the machine during manual mode in order to perform the measurement.

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.

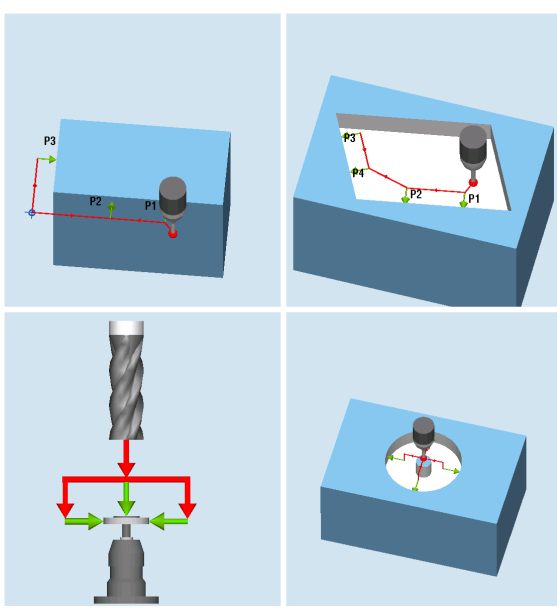
Measuring in AUTOMATIC (in-process measurement)

For **in-process measuring**, workpiece tolerances are determined within the production process and tool parameters are monitored. The nominal dimensions of the tool and workpiece are already known.

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

Examples of measuring cycles/functions

Measuring cycles/functions for all measuring tasks



The SINUMERIK features an extensive pool of practical measuring cycles/ functions for measuring tasks in JOG and AUTOMATIC modes. These measuring cycles/functions enable you to measure workpieces and tools using a graphically supported process.

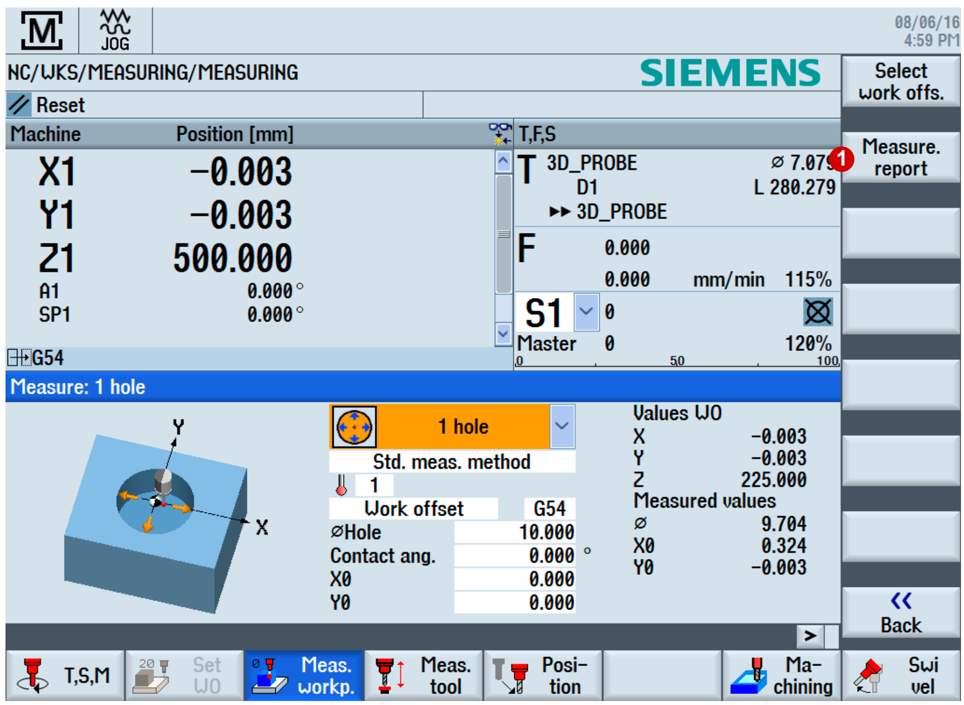
The measuring tasks are carried out with touch trigger probes and load cells or laser measuring systems.

When measuring in JOG, the measuring functions can also be used with edge probes or tools with known dimensions. During the measurement, the measuring points are manually approached and the current position is saved in the measuring function via a softkey.

2.6.1 Measurement logs in JOG

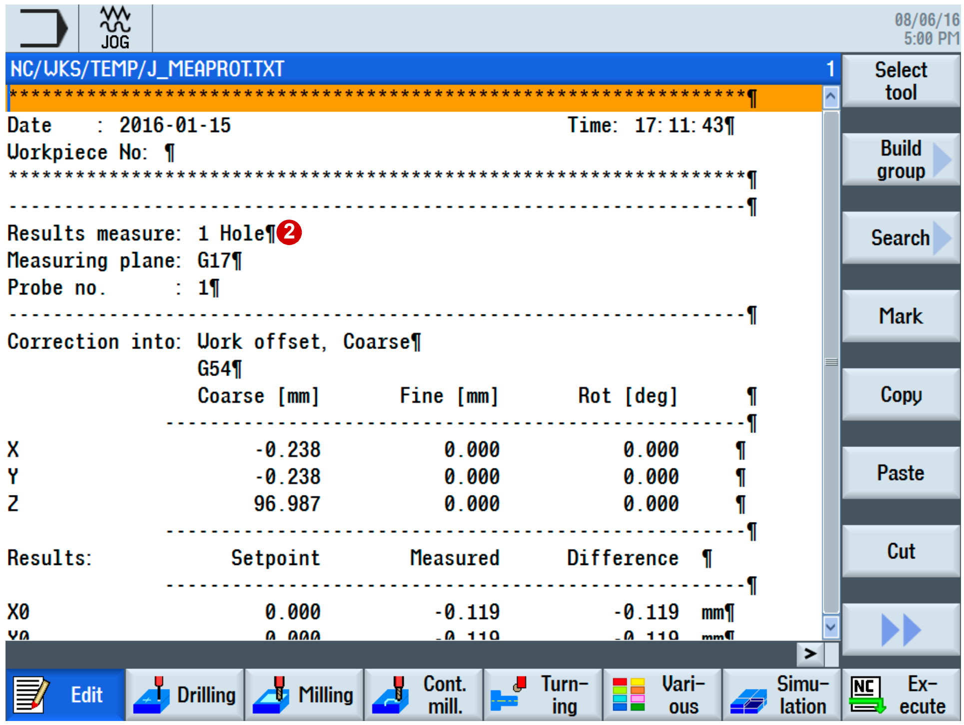
When measuring the workpiece and the tool zero, you can save the values determined in a standard report. For a standard report, you can first configure the report format (txt, csv), report data, archive location and the names.

After the measurement has been carried out, softkey **Measurement log** 1 becomes active - and you can save the measured values. Measurement logs can only be generated when using switching probes or load cells.



The following data are captured and logged:

- Date/time
- Log name with path
- Measuring version
- Input values
- Correction target
- Setpoints, measured values and differences



Example of a report file for workpiece measurement using measurement version **1 hole** ❷.

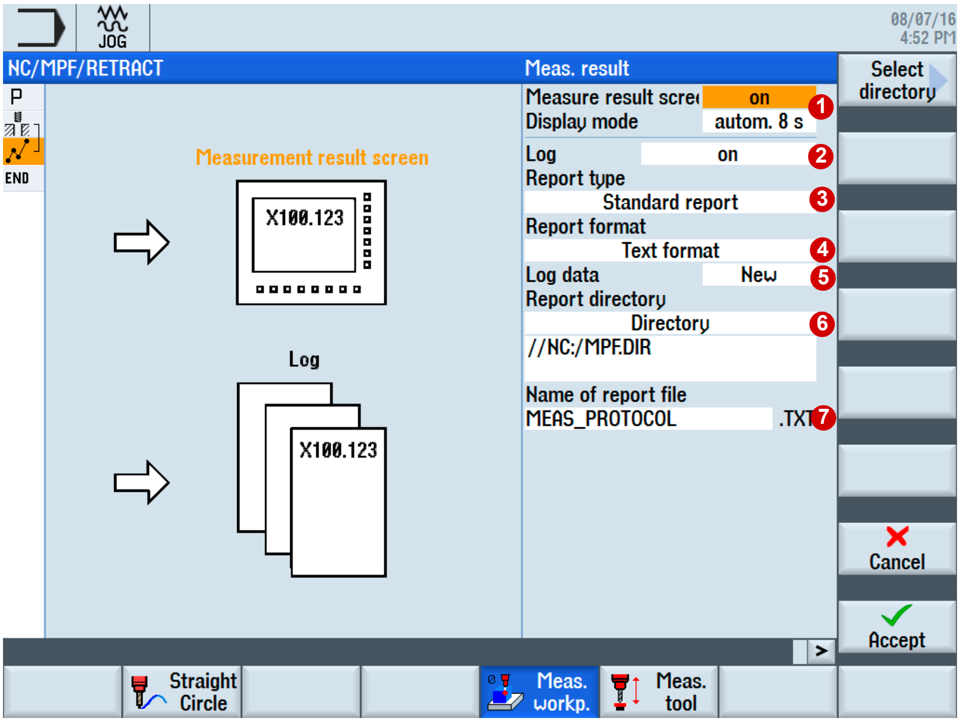
2.6.2 Measurement logs in AUTOMATIC

From SW 4.7 SP2 and higher, standard protocols can be used to save measurement results. Standard logs display the results from measuring cycles in a clear log structure. The output is possible in text or tabular format. Contents and structure are predefined.

The call of the control CYCLE150 is always at the start of the program. You can then optionally create user-defined texts and program the various measuring cycle calls.

In comparison to the standard report, in JOG additional data are captured:

- Name of the part program from which the measuring function was called
- Workpiece number
- Measuring point, measuring version and time of measurement



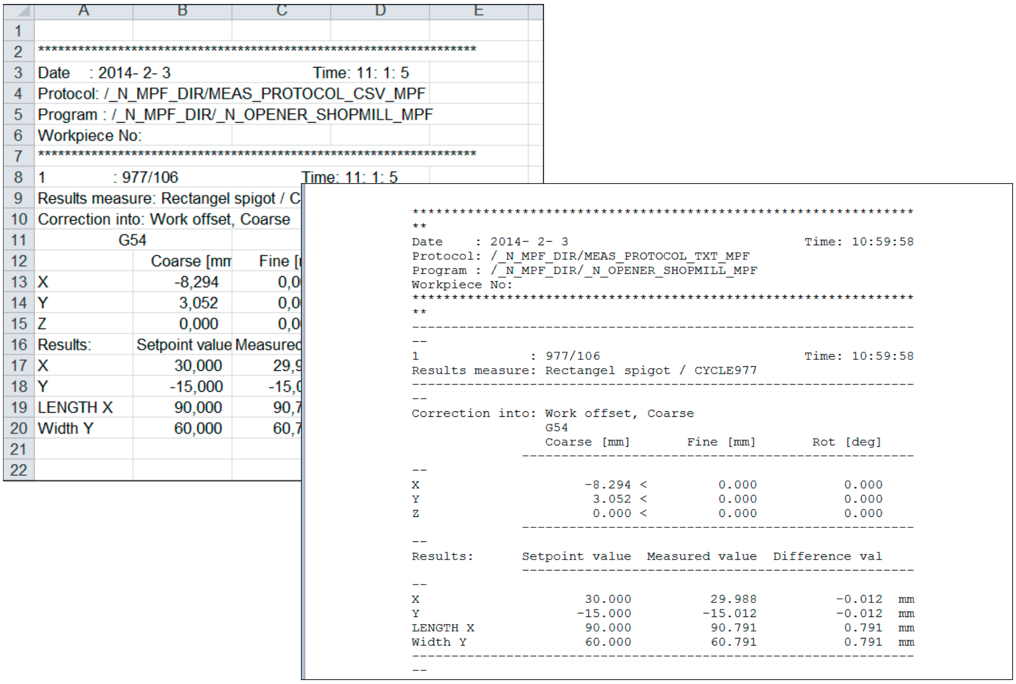
Procedure

- ▶ Select the display and display mode of the measurement result screen 1.
- ▶ Switch logging on 2 and select the standard log 3.
- ▶ Select the format (text, CSV) 4.
- ▶ Configure as to whether the report should be recreated after each measurement - or the measurements should be consecutively saved to the same file 5.
- ▶ Select the archive location (NC, USB...) 6 and the name of the log file 7.
- ▶ Parameterize the measuring functions and supplement optional user data with S_PROTTXT[.].

▶ If you have supplemented the standard log to include user data, then you must program CYCLE160 so that the user data are written to the log file.

Program example G code with call and a measurement function

```
...
N50 CYCLE150(10,1001,"MEAS_PROTOCOL.TXT")
N51 S_PROTTXT[0]=REP("") ; field, delete all data
N52 S_PROTTXT[1]=" Measure workpiece 1234" ; write user data
...
N70 CYCLE160 ; write user data to the log
N80 CYCLE997(109,1,1,10,1,5,0,45,0,0,0,5,5,5,10,10,10,0,1,,0,)
..
```



	A	B	C	D	E
1					
2	**				
3	Date : 2014- 2- 3		Time: 11: 1: 5		
4	Protocol: /_N_MPF_DIR/MEAS_PROTOCOL_CSV_MPF				
5	Program : /_N_MPF_DIR/_N_OPENER_SHOPMILL_MPF				
6	Workpiece No:				
7	**				
8	1 : 977/106		Time: 11: 1: 5		
9	Results measure: Rectangel spigot / C				
10	Correction into: Work offset, Coarse				
11	G54				
12		Coarse [mm	Fine [		
13	X	-8,294	0,0		
14	Y	3,052	0,0		
15	Z	0,000	0,0		
16	Results:	Setpoint value	Measured		
17	X	30,000	29,9		
18	Y	-15,000	-15,0		
19	LENGTH X	90,000	90,7		
20	Width Y	60,000	60,7		
21					
22					

```
*******************************************************************
**
Date     : 2014- 2- 3                                 Time: 10:59:58
Protocol: /_N_MPF_DIR/MEAS_PROTOCOL_TXT_MPF
Program : /_N_MPF_DIR/_N_OPENER_SHOPMILL_MPF
Workpiece No:
*******************************************************************
**
-------------------------------------------------------------------
--
1               : 977/106                             Time: 10:59:58
Results measure: Rectangel spigot / CYCLE977
-------------------------------------------------------------------
--
Correction into: Work offset, Coarse
                 G54
                 Coarse [mm]       Fine [mm]       Rot [deg]
              -----------------------------------------------------
--
X                     -8.294 <         0.000           0.000
Y                      3.052 <         0.000           0.000
Z                      0.000 <         0.000           0.000
              -----------------------------------------------------
--
Results:      Setpoint value  Measured value  Difference val
              -----------------------------------------------------
--
X                     30.000          29.988          -0.012  mm
Y                    -15.000         -15.012          -0.012  mm
LENGTH X              90.000          90.791           0.791  mm
Width Y               60.000          60.791           0.791  mm
-------------------------------------------------------------------
--
```

You can save the standard reports as csv or as text file.

TiP **You can use CYCLE150 - also without measuring functions - in order to display e.g. texts for the operator on the complete screen.**

2.7 Measuring workpiece/tool in JOG

2.7.1 Measure workpiece in JOG

Once the machine has been powered up and the reference point approached, the axis positions relate to the machine coordinate system. The work offset signals to the control the position of the workpiece in the machine coordinate system.

2.7.2 Workpiece measuring functions in JOG

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

The **measuring functions** facilitate the following functions:

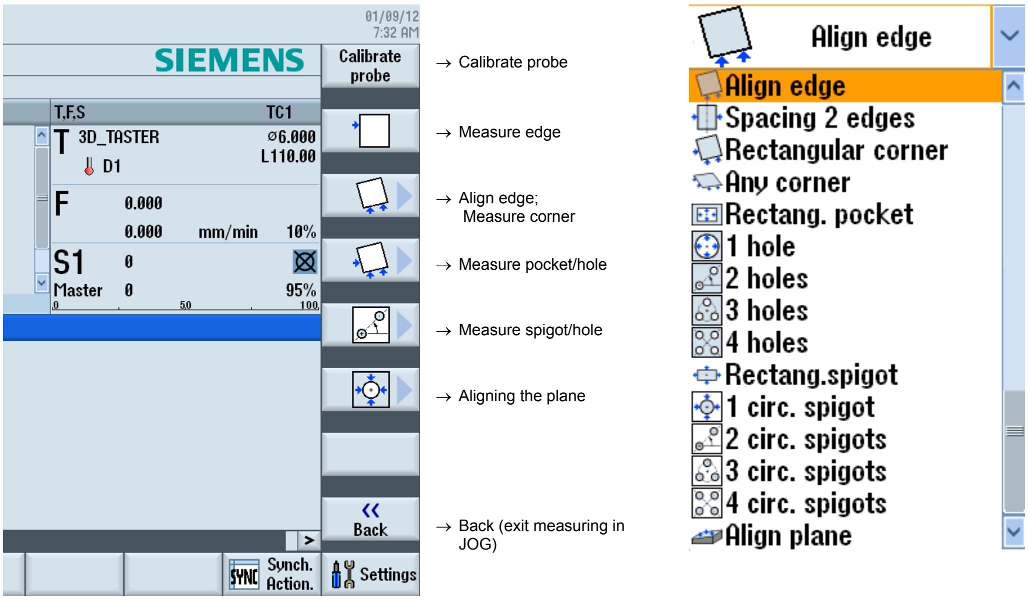
- Calibrating a probe
- Detecting dimensions and position of workpiece geometries, e.g. in order to set up the workpiece.

Requirements for using measuring functions

- The probe is calibrated and active; the tool offset is activated.

Practical measuring functions are provided to facilitate measurements

Measuring functions in JOG

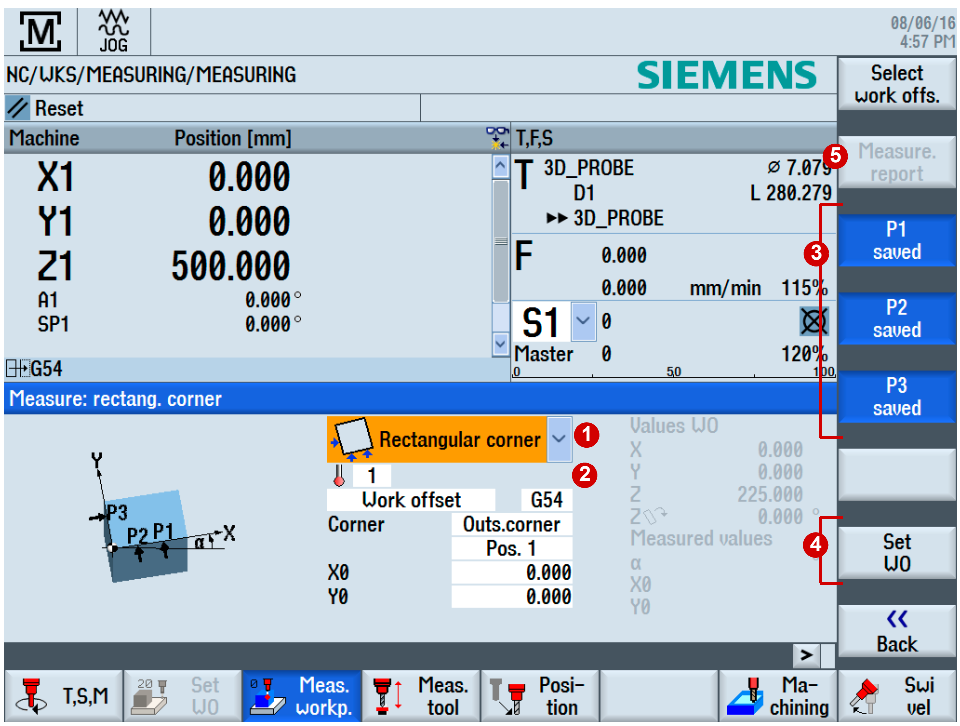


The measuring functions can be selected directly via the softkeys. The softkeys can be freely assigned as of the third softkey and the assignments are based on the last selected cycles. After selecting a measuring function, you can select additional measuring functions via a selection list (see figure on the right).

2.7.3 Example for measuring a workpiece in JOG

The workpiece is setup using the **Right-angled corner**measuring function. The compensation should be made in the work offset G54.

- Select softkey **Right-angled corner**. You can also select the cycle from the selection list ❶.
- Traverse the probe to measuring point **P1**.
- Select the work offset, e.g. **G54** ❷.
- When you press "NC Start", the respective measuring points P1, P2 and P3 ❸ are automatically approached starting from the manually assumed preliminary position. This means that the probe approaches the workpiece, is triggered and then retracts to the start position.
- After each measurement, you can save the measured values to a measurement to log ❺ - the measurement log can be configured in advance.
- After all of the measuring points have been approach, press **Set work offset** ❹.



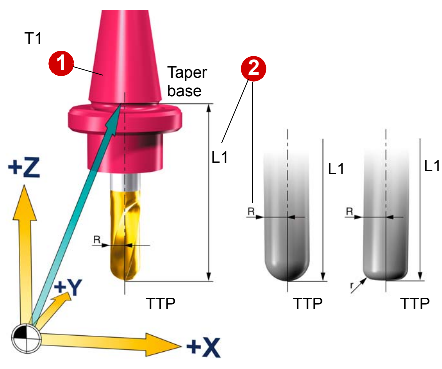
- The corner point now corresponds to the setpoint position. The calculated offset is stored in the work offset.

2.7.4 Measure tool in JOG

When executing a program, the various tool geometries must be taken into account. These are stored as tool offset data in the tool list. When the tool is called, the control considers the tool offset data.

You can determine the tool offset data (i.e. the length and radius or diameter) either via special tool pre-setting devices or with the aid of the measuring functions on the machine.

2.7.5 Tool reference point



TTP= Tool Tip Point

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

This means that to measure the length of the tool, you must use the same reference point (TTP) as the CAM system. For the purpose of determining the tool length, always remember to check the reference point the CAM programmer used to measure L1. The TTP is on the tool tip. The TCP (Tool Center Point) can be further up in the milling tool - e.g. for radius end mills at the center of the radius.

New tool – favorites

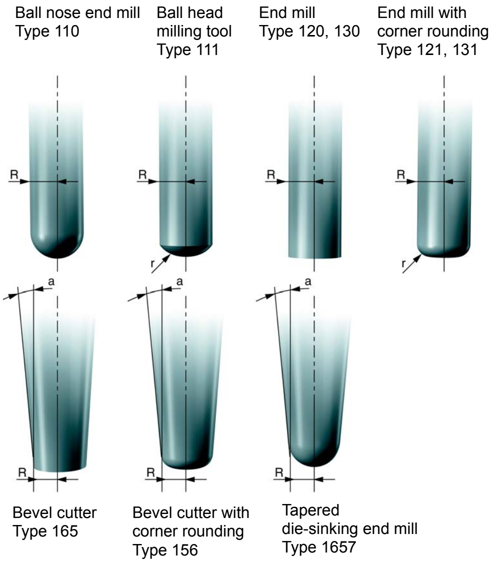
Type	Identifier	Tool position
120 –	End mill	
140 –	Facing tool	
200 –	Twist drill	
220 –	Center drill	
240 –	Tap	
710 –	3D probe	
711 –	Edge finder	
110 –	Ball nose end mill	
111 –	Conical ball end	
121 –	End mill corner rounding	
155 –	Bevelled cutter	
156 –	Bevelled cutter corner	
157 –	Tap. die-sink. cutter	

As is usual, the tool magazine is equipped, the tool numbers (T1...) or the plain text names (MILLING_12...) of the tools ❶ are entered in the tool table - and the tools assigned a tool offset D ❷ , comprising radius "R" and length "L1".

The typical milling tools are available for selection in the tool catalog. You only have to insert these and enter the necessary offset data.

TiP

CAM systems define the position of the reference point differently depending on the tool shape. Generally it is assumed that the output tool path refers to the tip of the milling cutter (TTP). If the CAM system specifies a different reference point, then this difference must be taken into account when specifying the tool length. You can perform this, for example, using the TOFFL command in the NC program.



Specify additional tool data depending on the tool type (e.g. tapered/conical tools).

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

Programmable tool offset (TOFFL, TOFFR)

You can use the TOFFL/TOFF or TOFFR to change the effective tool length (TOFFL/TOFF) or the effective radius (TOFFR) in the NC program without changing the tool offsets. Additional information on this topic is provided in the Programming Manuals.

Tool length offset:

TOFFL=<value>

Tool radius offset:

TOFFR=<value>

2.7.6 Example for measuring a tool in JOG

Function

You can execute the following functions using **"Measure tool"**:

- Calibrating a load cell
- Determining the tool length or the radius of milling tools or the tool length of drills, and entering this data into the tool offset memory.

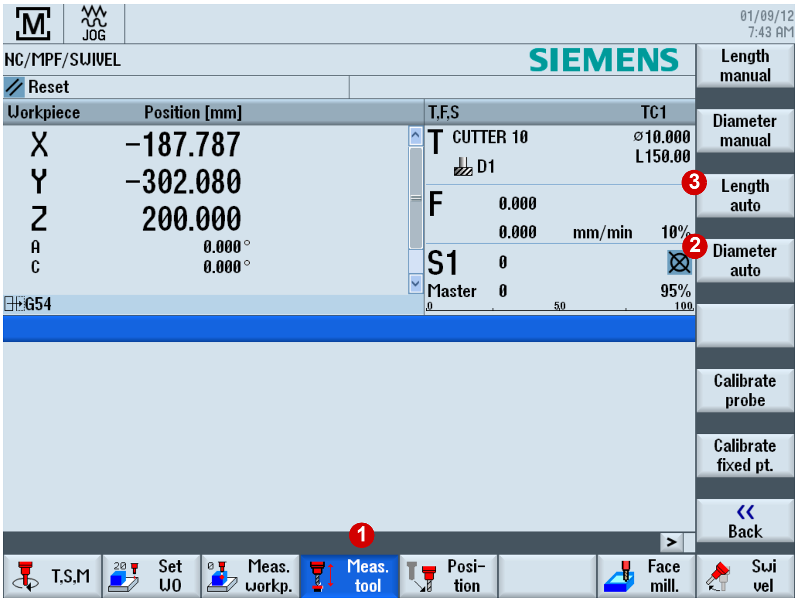
Requirements for using measuring functions

- The tool must have been loaded.
- The load cell is calibrated and active.

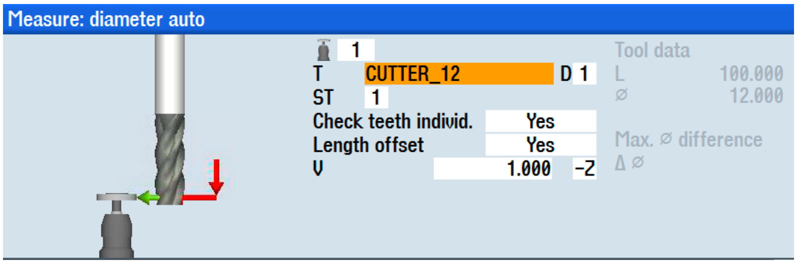
Procedure

- ▶ In the JOG operating mode, select softkey **Measure tool** (1). In the horizontal softkey bar, select whether you want to measure automatically or manually.
- ▶ Click on the appropriate softkey **Radius Auto** (2) or **Length Auto** (3), and for special tools - e.g. with rounded cutting edges - enter the offset.
- ▶ From SW 4.5 and higher, you can specifically measure individual teeth of the tool in order, for example, to identify a broken cutting edge. The single cutting edge measurement (4) can be used for remeasuring and initial measuring. A check is made as to whether the measured values of all cutting edges are within a defined tolerance range.
- ▶ From SW 4.7 and higher, a tool offset can be specified (5) in +/-direction. When measuring the tool in the automatic mode, you can create a measurement report (6) after each measurement.
- ▶ Click **NC Start** to initiate the measuring process; the tool offsets for radius and length 1 are entered in the active tool offset data.

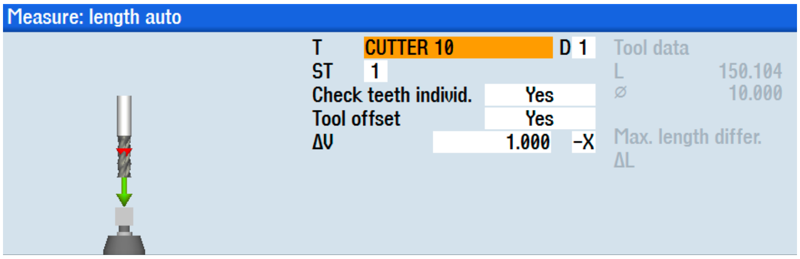
Measuring function - measure tool in JOG



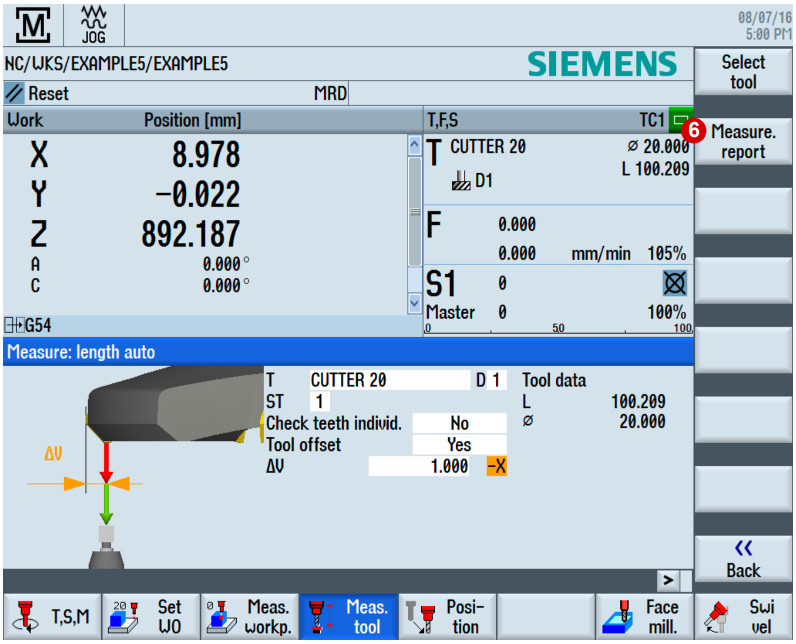
Measuring diameter/radius



Measure length



Measurement log



2.8 Measuring workpiece/tool in AUTOMATIC - in-process measuring

2.8.1 Measure workpiece in AUTOMATIC

For in-process measurements in theautomatic mode, measuring cycles are parameterized specifically for the measurement task. The input screens of the program editor are used for parameter assignment. The measuring points to be approached and the measuring task are automatically implemented in accordance with the measuring program.

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 the result of workpiece measurement:

- Measurement only without offsets (actual value is measured)
- Work offset compensation (setpoint - actual value deviation)
- Tool data offset (setpoint - actual value deviation)

2.8.2 Workpiece measuring cycles in AUTOMATIC

Practical measuring cycles are provided to facilitate in-process measurements.

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

Measuring cycles in AUTOMATIC

→ Calibrate probe

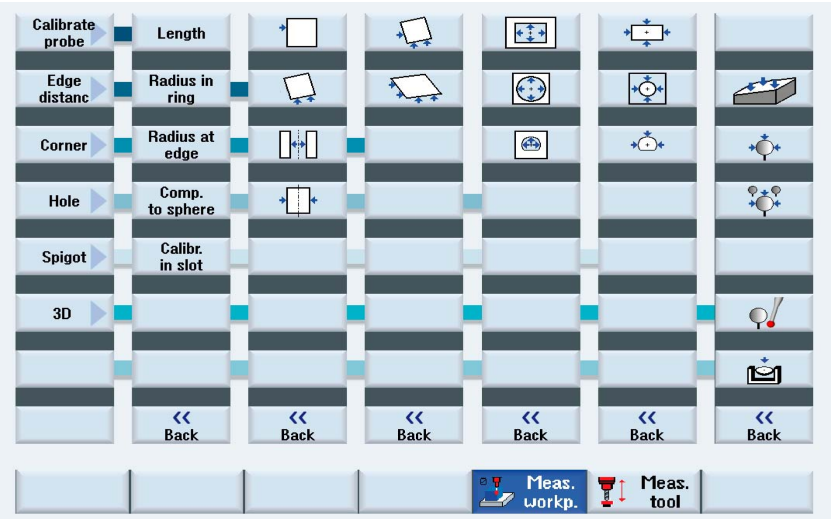
→ Edge, Measure distance

→ Measure corner

→ Measure hole

→ Measure spigot

→ plane, Angular deviation Measure kinematics



2.8.3 Example for measuring a workpiece in theAU-TOMATIC mode

The procedure is illustrated on the basis of the **Measure pocket** function. The cycle automatically measures the right-angled pocket and enters the values in the work offset.

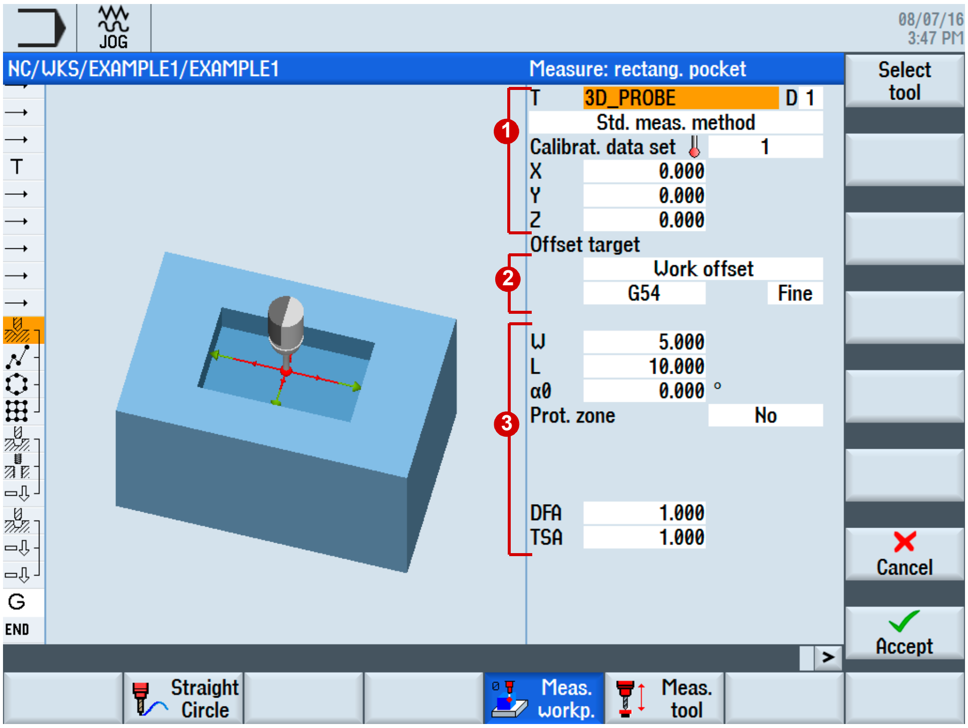
Requirements for using cycles

- The probe is calibrated and active; the tool offset is activated.

Determine work offset via Measure right-angled pocket:

- Create a new program for measuring the workpiece or edit an existing one.
- In the expanded softkey bar, select softkeys **Measure workpiece > Hole > Right-angled pocket** .
- Select the tool (3D probe) specify the measuring technique – and enter the starting point of the measurement ❶ .
- You can define whether the result of the measurement should be an offset compensation or merely a measuring operation ❷ .
  - Compensation in the work offset, specifying the WO
  - Compensation in the tool offset data
  - Measurement only

  As you are setting up the workpiece here, the compensation is made in the WO.
- Enter the setpoints of the right-angled (rectangular) pocket and parameterize additional cycle parameters ❸ .
  At the end of the measuring process, the measured values will be corrected in the active work offset frame.



2.8.4 Measure tool in AUTOMATIC - in-process measuring

A practical measuring cycle is provided to facilitate the process measurement of tools. The cycle determines the length and the diameter of the tool using a calibrated load cell.

▶ You can access the measuring cycles in the NC program by selecting the **Measure milling > Measure tool** softkeys from the expanded softkey bar.

Preconditions

- The load cell is calibrated
- The tool is clamped

In automatic mode, you can automatically measure the tool data or enter it as a tool offset.

2.8.5 Example for measuring tools in the AUTOMATIC mode

In the following example, you will generate a program that determines the tool length and the radius and enters this data into the tool offset.

Determining the tool length:

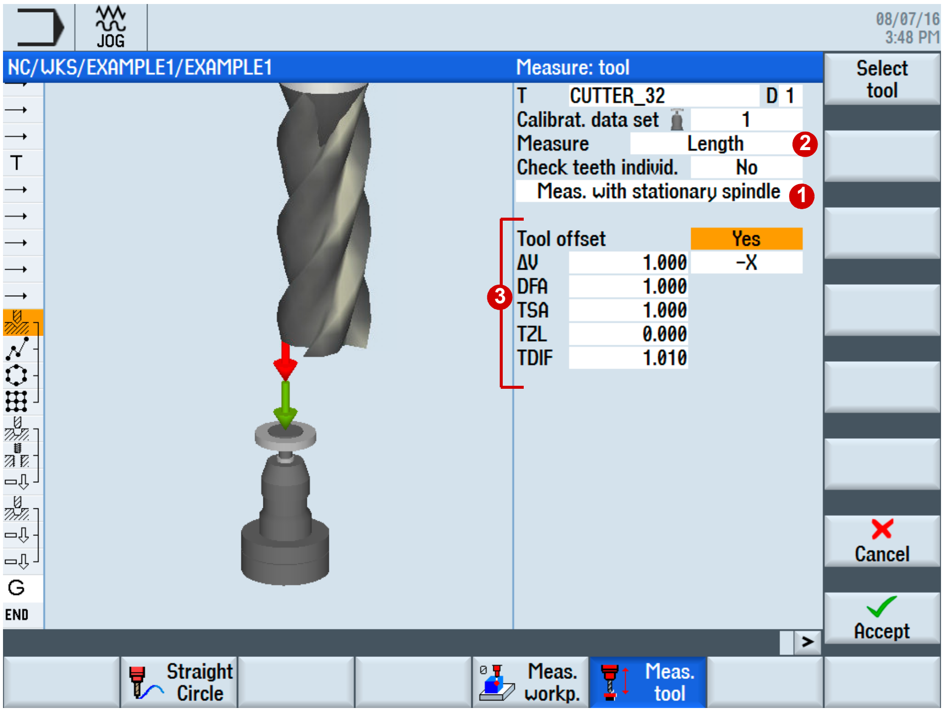
▶ Create a new program to measure the tool.

▶ Select measuring cycle **Measure tool**.

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

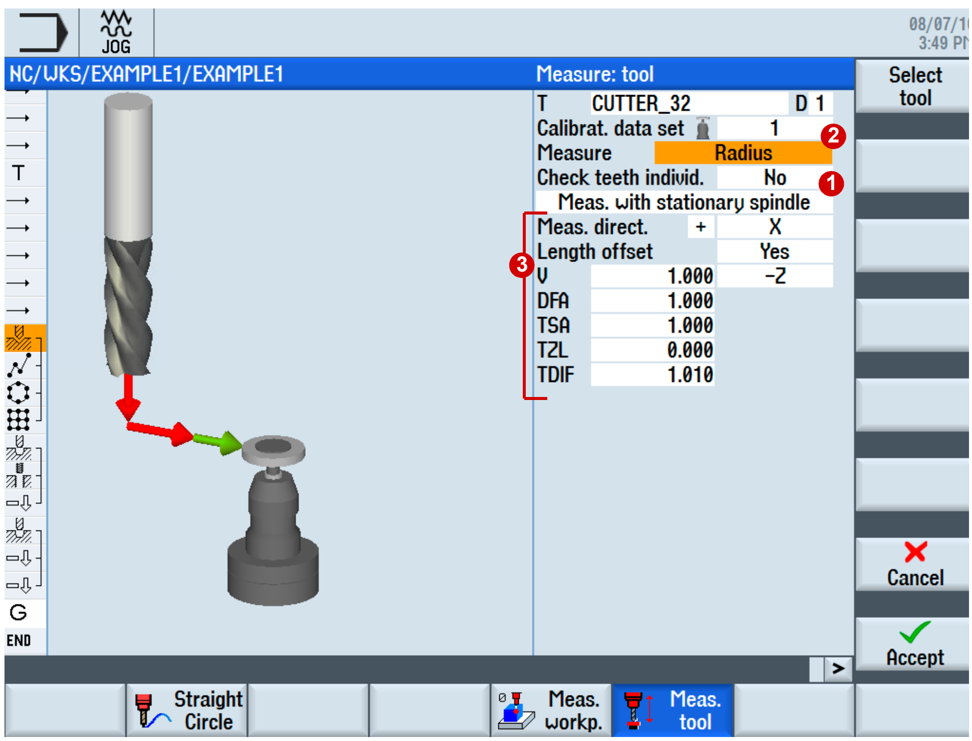
▶ Select the length as measured value ❷. Each tooth can be individually measured to check for tool breakage.

▶ Parameterize the measurement operation ❸.

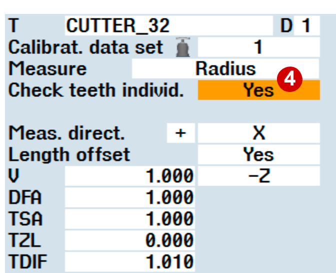


Determining the tool radius:

- The measurement is carried out with the spindle rotating - and the setpoint-actual value difference is optionally entered into the radius wear ❶ .
- Select the radius as measured value ❷ . Each tooth can be individually measured to check for tool breakage.
- Parameterize the measurement operation ❸ .



Single cutting edge measurement (as of SW4.5)



The single cutting edge measurement ❹ can be used for remeasuring (offset in the wear) and initial measuring (offset in the geometry). Milling tools with up to 100 cutting edges can be measured.

The cycle checks whether the measured values of all cutting edges are within a defined tolerance range. If the measured values are outside the tolerance range, an alarm is output.

If the measured value of the longest cutting edge is within the tolerance range, this is entered in the tool management.

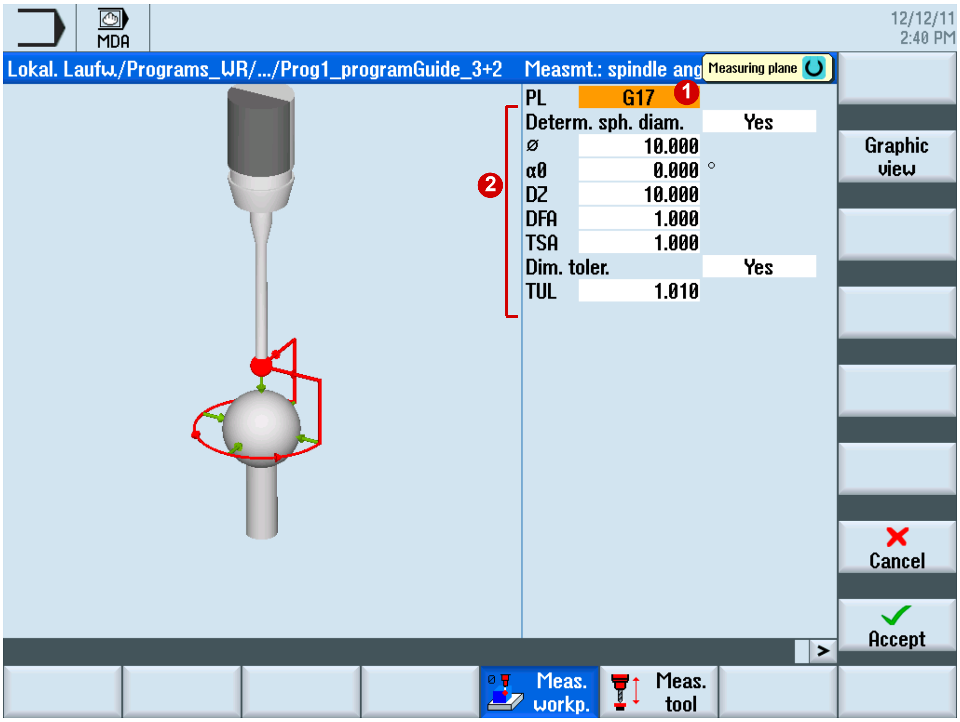
2.9 Checking and measuring machine/spindle

2.9.1 Spindle 3D angular displacement CYCLE995

With this measuring method the angularity of a spindle to the machine tool is measured on a calibration ball. The measurement is carried out by combining the measuring methods **sphere** CYCLE997 and **outer circle segment** CYCLE979.

Based on the measured values, the angular deviation of the spindle to the axis of the plane is calculated. With the measured angular deviations, the spindle can be mechanically aligned parallel to the tool axis or the corresponding tables for sag compensation can be updated.

If there are rotary axes, the determined angular data can be used to align the rotary axis. To do this, the result parameter (_OVR) of CYCLE995 must be used.



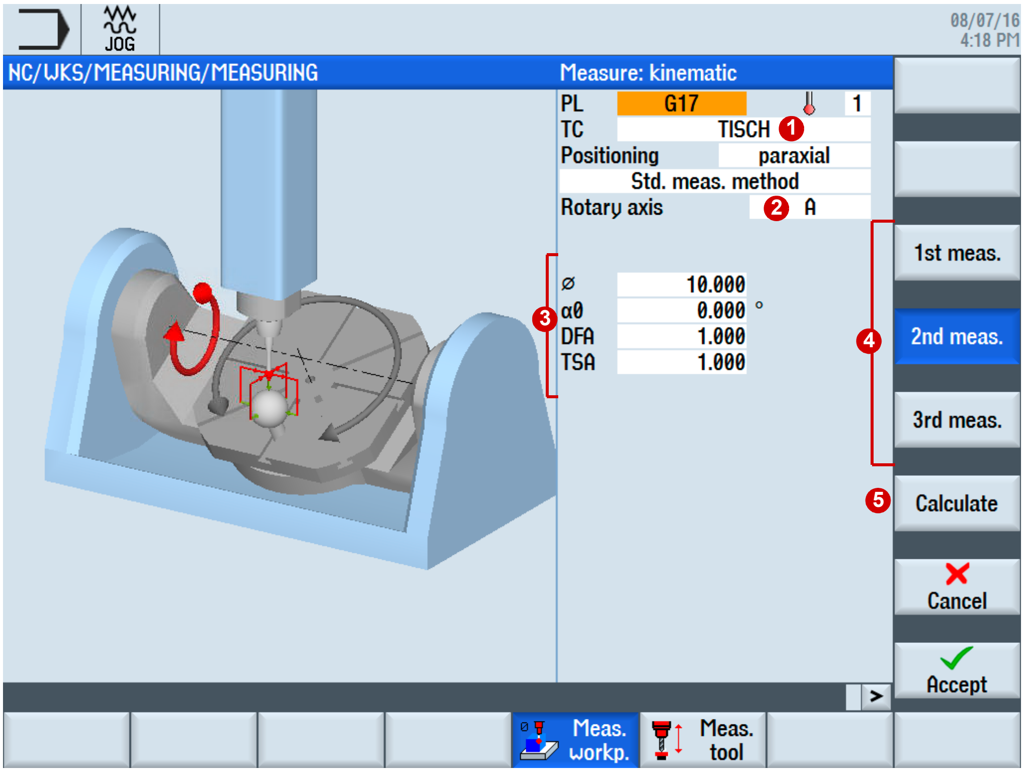
Checking the 3D angular displacement with CYCLE995

- ▶ You select the corresponding machining plane ❶ and the measuring parameters ❷.
- ■ The 1st The calibration ball is measured using CYCLE997 and the measurement is repeated. From 2 measuring points along the circumference and one measuring point at the "north pole" of the sphere (highest point), the center point (position of the ball) is determined. In addition, the diameter of the calibration ball can be determined.
- ■ The 2nd measurement is performed with CYCLE979 on the shank of the probe at a distance of DZ. The center point of the probe shaft in the plane is determined. The angular deviation in XY is calculated from the results of the two center points in XY and the distance between the two measurements in Z (for G17).

2.9.2 Checking/measuring the machine with the kinematics measuring cycle CYCLE996

The requirements associated with 5-axis machining can only be met by means of high-precision machines. As regard to the kinematic transformation, this means that the centers of rotation and the vectors for NC-controlled or manually alignable rotary axes must be determined and entered into the control. CYCLE996 is a special measuring cycle that has been designed specifically for this purpose. It uses a calibration sphere and a calibrated probe to measure the rotary axis vectors automatically. In practical applications, the cycle makes control easier and improves the quality of the process, as compensations of the rotary axis vectors (due, for example, to temperature fluctuations, or other influential factors) can be checked automatically.

For measurement purposes, a calibration sphere is mounted on the table. The sphere is measured with the probe at three separate swivel positions of the rotary axis and the values obtained are recorded. CYCLE996 must be called three times for each rotary axis using different rotary axis positions. The kinematics calculation is carried out in full once all the rotary axes have been measured.

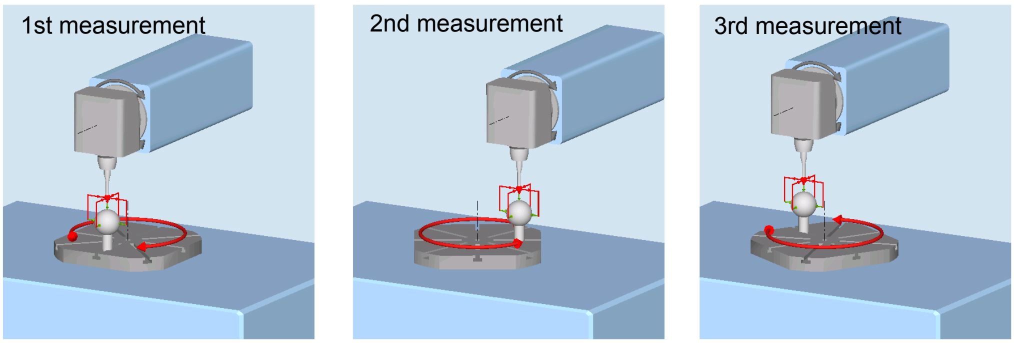


Checking the machine with CYCLE996:

- Create a new program to measure the kinematics.
- In the operating area Programs in the part program editor, select softkeys **>** > **MEASURE MILL** > **MEASURE WORKPIECE**> **3D** > **KINEMATICS**.
- Generate a new swivel data record, where the rotary axis is swiveled (1). Ideally, the swivel positions should create an equilateral triangle, i.e. each one should involve a swivel of 120°.
- Select the rotary axis you want to measure (2).
- Parameterize the measurement operation (3) for the 1st - 3rd measurement (4).
  Press softkey **Calculate** once all rotary axes have been measured (5).

The **Calculate kinematics** dialog is opened after you press the softkey. The following options are available in terms of how you then utilize the measurement results:

- Measure only (measure and calculate vectors)
- Enter (measure, calculate vectors, and enter vectors in swivel data record for correction purposes)



Caution when changing swivel data. These directly influence the kinematics and if an error is made with regard to the correction value, this can result in damage to the machine during operation.

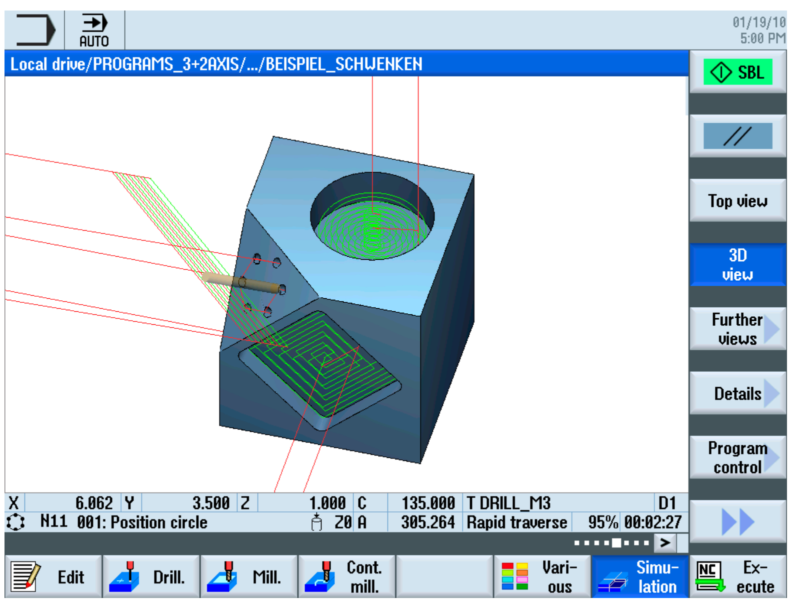
2.10 Workpiece visualization

2.10.1 Simulation

You can detect any programming errors even before production by means of the finished part simulation in the form of a 3D volume model or as a 3-plane view. Using the complete 3D simulation, you simulate the programs before production, even on swiveled planes and by means of 5-axis simultaneous simulation.

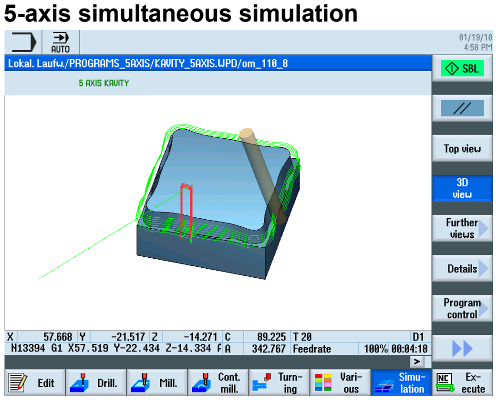
Opening simulation:

- Open the NC program in the program editor.
- Press softkey **Simulation.** The execution of the NC program is simulated.
- During simulation, you can change, turn and zoom the display. For a better display, you can place the cuts in the workpiece.
- Via the shortcuts SHIFT+[cursor keys], you can move or rotate the display or move the cutout and control the override using CTRL+[cursor keys].

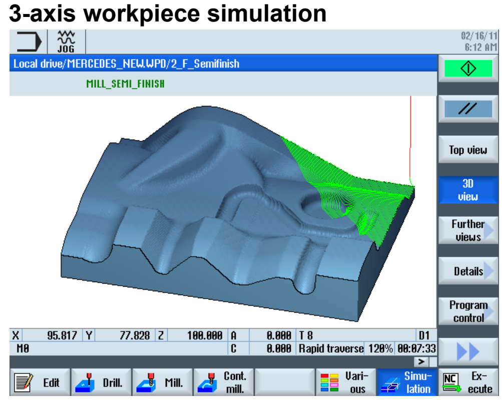


→ Start simulation
→ Reset
→ Top view
→ 3D view
→ Side views
→ Zoom, magnify, turn, cut
→ Override, single block
→ Display tool paths

5-axis simultaneous simulation



3-axis workpiece simulation



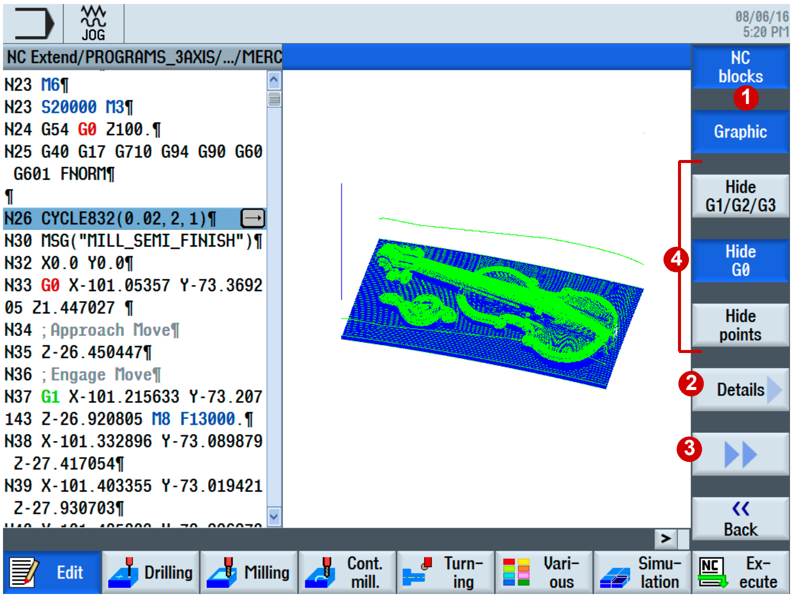
2.10.2 Quick mold making view - Quick Viewer

The mold-making quick view allows the visualization of the processing paths of large part programs, e.g. from CAM systems. By means of the quick view, you can get a quick overview of the program, e.g. the shape of the workpiece or traversing errors or check approach and retraction paths.

The mold-making view supports such blocks as those with G0, G1, G2 and G3, as well as the display of Victors such as a rotary axis or vector programming. Non-interpretable NC blocks are skipped and not displayed in the graphic.

Opening the mold-making view:

- Open the NC program in the program editor.
- Press softkey **>>** followed by **Mold making view**. The NC program and the graphic are displayed simultaneously.
- You can display and hide the views by pressing softkey **NC blocks** or **Graphic** ❶ .



→ Display NC blocks
→ Displaying a graphic
→ Hide G1, G2, G3
→ Hide G0
→ Hide points
→ Zoom and cutout
→ Select program block

Zoom and cutout:

- Press softkey **Details** ❷ followed by **Zoom+** or **Zoom-** to zoom in or zoom out.
- Press softkey **Details** ❷ followed by **Magnify+** or **Magnify-**, to increase or decrease the size of the cutout. Using the cursor keys, you can move the cutout.

Selecting an NC block with an error:

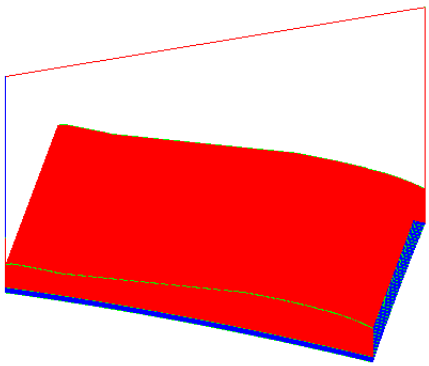
If there are errors in the graphic, you can directly select the NC block in the program.

- Press softkey **>>** ❸ and then **Select point.**
- Using the cursor keys, move the displayed cross-hairs to the appropriate position in the graphic.
- Press the "Select NC block" softkey. The erroneous NC block is highlighted in the editor.

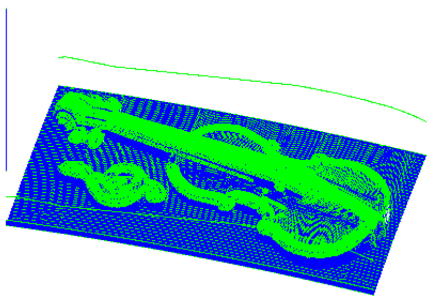
Hide G0, G1, G2, G3 lines:

▶ Using softkeys ❹ , you can hide G0, G1, G2, G3 or points so that the graphic lines are clearly shown. In the example, the rapid traverse movements G0 are hidden.

Lines G0, G1, G2, G3 displayed



Hide G0 lines

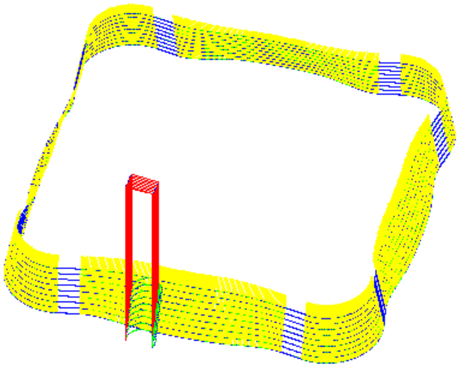


Display vectors, surfaces and curvature:

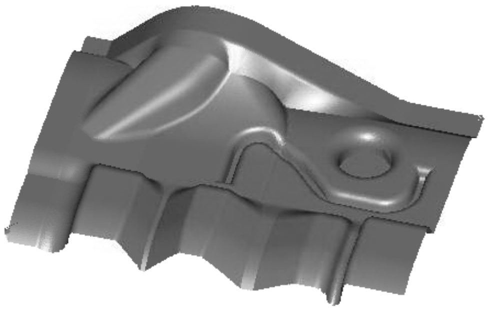
In addition to the classic view, for mold making programs, you can also display the rotary axis vectors and grid mesh (surface, mesh), for example.

▶ Press softkey **>>** ❸ and then **Vectors** or **Surface** or curvature. The appropriate view is displayed.

Vectors



Surface (mesh)

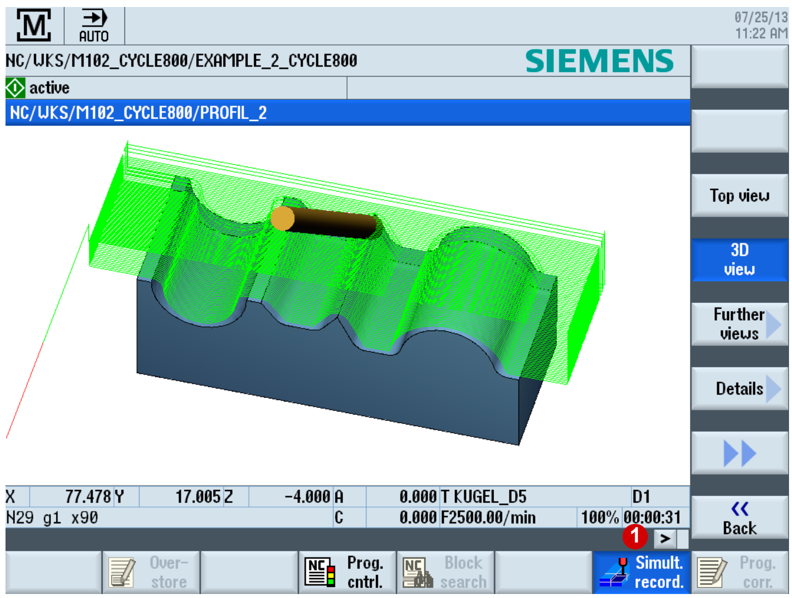


2.10.3 Simultaneous recording

In Automatic mode, i.e. during the program execution on the machine, the tool movements can be simultaneously recorded in the plan view, the three-side view or in the 3D view (with the 3D simulation option). This option is especially useful when the working area of the machine cannot be viewed clearly. During simultaneous recording, similar functions are available as for simulation.

Activate simultaneous recording:

- Before the NC program starts, press softkey **Simultaneous recording** ❶ .
- The execution of the program is simultaneously recorded after pressing CYCLE START. You can display the tool paths and the workpiece machining in parallel to the machining.



→ Top view

→ 3D view

→ Side views

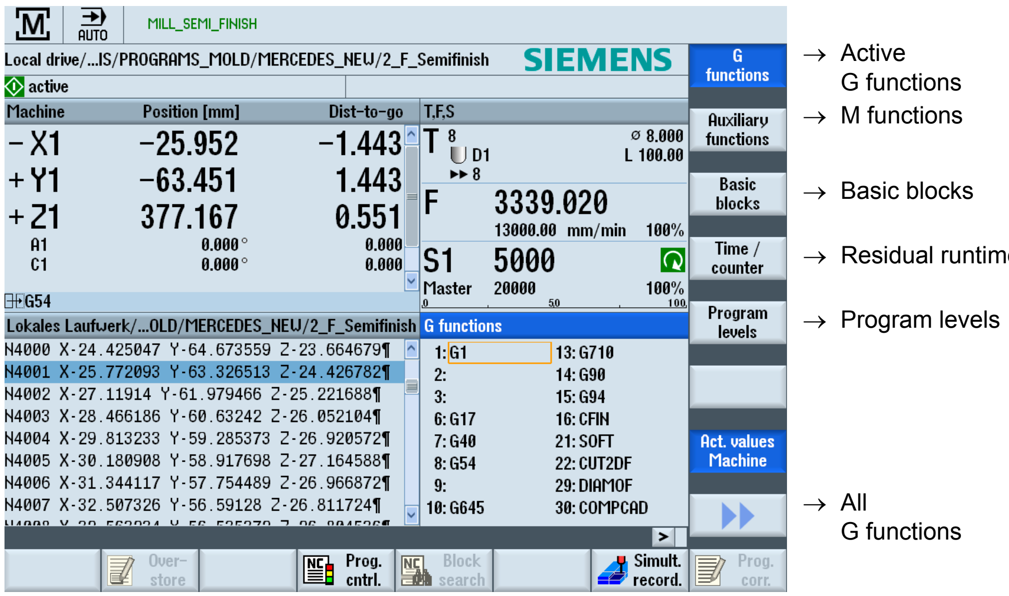
→ Zoom, magnify, turn, cut

→ Tool path displaying

2.11 Executing programs (AUTOMATIC)

SINUMERIK Operate provides functions that support the workflow during the execution of mold-making programs.

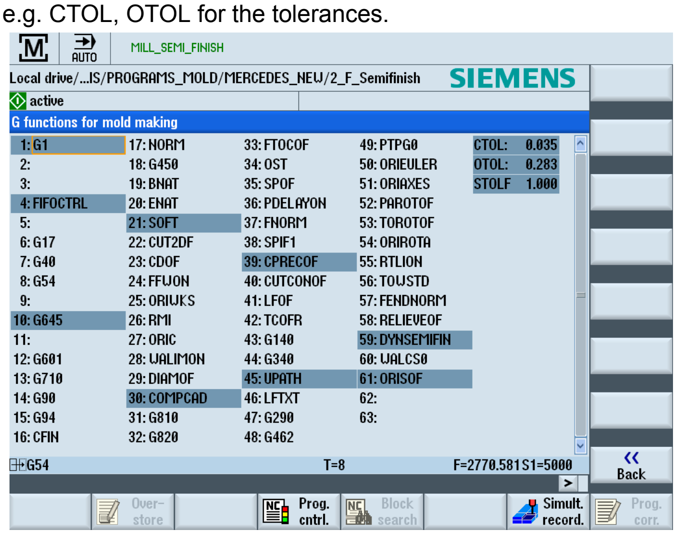
2.11.1 Display all relevant mold making functions



Selecting various displays during execution:

- Active G and M functions of up to 16 G groups.
- Basic blocks and residual runtime display of the program
- Program level for nested subprogram calls
- Current position and distance-to-go display to the target position

All relevant G code functions for mold making (Advanced Surface) can be displayed via the expanded softkey bar with the **All G functions** softkey. The G functions are highlighted in color, e.g. CTOL, OTOL for the tolerances.



2.11.2 Handwheel override

In the AUTOMATIC mode, while executing a program, small corrections and override feed of the tool in the tool direction are possible using a handwheel. When the orientation of the tool changes, the handwheel override that has been accumulated is also rotated. The manual correction acts in the form of override to the traversing motion from the NC program.

Handwheel override

Override with handwheel:

- Handwheel override of axes (DRF offset)
- Handwheel override in the tool direction (TOFFON function)
- Traverse axes in increments only

Active in the following states:

- NC/CYCLE stop
- Single block mode
- Override 0% and higher
- Programmed STOP M0

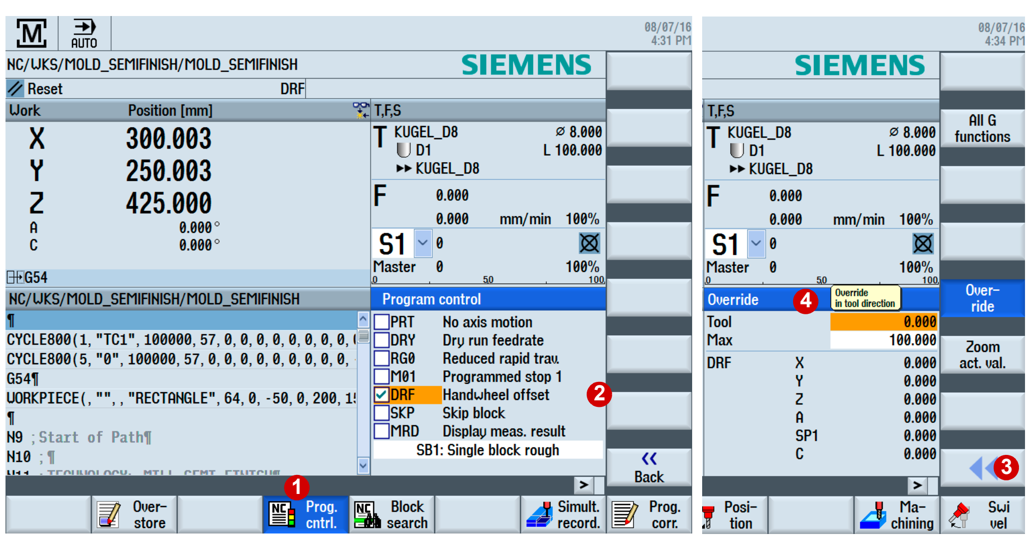
A typical application is, for example, changing the swivel head position (rotary axes) or the infeed depth in the tool direction, in order for example to mill a workpiece with tolerance.

Activating override

▶ In the AUTOMATIC mode, press softkey **Program control** ❶ and activate **DRF handwheel offset** ❷ .

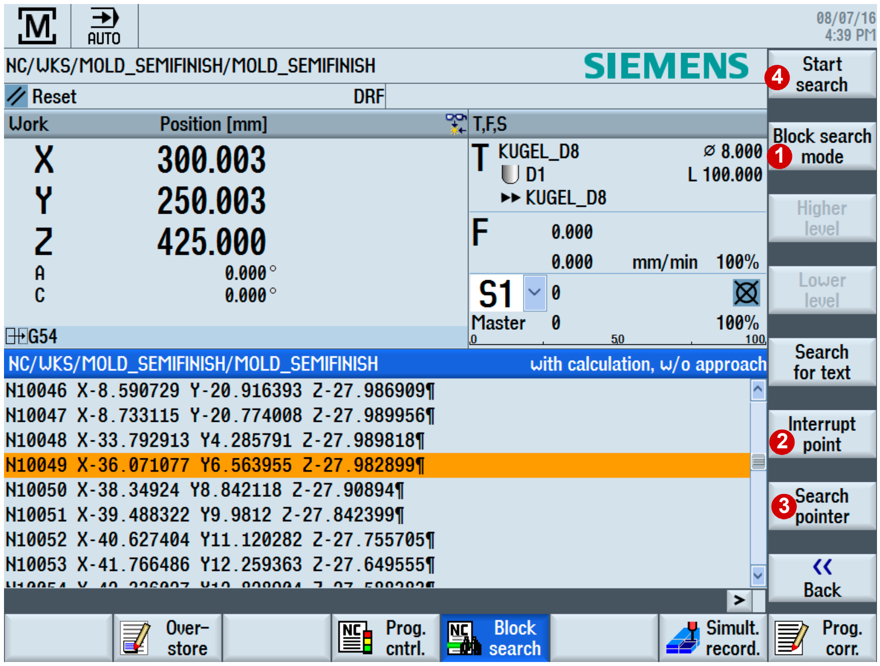
Displaying override

▶ Open the expanded softkey bar **>>** ❸ followed by softkey **Overrides** ❹ .



2.11.3 Block search

You can carry out a block search in the Reset machine state. The program data are prepared in such a way that all relevant parameters (tool, work offsets, M functions etc.) are available when accessing the program.

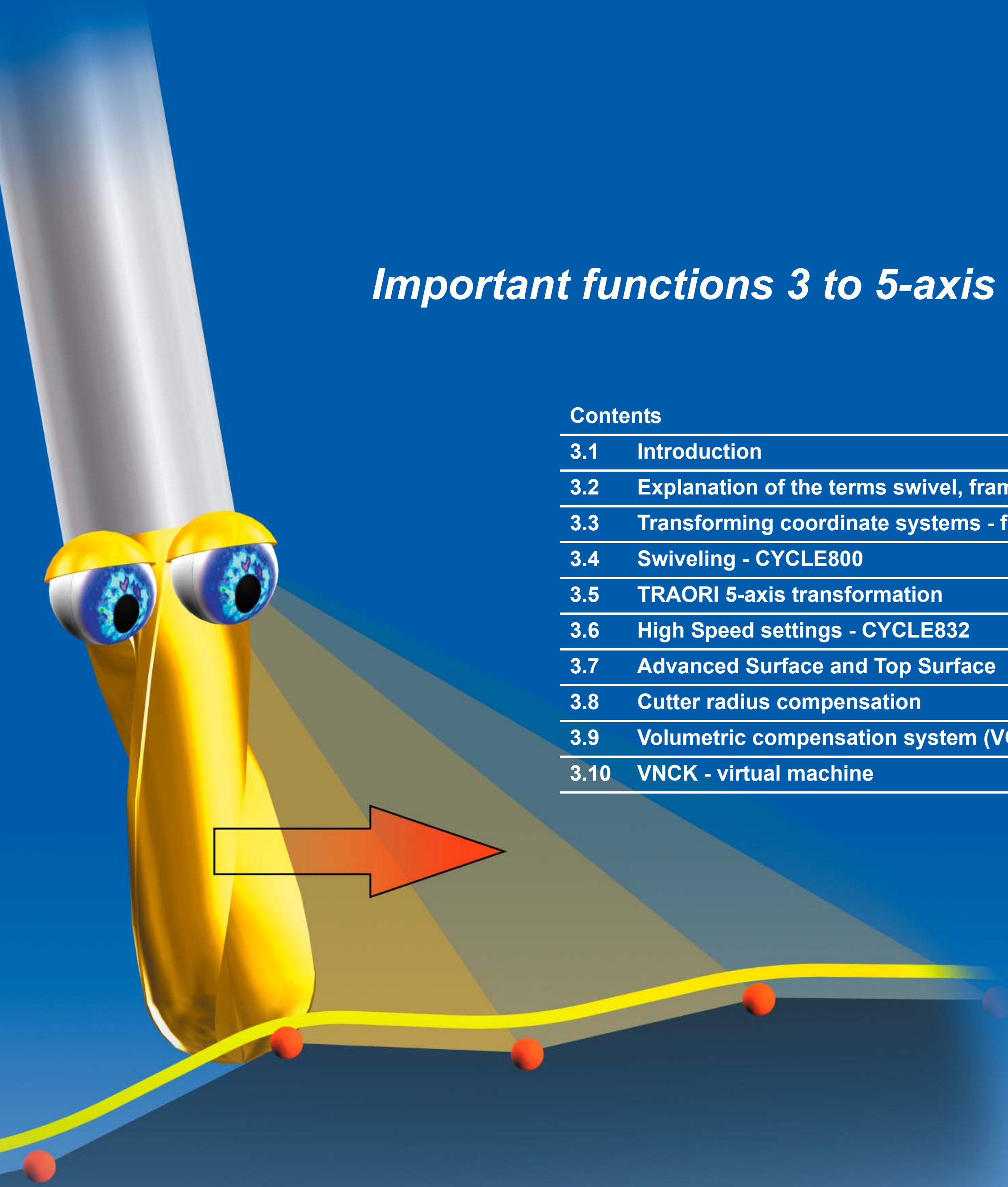


Possible such versions:

- Precise return to the point of interruption, even after power off
- To any CNC block in the DIN/ISO programs
- To any subprogram levels in DIN/ISO programs
- Machining step programs and position patterns in ShopMill
- Accelerated block search in large mold making programs

This is how you use the block search:

- ▶ You can individually configure the block search using softkey **block search mode** (1) e.g. with and without calculation or with/without approach.
- ▶ Using softkey **Interruption point** (2) , a jump is made to the last executed line of the NC program.
- ▶ Using the **search pointer** (3) , you can define the search target within an NC program, e.g. specific line number, mark or text.
- ▶ With **Start search** (4) , the search is carried out and using CYCLE START the program is executed at the search location.



Important functions 3 to 5-axis machining

Contents | Page

3

3.1 Introduction

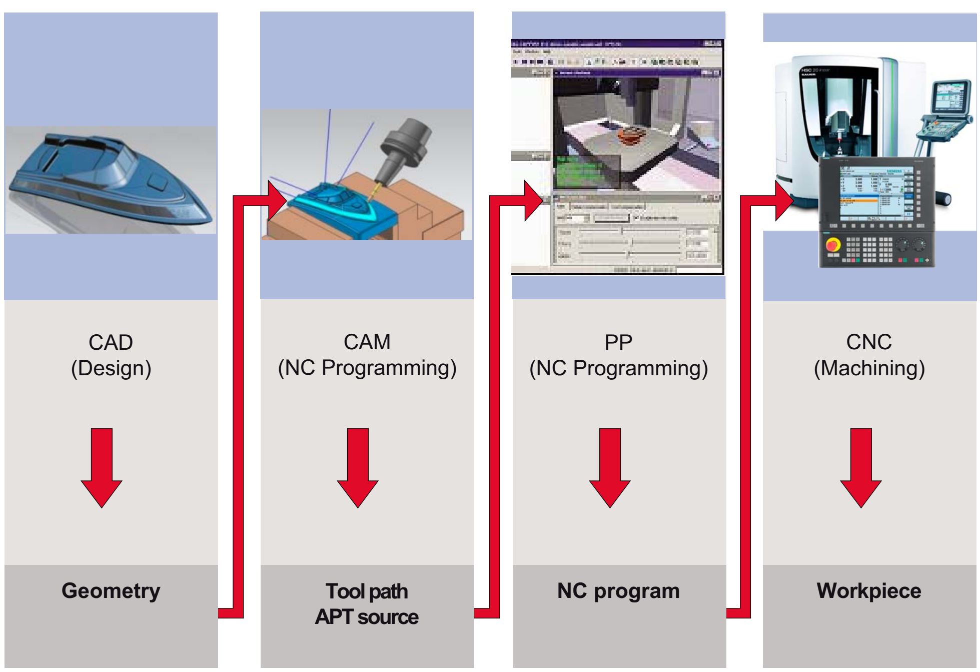
Especially within the context of mold making machining, the entire CAD/CAM/CNC process chain plays a major role in terms of ensuring optimum results on the machine.

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 into an executable NC code in the postprocessor.

The upstream postprocessor must support the programmable functions of the SINUMERIK control in order to use the performance capability of SINUMERIK controls to the full.

The postprocessor should ensure that the higher-order functions of SINUMERIK controls (as described in this section) are activated in the best possible way. An overview of all higher-order SINUMERIK functions is provided in the following sections.



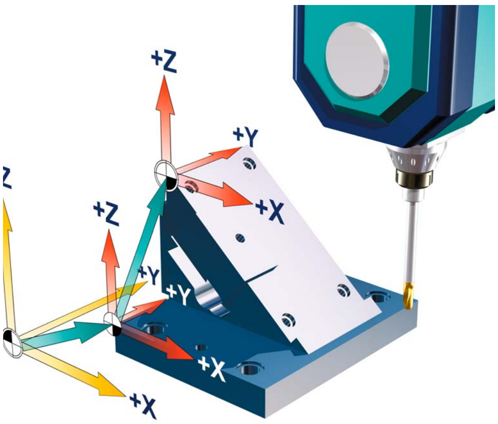
3.2 Explanation of the terms swivel, frames and TRAORI

The swivel (CYCLE800) and TRAORI features can be used to control rotary axes and angle the tool in relation to the machining surface. During swiveling, the rotary axes are positioned once and the linear axes X, Y, Z move during machining. The tool is oriented from a start position through to an end position and will then only work with the three linear axes. It is aligned in relation to the surface by means of a static process.

In contrast to swiveling, TRAORI is a dynamic process. The rotary and linear axes can be traversed simultaneously during machining. The orientation of the tool in space can be changed continuously. All axes (rotary and linear axes) are interpolated at the same time.

Frames only affect the coordinate system and only the coordinate system is modified, e.g. rotated, scaled, or moved. CYCLE800 takes the machine kinematics into account during swiveling, i.e. attention is paid to tool offsets and zero points. In the case of frames (e.g. ROT), these must be taken into account by the user.

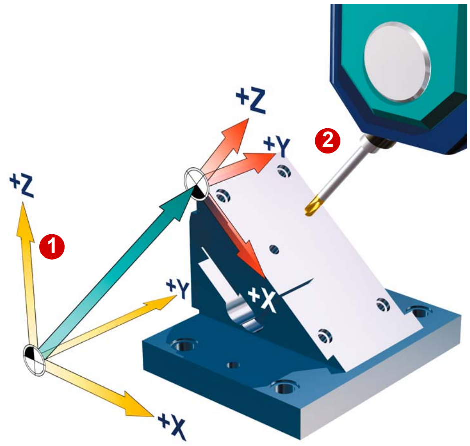
Frames



The frame is a self-contained arithmetic rule that transforms one Cartesian coordinate system into another Cartesian coordinate system.

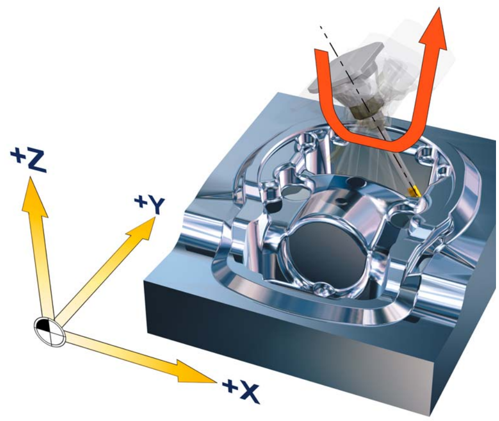
In the example, the coordinate system is twice subjected to translatory movement. This is the case, for example, if you set the zero point at the corner of the workpiece by means of a work offset.

Swiveling



The tool is aligned with the machining surface by moving the rotary axes. This example involves rotation of rotary axis B ❶ and the tool is positioned with respect to the XY plane ❷ Machining then takes place in this plane.

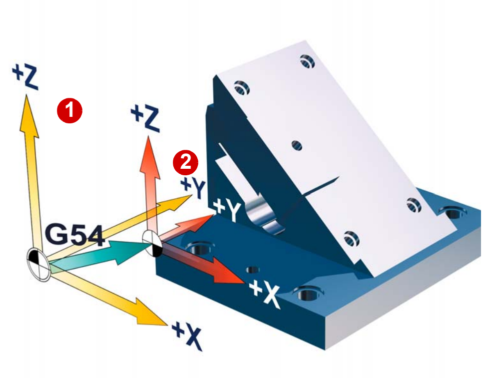
TRAORI



The tool is aligned with the machining surface dynamically during the milling process by means of linear and rotary axis interpolation. The tool length is taken into account and the kinematic compensating movements are initiated by the TRAORI function when the rotary axes are rotated.

3.3 Transforming coordinate systems - frames

Coordinate systems



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

Using frames, the coordinate systems can be shifted, rotated, mirrored and scaled so that they can be aligned with any workpiece surface. This allows the programming overhead to be reduced to a minimum.

Frames are used to describe the position of a target coordinate system by specifying coordinates and angles starting from the actual workpiece coordinate system ❷ .

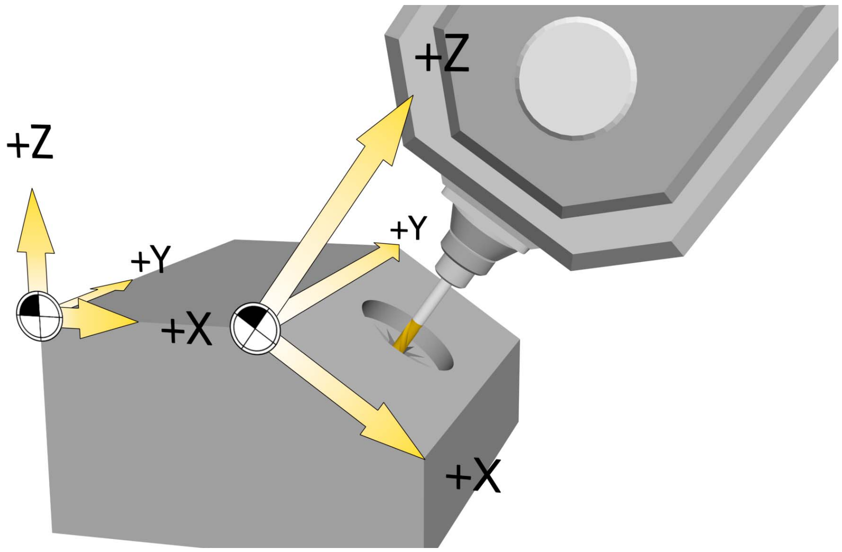
Possible Frames Include

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

Using frames it is possible to machine surfaces that can be shifted and rotated in space as required. All subsequent traversing commands now relate to the new workpiece coordinate system shifted using frames.

3.4 Swiveling - CYCLE800

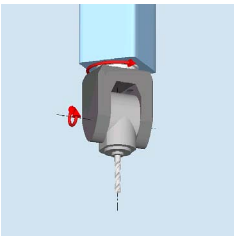
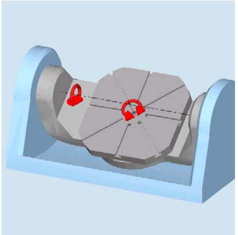
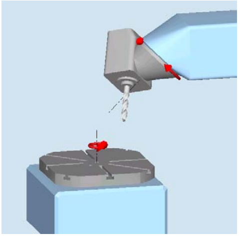
Function



You can use swivel heads or swivel tables to set up and machine inclined planes. Swiveling is possible in both JOG and AUTOMATIC modes. Swivel operation parameter assignment and programming are facilitated by the clearly laid out graphics. You can either program all the swivel axes directly on the machine (A, B, C) or you can simply specify the rotations 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 swivel axis of the machine during machining.

The swivel axes are always rotated in such a way that the machining plane is perpendicular to the tool axis for machining. During machining, the machining plane is permanently set.
When the axes are swiveled, the active zero points and tool offsets are automatically converted for the swiveled state, resulting in a new coordinate system.

Machine kinematics

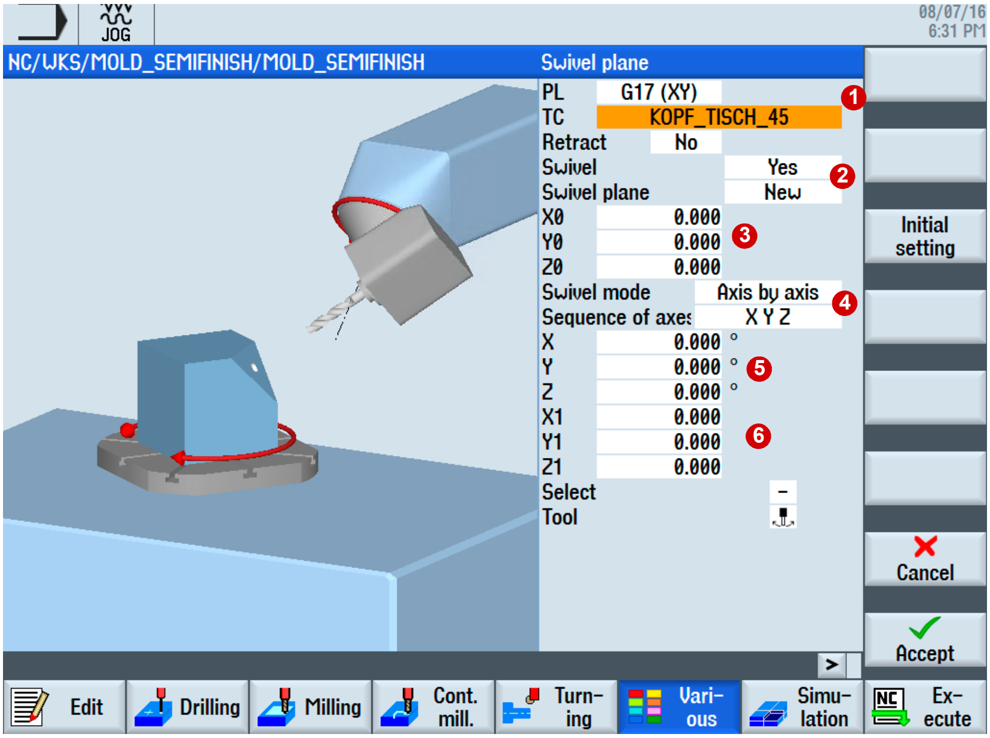
Swivel head (type T)	Swivel table (type P)	Swivel head + swivel table (type M)
Swiveling tool carrier	Swiveling workpiece holder	Mixed kinematics
		

Procedure for programming swivel motion and subsequent machining:

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

Basic procedure for using the swivel cycle

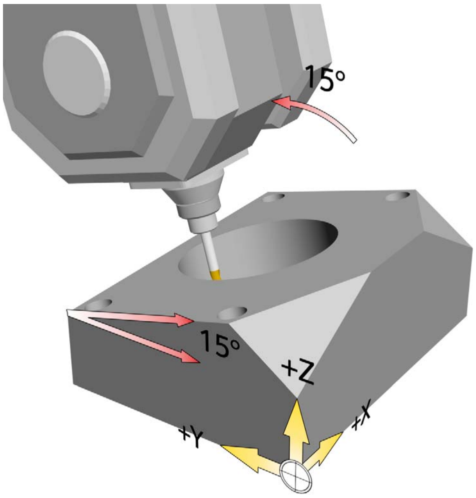
- Call the **swivel** function in the program.
- Select the name of the swivel data set (1).
- Select "yes" for swivel if you wish to perform a swivel movement. Select "new" as swivel movement if you wish to perform a new swivel movement, or "additive" if you wish to base the movement on a previous swivel movement (2).
- Specify the reference point before rotation (X0, Y0, Z0) (3).
- Select the "axis-by-axis" swivel mode directly or via the projection 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)
- Shift the zero point on the swiveled plane (6).

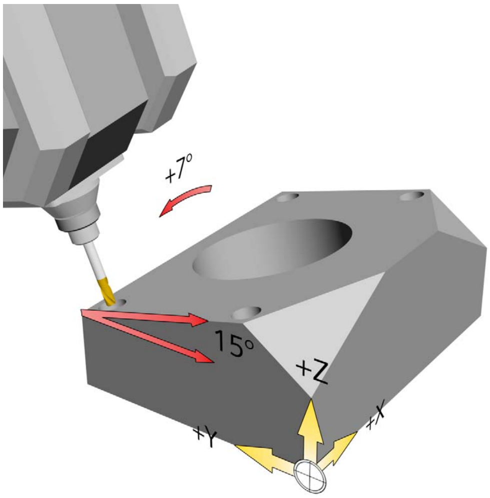


Program example for swiveling in programGUIDE

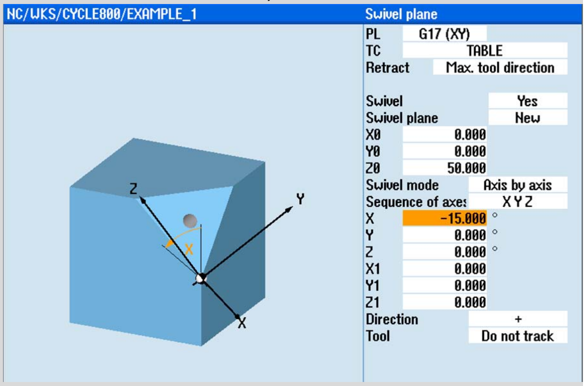
Standard milling and drilling cycles are applied to swiveled machining surfaces in the following example.

- Face milling of the workpiece.
- Swiveling the machining plane by X=-15 degrees and milling a circular pocket.
- Swiveling through Y=-7 or +7 degrees, face grooving with milling cutter for the holes and machining two holes at angles of +7 degrees and two holes at -7 degrees.



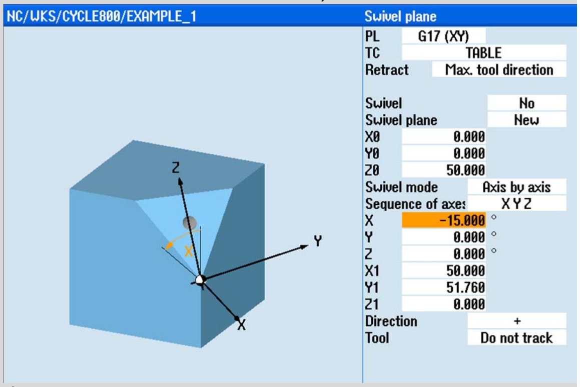



```
N100   ; Swiveling the plane to the basic position
N110   CYCLE800(4,"TABLE",200000,57,0,0,0,0,0,0,0,0,0,1,,1)
N120   ; Raw part definition for simulation
N130   WORKPIECE(,,,"BOX",112,0,51,-80,-2.5,-2.5,102.5,102.5)
N140   T10 D1;  T="MILLER_D32"
N150   M06
N160   S5000 M03
N170   G0 G54 X0 Y0
N180   TRANS Z50 ;Work offset in Z workpiece upper edge
N190   ;Face milling of workpiece in basic position
N200   CYCLE61(10,1,5,0,0,0,100,100,1,20,0,2000,32,0,1,0)
N210   ;Swiveling into the plane_1  X=-15 DEGREES
N220   CYCLE800(4,"TABLE",200000,57,0,0,50,-15,0,0,0,0,0,1,,1)
```

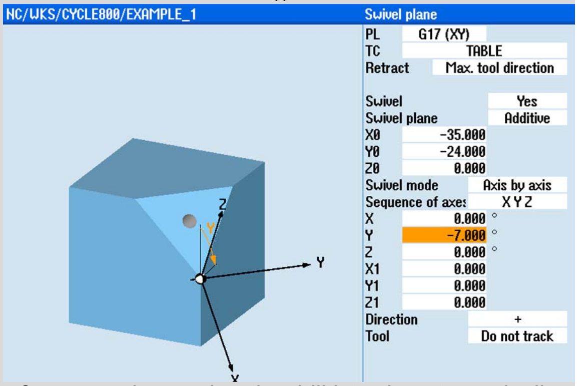



```
N230   ;Face milling of workpiece in Plane_1
```

```
N240  CYCLE61(35,25.8,5,0,0,0,100,103.6,5,20,0.2,2000,31,0,1,0)
N250  T11 D1; T="CUTTER_D16"
N260  M06
N270  S8000 M03
N280  G54 X50 Y51.76 M08; Pre-position on center of pocket
N290  ; Circular pocket roughing and finishing Plane_1
N300  POCKET4(10,0,2,-
      15,40,50,51.76,5,0.1,0.1,2000,2000,0,21,80,0,,10,2.5,0,,,10100,111,10)
N310  POCKET4(10,0,2,-
      15,40,50,51.76,15,0.1,0.1,1000,1000,0,22,80,0,,5,2.5,0,,,10100,111,10)
N320  T12 D1;  T="CUTTER_D10"
N330  M06
N340  S6000 M03 F500
N350  G54 X0 Y0 M08
N360  ; Calculate swiveling Plane_1 with new center point X1, Y1 X=-15 degrees
N370  CYCLE800(4,"TABLE",220000,57,0,0,50,-15,0,0,50,51.76,0,0,,1)
```

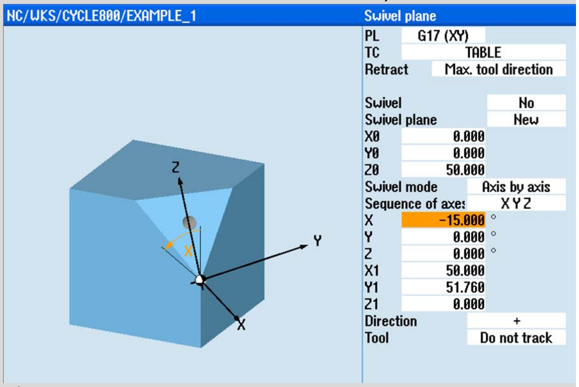



```
N380  ; Swiveling Plane_1 additive Y=-7 degrees
N390  CYCLE800(4,"TABLE",200001,57,-35,-24,0,0,-7,0,0,0,0,1,,1))
```

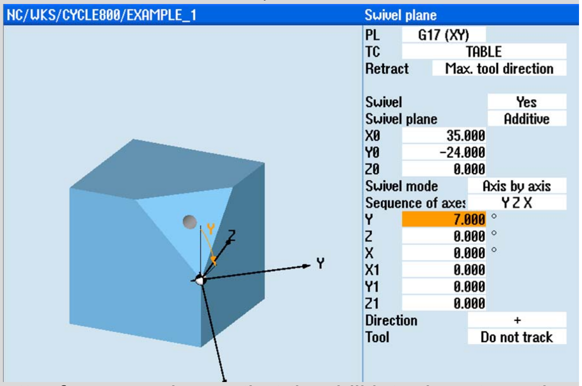



```
N400  ; Area -7 degrees with milling cutter face grooving so that the drill later inserts vertically
N410  MCALL CYCLE82(10,0,5,-3,,1,0,1,12)
N420  G0 X0 Y0 M8
N430  X0 Y60
N440  MCALL
N450  ; Calculate swiveling Plane_1 with new center point X1, Y1 X=-15 degrees
```

N460 N141 CYCLE800(4,"TABLE",220000,57,0,0,50,-15,0,0,50,51.76,0,0,,1)

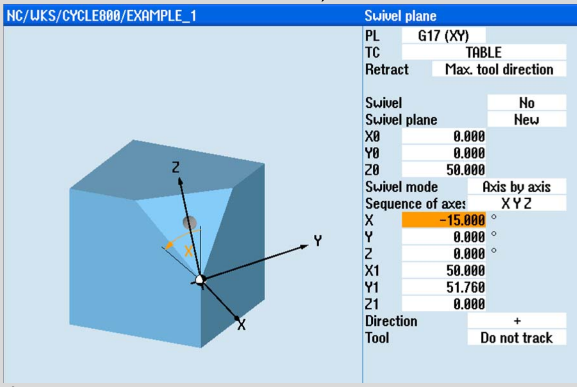


N465 ; Swiveling Plane_1 additive Y=+7 degrees
N470 CYCLE800(4,"TABLE",200001,30,35,-24,0,7,0,0,0,0,0,1,,1)

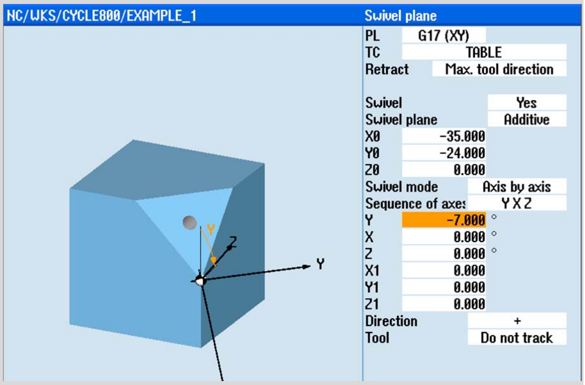


N480 ; surface +7 degrees with milling cutter face grooving so that the drill later inserts vertically
N490 MCALL CYCLE82(10,0,5,-3,,1,0,1,12)
N500 G0 X0 Y0 M8
N510 X0 Y60
N520 MCALL
N530 T13 D1; T="DRILL_D8.5"
N540 M6
N550 S4000 M3 F500 D1
N560 G54 G0 X0 Y0 M8
N570 ; Deep-hole drilling D8.5 Y=+7 degrees
N580 MCALL CYCLE83(10,0,2,-20,,-5,,054,0,100,0,0,4,1,0,0,0,1,11121112)
N590 G0 X0 Y0
N600 X0 Y60
N610 MCALL
N615 ; Calculate swiveling Plane_1 with new center point X1, Y1 X=-15 degrees

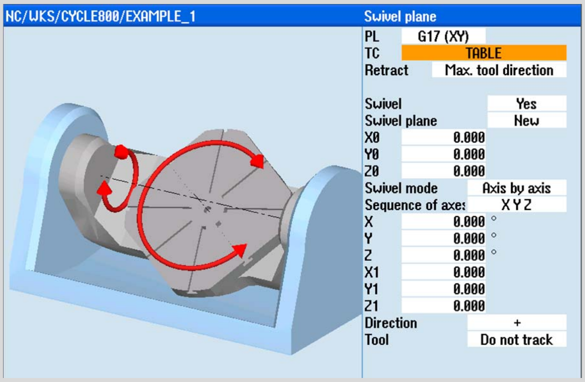
```
N620  CYCLE800(4,"TABLE",220000,57,0,0,50,-15,0,0,50,51.76,0,0,,1)
```




```
N625  ; Swiveling Plane_1 additive Y=-7 degrees
N630  CYCLE800(4,"TABLE",200001,54,-35,-24,0,-7,0,0,0,0,0,1,,1)
```




```
N640  ; Deep-hole drilling D8.5 with -7 degrees
N650  MCALL CYCLE83(10,0,2,-20,,-5,,054,0,100,0,0,4,1,0,0,0,1,11121112)
N660  G0 X0 Y0
N670  X0 Y60
N680  MCALL
N690  ; Swiveling the plane to the basic position
N700  CYCLE800(2,"TABLE",200000,57,0,0,0,0,0,0,0,0,0,1,,1)
```




```
N710  ; Deselection CYCLE800
N720  CYCLE800(0,"0",110000,57,,,,0,0,0,,,,0,,0)
N730  M30; end of program
```

3.5 TRAORI 5-axis transformation

Inclined planes can be machined with the aid of CYCLE800. The tool is inclined relative to the workpieces so that inclined workspaces can be machined. The inclination is performed at the start of the machining and is not changed during the machining of the inclined plane. However, in many case this static inclined position is not sufficient. A constantly changing tool orientation relative to the workpiece is required. So that this can be performed in a clear and user-friendly way, the 5-axis transformation, which is activated by the TRAORI command, has been introduced.

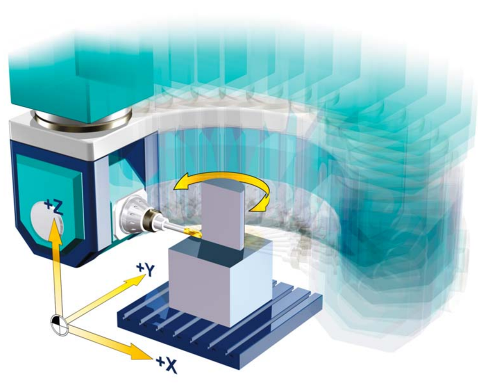
3.5.1 Basics of simultaneous 5-axis machining

The various kinematics of 5-axis machines were described in Section 1.5. A distinction is made between three basic types.

- Machines with a rotatable tool which is swiveled by means of two rotary axes.
- Machines with a rotatable workpiece which is swiveled while the orientation of the tool remains fixed in space.
- Machines of the "mixed" type on which one rotary axis rotates the tool and the second rotary axis swivels the workpiece.

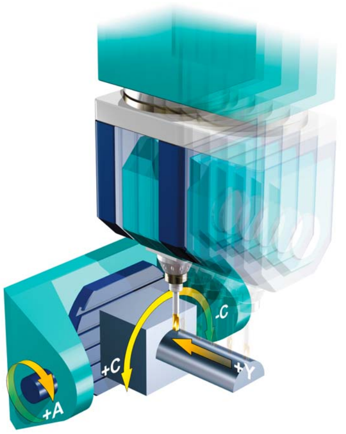
Machine kinematics and machine motion

Motion sequence for head/head kinematics



A semicircle must be described in the X/Y plane for one cycle. At the same time, the tool must be rotated through 180° around Z if the tool is always to be perpendicular to the surface. Therefore, for the machining of one cycle, 3 axes, X, Y and C, must move.

Motion sequence for table/table kinematics

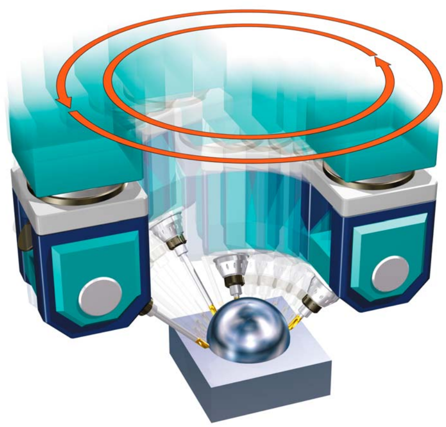


At the start, the A axis must be rotated through 90°. The tool is then perpendicular to the surface. For one cycle, the C axis swivels from +90° to -90°. Therefore, for one cycle (a semicircle), only the C axis moves.

It is clear from the example that totally different machine movements produce the same machining result. If we look at the movements more exactly, we can see that in both cases the movement of the tool tip on the workpiece and the tool orientation relative to the surface are identical.

Effect of the tool length on machine movement

To investigate the effect of the tool length on the machine movement, let us consider a further example. The base circle of the half sphere shown in the graphic is to be machined with an inclined tool.



In the graphic, you can see that different tool lengths result in different machine movements. To machine the circle on the lower part of the half sphere, traversing must be performed in a circle in the X-Y plane and also the C axis must be rotated through 360° during the circular motion. Whereas the circle for the tool tip always remains the same size irrespective of the tool length, the circle that the machine axes must describe increases with the tool length. This is indicated by the two red circles that are produced with different tool lengths.

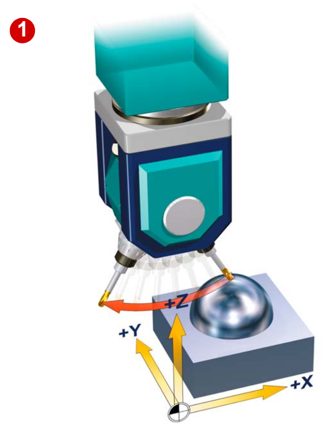
It is clear that when programming machine axis positions, a different program must be created for each tool length, while the movement of the tool tip always remains the same.

Rotary axis movement and tool orientation

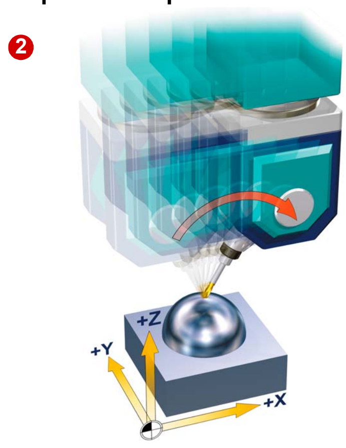
The rotary axes in 5-axis machines are used to enable the various orientations of the tool relative to the workpiece. The movement of a rotary axis normally not only changes the tool orientation, but also the position of the tool tip in relation to the workpiece surface if this is not prevented by a suitable mechanism.

.

Tool tip moves



Tool tip fixed in space

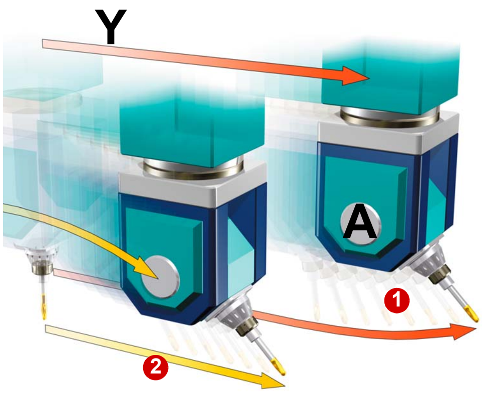


The example ❶ shows that the movement of the A axis not only changes the tool orientation to the workpiece. At the same time, the tool tip moves in space, a circular path in the Z-X plane in our example. This is an unwanted side effect.

Generally, the behavior shown in ❷ is desirable, where compensatory movements of the linear axes ensure that the tool tip is fixed in space during rotary axis motion.

Simultaneous movement of linear/rotary axes

This effect is also shown during simultaneous movement of linear and rotary axes.



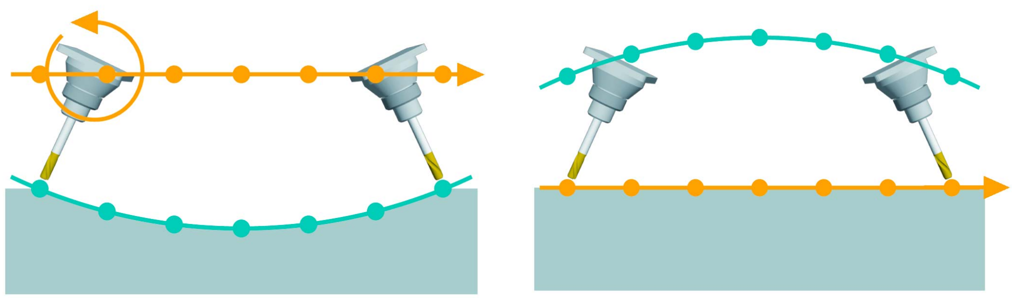
The graphic shows the simultaneous movement of the A axis and the Y axis. In situation ❶ both axes move in synchronism; they are linearly interpolated with one another. A straight path of machine axes Y and A has been programmed. Such a motion command **G1 Y… A…** produces the red, curved path for the tool tip.

As a rule however, the behavior as described in situation ❷ is desirable. A straight line with regard to the tool tip is wanted for the G1 programming even when the A axis moves simultaneously. In this case, the machine axis movement follows the curved path. The Z axis also moves to hold the tool tip in a straight line.

Chord error

The following graphic illustrates the so-called chord error which occurs when movements that should be defined with regard to the tool tip, are programmed for the machine axes.

.



The graphic on the left shows the result of the simultaneous change of the tool orientation through movement of the A axis and the Y axis via linear interpolation of both axes. The orange line is the programmed straight line in the Y direction. If the A axis is also moved, this results in the green line for the movement of the tool tip. If a certain path is to be maintained for the tool tip as in the example on the right-hand side, then position changes to the tool tip that result from movements of the rotary axis must be compensated.

Summary of simultaneous 5-axis machining

The simultaneous 5-axis machining enables the adaptation of the tool orientation synchronously to the linear movement in space. Without the additional control functionality of the 5-axis transformation, this produces some unwanted side effects.

- New NC programs for identical workpieces must be created for each machine type.
- The NC program must be adapted for changes to the tool length.
- During the programming of the machine axes, overlaid movements of the rotary and linear axes occur which cause a linearization error for the path of the tool tip, the chord error.
- Tool orientations are programmed and changed via the positions of the swivel axes. An orientation programming and interpolation in workpiece coordinates is not possible without suitable control functions.
- Technologically, feedrates should always refer to the relative movement between the workpiece and tool, which is not the case without 5-axis transformation. The programmed feedrates refer to the velocity of the machine axis movements.

Normally, NC programs are created in relation to the workpiece, i.e. all the tool positions relate to the workpiece coordinate system (WCS). In order for an NC program to be executed on the machine, the positions must be transformed into axis movements, i.e. converted into the machine coordinate system (MCS). The SINUMERIK is equipped with the TRAORI function to enable this kind of transformation.

3.5.2 The TRAORI command

The TRAORI command activates the 5-axis transformation. This enables the simultaneous 5-axis machining without the disadvantages described in the previous section. The programming of TRAORI changes the reference variables in the NC program. The coordinates of the X, Y, Z linear axes now refer to the position of the tool tip.

If orientation changes are programmed in addition to the linear positions, the effects of the rotary axis movements on the position of the tool tip are compensated by movements of the linear axes.

In addition to the explicit programming of swivel axis positions, TRAORI enables the use of orientation information which allows also the orientation to be programmed and interpolated independently of the kinematics.

If tools are used with a different length, the new tool data is automatically taken from the tool table, calculated and the programmed feedrate always refers to the movement of the tool tip relative to the workpiece.

Programming

```
TRAORI(n)        ; transformation on
TRAFOOF          ; transformation off
```

Explanation of the commands

TRAORI	Activates the first configured orientation transformation.
TRAORI(n)	Activates the orientation transformation configured with n.
n	The number of the transformation (n = 1 or 2), TRAORI(1) corresponds to TRAORI.
TRAFOOF	Deactivate transformation

TIP

Depending on the configuration (depending on the machine manufacturer), TRAORI can reset the active work offset (e.g. G54) and the tool edge compensation (D1). It is therefore recommended that the work offset and the tool edge compensation be activated again after TRAORI is called up.

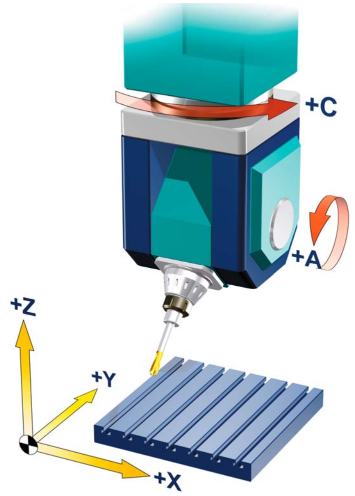
3.5.3 Programming the tool orientation

For an active 5-axis transformation, the orientation of the tool relative to the workpiece must be programmed in addition to the position of the tool tip. This can be performed in several ways.

Rotary axis positions (A=<value> B=<value> C=<value>)

The most common way is to program the positions of the swivel axes directly. In this way, the tool orientations are always specified implicitly, which however are directly dependent on the machine kinematics.

Programming rotary axes



The tool orientation in space is clearly defined through the specification of the rotary axis positions for the C and A axes.

However, this also depends on the kinematics type.

The NC program would then appear as follows:

N020 TRAORI

…

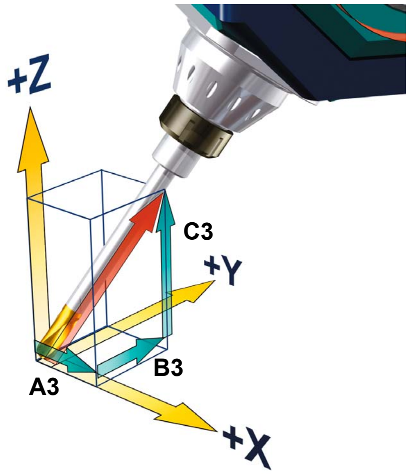
N140 G1 A-45 C0

A commonly used machine type is shown in the graphic. However, there are also other basic kinematics with the C and A axes as table axes or in a mixed arrangement, and in each of these machine types different tool orientations are associated with the specified pair of C and A axis positions. This means that the same problem exists with regard to the tool orientation as for the linear movements. The solution there was that TRAORI enable the position of the tool tip to be programmed independently of the machine kinematics. In the same way, kinematic-independent options have been introduced for the tool orientation programming.

Direction vector (A3=<value> B3=<value> C3=<value>)

The components of a vector in space are programmed during the programming of direction vectors. Whereby A3 is associated with the X component, B3 is associated with the Y component and C3 is associated with the Z component; the length of the vector is of no significance. For example, A3=1 B3=1 C3=1 defines a vector with the components (1,1,1) which points in the direction of the spatial diagonal of the coordinate system.

Programming direction vector



The components of the direction vector are programmed with A3, B3, and C3. The vector points towards the tool adapter; the length of the vector is of no significance. Vector components that have not been programmed are set equal to zero.

```
N020 TRAORI
N035 G54
N040 G1 X0 Y0 Z0 A3=1 B3=1 C3=1 F10000
...
```

The example shows the tool tip in the position (0,0,0), and the tool shank is the diagonal of a cube (35.26° in relation to X-Y plane).

The vectors clearly correspond to the I, J, K vectors used in CAM systems. However, as I, J, K are reserved for the circle parameters in the CNC standard, the addresses A3, B3 and C3 are used for the vector components in SINUMERIK.

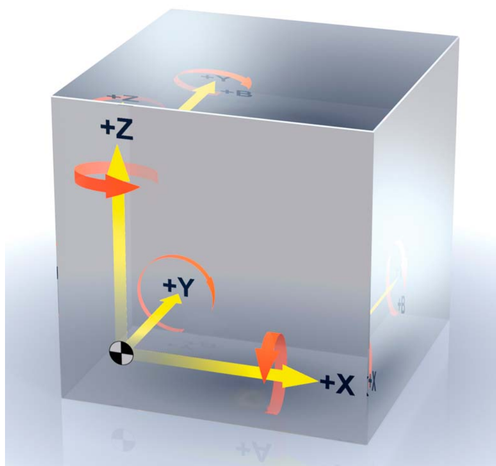
TIP

It is recommended that the direction vector is used. The precision should be selected as high as possible. As far as 5-axis programs are concerned, practical experience has shown that good results can be achieved by using 5 decimal places for linear axes and 6 decimal places for the direction vector.

Use of Euler or RPY angles (A2=<value> B2=<value> C2=<value>)

Instead of direct vector programming, abstract rotation angles can be used for the orientation description. Normally, Euler angles or RPY angles (roll, pitch, yaw) are used to rotate coordinate systems around the coordinate axes.

Euler and RPY

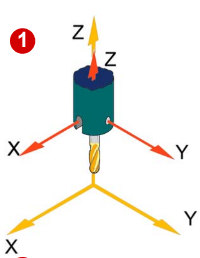


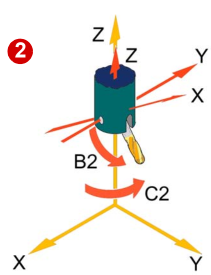
With Euler angles, rotation is first around the Z axis, then around the rotated X axis and finally around the resulting Z axis. There are different definitions for RPY angles.

The following is used in SINUMERIK: The first rotation is around the Z axis, the second around the rotated Y axis and the third around the rotated X axis.

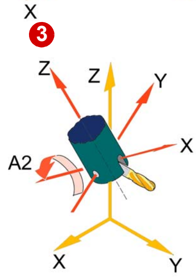
They can also be used accordingly for the definition of a vector. This is based on a vector in the Z direction of the coordinate system that is rotated using Euler or RPY angles. The resulting vector is the new tool vector. The NC addresses A2, B2 and C2 are used to program the rotation angles. The meaning of the angles, i.e. whether they are Euler angles or RPY angles, is specified via the G codes ORIEULER and ORIRPY. This results in the following definitions:

Programming RPY angle





with C2=90° rotated around Z axis with B2=45° rotated around rotated Y axis



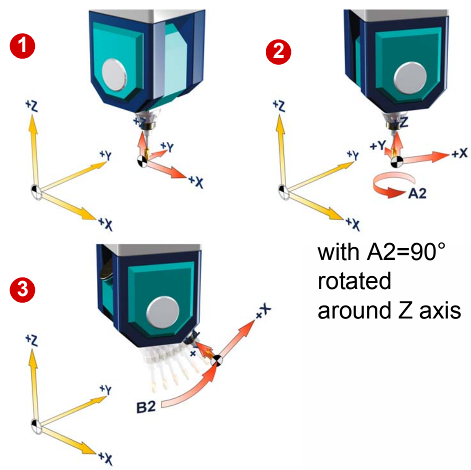
with A2=30° rotated around rotated X axis

The values programmed with A2, B2, C2 for orientation programming are interpreted as an RPY angle (in degrees).

Originating from the (1) basic position: The orientation vector results from turning a vector in the Z direction firstly with C2 around the Z axis (2), then with B2 around the new Y axis (3) and finally with A2 around the new X axis **Z, Y', X**"). In contrast to Euler angle programming, all three values here have an effect on the orientation vector.

Example:
N020 TRAORI
N030 G54
N040 G0 X0 Y0 Z0
N050 C2=0 B2=0 A2=0 F10000
N060 C2=90 B2=45 A2=30
N070 ...

Programming Euler angle



with A2=90° rotated around Z axis

with B2=45° rotated around rotated Y axis

The values programmed during orientation programming with A2, B2, C2 are interpreted as Euler angles (in degrees).

Originating from the ❶ basic position: The orientation vector results from turning a vector in the Z direction firstly with A2 around the Z axis, ❷ then with B2 around the new X axis ❸ and lastly with C2 around the new Z axis (**Z, X', Z''**).

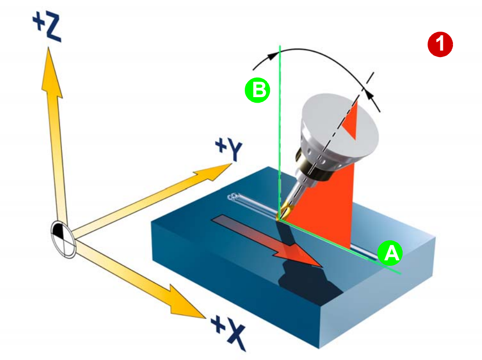
Example:
N020 TRAORI
N030 G54
N040 G0 X0 Y0 Z0
N050 G1 A2=0 B2=0 F1000
N060 G1 A2=90 B2=45
N070 ...
In this case, the value of C2 (rotation around Z axis) is irrelevant and does not need to be programmed.

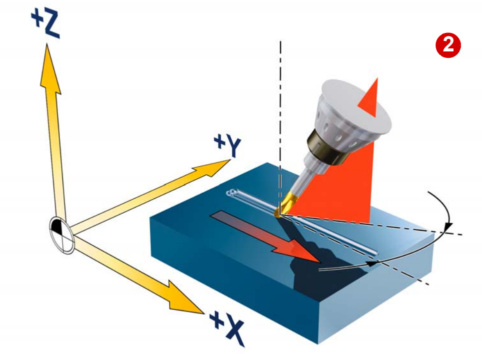
Lead and tilt angles (LEAD/TILT)

In conjunction with LEAD and TILT,the orientation is programmed via a lead angle and a tilt angle. This is the only orientation programming variant that is linked to a certain orientation interpolation for the type of orientation definition, namely ORIPATH.

LEAD	**Lead angle** for programming tool orientation. Angle relative to the surface normal vector in the plane put up by the path tangent and the surface normal vector.
TILT	**Tilt angle** for programming the tool orientation. The TILT angle defines the rotation of the lead angle around the surface normal vector

LEAD and TILT





The resultant tool orientation is determined from:

- Path tangent A
- Surface normal vector B
- Lead angle LEAD 1
- Tilt angle TILT at end of block 2

LEAD describes the angle between the surface normals and the new tool orientation in the plane set up by the path tangent and the surface normals 1 .

The tilt angle corresponds to a rotation of the tool in the plane perpendicular to the surface normals 2 .

The lead angle is often used to orient the tool at a fixed setting angle to the machining plane. This increases the cutting performance. For example, a ball mill touches the surface to be machined precisely with the tool tip when the orientation is perpendicular. However, the rotation and therefore the cutting speed is 0 at this point of the milling cutter.

The following program sequence corresponds approximately to that shown in the graphic.

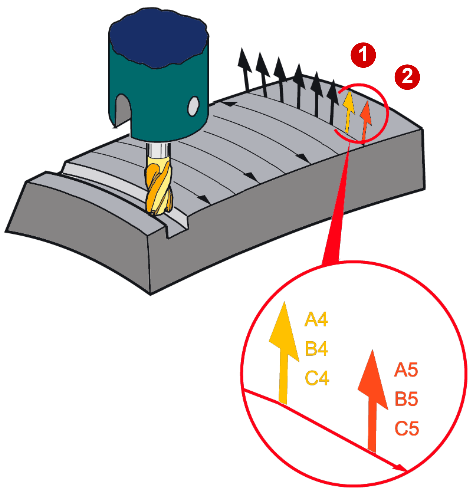
N10 TRAORI

N20 G1 X=0 Y0 A3=0 B3=0 C3=1

N30 ORIPATH

N40 X10 LEAD=30 TILT=10 C5=1

As already mentioned, the lead and tilt angles are only defined in conjunction with the surface normal vector. This is defined in the same way as the orientation vector with the components A4, B4, C4 or A5, B5, C5: The first designate the normal vector at the start of the block, the latter at the end of the block.



The surface normal vector is perpendicular

to the machining surface.

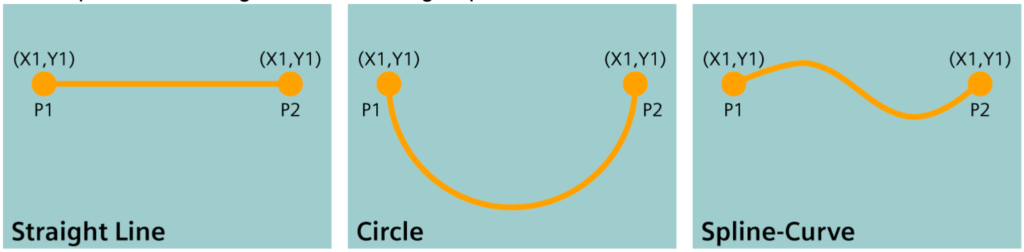
The start vector at the **start of the block** is programmed with A4, B4, C4. The vector at the **end of the block** is programmed with A5, B5, C5.

3.5.4 Orientation interpolation and orientation reference

In the NC program, the tool orientation relative to the workpiece is specified block-by-block by vectors (A3, B3, C3) or angles ((axis positions A, B, C), Euler or RPY angles (A2, B2, C2) or (LEAD/TILT)). In 5-axis simultaneous machining, these normally change in each block. If the machine is now to be moved in accordance with the specifications of the NC program, the orientation must be continuously changed synchronous to the path motion.

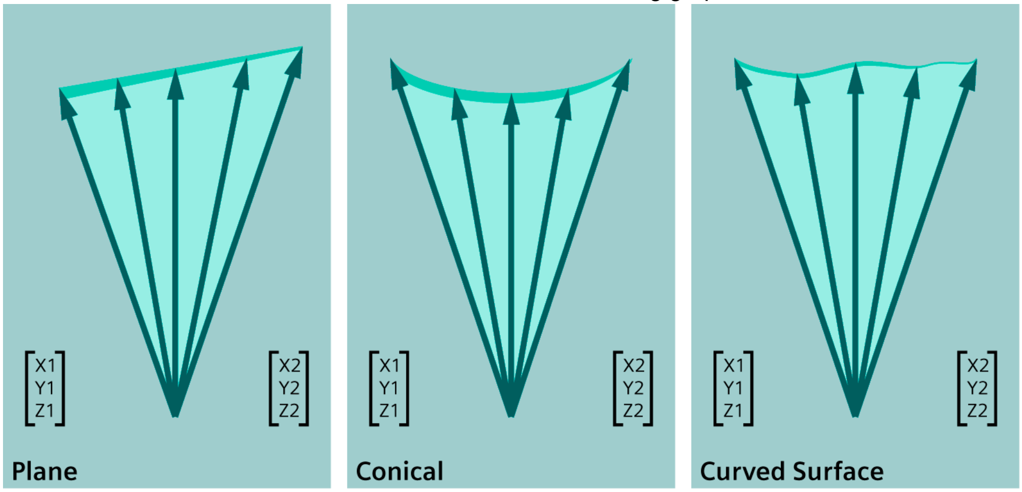
Motion interpolation in the 3-axis area

New X, Y and Z coordinates are specified block-by-block. However, there are obviously numerous ways to move a tool from one position to the next. This can be performed along a straight line (G1), on an arc (G2, G3) or also on a spline curve (ASPLINE, BSPLINE, CSPLINE). In each case, there is a different movement although the programmed coordinates are identical. Therefore, to clearly describe the desired movement, it is not sufficient to only specify the coordinates, the way in which traversing is performed from one coordinate to the next must also be specified. This is performed using the G codes of group 1.



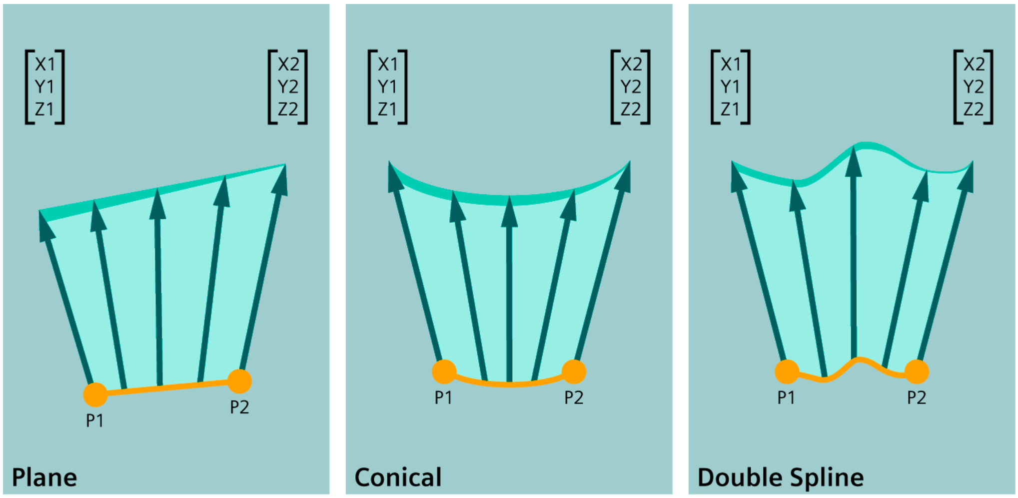
Interpolation of the tool orientation in a point

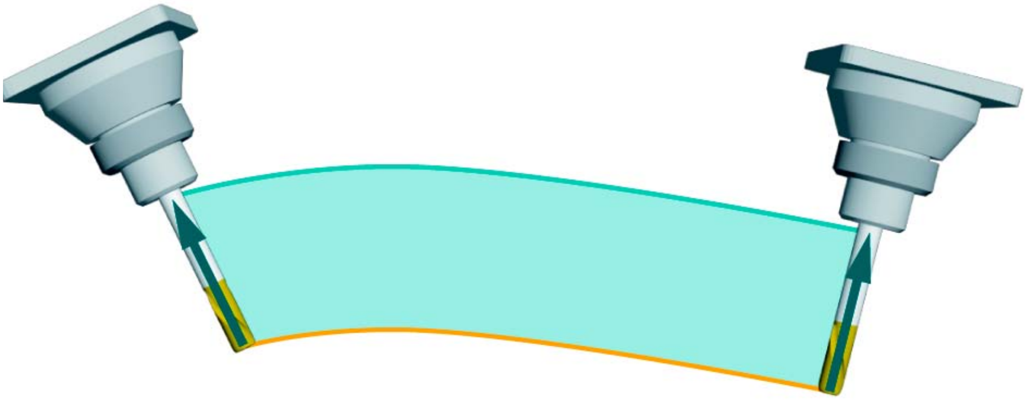
In the same way, there are numerous ways to move form one tool orientation (X1, Y1, Z1) to the next (X2, Y2, Z2). This can be performed by moving the vector in a plane, on a conical surface or an otherwise curved surface. These are shown in the following graphics.



Tool orientation in the 5-axis area

To change a tool orientation from the start orientation of the start block (X1, Y1, Z1) to the target orientation of the next block (X2, Y2, Z2), it is also not sufficient to only specify the coordinates. How the vectors move must also be defined. In a plane, on a conical surface or an otherwise curved surface. The G codes of group 51, ORIVECT, ORIAXES, ORICONXX, ORICURVE, etc. are used for this purpose.



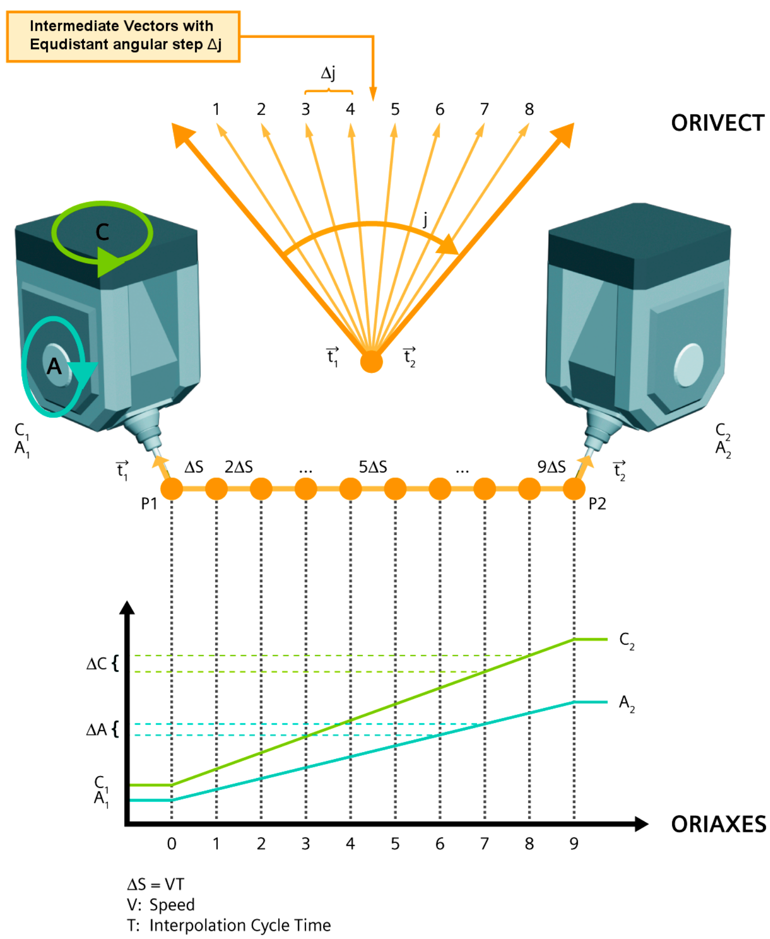


This is best seen when the position of the tool tip and the tool orientation are changed simultaneously. This can generally be described by two curves. The orange line describes the movement of the tool tip while the green line defines the movement of a second point on the tool. These two curves clearly describe the surface on which the tool moves. This fact is used for the 3D spline interpolation ORICURVE by displaying the orange and the green lines through spline curves.

Comparison of ORIVECT and ORIAXES

To illustrate the differences more clearly, let us consider the two most commonly used cases ORIVECT and ORIAXES. As already mentioned, ORIAXES is the standard interpolation type that is most commonly used. The rotary axes are interpolated linearly synchronous to the movement of the tool tip. Let us consider the following situation to illustrate this type.

The following figure shows the movement of a fork head machine with C and A axes along an NC block.



We will begin at the start of the block with position P1, the tool vector t1 and the corresponding rotary axes positions C1 and A1. The position, the tool vector and the rotary axes positions at the end of the block are specified with P2, t2, C2 and A2. For the interpolation of the tool tip with the programmed speed, the distance to the end point is divided into sections of equal size, whose length ΔS corresponds to the interpolation cycle and the programmed feedrate. The increment is $\Delta S = VT$, whereby V is the velocity and T the interpolation cycle time. At constant speed, the entire traversing distance D between P1 and P2 is divided into 9 equidistant interpolation increments.

In the same way with ORIAXES, the difference of the rotary axis positions C2-C1 and A2 - A1 is divided into 9 equidistant increments C and A. This means that the curve of the rotary axis positions linearly follows the path motion.

The circumstances are different with ORIVECT. The vector is interpolated and not the rotary axes. For this purpose, the differential angle J between the start vector t1 and the end vector t2 is considered. This is divided into 9 equidistant increments ∆J in accordance with the path motion. The intermediate vectors defined as a result are all in a plane set up by t1 and t2. A surface is therefore defined in workpiece coordinates, in this case a plane, on which the orientation vector moves. The possible orientation interpolations are described in the following section.

Orientation reference of the coordinate system (ORIMKS, ORIWKS)

The reference for the rotary axis interpolation is defined using the G code commands ORIMKS/ ORIWKS.

ORIMKS **Tool orientation in the machine coordinate system.**

With **ORIMKS**, the programmed orientation relates to the coordinate system defined by the machine axes. Rotations through frames are not taken into consideration here.

ORIWKS **Tool orientation in the workpiece coordinate system.**

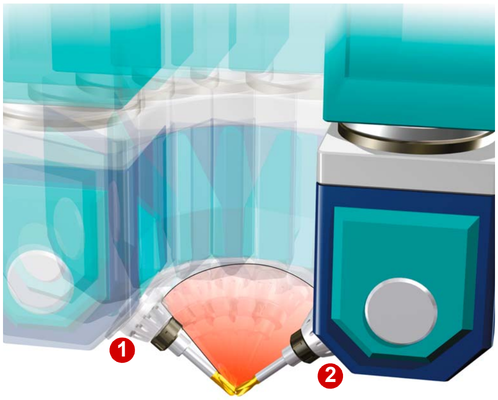
For **ORIWKS**, the programmed orientation relates to the workpiece coordinate system which may be rotated in relation to the machine coordinate system using a frame.

Orientation reference	
ORIMKS	The reference system for the orientation vector is the machine coordinate system.
ORIWKS (recommended)	The reference system for the orientation vector is the workpiece coordinate system. Machine data is used to determine precisely what happens.

Orientation interpolation of the axes

Linear interpolation (ORIAXES)

Linear interpolation (ORIAXES)

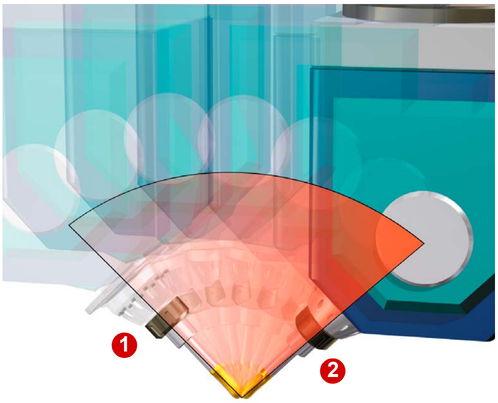


The rotary axes are interpolated linearly synchronous to the path motion. In this interpolation type, the orientation curve depends on the machine kinematics. It can be used in all cases in which it is not necessary that the tool moves on a precisely defined surface in the workpiece coordinate system. This is the case, for example, in molding making when ball mills are used.

	Axis/linear interpolation
ORIAXES (recommended)	Linear interpolation of the machine axes

Vector interpolation (ORIVECT/ORIPLANE)

Vector interpolation Large circle interpolation (ORIVECT/ORIPLANE)

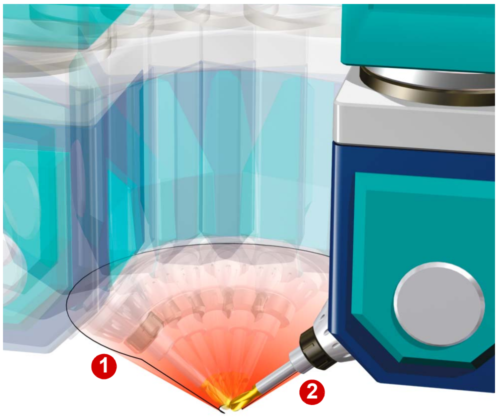


The tool vector always moves in a plane set up by the start and end vectors. The interpolation type is frequently used in the milling of aircraft structures because the pockets mostly have flat and inclined walls. It is also recommended when using toroidal millers for face milling.

	Vector interpolation
ORIVECT	Interpolation of the orientation vector in a plane (large circle interpolation)
ORIPLANE	Interpolation in a plane (large circle interpolation), identical to ORIVECT

Cone surface interpolation (ORICONxx)

Cone surface interpolation (ORICONCW)

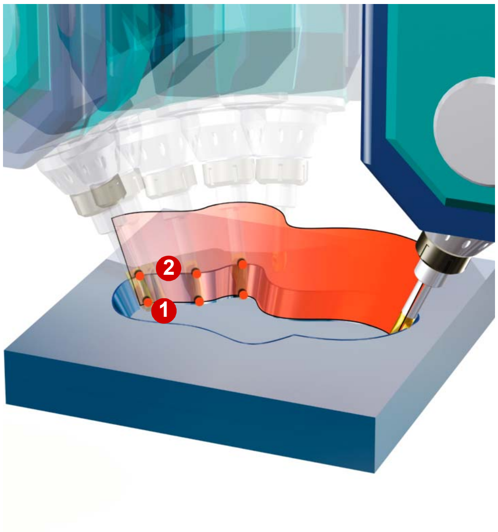


With cone surface interpolation, during reorientation the tool moves on a programmable peripheral surface of a cone located anywhere in space.

	Vector interpolation
ORICONCW	Interpolation on the peripheral surface of a taper in the clockwise direction Programming: Circle end point with radius G2 X.. Y.. Z.. CR= End orientation vector: A3= B3= C3= or A2= B2= C2= Vector of the cone axis of rotation: A6= B6= C6= Opening angle of the cone (PSI), value 0 -180 degrees: GROOVE=
ORICONCCW	Interpolation on a peripheral surface of a cone in the counterclockwise direction Programming: Circle end point with radius G3 X.. Y.. Z.. CR= End orientation vector: A3= B3= C3= or A2= B2= C2= Vector of the cone axis of rotation: A6= B6= C6= Opening angle of the cone (PSI), value 0 -180 degrees: GROOVE=
ORICONIO	Intermediate orientation via A7=… B7=…, C7=…. Programming: Circle end point: CIP X.. Y.. Z.. Circle intermediate point I1=.. J2=.. K1=.. Intermediate orientation vector: A7=.. B7=.. C7=.. End orientation vector: A3= B3= C3= or A2= B2= C2=
ORICONTO	Interpolation on a peripheral surface of a cone with tangential transition. Programming: Circle end point: CT X.. Y.. Z.. End orientation vector: A3= B3= C3= or A2= B2= C2=

Spline interpolation (double spline ORICURVE)

Spline interpolation
Curve interpolation
(ORICURVE)



With spline interpolation, the movement of the orientation vector is defined by the tool tip's path 1 and a path of a second tool tip 2.

If, for example, you wanted to use circumferential milling to create an inclined surface, you would define the surface on which the milling cutter should move using two spline curves at the top and bottom ends of the tool (1, 2).

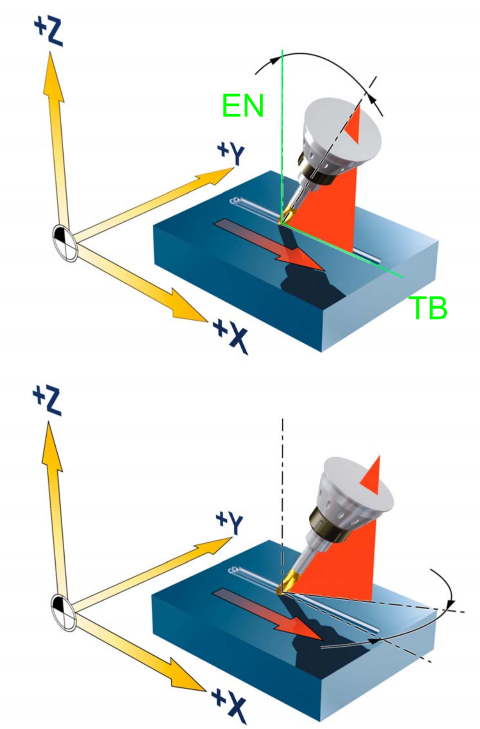
The advantage of this is that it enables you to define a wide variety of surfaces for machining with a high degree of precision.

Spline interpolation is the best type of interpolation available, but it is also the most involved and calls for special syntax in the NC program that must be supported by the CAM system.

	Spline / double spline interpolation
ORICURVE	Orientation interpolation with specification of the tool tip motion and that of a second point on the tool. The path of the second point is defined via XH=… YH=… ZH=…, in conjunction with BSPLINE as a control polygon with POLY as polynomial: PO[XH] = (xe, x2, x3, x4, x5) PO[YH] = (ye, y2, y3, y4, y5) PO[ZH] = (ze, z2, z3, z4, z5) If the BSPLINE or POLY additional information is omitted, straightforward linear interpolation will be performed accordingly between the start and the end orientation.

Path-related interpolation (ORIPATH)

Path-related interpolation



The orientation vector is set relative to the path and to the surface normals. The programming is performed using the LEAD angle and the TILT angle in conjunction with the surface normal vector. LEAD describes a rotation around the direction perpendicular to the tangents and normals while TILT describes a rotation of the orientation around the surface normal vector.

The orientation interpolation with ORIPATH is recommended when programming on the CAM system during the machining of free-form surfaces in tool and mold-making. The prerequisite for this is support of the issuing of surface normals of your CAM system.

	Path-related interpolation
ORIPATH	Tool orientation in relation to the path. This is used with the commands LEAD and TILT and the specification of the surface normals with A4, B4, C4 and A5, B5, C5.

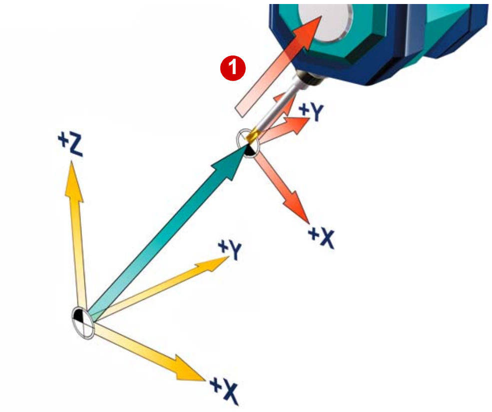
3.5.5 Example application involving TRAORI and frames

Example TOROT - retraction from a slanted hole

When 5-axis transformation is active, TOROT generates a frame whose Z axis coincides with the current tool orientation. This function can be used with a 5-axis program, for example, to retract the tool after a tool breakage without collision, simply by retracting the Z axis. Following tool orientation with TOROT, all the geometry axis movements programmed will relate to the frame generated as a result of this function.

Programming TOROT in MDI

N110	TRAORI	; Activate TRAFO
N120	TOROT	; Calculate and select retraction frame
N130	G1 G91 Z50 F500	; Retraction by 50 mm in Z direction in a straight line
N140	M17	; end of subprogram



A frame containing the current tool orientation in the Z direction is generated (1). In the JOG mode, the tool can be retracted from the workpiece in the Z direction.

As an alternative to traversing incrementally in MDI mode, in JOG mode you can use the direction key to enable retraction in the tool direction.

Notice:
Retraction is only possible in JOG mode if the machine has been configured accordingly (Z axis serves as the geometry axis).

TOROT after machining plane

TOROT for G17

TOROTX for G18 -> tool axis X

TOROTX for G19 -> tool axis X

TiP

TOROT must be deselected prior to the next program start: TOROTOF.

TiP

A similar function is available in JOG with manual retraction. After an interruption of a tapping operation (G33/G331/G332) or a general drilling operation (tools 200 to 299) due to power loss or a RESET at the machine control panel, you have the possibility to retract the tool in the tool direction without damaging the tool or the workpiece.

3.6 High Speed settings - CYCLE832

Application

You can influence the sequence of CAM programs using SINUMERIK CYCLE832. It provides technological support for 3- and 5-axis machining in the high-speed cutting (HSC) range.

When executing CAM programs in the HSC range, the control needs to achieve high machining feedrates. By applying different machining strategies, you can use CYCLE832 to fine-tune the program for execution.

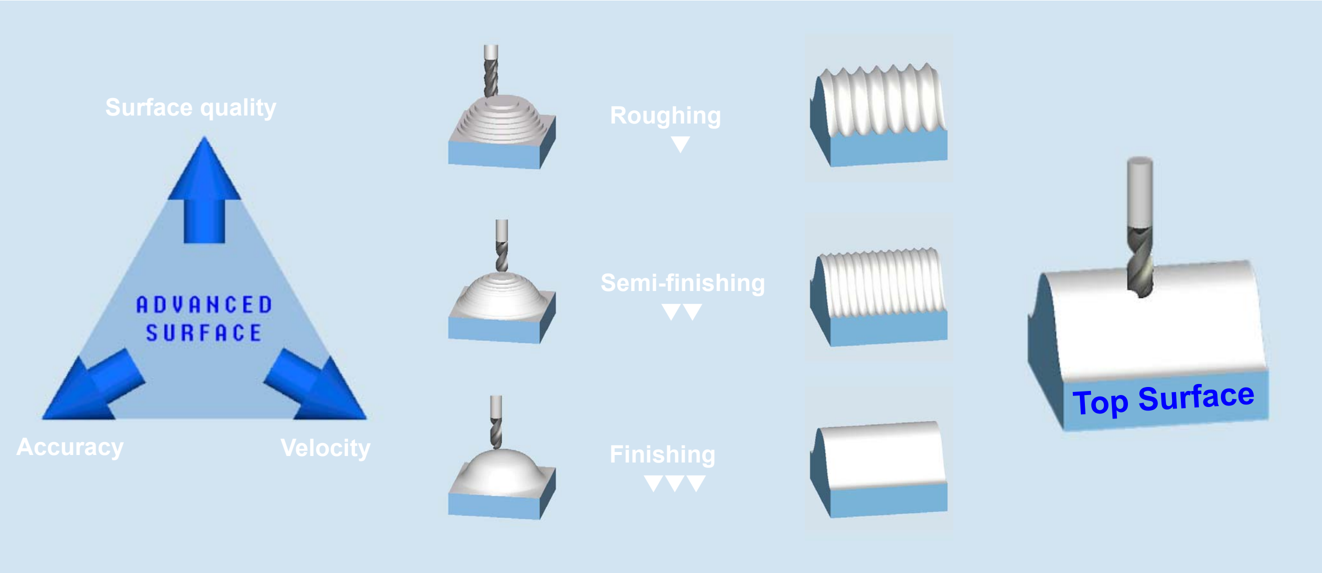
- For **roughing** by blending the contour, the emphasis is placed on speed.
- For **finishing,** emphasis is placed on surface quality and precision.

In both cases, specifying a tolerance ensures that the correct machining contour is achieved in order to obtain the desired surface quality and accuracy. Generally, for roughing the tolerance is greater than for finishing.

Advanced Surface and Top Surface

For CYCLE832, from software release V4.7 SP2, you can select between **Advanced Surface** and option **Top Surface**.

Top Surface is an option of CYCLE832 - and is based on the COMPSURF compressor. This has been improved for machining demanding CAD/CAM programs. The new compressor especially compensates the potential negative impact on the surface quality for an inhomogeneous point distribution, as is the case for inclined line-by-line finishing programs and for bidirectional milling. The result is a uniformly perfect surface, which is essentially independent of the quality of the CAD/CAM data.



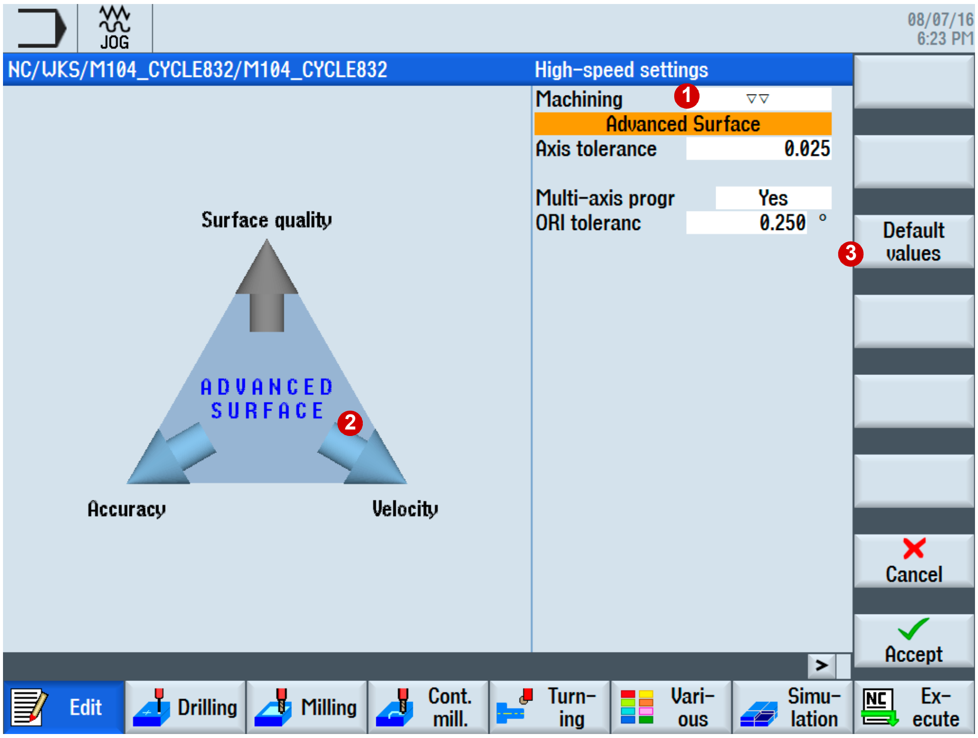
3.6.1 Parameters for CYCLE832

CYCLE832 can be set by the machine operator or, within the context of NC program generation, by the postprocessor or programmer. Dynamic values and NC commands are adapted to address specific user requirements and they depend on the settings of the machine data (machine builder).

In CYCLE832, the operator can select between the four different types of machining of the technology group Dynamic G group 59 and activate their dynamic parameters.

CYCLE832 with Advanced Surface based on the well proven COMPCAD compressor. The new COMPSURF compressor is used for Top Surface. Top Surface when milling inclined line-by-line contours.

With innovated lookahead functionality, as well as harmonizing velocity profiles of adjacent paths and optimized acceleration and jerk behavior, CYCLE832 guarantees extremely good surface quality with short machining times.

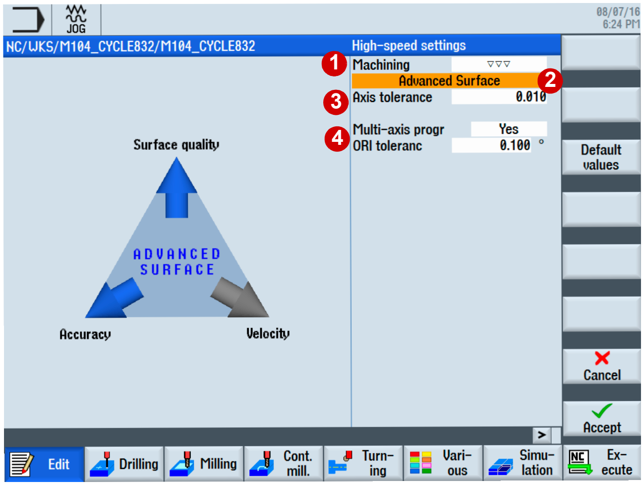


By selecting the machining strategy, the CAM program execution is optimized for the machine.

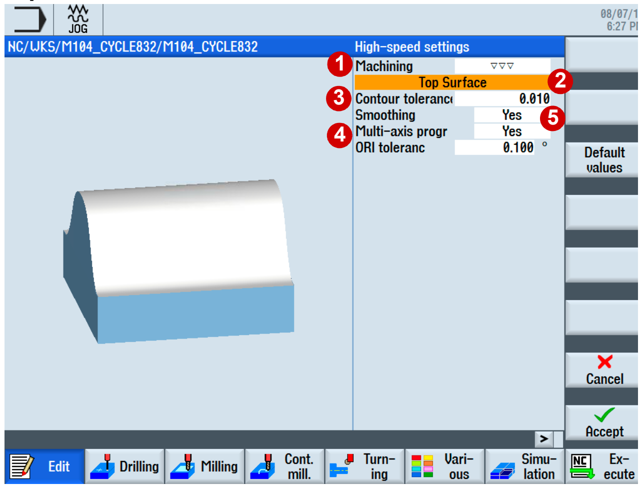
Corresponding to the selected machining, 1 the blue arrows 2 either point in the direction **Speed**, **Surface quality** or towards **Accuracy**. The cycle parameters are preassigned, depending on the selected machining, using softkey **Default values** 3 .

Parameters for the High-Speed setting cycle

Advanced Surface



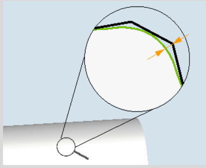
Top Surface

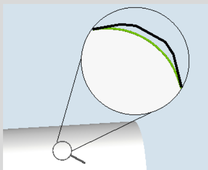


Parameterizing CYCLE832

- Under Machining, select between parameter deselection, finishing, semi-finishing and roughing.
- If the Top Surface option is activated, then you can select between the two motion controls. Advanced Surface is set as default setting.
- Enter a value in field **Axis/contour tolerance**.
- For **multi-axis programs,** from SW4.5, you can smooth the orientation characteristic in the cycle and apply an **ORI tolerance**.
- For Top Surface, you can additionally activate **smoothing**. When smoothing, inhomogeneous areas long the path are smoothed.

CYCLE832 parameters (as of SW4.7 SP2)

Machining	■ Finishing (_FINISH) ■ Semi-finishing (_SEMIFIN) ■ Roughing (_ROUGH) ■ Deselection (_OFF)	Selection for **3-axis programs** with plain text. Choosing "Deselection" resets the values to the default values.
	■ Finishing (_ORI_FINISH) ■ Semi-finishing (_ORI_SEMIFIN) ■ Roughing (_ORI_ROUGH)	Selection for **multi-axis programs** with orientation tolerance in plain text.
Tolerance_tol. 	■ Chord tolerance, contour tolerance, axis tolerance (Chord tolerance must be imported from the CAM system 1:1)	Tolerance of the linear axes

Smooth-ing 	■ Yes (_TOP_SURFACE_SMOOTH_ON) ■ No (_TOP_SURFACE_SMOOTH_OFF)	**Yes**: Optimized path within the contour tolerance Inhomogeneous areas in the path in the standard tolerance range of 10µm are reliably smoothed. The workpiece assumes a shiny appearance. **No**: Path close to contour The specified contour is precisely followed. Engraving and geometries with a depth <= 0.005 mm = 5µm are visible on the workpiece surface.
Multi-axis program	■ No ■ Yes	**No**: For 3-axis programs without rotary axes. The value 1 is entered automatically **Yes**: For multi-axis programs with rotary axes. The orientation smoothing is activated; ORI tolerance 0° must be entered.
ORI tolerance	■ OTOL tolerance (approx. factor 10 of the linear tolerance)	Has two meanings: ■ Smoothing tolerance of the rotary axes for 5-axis transformation TRAORI ■ Orientation tolerance for the smoothing of vectors with active ORISON

NOTE

CYCLE832 is based on the use of G1 blocks, also G0, G2 and G3, depending on the machine setting.

NOTE

When using the ORI tolerance, only for Advance Surface, after calling CYCLE832 does orientation smoothing activation have to be programmed using ORISON. You should not program ORISON for 5-axis hobbing/circumferential milling as this can result in contour errors. For Top Surface, ORISON is not programmed after calling CYCLE832.

CYCLE832 structure

CAM systems frequently specify via the postprocessor that CYCLE832 is output directly in the geometry program. Therefore, make sure that CYCLE832 is programmed in the geometry program for your start program. CYCLE832 is then automatically selected and deselected in the subprogram. The start program then only contains the calls of the subprograms with the geometry data.

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 free-form surfaces.

NOTE

Program CYCLE832 after the tool change. After calling CYCLE832, only the geometry data should still be programmed.

CYCLE832(tolerance, technology, ORI tolerance)

Programming the cycle:

- Tolerance (chord tolerance)
- Technology
  0 = Deselection
  1 = finishing (_FINISH)▼▼▼
  2 = semi-finishing (_SEMIFIN)▼▼
  3 = roughing (_ROUGH)▼
  11 = finishing multi-axis program (_ORI_FINISH)▼▼▼
  12 = semi-finishing multi-axis program (_ORI_SEMIFINISH)▼▼
  13 = roughing multi-axis program (_ORI_ROUGH)▼
  1000000 = Top Surface without smoothing
  2000000 = Top Surface with smoothing
- ORI tolerance
  Orientation tolerance or version identifier CYCLE832. Is required when executing an HSC program on machines with dynamic orientation transformation (e.g. 5-axis machining). Parameter S_OTOL must be programmed. This also applies for applications on 3-axis machines for programs without orientation of the tool (S_OTOL = 1).

CYCLE832(0.05,12,0.5)

CYCLE832 for semi-finishing with tolerance 0.05 with orientation smoothing and ORI tolerance 0.5.

CYCLE832(0.01,_TOP_SURFACE_SMOOTH_OFF+_ORI_FINISH,0.1)

CYCLE832 for finishing with 0.01 tolerance, Top Surface activated, smoothing off and orientation smoothing with ORI tolerance 0.1

Programming example, CYCLE832 Advanced Surface

N10	T1 D1	; tool selection
N20	G54	; Select tool zero
N30	M3 S1200	; Clockwise spindle rotation and speed
N40	**CYCLE832(0.1,3,1)**	; **3-axis program**, tolerance 0.1
N50		; [3] = Roughing, [1] = Standard without ORI
N60	EXTCALL "CAM_ROUGH_0"	; call subprogram CAM_ROUGH_0
N65	CYLE832(0,0,1)	; deselect CYCLE832
N70	**CYCLE832(0.1,_ROUGH,1)**	; **3-axis program**, tolerance 0.1
N80		; [_ROUGH] = roughing as **plain text**, [1] = standard without ORI
N90	EXTCALL "CAM_ROUGH_1"	; call subprogram CAM_ROUGH_1
N95	CYLE832(0,0,1)	; deselect CYCLE832
N100	**CYCLE832(0.01,11,0.1)**	; **5-axis program** (with orientation), tolerance 0.01
N110		; [11] = finishing, [0.1]= ORI tolerance 0.1
N120	**ORISON**	; Activation of the orientation smoothing
N130	EXTCALL "CAM_FINISH_0"	; call subprogram CAM_FINISH_0
N135	CYLE832(0,0,1)	; deselect CYCLE832
N140	**CYCLE832(0.005,_ORI_FINISH,0.05)**	; **5-axis program** (with orientation), tolerance 0.005
N150		; [_ORI_FINISH] = finishing as **plain text**, [0.05]= ORI tolerance 0.05
N160	**ORISON**	; Activation of the orientation smoothing
N170	EXTCALL "CAM_FINISH_1"	; call subprogram CAM_FINISH_1
N180	CYLE832(0,0,1)	; deselect CYCLE832
N200	M30	; end of program

Programming example, CYCLE832 Top Surface

N10	T1 D1	; tool selection
N20	G54	; Select tool zero
N30	M3 S1200	; Clockwise spindle rotation and speed
N40	**CYCLE832(0.1,_TOP_SURFACE_SMOOTH_OFF+_ROUGH,1)**	
N50	; **3-axis program**, tolerance value 0.1, [_TOP_SURFACE_SMOOTH_OFF] = Top Surface active, smoothing off	
N55	; [_ROUGH] = roughing, [1] = standard without ORI	
N60	EXTCALL "CAM_ROUGH_0"	; call subprogram CAM_ROUGH_0
N65	CYLE832(0,0,1)	; deselect CYCLE832
N70	**CYCLE832(0.05,_TOP_SURFACE_SMOOTH_OFF+_SEMIFIN,1)**	
N80	; **3-axis program**, tolerance value 0.05, [_TOP_SURFACE_SMOOTH_OFF] = Top Surface active, smoothing off	
N85	; [_SEMIFIN] = rough finishing, [1] = standard without ORI	
N90	EXTCALL "CAM_SEMIF_1"	; call subprogram CAM_SEMIF_1
N95	CYLE832(0,0,1)	; deselect CYCLE832
N100	**CYCLE832(0.001,_TOP_SURFACE_SMOOTH_ON+_ORI_FINISH,0.01)**	
N110	; **5-axis program**, tolerance 0.001, [_TOP_SURFACE_SMOOTH_ON] = Top Surface active, smoothing on	
N120	; [_ORI_FINISH] = 5-axis finishing as **plain text**, [0.01]= ORI tolerance 0.01	
N130	EXTCALL "CAM_FINISH_0"	; Call subprogram CAM_FINISH_0
N135	CYLE832(0,0,1)	; deselect CYCLE832
N200	M30	; end of program

TiP

Before the functions listed here can be used, the machine manufacturer must have optimized the CNC machine correctly.

CYLCE832 in older software releases

Because of the continuous further development of CYCLE832, the syntax and also the functional scope of the cycle differs between the software releases. Therefore, note the new syntax of CYCLE832 when using older programs. A compatibility mode can be set via a machine data item. Contact the machine manufacturer if you have any questions on compatibility.

Software release	NC code
SINUMERIK up to SW 7.5	CYCLE832(tolerance, technology) `CYCLE832(0.01,112101)`
SINUMERIK Operate SW 2.6	CYCLE832(tolerance, machining, version ID) `CYCLE832(0.1,3,1)`

SINUMERIK Operate
SW 2.7 to SW 4.4
	CYCLE832(tolerance, machining, version ID) and ORISON can be programmed separately
	`CYCLE832(0.005,1,1)`
	`ORISON`
	`OTOL=0.05`
	When using the ORI tolerance, the activation of the orientation smoothing with ORISON must be explicitly programmed after the call of CYCLE832.

SINUMERIK Operate
SW 4.5
	CYCLE832(tolerance, machining, ORI tolerance) and ORISON can be separately programmed
	`CYCLE832(0.005,_ORI_FINISH,0.5)`
	`ORISON`
	When using the ORI tolerance in the CYCLE832, depending on the configuration of CUST_832, after calling CYCLE832, the activation of orientation smoothing must be programmed with ORISON.

3.6.2 CYCLE832 and CUST_832

The manufacturer cycle CUST_832 is available for individual adaptation of CYCLE832. This is individually adapted according to the machine by the machine manufacturer. Special NC commands for machining are also set in CUST_832. As a consequence, the parameters in the CYCLE832 dialog can deviate from the descriptions in Chapter 3.6.1 if the machine manufacturer has made specific adaptations.

NC commands in connection with Advanced Surface and Top Surface

The following NC code commands are preset in CUST_832.SPF and are activated when the technology groups are selected in CYCLE832:

- DYNNORM, DYNROUGH, DYNSEMIFIN, DYNFINISH (G code group 59).
- COMPCAD makes it possible to combine part programs with short linear blocks (G1), with the associated tolerance, using polynomials.
- COMPSURF, compressor ensures a better workpiece surface for inclined line-by-line finishing programs, poor data quality - or for example, irregular point distribution in NC programs from the CAD/CAM system.
- SOFT (G code group 21) activates the jerk-limited velocity control.
- G645 (G code group 10) switches to the continuous path mode (Look Ahead).
- FIFOCTRL (G code group 4) switches to the automatic pre-processing memory control.
- FFWON (G code group 24) switches to the parameterized precontrol (speed or acceleration precontrol). Can be optionally activated in CUST_832.

Important NC commands for 5-axis machining

In CUST_832.SPF, the following NC code commands can be preset by the machine manufacturer.

- TRAORI activates the 5-axis transformation set in the transformer machine data and must be programmed alone in the block.
- UPATH (G code group 45) ensures synchronous movements of the rotary and linear axes in 5-axis spline interpolation, as performed with active compressor, for example.
- ORIAXES (G code group 51) linearly interpolates the orientation axes in the block up to the end of block.
- ORIWKS (G code group 25) defines the workpiece coordinate system for orientation programming.
- ORISON (G code group 61) activates the orientation smoothing for 5-axis machining with active 5-axis transformation (TRAORI).

TiP

You can see which functions of CYCLE832 are currently active during processing in the "All G functions" display. See "Executing programs (AUTOMATIC)" on Page 54.

In the following sections, you will find the relevant machine functions such as Compressor and Look Ahead, which are only explained briefly here, because they are automatically called up with optimal values by CYCLE832 or CUST_832.

3.7 Advanced Surface and Top Surface

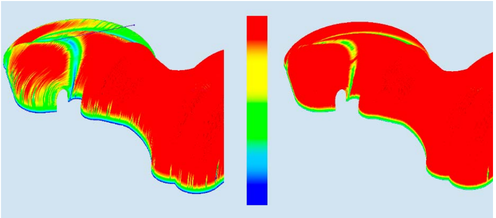
SINUMERIK bundles a range of special mold making functions under Advanced Surface and Top Surface. This intelligent motion control means optimum workpiece surfaces while at the same time providing maximum machining velocity. Advanced Surface or Top Surface are automatically activated when you work with CYCLE832.

When the new motion control is used, an optimized **Look Ahead** function contributes to perfect surface quality through reproducible results in adjacent milling paths, accuracy and increased speed. The compressor ensures precise contour accuracy and maximum machining velocity. Intelligent jerk limitation reduces the stress on the machines mechanical system and allows soft acceleration and braking of the axes, therefore extending the machine service life.

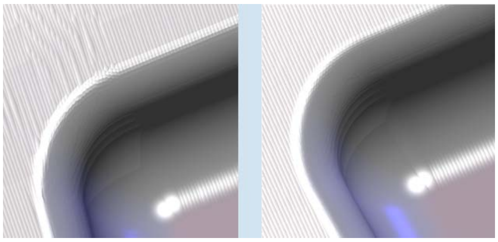
One important advance is the automatic harmonization of the velocity profiles on adjacent milling paths by means of CNC. It also works for the forward/backward line-by-line milling of contours and free-form surfaces and it directly leads to increased surface quality.

Top Surface is based on compressor COMPSURF. This has been improved for machining demanding CAD/CAM programs.

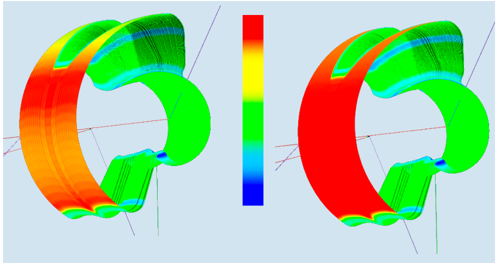
Without Advanced Surface Top Surface | **With Advanced Surface Top Surface**



Increased machining velocity due to an improved velocity profile. The red areas show the maximum velocity.

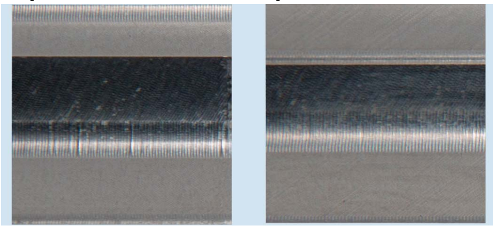


Perfect surface quality thanks to reproducible results in adjacent milling paths. The surface is much more homogeneous.



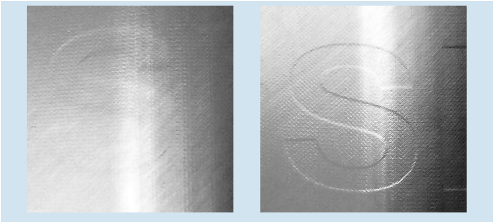
The compressor does not only work via G1 blocks. It also compresses rapid traverse movements G0 using blending. The velocity remains at a constant high level during the entire machining process (red areas).

Without Top Surface | **With Top Surface**



Significantly better workpiece surfaces for bidirectional milling using direction-independent identical smoothing of the milling paths - and essentially independent of the CAM data quality.

Top Surface Smoothing on | **Top Surface Smoothing off**



When smoothing is active, unevenness in the micrometer range is smoothed within the tolerance.

If smoothing is set to "no", then the specified contour is precisely machined.

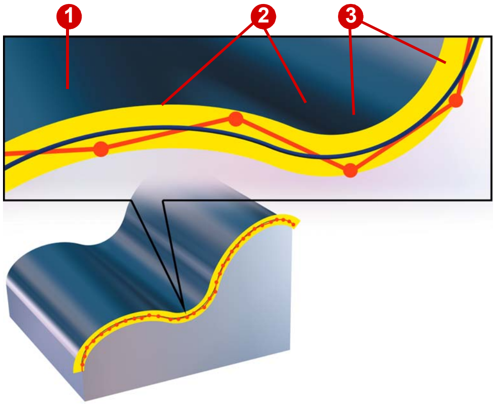
3.7.1 Compressor (COMPCAD, COMPCURV, COMPSURF)

The compressor is called in CYCLE832. If it is to be programmed separately, then proceed as described below. The common objective of compressor functions is to optimize the surface quality and machining speed by achieving continuous block transitions and increasing the path length for each block. You can use machine data to set whether only G1 blocks or G2/G3 or G0 blocks should also be compressed.

Explanation of the commands

COMPOF	Compressor off
COMPCAD (standard for Advanced Surface)	Compressor on; surface quality and speed are further optimized. COMPCAD smooths the point characteristic before approximation. (B spline) and offers, at a high path velocity, the highest degree of accuracy with transitions that have a constant acceleration rate. Preferred method for milling **free-form surfaces (recommended).**
COMPCURV	Compressor on. Blocks are approached using a polynomial. Block transitions are jerk-free. Preferably used for **circumferential milling**.
COMPSURF (standard for Top Surface)	Compressor on. The COMPSURF compressor ensures a significantly improved workpiece surface for inclined line-by-line finishing programs, poor data quality - or for example, irregular point distribution in NC programs from the CAD/CAM system. Further, COMPSURF improves complying with acceleration and jerk limits. As a consequence, machine-specific dynamic parameters can be increased.

Operating principle of the compressor



The compressor combines a sequence of path commands in accordance ❷ with the set tolerance range ❶ and compresses them into a spline ❸ that can be directly executed by the control system. A new contour is created whose characteristic lies within the specified tolerance range.

The compressor generates smooth paths and paths with constant curvature. The constant curvature results in a steady velocity and acceleration characteristic, meaning that the machine can run at higher speeds, thereby increasing productivity.

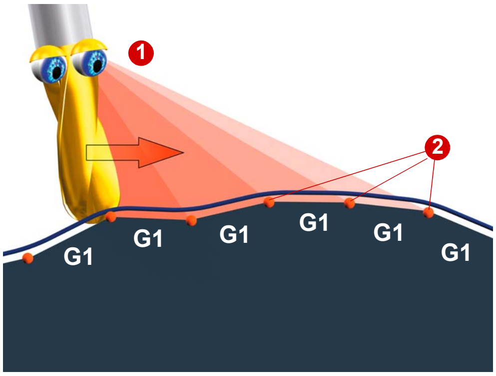
3.7.2 Continuous-path mode, Look Ahead – G64, G645

In continuous-path mode, the path velocity at the end of the block (for the block change) is not decelerated to a level which would permit the fulfillment of an exact stop criterion. The objective of this mode is, in fact, to avoid rapid deceleration of the path axes at the block-change point so that the axis velocity remains as constant as possible when the program moves to the next block. To achieve this objective, the **Look Ahead** function is also activated when the continuous-path mode is selected.

Explanation of the commands

G64	Continuous-path mode – **Look Ahead** where the axis only brakes at corners
G645 (recommended)	Continuous-path mode with blending and tangential block transitions within the defined tolerances. With G645, the smoothing motion is defined so that the acceleration of all axes involved remains smooth (no jumps) and the parameterized maximum deviations from the original contour are not exceeded. **In conjunction with Advanced Surface, it is recommended that you only work with G645.**

Using G64, G645



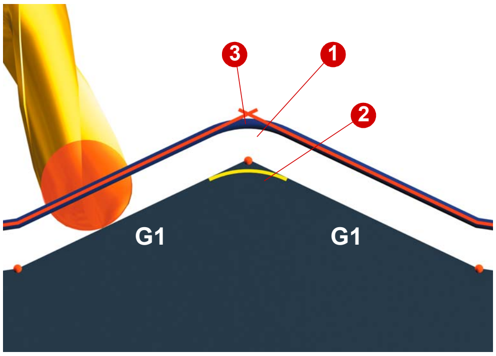
The objective of continuous-path mode is to increase the speed and harmonize the traversing behavior. This is implemented by two functions.

Look Ahead – anticipatory velocity control 1

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

Rounding corners 2

The Look Ahead function also means that the control system is able to smooth the corners that it identifies. This means that the programmed corner points are, therefore, not approached exactly. Sharp corners can be 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 off ❸ sharp edges, for example, the continuous-path command **G645** forms transition elements ❶, ❷ at the block limits. The continuous-path commands differ in terms of how they form these transition elements.

With **G645**, rounding blocks are also generated on tangential block transitions if the curvature of the original contour exhibits a jump in at least one axis.

TIP

We recommend G645 for free-form surface applications.

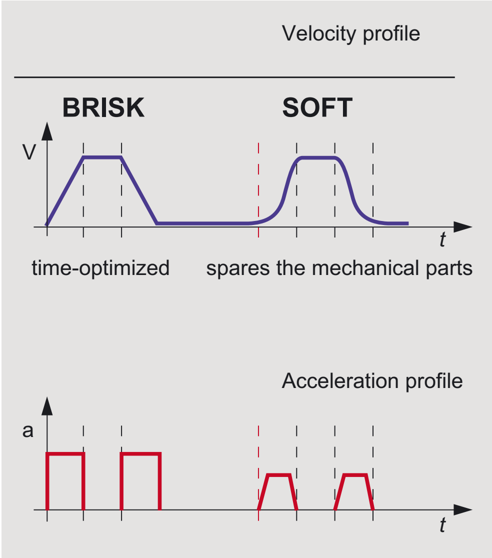
3.7.3 Precontrol and jerk limitation – FFWON, SOFT, ...

Precontrol and jerk limitation are called in CUST_832 on a combined basis. This is because this combination offers ideal conditions for free-form surface milling. These functions are set by the machine manufacturer.

Explanation of the commands

FFWON (recommended)	control "on"
FFWOF	Precontrol "off"
BRISK (not recommended)	**Without jerk limitation** Abrupt acceleration of path axes
SOFT (recommended)	**With jerk limiting** Jerk limited acceleration of the path axes Axial jerk limitation

Jerk limiting function



To ensure that the machine accelerates as smoothly as possible, the acceleration profile of the axes can be influenced using the commands **Soft** and **Brisk**. If **Soft** is activated, the acceleration behavior does not change abruptly, but is increased in the form of a linear characteristic. This helps to reduce the stress on the machine and improves the surface quality of workpieces, as much less machine resonance is generated.BRISK.

BRISK:

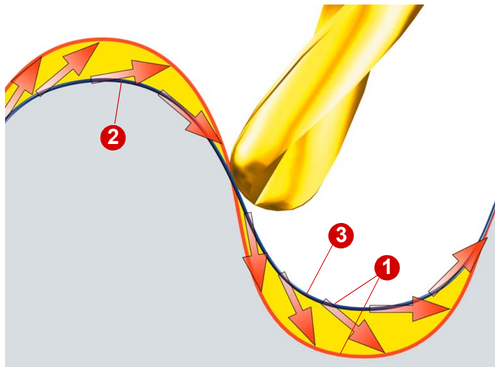
Acceleration behavior: The path axes accelerate abruptly according to the machine data that has been set

The axis slides travel with maximum acceleration until the feedrate is reached. BRISK enables time-optimized machining; however, with steps in the acceleration characteristic..

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 lower stress on the machine.



Precontrol function. In cases where axes are not precontrolled, the following error results in a contour error whose severity is determined by the velocity 1. Generally, this will manifest itself in the form of a narrowing of the radius 3 at curved contours. The following error depends on the servo gain factor (Kv factor) that is set (dependent on the mechanical system) and the axis velocity.
Precontrol **FFWON** reduces the speed-dependent following error when contouring towards zero. Traversing with precontrol permits higher path accuracy and thus improved machining results.

3.7.4 Smoothing tolerance CTOL, OTOL, ATOL

The tolerance for linear and rotary axes for smoothing was previously set with CYCLE832. As of SW 2.7 this is implemented via the functions CTOL (contour tolerance), OTOL (orientation tolerance), and ATOL (axis-specific tolerance).

The CTOL, OTOL, and ATOL commands can be used to adapt the machining tolerances defined for the compressor functions (COMPCAD), the smoothing types G645, and the orientation ORISON using machine and setting data in the NC program.

The tolerance value for CTOL is set by CYCLE832. OTOL is calculated in the cycle. Since OTOL also has an effect on ORISON by default, it is important when activating ORISON in the cycle (or manually) to ensure that the orientation tolerance is not too great, because otherwise too much orientation smoothing will result. In most cases, over-smoothing occurs during roughing operations.

3.7.5 Path reference UPATH/SPATH

During polynomial interpolation, there may be a requirement for two different relationships between the velocity-determining FGROUP axes and the other path axes. The path axes that are not included in the FGROUP should either be routed synchronously to the path of the FGROUP axes or to the curve parameters. Therefore, for the axes that are not contained in FGROUP, there are two ways to follow the path:

- SPATH synchronous to path S
- UPATH synchronous to the curve parameter

Both types of path interpolation are needed for different applications and can be changed over via G codes SPATH and UPATH.

NOTE

UPATH is recommended for programming with active 5-axis transformation (TRAORI) and is preset in the manufacturer cycle CUST_832.

3.7.6 Smoothing the orientation characteristic (ORISON, ORISOF)

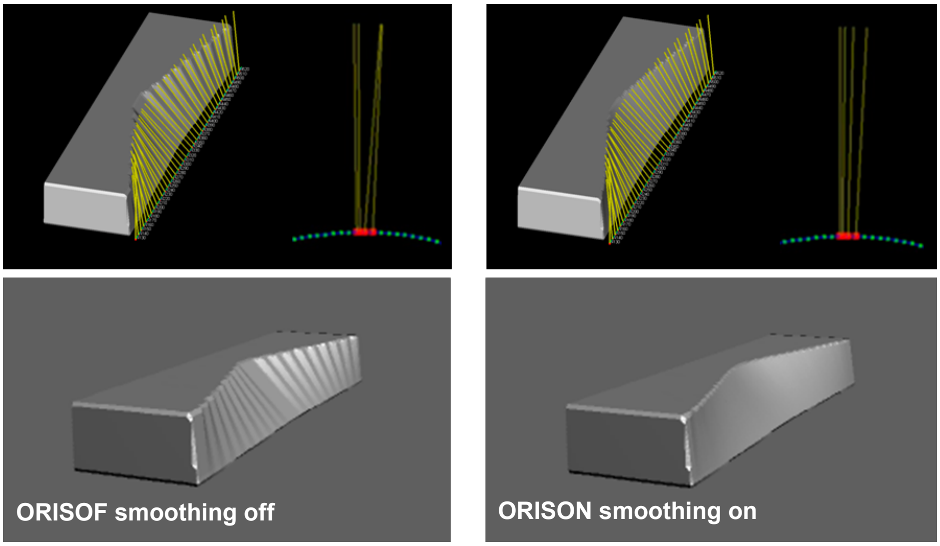
The ORISON function can be used to smooth fluctuating orientation across several blocks. The aim is to achieve a smooth characteristic for the orientation and therefore a more harmonious movement of the axes.

For 5-axis programs generated by CAD-CAM systems in which the milling paths and direction vectors are specified for the tool, the programs often contain inharmonic changes of the tool orientation which cause sudden movements of the rotary axes and therefore, because of the compensatory movements, also of the linear axes. Even if these deviations are only minimal, they will lead to compensatory movements in the linear axes, which will manifest themselves in slowed movements or even stopping in the path. The consequences are visible traces on the workpiece surface and longer machining time.

With ORISON, the orientation is smoothed irrespective of the contour because the movement of the rotary axes and the linear axes is much more harmonious. This results in higher machining speeds and shorter machining times.

NOTE

Up to SW 4.5, ORISON orientation smoothing is not part of CYCLE832 and it must therefore be programmed separately with the desired vector smoothing in the NC program in the case of a 5-axis program with orientation interpolation. If the ORISON function is to be automatically activated with active 5-axis transformation, then this setting can be made in CUST_832 by the OEM. Depending on the orientation tolerance, the NC command ORISON is activated in the manufacturer cycle CUST_832, and is then also available as **multi-axis program** parameter with **ORI tolerance** in CYCLE832.



ORISOF smoothing off

ORISON smoothing on

NOTE

ORISON is automatically deactivated when using Top Surface (COMPSURF). Therefore, after CYCLE832 with Top Surface do not program ORISON.

ORISON programming (without CYCLE832)

```
N110   TRAORI                          ; Activation of orientation transformation.
N120   CYCLE832(0.005,1,1)
N130   ORISON                          ; Activation of orientation smoothing
N140   OTOL=0.5                        ; Specify tolerance
N150   G1 X10 A3=1 B3=0 C3=1           ; Geometry program
       ...
N990   ORISOF                          ; Deactivation of orientation smoothing
```

ORISON programming (with CYCLE832)

```
N110   TRAORI                          ; Activation of orientation transformation.
N120   CYCLE832(0.005,_ORI_FINISH,0.5)
N130   ORISON                          ; Activation of the orientation smoothing
N140   G1 X10 A3=1 B3=0 C3=1           ; Geometry program
       ...
N990   M30                             ; end of program
```

3.7.7 Technology G groups

Using the "Technology" G group, the appropriate dynamic response can be activated on the machine for 5 varying technological machining operations. The dynamic values and G codes are configured and set by the machine manufacturer.

Five dynamic response settings are available in the technology G code group:

- DYNORM for standard dynamic response
- DYNPOS for positioning mode, tapping
- DYNROUGH for roughing
- DYNSEMIFIN for semi-finishing
- DYNFINISH for finishing

The G groups are switched and activated automatically when the machining method (e.g. finishing or roughing with CYCLE832 or CUST_832) is selected.

Dynamic parameters can be adapted to the respective machining operation with the technology G groups. Using the commands of technology G group 59, the value of channel and axis-specific machine data is activated using the corresponding array index. These are, for example, jerk and acceleration values.

3.7.8 Preprocessing memory control FIFOCTRL

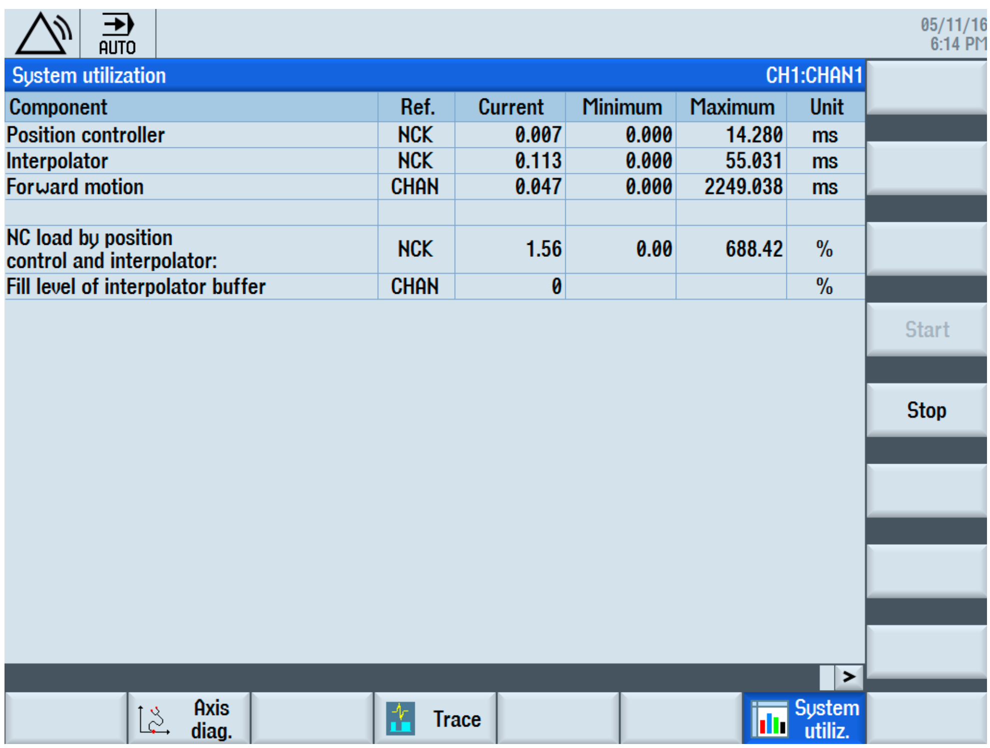
Mold-making programs are generally executed directly from the hard drive or an external storage medium (e.g. USB flash drive, CF card). The maximum number of NC blocks that are loaded in the NC memory can be set via machine data.

During processing of these NC programs, the interpolation memory may run out. In this case, the NC will stop until data is reloaded. To counter this, the pre-processing memory control FIFOCTRL is used. For very small distances between points in a mold-making program, it is necessary to keep the fill level as high as possible in order prevent dropping to 0% (interpolation dip).

The pre-processing memory control FIFOCTRL is automatically called up by CYCLE832 or CUST_832. It is deselected with STARTFIFO.

This is how you can open the fill level of the interpolation buffer:

▶ In the **Diagnostics>** menu, select softkey **System utilization**.

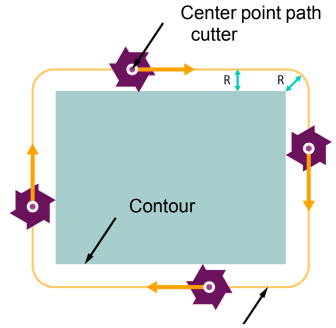


3.8 Cutter radius compensation

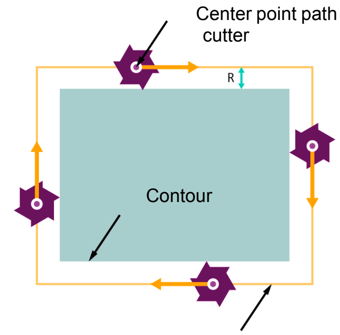
In the following, we will consider the expansion of the conventional 2-dimensional cutter radius compensation to three dimensions. Normally the compensation is activated by the G codes G41/ G42 (left or right) and deactivated by G40. After activation, the control calculates an offset curve with an offset the size of the cutter radius.

2-dimensional cutter radius compensation

.



G41 / G450



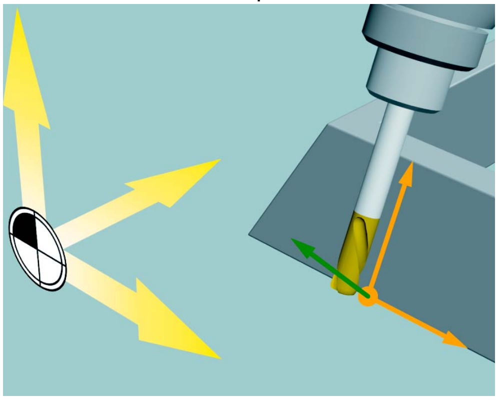
G41 / G451

The 3-dimensional case is more complicated. In two dimensions, it is implicitly clear that the tool is aligned perpendicular to the compensation plane and therefore the compensated path is always perpendicular to the path tangent in the X-Y plane. In three dimensions, the tool orientation normally changes continuously and therefore the compensation direction must also change. It is now defined by a vector in space. We must distinguish between two situations. On the one hand, the circumferential milling and on the other hand, the face milling which only requires a constant compensation value for the Z axis in the 2-dimensional case.

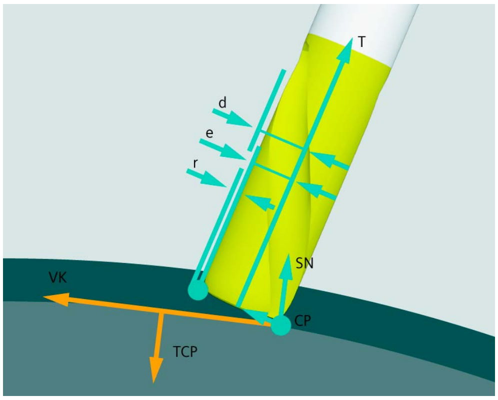
Cutter radius compensation for circumferential and face milling

3-dimensional cutter radius compensation

CUT3DC circumferential milling,
Green: Direction of compensation



CUT3DF face milling



CUT3D circumferential milling

In circumferential milling, the direction of compensation is always perpendicular to the plane on which the cutter is moving. This is always defined by the current path tangent and the tool vector and normally changes in each interpolation increment. It is defined with CUT3DC and activated with G41/G42 (left/right).

CUT3DF face milling

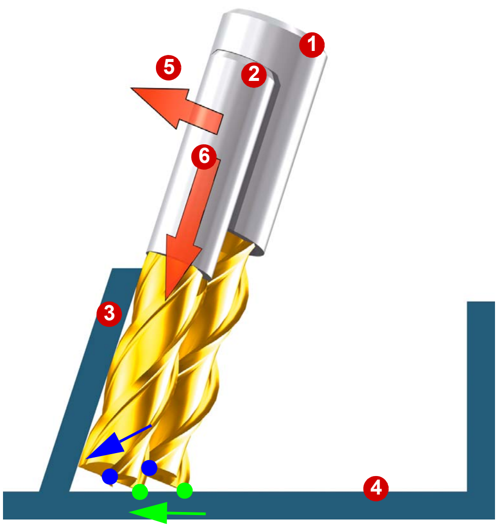
The situation is also more complicated with face milling. Since the tool is not always perpendicular to the plane to be machined as with two dimensions, a constant offset is no longer sufficient. The compensation value and the compensation direction now depend on the tool radius, the rounding radius and of course on the tool orientation relative to the workpiece surface. This means that we require additional information about the surface. This is defined through the programming of the normal vectors A4, B4, C4 (start of block) and A5, B5, C5 (end of block).

The compensation is defined with CUT3DF and activated with G41/G42, whereby there is no difference between G41 and G42 in this case.

Influence of tool radius compensation with 5-axis circumferential milling, taking into account the limiting surface (CUT3DCC)

A combination of these two methods is used when machining special structural components in the aircraft industry. Pockets are machined where the wall inclination forms an acute angle to the floor. The wall is machined with circumferential milling with simultaneous contact with the floor. If a large cutter is used in this situation, compensation must be performed simultaneously perpendicular to the wall and along the cutter axis in order to avoid contour violations of the floor.

Circumferential milling with limiting surfaces



1. Standard tool (tool from CAM)
2. Tool with smaller radius
3. Machining surface, inner surface
4. Limiting surface of pocket floor
5. Compensation in relation to machining surface
6. Compensation in relation to limiting surface

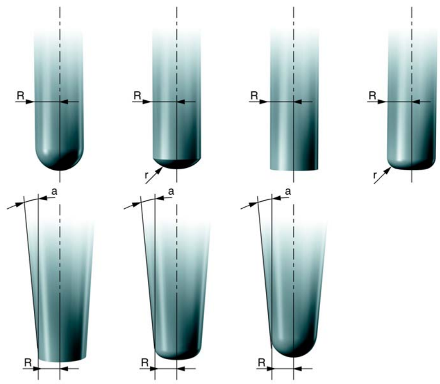
The control system not only takes into account that a correction must be made in the direction of the machining surface, 5 but also that feed is required in the tool direction 6 so that the point of action (green) is at the same level as the pocket floor. This results in a shift in the cutter tip (blue) in the direction of the pocket floor.

This compensation version is known as **circumferential milling with limiting surfaces**. It is selected via the code CUT3DCC or CUT3DCCD and activated with G41/42 as for CUT3DC.

With CUT3DCC, the coordinates in the NC program refer to the contour and consequently the full miller radius is compensated. For the compensation radius this results in R = $TC_DP6 plus the programmable offset that is defined by OFFN=…., and for the face milling on the floor, the corner radius $TC_DP7. This results in R' = $TC_DP6 + $TC_DP15 + OFFN for the circumferential milling component and r' = $TC_DP7 + $TC_DP16 for the face milling with the other cutter side.

For CUT3DCCD, the programmed path refers to the standard tool (zero tool). The compensation value is then the difference between the zero tool and the reground current tool, i.e. the tool wear value from the $TC_DP15 tool data table plus the offset OFFN=….

As with face milling, the normal vector of the limiting surface must be programmed via A4, B4, C4 or A5, B5, C5. If no normal vector is programmed, the Z direction of the current coordinate system is assumed for the CUT3DF.



The following figure shows the tool types that are permitted in conjunction with the 3D compensations. For the circumferential milling (also with limiting surfaces), only cylindrical types (top row), for face milling, also the conical types of the bottom row.

Explanation of the commands

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 method at external corners (all compensation types)
2 ½D circumferential milling	
CUT2D	2 1/2D COMPENSATION with compensation plane determined using G17 – G19
CUT2DF	2 1/2D COMPENSATION with compensation plane determined using a frame
3D circumferential milling	
CUT3DC	Compensation perpendicular to path tangent and tool orientation
CUT3DCD	Activating the 3D radius compensation referred to a differential tool for circumferential milling
ISD	The clearance between the milling tool tip (FS) and the milling tool construction point (FH) are specified using the command ISD.
ORID	No changes in orientation in inserted circular blocks at external corners. Orientation motion is performed in the linear blocks.
ORIC	Travel path is extended by means of circles. The change in orientation is also performed proportionately in the circle.

Face milling	
CUT3DFS	Constant orientation (3-axis). Tool points in the Z direction of the coordinate system defined via G17 - G19. Frames do not have any effect.
CUT3DFF	Constant orientation (3-axis). Tool in Z direction of the coordinate system currently defined via the frame.
CUT3DF	5-axis with variable tool orientation
3D circumferential milling with limiting surface (combined circumferential/face milling)	
CUT3DCC	CNC program relates to the contour on the machining surface.
CUT3DCCD	The CNC program relates to the tool center point path.

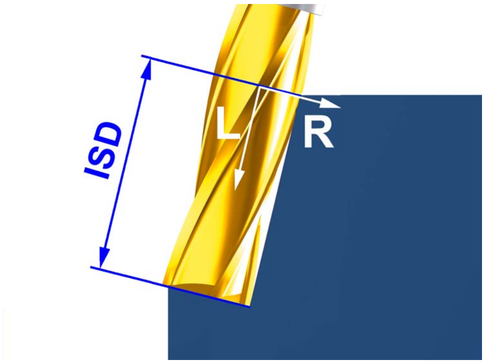
NOTE

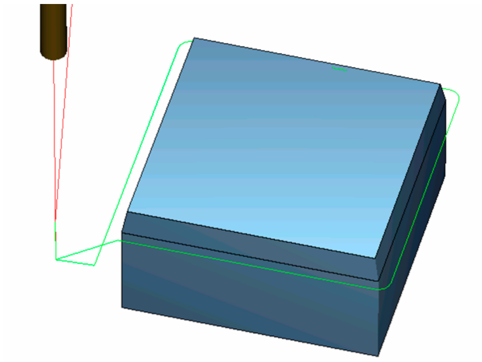
For 3D tool radius compensation when face milling (CUT3DF..), in conjunction with CYCLE832, the compressor with COMPOF should be deactivated before programming 3D tool radius compensation.

Programming example for circumferential milling

A workpiece contour needs to be milled at the circumference. Programming is to be from the top edge of the workpiece and the insertion depth ISD is to be taken into account. In the example, compensation is performed to the right of the contour with an ISD of 20.

Example involving CUT3DC





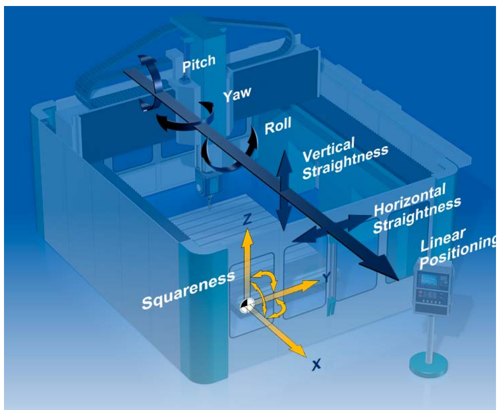
```
N100    CYCLE800(2,"TABLE",200000,57,0,0,0,0,0,0,0,0,0,1,,1)
N101    CYCLE800()
N102    G54
N103    WORKPIECE(,,"","BOX",112,0,-50,-80,0,0,100,100)
N104    T="MILLER_12"
N105    M6
N106    S6000 M3
N108    TRAORI; activate 5-axis transformation
N109    ORIWKS; orientation reference is the WCS
N110    ORIVECT
N111    CUT3DC; 3D tool radius compensation for circumferential milling
N112    G54 G0 X-20 Y-20 D1
N113    G0 Z10
N114    G1 Z-10 F1000 M8
N115    G41 X0 A3=1.7632698 B3=0 C3=10; activate tool radius compensation to the right of the contour and approach
N116    G1 Y100
N117    G1 A3=0 B3=-1.7632698 C3=10
N118    G1 X100
N119    G1 A3=-1.7632698 B3=0 C3=10
N120    G1 Y0
N121    G1 A3=0 B3=1.7632698 C3=10
N122    G1 X0
N123    G40 X-20 Y-20 A3=0 B3=0 C3=1; deactivate tool radius compensation and retract
N124    G0 Z100
N125    TRAFOOF
N126    M30
```

3.9 Volumetric compensation system (VCS)

As far as the production of large workpieces is concerned, e.g. structural parts on gantry-type milling machines, it is very difficult to achieve the necessary level of machine positioning accuracy due to the large dimensions of the working area. Errors due to sagging, buckling, etc. in particular can only be remedied mechanically with a great deal of effort.

The volumetric compensation system for the SINUMERIK is a tool for correcting geometric distortions of the machine mechanics. It reduces the effects of machine errors on the tool center point (TCP) systematically, thereby increasing the accuracy of the machine.

VCS compensations



Below are some examples of the kinds of error source that can be compensated:

- Linear position deviation
- Straightness of axes
- Unintentional axis rotations
- Roll, pitch, and yaw
- Squareness of axes in relation to one another
- Tool orientation errors involving swivel heads

As part of this process, the machine errors are detected on the basis of measurements and an error file is generated. VCS then implements the compensation values automatically. To increase the level of accuracy, calibration and testing with the compensation values can be performed using an iterative method. VCS automatically compensates the detected errors in conjunction with TRAORI.

TIP

Please contact your machine manufacturer regarding the VCS commissioning process and machine calibration/measurement.

3.10 VNCK - virtual machine

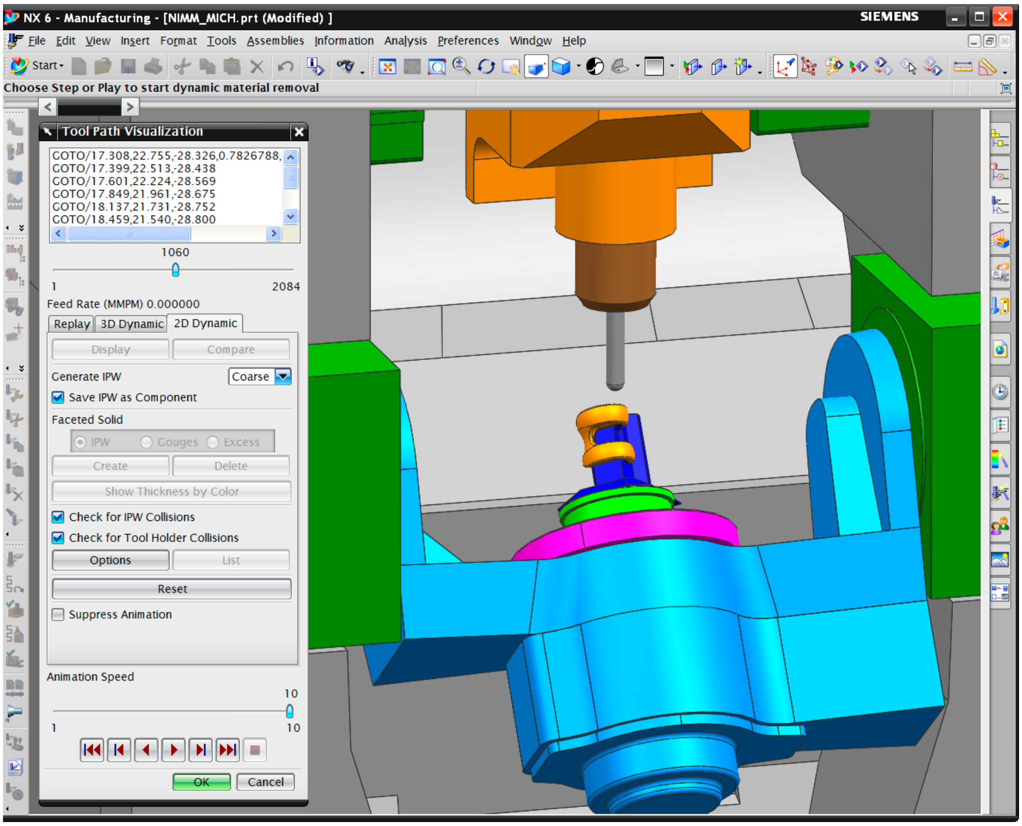
Due to the level of complexity and machining time associated with 5-axis milled workpieces, a deliberate effort is made to check that the programs are free of errors before actual production commences. To ensure that the data obtained is as realistic as possible, virtual models of the machine and control are created and simulated. Siemens provides the following basic modules for this purpose:

- The virtual NC kernel (VNCK)

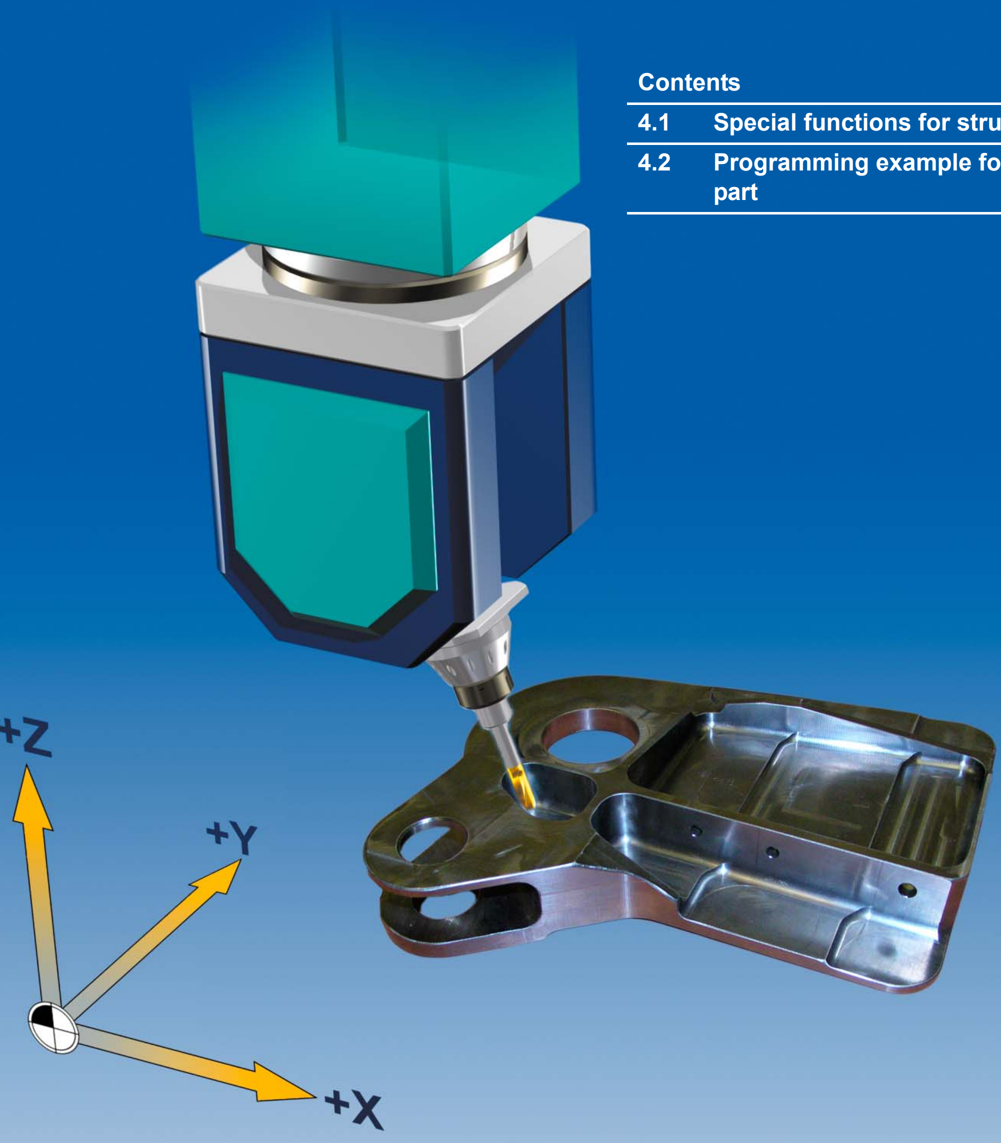
Using these basic modules and other components, such as the CAD data for the real machine, the machine manufacturer or CAM system manufacturer can create a virtual machine that resembles the real machine as closely as possible.

Using the virtual machine in conjunction with the SIEMENS VNCK offers many advantages:

- Programming errors are detected immediately.
- Program simulation with calculation of the actual time so that production effort can be estimated more easily.
- Collision checking with actual tools, chucking devices, and machine geometries.
- While the current production process is underway, the workpiece can be programmed, optimized and then implemented on the machine immediately.
- Shorter setup times.
- Can be used for training and instruction. New machines can be programmed without any risk.



Aerospace, structural parts



Contents

4

4.1 Special functions for structural parts

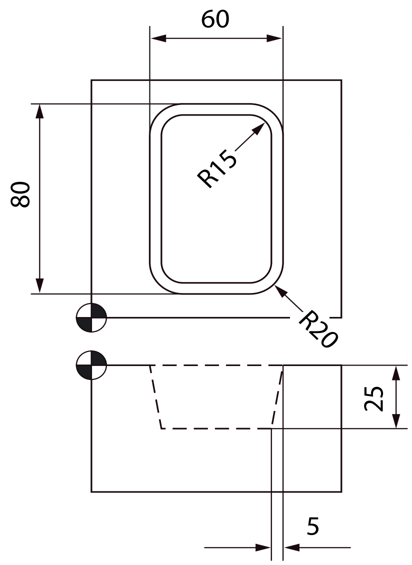
Structural parts are frequently used within the aircraft industry and take the form of load-bearing aircraft parts, e.g. the parts used to assemble the fuselage or wings. One of the key features of structural parts is the way in which they use only a small amount of material due to their geometry, but offer increased load-bearing capacity. For safety reasons, structural parts are milled from solid material and stock removal rates of up to 97% are by no means unusual. As a result, special functions are required throughout the entire machining process.

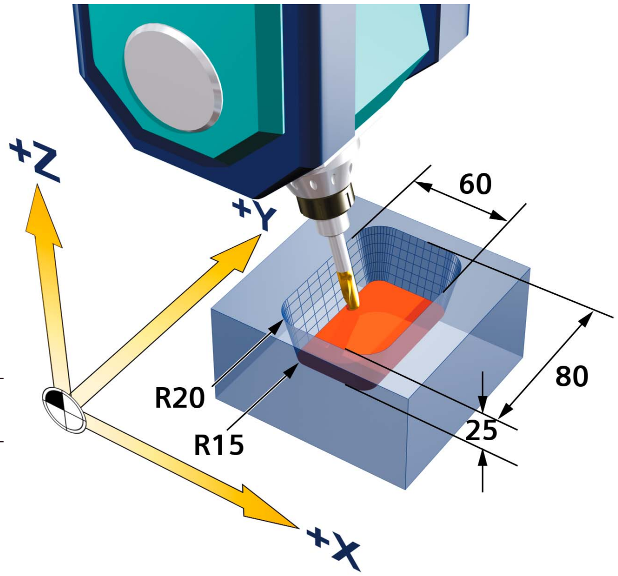
Important functions for the machining process for structural parts:

- VNCK,
  as the high level of material usage calls for a form of simulation that resembles the control exactly.
- VCS,
  as maximum accuracy is required in conjunction with extremely large workpieces.
- CYCLE832
  Easy preselection of the key machine settings for roughing and finishing.
- ORIVECT,
  as this is the only kind of orientation interpolation that will ensure the necessary precision when creating inclined walls.
- 3D tool radius compensations,
  as this even allows the use of reground tools without having to rebuild the NC program.
- Integrated process chain from generation in CAD through to execution on the CNC.

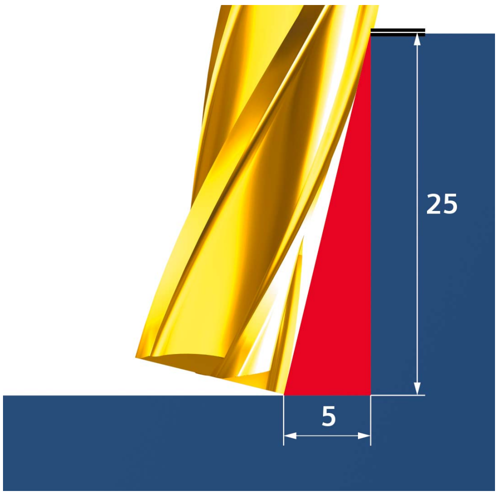
4.2 Programming example for the pocket on a structural part

Inclined inner and outer walls are a typical feature of structural parts. The most effective way to create these is to use the circumferential milling technique. The following programming example relates to the milling of a pocket with inclined walls. It illustrates the functions required to achieve this, such as orientation interpolation and tool radius compensation, by showing what actually happens in practice.





ISD pocket floor
Workpiece surface



The pocket contour can be programmed on the basis of the pocket floor, in which case the ISD is 0.

Alternatively, the contour can be programmed in relation to the workpiece surface and in this case the insertion depth ISD is 25.9807 (length of pocket wall). The radii will need to be adjusted.

The adjustment can be calculated using Pythagoras' theorem.

$$ISD = \sqrt{(25)^2 + (5)^2} = (25,9807)$$

Program example with RPY angle programming

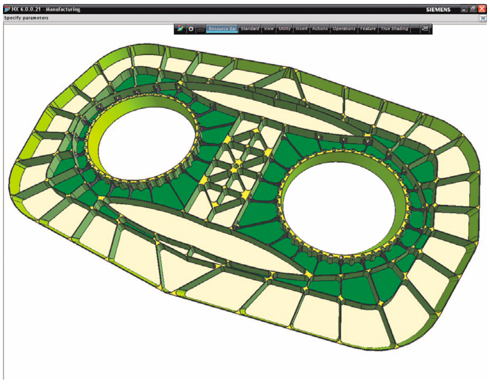
In the example, the pocket is rough machined and the pocket walls machined via circumferential milling with large circle interpolation ORIVECT and ORICONxx. Programming is with RPY angles A2=,B2=,C2= (upper profile) with automatic angle calculation and insertion depth ISD

```
EXAMPLE_4_POCKET_RPY_2.MPF
DEF REAL _VECT1, _VECT2
_VECT1=5
_VECT2=25
R10=ATAN2(_VECT1,_VECT2); ISD is calculated
;*** 2D POCKET ROUGHING ***
N100 CYCLE800(0,"TABLE",200000,57,0,0,0,0,0,0,0,0,0,1,,1)
N101 CYCLE800()
N103 G54 G17
N104 WORKPIECE(,,,"BOX",112,0,-50,0,0,0,100,100)
N105 T="CUTTER_D14"
N106 M6
N107 S8000 M3
N108 CYCLE832(0.01,0,1) ; Deselect HSC setting (DYNNORM)
N109 TRANS X50 Y50 Z50 ; WO at the pocket center, upper workpiece edge
N110 G0 X0 Y0 Z20
N111 CUT2DF ; CUTTER COMP 2D IN ACTIVE FRAME
N112 POCKET3(20,0,2,-25,70,50,15,0,0,90,2,0,0,2000,0.1,0,21
,60,8,3,15,6.5,1,0,1,2,11100,11,110)
;*** 3D CIRCUMFERENTIAL MILLING TO DEPTH ***
N113 CYCLE832(0.01,1,1) ; HSC setting finishing (DYNFINISH)
N114 COMPOF ; for 3D tool radius compensation, the compressor must be deacti-
vated
N116 TRAORI
N117 ORIWKS
N118 ORIVECT
N119 CUT3DC ; CUTTER COMP 3D CIRCUMFERENTIAL
N120 G54 G0 X0 Y0 Z10
N122 G1 A2=R10 B2=0 C2=0 F1000
N123 ISD=25.9807 ; INSERTION DEPTH
N124 G1 Z0
N125 G1 G41 X0 Y-40
N126 G1 X10
N127 ORICONCCW
N128 G3 X30 Y-20 CR=20 A2=0 B2=R10 C2=0 NUT=R10*2 ; GROOVE=opening angle of the
cone
N129 ORIVECT
N130 G1 Y20
N131 ORICONCCW
N132 G3 X10 Y40 A2=-R10 B2=0 C2=0 CR=20 GROOVE=R10*2
N133 ORIVECT
N134 G1 X-10
N135 ORICONCCW
N136 G3 X-30 Y20 A2=-0 B2=-R10 C2=0 CR=20 GROOVE=R10*2
N137 ORIVECT
N138 G1 Y-20
N139 ORICONCCW
N140 G3 X-10 Y-40 A2=R10 B2=0 C2=0 CR=20 GROOVE=R10*2
N141 ORIVECT
N142 G1 X0
N143 G40 X0 Y0 Z=IC(1) A2=0 B2=0 C2=0
N144 G0 Z100
N145 TRAFOOF
N146 G0 A0 C0
N147 M30
```

Special functions in the CAM system

CAM systems provide support for special workflows. Within the context of 5-axis machining in particular, they provide methods that meet the demands associated with the programming of structural parts, thereby creating the perfect conditions for ensuring optimum results on the machine.

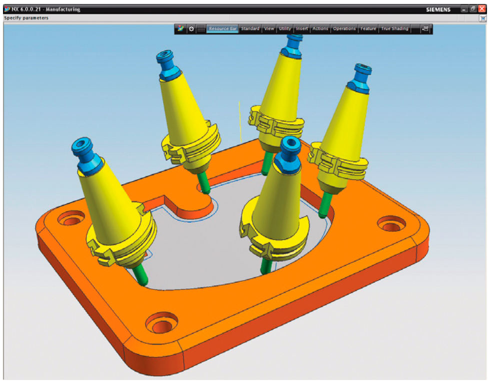
Support
Ribs and pockets



CAM systems enable roughing and finishing to be performed quickly and precisely on highly complex parts such as those typical of the aircraft industry.

The geometry selection process (e.g. for frequently occurring parts such as pockets and fins) has been highly automated to enable quick and easy programming.

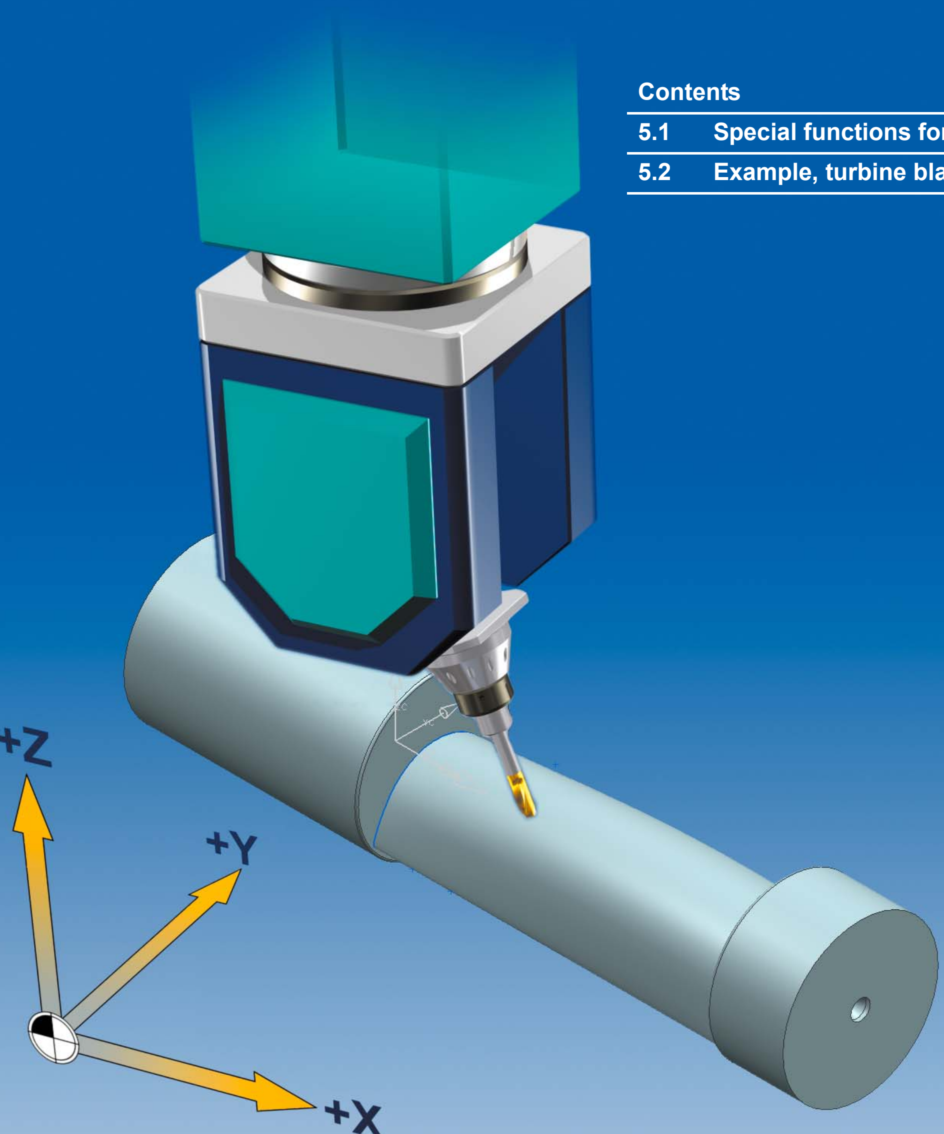
Angled tools
when milling profiles



Automatic profile milling with variable axes speeds up the programming process. CAM systems offer a range of automatic tool position and tool axis settings for 5-axis milling on walls and other profiles.

The tool paths may involve following pocket floors, the edges of walls or offsets.

Engine and turbine components



Contents | Page

5

5.1 Special functions for engine and turbine components

Turbine impellers or blades are subject to the toughest of requirements in terms of surface quality and contour accuracy. As a result, the NC is expected to process large quantities of data within a very short period. Even the smallest jumps in deceleration and acceleration can result in surface defects (e.g. chatter marks).

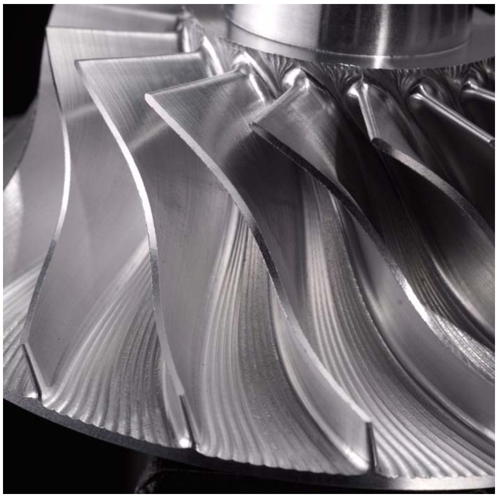
Turbine blades are often made from high-strength nickel alloys or titanium, which means that suitable methods have to be employed. This makes SINUMERIK the perfect solution, as it is a complete package offering a highly dynamic drive combined with a control system.

Important functions for the machining process for parts and components of engines and turbines:

- High-speed settings CYCLE832
  as optimum data compression within the tolerance band combined with precontrol and jerk limitation ensure the required surface quality and contour accuracy.
- Spline interpolation for hobbing (face/circumferential milling) impeller blades.
- TRAORI,
  for 5-axis transformation that is independent of the kinematics.
- Integrated process chain from generation in CAD through to execution on the CNC.

Impeller

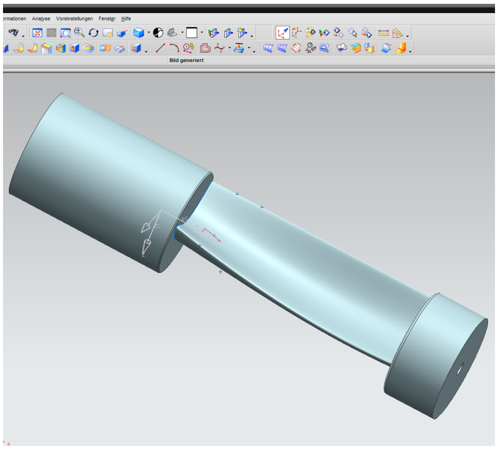
.



5.2 Example, turbine blade

This example relates to the milling of a turbine blade. The manta ray is modeled using a CAD/CAM system.

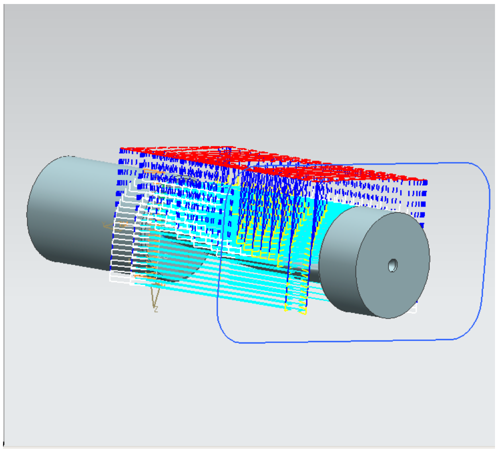
Turbine blade



At the modeling stage, it is essential to ensure that the machining strategies take account of the chucking conditions that will apply during production.

As a general rule, the contours of turbine blades are milled in a helical path, i.e. a full rotation is performed about the Z axis using a suitable chucking device.

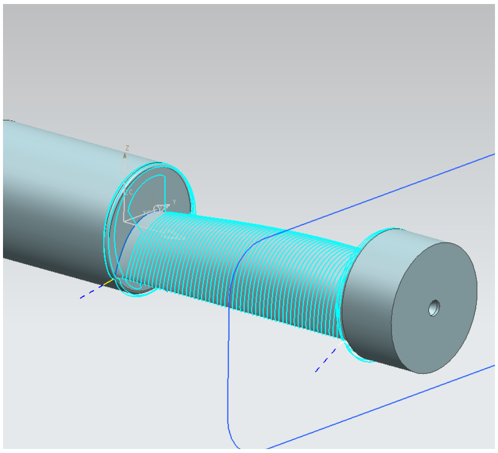
Plane roughing



Roughing was carried out by machining the upper and lower surfaces with the plane roughing method. This figure shows the tool paths involved in machining the upper surface.

From the point of view of ensuring optimum performance and surface quality, constant Z plane roughing is a highly effective approach and allows good control over the level of stress to which the tool is subjected.

5-axis copy milling
Face finishing



The 5-axis copy milling method was used for finishing purposes, as this allowed face milling in the form of helical finishing to be performed in accordance with axis selection. The tool is positioned at a lead angle.

Turbine blade during machining. Face milling on the other side with rotation about the X axis.

Program code example

The key aspects of driving gear and turbine components production are illustrated below on the basis of the start program and a finishing program.

Start program example

It is recommended that all of the technology parameters such as feedrate, speed, HSC settings (CYCLE832), vector smoothing (ORISON), and special M codes be defined in the main program. All feedrate values are defined here as R parameters. This allows greater flexibility when optimizing the feedrate values for the machine operator.

The subprogram should only contain geometry information. For testing the program, it is recommended that the subprograms be structured in such a way that they can be processed individually. To this end, jump marks should be set in the main program for jumping to a specific subprogram. Experience has shown that this is faster and more effective than a block search.

```
N100    GOTOF OP_1 ; Jump mark for the processing step
N105    OP_1:
N110    T="TOROID_D16_R3"     ; Tool call
N115    M6                    ; Load tool
N120    S10000 M3
N125    R1=4000      ; Milling feed
N130    R2=4000      ; Approach feed
N135    G54 G0 X0 Y0 C0 A0 D1
N140    G0 Z100
N145    CYCLE832(0.05,3,1)   ; High Speed Settings(DYNROUGH)
N150    EXTCALL "ROUGHING_1"
N155    CYCLE832()           ; Deselection of High Speed Settings
...     ....
```

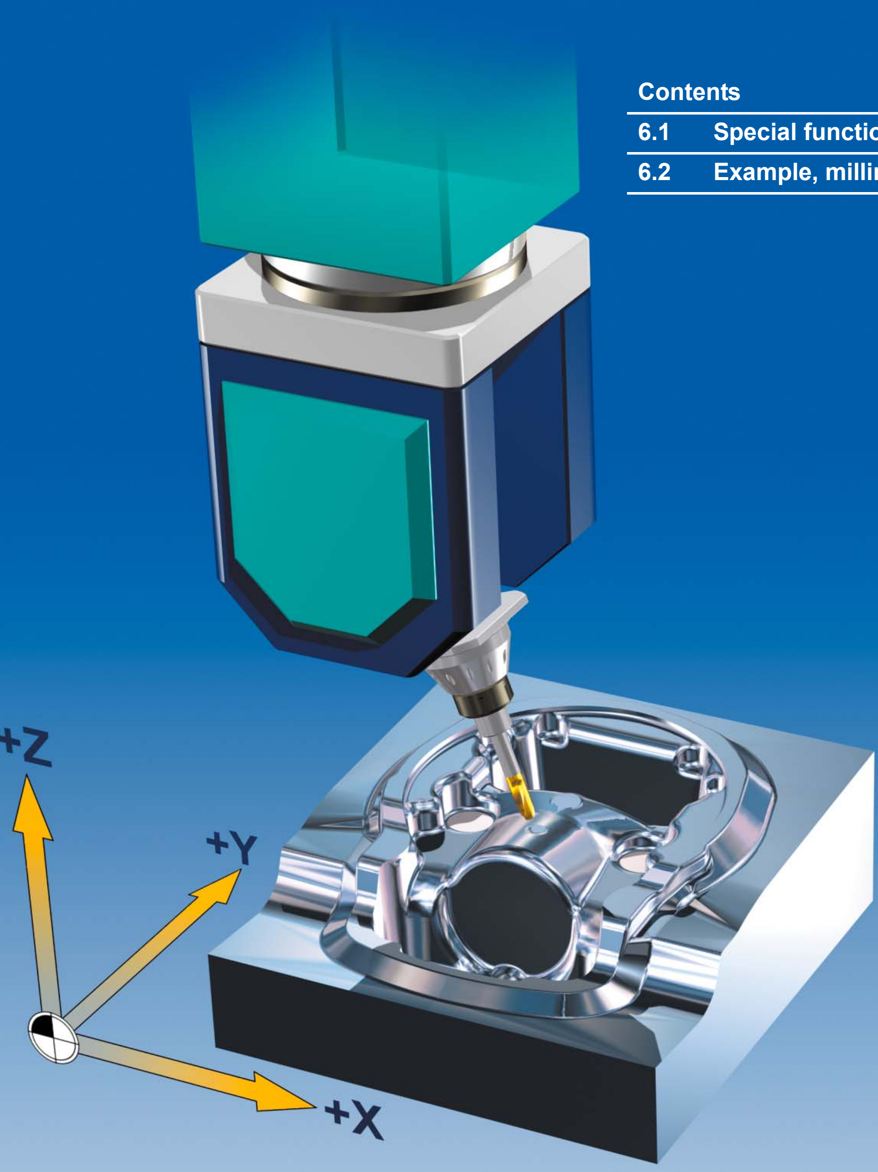
```
N160   OP_5:
N165   T="SPHERE_D6"
N170   M6
N175   M25          ;Release C-axis clamping (OEM-specific)
N180   M27          ;Release B-axis clamping (OEM-specific)
N185   S15000 M3
N190   R1=2000      ;Milling feed
N195   R2=1000      ;Approach feed
N200   G54 G0 X0 Y0 C0 A0 D1
N205   G0 Z100
N210   CYCLE832(0.01,1,1)   ;High Speed Settings (DYNFINISH)
N215   ORISON
N220   OTOL=0.5
N225   EXTCALL "FINISH_1"
N230   CYCLE832()           ; Deselection of High Speed Settings
N235   SUPA G0 Z0 D0        ; Max. retraction in Z (MCS)
N240   SUPA G0 X0 Y0 D1     ; Max. retraction in X and Y (MCS)
N245   M30
```

Subprogram example *FINISHING_1*

The subprogram contains the NC blocks for the geometry and the transformation.

```
N100   TRAORI                              ; Transformation ON
N105   ORIWKS                              ;
N110   ORIAXES
N115   G54
N120   MSG("FINISHING")
N125   ;Positioning movement
N130   G0 X-52.73538 Y-17.80536 Z31.9 A3=-.39485858 B3=.49800333 C3=.77206177 M3
N135   ; Approach movement
N140   G1 Z13.37361 A3=-.39485858 B3=.49800333 C3=.77206177 F=R2
N145   X-47.99708 Y-23.7814 Z4.10887 A3=-.39485858 B3=.49800333 C3=.77206177
N150   ; Start motion
N155   G1 X-47.84399 Y-23.76942 Z3.80953 A3=-.39485858 B3=.49800333 C3=.77206177
       M8 F=R1
N160   X-47.69248 Y-23.71986 Z3.51328 A3=-.39485858 B3=.49800333 C3=.77206177
N165   ...
...
N6555  X-103.02652 Y-5.64791 Z4.55886 A3=.55592652 B3=.22406464 C3=.80046283
N6560  X-102.837 Y-5.70081 Z4.83174 A3=.55592652 B3=.22406464 C3=.80046283
N6565  ;Retraction
N6570  X-102.6453 Y-5.71617 Z5.10778 A3=.55592652 B3=.22406464 C3=.80046283
N6575  ;Retraction movement
N6580  G0 X-95.97418 Y-3.0274 Z14.71333 A3=.55592652 B3=.22406464 C3=.80046283
N6585  Z31.9 A3=.55592652 B3=.22406464 C3=.80046283
N6590  G0 X0 Y0 Z100 A3=0 B3=0 C3=1
N6595  TRAFOOF                             ; Transformation OFF
N6600  SUPA G0 Z0.0 D0                     ; Max. return in Z (MCS)
N6605  SUPA G0 X0.0 Y0.0 A0.0 C0.0 D1      ; Max. return in XY (MCS)
N6610                                      ; and move rotary axes to basic position
N6615  M17                                 ; End of subprogram
```

Complex free-form surfaces



Contents

6

6.1 Special functions for free-form surfaces

When machining free-form surfaces, surface quality is the top priority. This calls for a correspondingly high level of accuracy for the CAM data with small tolerances and a large number of intermediate points.

The large number of points results in even distribution, thereby ensuring a high level of accuracy and an extremely smooth surface. The integrated high-speed setting cycle (CYCLE832) ensures a high machining speed. This cycle is responsible for activating all the functions that are required for milling free-form surfaces. This includes the Look Ahead function featuring blending and jerk limitation, which anticipates a configurable number of traversing blocks so that the machining speed can be optimized.

In addition, precontrol has been implemented to ensure that machining is free from following errors. This is supplemented by the COMPCAD online compressor, which is specifically recommended for free-form surface milling applications. This brings together a sequence of G1 commands in accordance with the tolerance set in CYCLE832 and compresses them into a spline that can be executed by the control directly.

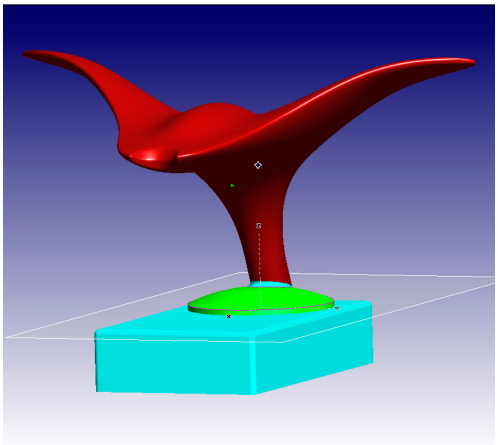
Important functions for the machining process of free-form surfaces:

- High-speed settings CYCLE832
  as optimum data compression within the tolerance band, combined with precontrol and jerk limitation, can be set for the required surface finish without contour deviation.
- TRAORI,
  as it has an integrated 5-axis transformation feature for all kinds of machine kinematics, enabling the tool to be perfectly oriented in relation to the surface so that no contour or surface defects can occur.
- VCS,
  as maximum accuracy is required in conjunction with extremely large workpieces. Particularly applicable when making compression molds and templates in an automotive engineering context.
- Integrated process chain from generation in CAD through to execution on the CNC.

6.2 Example, milling a manta ray

This example involves milling a manta ray from a free-form surface model. The manta ray is modeled using a CAD/CAM system.

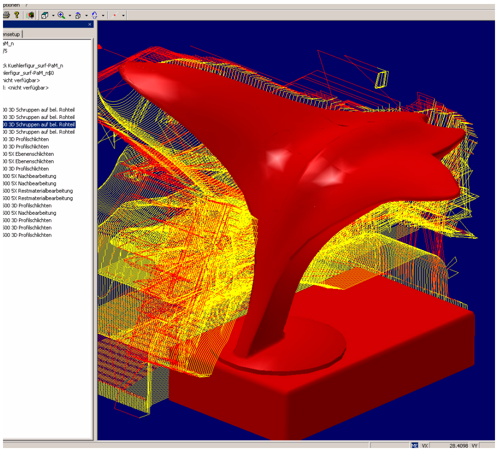
Manta ray in the CAM system



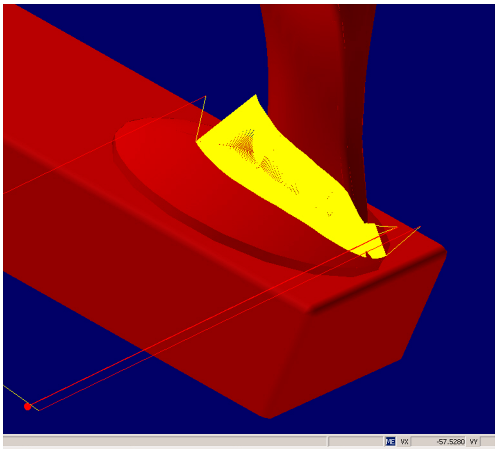
The manta ray is modeled as a free-form surface in the CAM system using a digitized scatterplot. The machining strategies included 3D plane roughing and several 3-axis and 5-axis semi-finishing and finishing operations.

The face was machined, for example, using a line-by-line finishing operation with axes permanently set at an angle. This technique enabled optimum surface quality to be achieved given the extreme level of curvature involved.

Plane roughing ***with 3 axes***



5-axis residual material machining



Strategies for 5-axis residual material machining were used to finish the residual material, e.g. undercutting without taking off the tool.

Program code example

The NC programs for producing the manta ray involve a number of roughing, semi-finishing, and finishing strategies. The key components of the NC programs are illustrated below on the basis of the start program and a roughing program.

Start program example

Within the start program, it is sometimes a good idea to make all the settings in the geometry subprograms, particularly as far as individual part production and the test phase are concerned. The start program will only call those subprograms that contain all the specifications, such as tool, CYCLE832. This is particularly helpful in cases where the postprocessor is perfectly attuned to the SINUMERIK system and all higher-order functions have already been integrated. In test mode, it is advisable to execute the subprograms individually; jump labels can be used within the start program, for example, for the purpose of launching the required subprogram.

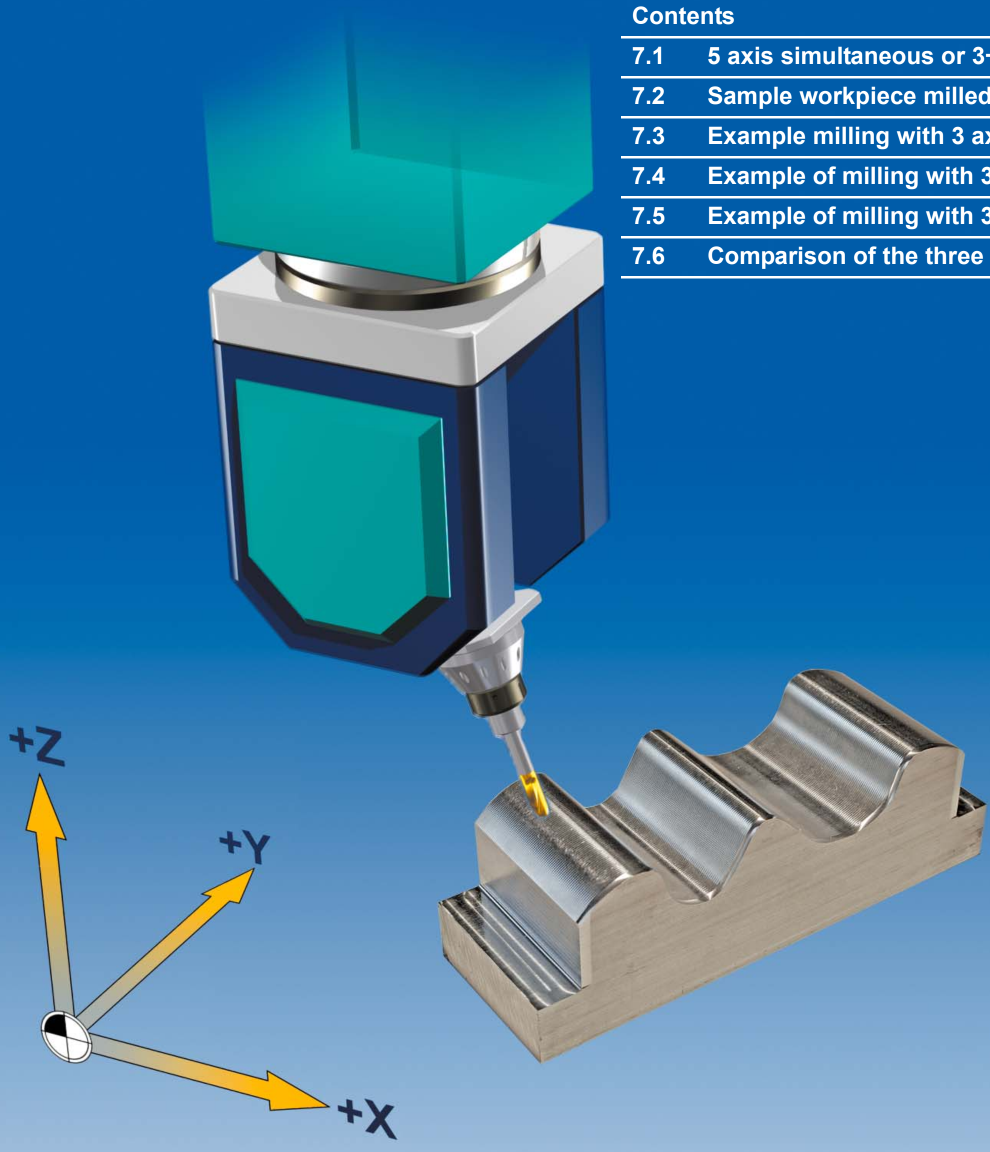
```
N100   G90 G17 G54            ; Absolute dimension specification, select working plane
                              ; and work offset
N105   ORIWKS ORIAXES         ; Workpiece coordinate system, axis interpolation
N110   GOTOF _ROUGH_01        ; Jump mark to the subprogram call for roughing using
                              ; program ROUGH_01.MPF
                              ; This program is explained in greater detail on the next
                              ; page.
N120   ;GOTOF _ROUGH_02       ; Unused jump labels are
                              ; commented out for the test phase.
       ...                    ;
N210   ;GOTOF _FINISH_05      ;
N220   _ROUGH_01:             ; Jump destination for GOTOF
N230   EXTCALL "ROUGH_01"     ; Call for roughing 01 subprogram
N240   STOPRE                 ; Preprocessing memory stopped, i.e. the subsequent NC
                              ; blocks will only be read in once all the previous NC
                              ; blocks have been executed.
N250   M00                    ; Program stop
N280   ...
N360   _FINISH_05:            ;
N370   EXTCALL "FINISH_05     ; Subprogram call for the last finishing program
N380   STOPRE                 ;
N390   M00                    ;
N400   M30                    ; End of program
```

Subprogram example
Roughing ROUGH_01

The subprogram contains the NC blocks for the geometry and all the data required for production. Assuming that your postprocessor has been optimized, all this data should be listed in the subprogram. All subprograms are structured in a similar fashion. They only differ in terms of the tool data, technology data, CYCLE832 parameters, and of course the NC blocks.

```
N100   ; TOOL                      ; Tool specification in the form of a comment
N110   ; T1 radius milling tool D32 ; Tool dimensions
       R2
N120   G90 G17 G54                 ; Absolute dimension specification, select working plane
                                   ; and work offset
N130   TRAFOOF                     ; Deactivate all active transformations and frames
N140   CYCLE800(1,"TC1",0,57,0,0,0,0,0,0,0,0,0,-1,)
N145                               ; Swivel all axes to the basic position
N150   CYCLE800()                  ; Resetting of the swiveled planes for defined original
                                   ; position
N160   T1                          ; Call tool T1
N170   M6                          ; Change tool in spindle
N180   R2=10000                    ; R2 as parameter for feedrate in the XY plane
                                   ; Feedrate is programmed in NC block as R2. In this way,
                                   ; the feedrate value can be modified quickly for the test
                                   ; phase.
N190   R1=10000                    ; R1 as feedrate in Z direction
N200   R3=4500                     ; Reduced feedrate
N210   S10000 M3 M8                ; Spindle speed, clockwise rotation, cooling on
N220   CYCLE800(0,"TC1",0,57,-36,0,-105,0,0,0,0,0,0,-1)
N225                               ; Pre-positioning of the tool in relation to the workpiece. In
                                   ; each subprogram, a fixed position should first be
                                   ; approached/swiveled into so that there is a defined orig-
                                   ; inal position at the start of machining. This means that if
                                   ; TRAORI is active, the way the workpiece is approached
                                   ; may vary under certain circumstances. Pre-positioning
                                   ; without TRAORI.
N230   CYCLE832(0.13,3,1)          ; Define high speed settings with 0.13 tolerance for
                                   ; roughing.
                                   ; 3 roughing
                                   ; 1 from SW 2.6
N240   G0 X133.1221 Y1.2413        ;
N250   G0 Z125                     ;
N260   G0 Z108.1501                ;
N270   G1 Z103.1501 F=R1           ; The programmed feedrate R1 is used here.
N280   X126.5626 Y1.1611 F=R2      ; The programmed feedrate R2 is used here
N290   ...                         ; NC blocks for geometry
...    ...
N4580  G0 Z150                     ; Retraction in Z
N4590  CYCLE800(1,"TC1",0,57,0,0,0,0,0,0,0,0,0,-1,)
N4595                              ; Swivel to original position
N4600  CYCLE832(0.02,0,1)          ; Set CYCLE832 to default values
N4610  CYCLE800()                  ; Resetting of the swiveled planes
N4620  M17                         ; End of subprogram
```

Difference 3, 3+2, 5 axes



7

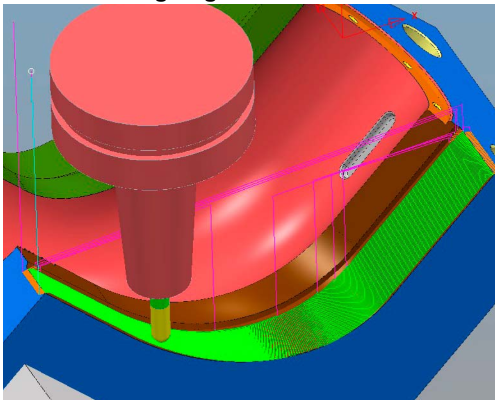
7.1 5 axis simultaneous or 3+2 axes

In this chapter, you will learn more about the differences when milling with 3 or 5 axes. The cost-effectiveness must always be taken into account for 5-axis machining? This means that the following procedure is recommended for the production, if permitted by the workpiece geometry and the machine kinematics:

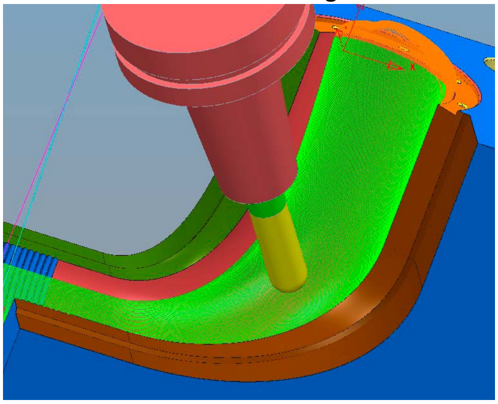
- As much as possible 3-, 3+1- and 3+2-axis (set at an angle) roughing/semi-finishing
- 5-axis simultaneous milling only for the finishing

The advantages and disadvantages of the 3+2-axis and the 5-axis simultaneous machining are briefly compared in the following tables.

Semi-finishing angled 3+2



5-axis simultaneous finishing



3+2-axis machining

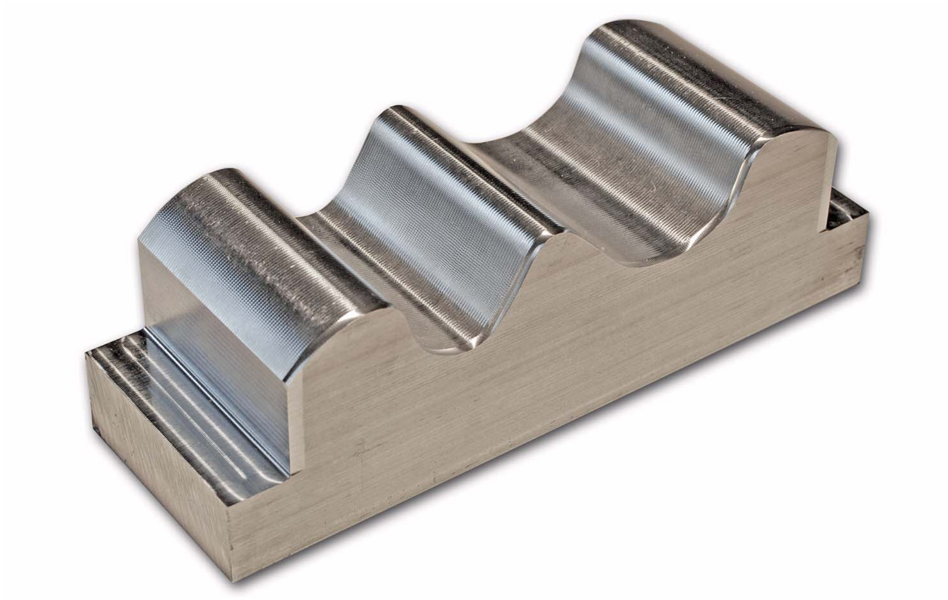
Advantages	Disadvantages
■ Less programming work ■ No limitation of the dynamic response as only linear axes are used ■ Greater stiffness of the moving axes with 3+2-axis machining (depending on the machine concept). As a result, longer tool life and higher surface quality	■ Limiting of the workpiece the geometry, e.g. undercut cavities ■ Longer tools may be required ■ Several setting positions (3+2-axis) required, possibly longer machine times and visible machining transitions

5-axis simultaneous machining

Advantages	Disadvantages
■ Deep cavities and undercuts can be machined in one clamping operation ■ Shorter tools with compact clamping ■ More uniform surfaces - no transitions ■ No special tools necessary	■ More programming work ■ Higher risk of collision ■ Frequently longer machining times due to compensatory movements of the kinematics ■ Tolerances of the kinematics can add up because more axes are being used

7.2 Sample workpiece milled using 3 to 3+2 axes

Using an example shaft-shaped workpiece, the programming is analyzed with 3-, 3+1- and 3+2-axis. The procedure and results of the different milling operations are shown based on short program excerpts.



Workflow start program, roughing and semi-finishing

For machining, a start program was created in each case which calls up the geometry programs. This is identical for all four variants. The programs for the roughing and semi-finishing are the same for the three workpieces. The differences are only to be found in the finishing programs.

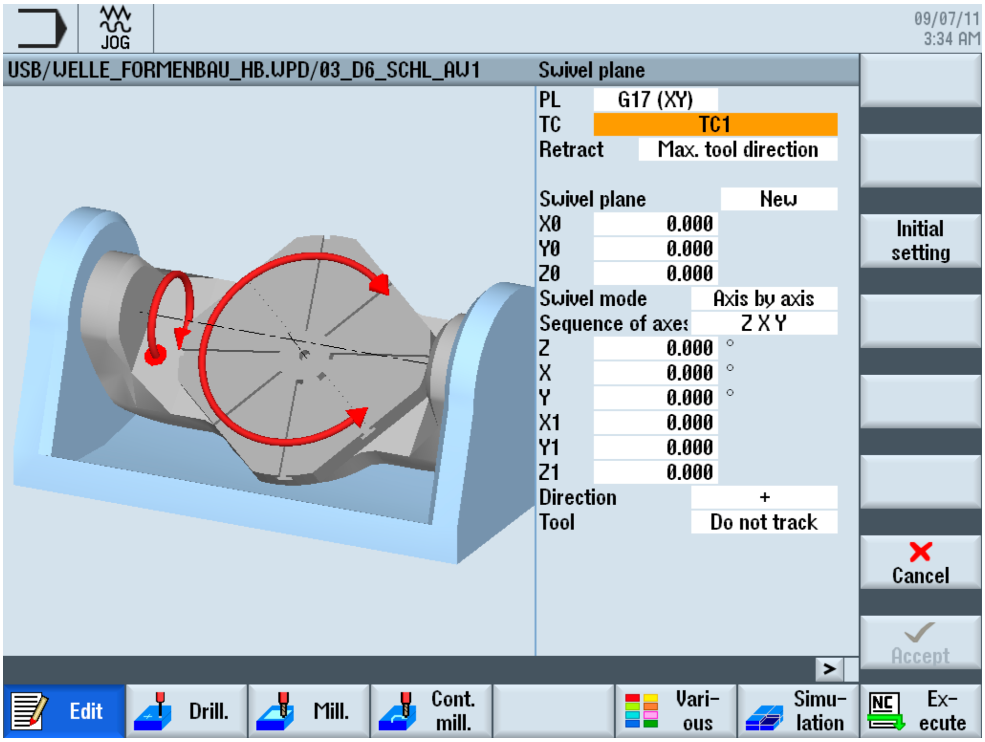
The distance between the individual milling paths in the finishing programs has been set to be relatively large so that the uniformity of the individual paths can be evaluated more accurately and errors can be seen.

Prepositioning the milling tool at the workpiece

In the geometry programs, the milling cutter should first be pre-positioned at the workpiece, because if TRAORI is active, a collision with the workpiece at any given position in the milling machine's interior or overtraveling of the software limit switch cannot be ruled out during orientation of the rotary axes. You can define the rotary axis positions either via CYCLE800 or an ORIRESET.

Prepositioning with CYCLE800

N100	CYCLE800(4,"TABLE",200000,39,0,0,0,0,0 ,0,0,0,0,1,,1)	; Deselect CYCLE800, move to basic ; position
N105	...	;



Basic position of the tool orientation with ORIRESET

N100	TRAORI	; Activate TRAORI
N105	ORIRESET (90, 45)	; Example of machine kinematics CA ; (channel axis names C, A) ; C to 90 degrees, A to 45 degrees

By programming ORIRESET (A, B, C), the orientation axes are moved in linear and synchronous motion from their current position to the specified basic position. ORIRESET is only possible when TRAORI is active. If a basic position is not programmed for an axis, a defined position from the associated machine data $MC_TRAFO5_ROT_AX_OFFSET_1/2 is used.

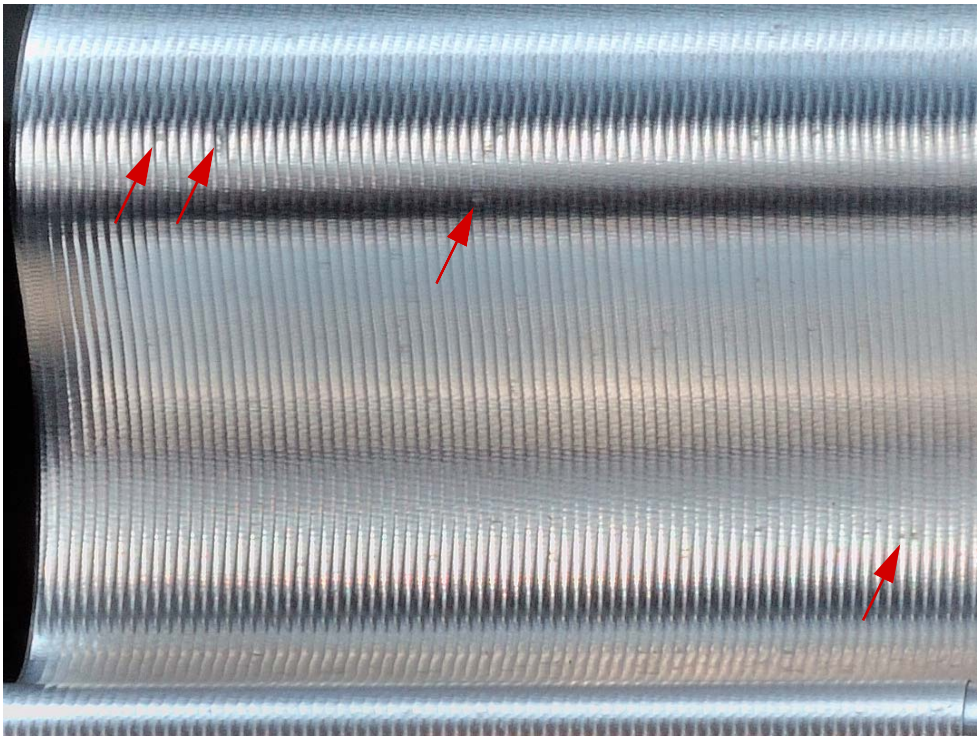
7.3 Example milling with 3 axes

In the first example, the shaft was only milled with the three linear axes.

3-axis finishing

N100	T="K_D6"	; Selection of the ball mill with D=6
N110	M6	; Loading of tool
N120	R1=8000	; R1 to R3 are variables for feedrates
N130	R2=8000	;
N145	R3=8000	;
N150	S18000	;
N160	M03 M8 G54 G17 G90	;
N170	CYCLE832(0.005,1,1)	; High Speed Settings finishing with tolerance 0.005
N180	G00 X-3.9247 Y-5.5063 Z10	; Approach with G0
N190	G00 X-3.9247 Y-5.5063 Z-6.7226	;
N200	G01 X-3.9247 Y-5.5063 Z-11.7226 F=R1	; Begin machining
N210	G01 X-3.9224 Y-5.503 Z-11.7235 F=R2	;
N290	...	;
N4580	G00 X102.5039 Y46.3472 Z10	;
N4600	CYCLE832(0,0,1)	; Deselect High Speed Settings
N4610	M5	;
N4620	M30	; End of program

Image excerpt of the upper shaft section



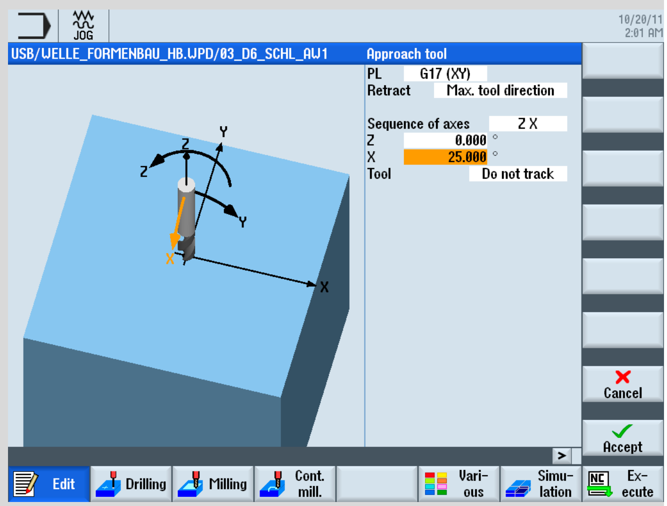
The milling paths are very uniform. Pressure points / matt points are only visible on narrow curves due to the pressing cut of the ball mill. These are caused by the milling cutter not being able to cut freely and the material being pressed out.

7.4 Example of milling with 3+1 axes

In the second example, the X axis has been adjusted by 25°.

3+1 axis finishing

```
N100   T="K_D6"                                ; Selection of the ball mill with D=6
N110   M6                                      ; Loading of tool
N120   R1=8000                                 ; R1 to R3 are variables for feedrates
N130   R2=8000                                 ;
N145   R3=8000                                 ;
N150   S18000                                  ;
N160   M03 M8 G54 G17 G90                      ;
N170   CYCLE800(4,"TABLE",200000,39,0,0,0,     ; Swivel to basic position
       0,0,0,0,0,0,1,,1)
N180   CYCLE800(4,"TABLE",101,39,,,,0,25,,,,,  ; Swivel around the X axis through 25°
       -1,100,1)
```



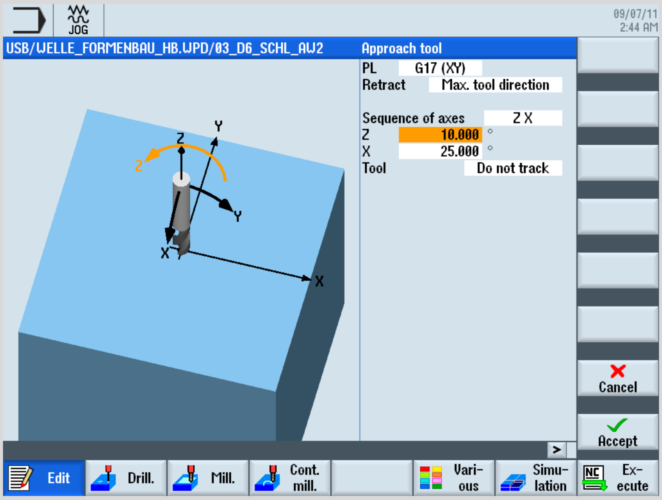

```
N190   CYCLE832(0.005,1,1)                     ; High Speed Settings finishing with toler-
                                                 ance 0.005
N200   G00 X-3.9247 Y-5.5063 Z10               ; Approach with G0
N210   G00 X-3.9247 Y-5.5063 Z-6.7226          ;
N220   G01 X-3.9247 Y-5.5063 Z-11.7226         ; Begin machining
       F=R1
N230    G01 X-3.9224 Y-5.503 Z-11.7235 F=R2    ;
N290   ...                                     ;
N4580  G00 X102.5039 Y46.3472 Z10              ;
N4600  CYCLE832(0,0,1)                         ; Deselect High Speed Settings
N4610  CYCLE800(4,"TABLE",110000,39,0,0,0,     ; Swivel to basic position
       0,0,0,0,0,0,0,,1)
N4620  CYCLE800(4,"0",220000,39,0,0,0,0,0,0,   ; Deselect swivel
       0,0,0,0,,1)
N4630  M5                                      ;
N4640  M30                                     ; End of program
```

7.5 Example of milling with 3+2 axes

In the third example, the X axis has been adjusted by 25° and the Z axis has been adjusted by 10°.

3+2-axis finishing

N100	T="K_D6"	; Selection of the ball mill with D=6
N110	M6	; Loading of tool
N120	R1=8000	; R1 to R3 are variables for feedrates
N130	R2=8000	;
N145	R3=8000	;
N150	S18000	;
N160	M03 M8 G54 G17 G90	;
N170	CYCLE800(4,"TABLE",200000,39,0,0,0,0,0,0,0,0,0,1,,1)	; Swivel to basic position
N180	CYCLE800(4,"TABLE",101,39,,,,10,25,,,,,-1,100,1)	; Swiveled around the X axis through 25° and around the Z axis through 10°

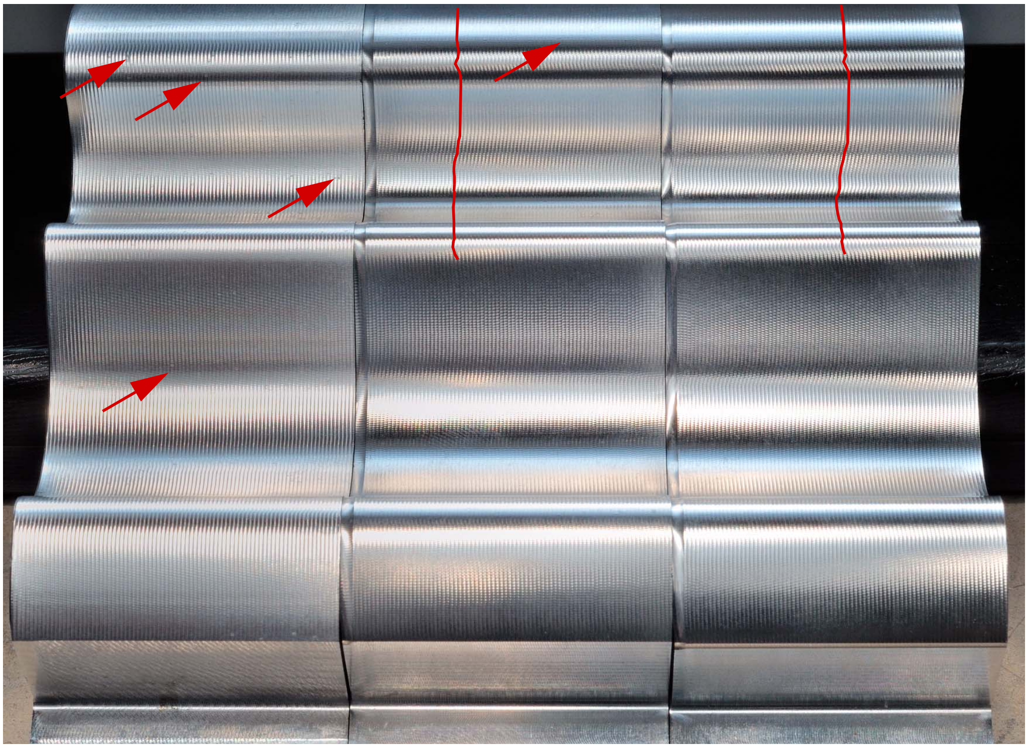


N190	CYCLE832(0.005,1,1)	; High Speed Settings finishing with toler-ance 0.005
N200	G00 X-3.9247 Y-5.5063 Z10	; Approach with G0
N210	G00 X-3.9247 Y-5.5063 Z-6.7226	;
N220	G01 X-3.9247 Y-5.5063 Z-11.7226 F=R1	; Begin machining
N230	G01 X-3.9224 Y-5.503 Z-11.7235 F=R2	;
N290	...	;
N4580	G00 X102.5039 Y46.3472 Z10	;
N4600	CYCLE832(0,0,1)	; Deselect High Speed Settings
N4610	CYCLE800(4,"TABLE",110000,39,0,0,0,0,0,0,0,0,0,,1)	; Swivel to basic position
N4620	CYCLE800(4,"0",220000,39,0,0,0,0,0,0,0,0,0,,1)	; Deselect swivel
N4630	M5	;
N4640	M30	; End of program

7.6 Comparison of the three versions

The figure below shows a comparison of the three milling variants. From left to right, 3-axis, 3+1-axis, and 3+2-axis.

Shaft milling versions



3-axis

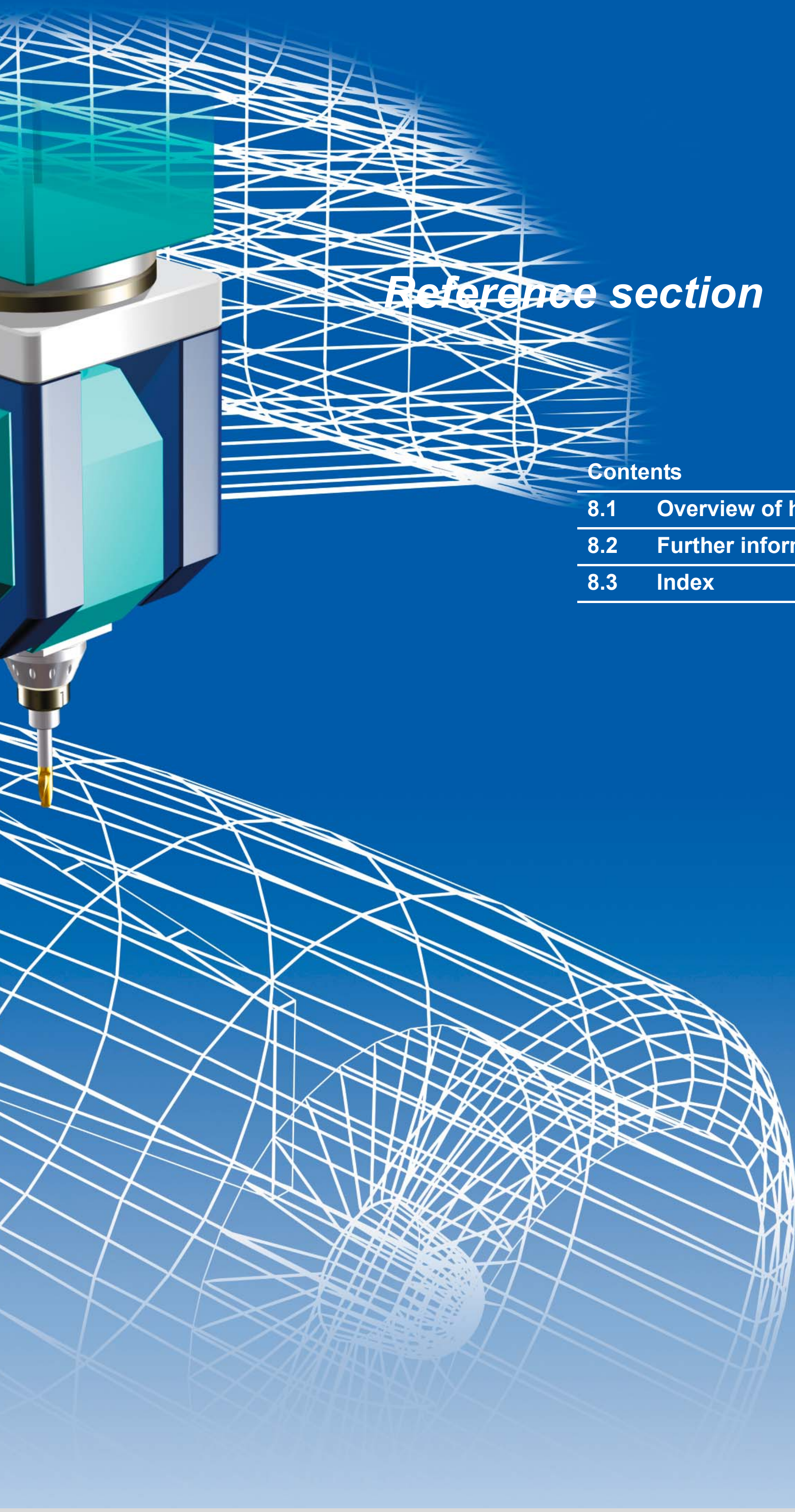
The milling paths are very uniform. Pressure points / matt points are only visible on narrow curves due to the pressing cut of the ball mill. These are caused by the milling cutter not being able to cut freely and the material being pressed out.

3+1 axis

The milling paths are very uniform. Hardly any pressure points / matt points are now visible on narrow curves due to the pressing cut of the ball mill. Due to the positioning, the blade of the ball mill can now cut freely. However, distortions of the milling paths are visible due to the positioning. The cutting point drifts due to the positioning on the ball mill. The milling paths no longer run parallel to the body edges.

3+2 axis

The milling paths are very uniform. No pressure points / matt points are now visible on narrow curves due to the pressing cut of the ball mill. Due to the positioning, the blade of the ball mill can now cut freely. However, even greater distortion of the milling paths is visible due to the positioning.



Reference section

Contents Page

8

8.1 Overview of higher-order functions

Higher-order functions of the SINUMERIK control system 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 5-axis machining.

Motion commands

Language elements with circular interpolation programming

CIP	Circular interpolation through intermediate point CIP X... Y... Z... I1=... J1= K1=..
CT	Circle with tangential transition CT X... Y... Z...
TURN	Number of full circles to be traversed G3 X… Y… I… J… TURN =
CR=	Additional parameters Circle radius
I1, J1, K1	Intermediate points in Cartesian coordinates (in X, Y, Z direction)
AP=	End point in polar coordinates, polar angle, also in the case of linear interpolation
RP=	End point in polar coordinates, polar radius, also in the case of linear interpolation
AR=	Opening angle

840D spline versions

CSPLINE	Activation of cubic interpolating spline
ASPLINE	Activation of Akima spline
	Start and end condition BNAT / ENAT zero curvature BTAN / ETAN tangential transition BAUTO/EAUTO C3-constant at first and last spline segment transition

BSPLINE **SD=...** **PL=...** **PW=...**	B spline activation B spline (max. 3) Interval length (node vector), "non uniformity" Weightings, i.e. denominator of rational B spline with polynomial representation **Example** N20 BSPLINE X... Y... SD=... PL=... PW=...

POLY **SD=...** **PL=...**	Activation of polynomial interpolation, B spline representation in polynomial format B spline order (max. 5!! -> difference to BSPLINE) Interval length (node vector), "Non-uniformity" **Syntax** PO[axis] = (block end position, a2 (square coefficient), a3 (cube coefficient), a4, a5) -> numerator polynomial PO[ ] = (block end position, b2, b3, b4, b5) -> denominator polynomial **Example** N10 POLY PO[X] = (0.25,0.5,0) PO[Y] = (0.433,0,0) PO[] = (1,1,0)

Compressor

COMPCAD	Surface-optimized compressor (constant acceleration) Also see CYCLE832
COMPSURF	Optimized compressor for inclined line-by-line finishing programs with poor data quality or irregular point distribution Only in conjunction with Top Surface
COMPCURV	Transitions with constant acceleration and jerk-free transitions
COMPON	Transitions with constant velocity

UPATH	Additional commands for combining path and synchronized axes. Parameter assignment for synchronized axes corresponds to path axes, i.e. the following applies to the motion of a synchronized axis A: A = f(u), where u denotes the path parameter for the path motion.
SPATH	Parameter assignment of the synchronized axes follows the arc length for the path axes, i.e. the following applies to the motion of a synchronized axis A: A = f(s), where s denotes the arc length for the path motion.

Dynamic response

Technology G groups

DYNNORM	Standard dynamic response, as before
DYNPOS	Positioning mode, tapping
DYNROUGH	Roughing
DYNSEMIFIN	Finishing
DYNFINISH	Smooth finishing

Look Ahead

G64	**Overrun of block end (LOOK AHEAD)**
	Blending
G641	ADIS = … smoothing distance ADISPOS =… smoothing distance for G0, constant velocity
G642	Blending with single-axis tolerances or ADIS, ADISPOS with intermediate blocks, constant-acceleration
G645	Continuous-path mode with blending and tangential block transitions within defined tolerances
G60, G64, G645	G code group 10

Velocity programming

	Conventional block-by-block (non-modal) velocity programming in
G94	inch/min or mm/min
G93	inverse time
G95	inch, mm per spindle revolution
G96	Constant cutting rate
	Programming velocity/feed profiles To permit flexible definition of the feed characteristic, the feed programming according to DIN 66025 has been extended by linear and cubic characteristics. The cubic characteristics can be programmed either directly or as interpolating splines. These additional characteristics make it possible to program continuously smooth velocity characteristics depending on the curvature of the workpiece to be machined. As a result, limiting acceleration changes can be programmed, enabling uniform workpiece surfaces to be produced.

FNORM	Basic setting. The feed value is specified as a function of the traverse path of the block and is then valid as a modal value.
FLIN	Path velocity profile linear: The feed value is approached linearly via the traverse path from the current value at the block beginning to the block end and is then valid as a modal value.
FCUB	Path velocity profile cubic: The blockwise programmed F values (relative to the end of the block) are connected by a spline. The spline starts and ends at a tangent to the previous or subsequent feedrate function. If the F address is missing from a block, the last F value to be programmed is used. Path velocity profile via polynomial:
F=FPO(..)	The F address defines the feed characteristic via a polynomial from the current value to the block end. The end value is valid thereafter as a modal value.
endfeed	Feedrate at block end
quadf	Quadratic polynomial coefficient
ubf	Cubic polynomial coefficient
	Path reference
FGROUP(X, Y, Z,...)	Defines the path axes in relation to the feedrate, i.e. the overall feedrate relates to the axes defined here. Example: FGROUP(X, Y), then the following applies: $F = \sqrt{F_x^2 + F_y^2}$

Jerk

SOFT	Jerk limiting
BRISK	Acceleration limitation

Precontrol

FFWON	Precontrol on
FWOF	Precontrol OFF

5-axis functionality

Transformation

TRAORI	Activate transformation 1
TRAORI(1)	Activate transformation 1
TRAORI(2)	Activate transformation 2
TRAORI(1, ..., ..., ...)	Activate transformation 1, generic transformation, additional 3 parameters for vector of the basic orientation
TRAORI(2, ..., ..., ...)	Activate transformation 2, generic transformation, additional 3 parameters for vector of the basic orientation
TRAFOOF	Deactivate transformation

Orientation programming

ORIEULER	Orientation programming on the basis of Euler angles (default)
ORIRPY	Orientation programming using RPY angle Otherwise, specified via machine data.
A2=... B2=... C2=...	Euler or RPY angle
A3=... B3=... C3=...	Cartesian orientation vector
XH=..., YH=..., ZH=...	For ORIVECT or ORIPLANE synonymous with A3=... etc. More extensive meaning in conjunction with ORICURVE; in this case either serves as a control polygon with BSPLINE, or in conjunction with POLY has a polynomial definition, otherwise linear interpolation for the upper straight line, geometric large circle, but not in terms of velocity.
LEAD, TILT	Lead/tilt angle relative to normal vectors and path tangent. The normal vectors at the start of the block and at the end are defined by A4=... B4=... C4=... and A5=... B5=... C5=... Only in conjunction with ORIPATH.

Orientation reference

ORIMKS	The reference system for the orientation vector is the basic coordinate system. Notice: Response of ORIMKS can be set via machine data.
ORIWKS	The reference system for the orientation vector is the workpiece coordinate system. Notice: Response of ORIWKS can be set via machine data.

Orientation interpolation

	The following G codes are only active if the corresponding machine data is set:
	Axis interpolation
ORIAXES	Linear interpolation of the machine axes or interpolation of the rotary axes using polynomials (with active POLY)
	Vector interpolation
ORIVECT	Interpolation of the orientation vector in a plane (large circle interpolation)
ORIPLANE	Interpolation in a plane (large circle interpolation), synonymous with ORIVECT
ORIPATH	Tool orientation in relation to the path. A plane is created from the normal vector and path tangent that defines the meaning of LEAD and TILT at the end point. In other words, the path reference only applies to the definition of the end orientation vector. Large circle interpolation is performed between the start and end orientations. LEAD and TILT do not merely provide the lead and tilt angles. They are defined as follows: LEAD defines the rotation in the plane created by the normal vector and path tangent. TILT then defines the rotation around the normal vector. In other words, they correspond to theta and phi in a sphere coordinate system, with the normal vector serving as the Z axis and the tangent as the X axis.
ORICONCW	Interpolation on the peripheral surface of a taper in the clockwise direction
ORICONCCW	Interpolation on a peripheral surface of a cone in the counterclockwise direction In both cases, additionally required: A3=... B3=... C3=... or XH=., YH=... , ZH=... End orientation, cone axis of rotation: A6, B6, C6 Opening angle: NUT=.
ORICONIO	Interpolation on a peripheral surface of a cone with an intermediate orientation specified via A7=… B7=... C7=.... Also required: A3=... B3=... C3=... or XH=..., YH=..., ZH=... End orientation

ORICONTO	Interpolation on a peripheral surface of the cone with tangential transition Also required: A3=... B3=... C3=... or XH=..., YH=..., ZH=... End orientation With POLY, PO[PHI] = ..., PO[PSI]=... can also be programmed in these cases. This is a more generalized form of large circle interpolation, whereby polynomials are programmed for the lead and tilt angles. With cone interpolation, the polynomials have the same significance as with large circle interpolation for the given start and end orientations. The polynomials can be programmed with ORIVECT, ORIPLANE, ORICONCW, ORICONCCW, ORICONIO, ORICONTO.
ORICURVE	Orientation interpolation with specification of the tool tip motion and that of a second point on the tool. The path of the second point is defined via XH=... YH=... ZH=... defines, in conjunction with BSPLINE as a control polygon with POLY as a polynomial:' PO[XH] = (xe, x2, x3, x4, x5) PO[YH] = (ye, y2, y3, y4, y5) PO[ZH] = (ze, z2, z3, z4, z5) If the BSPLINE or POLY additional information is omitted, straightforward linear interpolation will be performed accordingly between the start and the end orientation.

ORISON	Smoothing of the orientation characteristic ON. The "Smoothing the orientation characteristic (ORISON)" function can be used to smooth oscillations affecting orientation over several blocks. The aim is to achieve a smooth characteristic for both the orientation and the contour.
ORISOF	Smoothing of the orientation characteristic 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 method at external corners (all compensation types)

2½D circumferential milling

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

3D circumferential milling

CUT3DC	Compensation perpendicular to path tangent and tool orientation
CUT3DCD	Activating the 3D radius compensation referred to a differential tool for circumferential milling
ISD	The clearance between the milling tool tip (FS) and the milling tool construction point (FH) are specified using the command ISD.
ORID	No changes in orientation in inserted circular blocks at external corners. Orientation motion is performed in the linear blocks.
ORIC	Travel path is extended by means of circles. The change in orientation is also performed proportionately in the circle.

Face milling

CUT3DFS	Constant orientation (3-axis). Tool points in the Z direction of the coordinate system defined using G17-G19. Frames do not have any effect.
CUT3DFF	Constant orientation (3-axis), tool in Z direction of the coordinate system currently defined via the frame
CUT3DF	5-axis with variable tool orientation

3D circumferential milling with limiting surface - combined circumferential/face milling

CUT3DCC	NC program relates to the contour on the machining surface.
CUT3DCCD	The NC program relates to the tool center point path.

FRAMES

Programmable frames

TRANS X… Y… Z…	Absolute offset
ATRANS X… Y… Z…	Incremental offset, relative to a frame that is already active
ROT X… Y… Z…	Absolute rotation
AROT X… Y… Z…	Incremental rotation, relative to a frame that is already active
ROTS X… Y…	Absolute rotation, which is defined using two angles. The angles are the angles of the lines of intersection between the inclined plane and the main planes with respect to the axes.
AROTS X… Y…	Incremental rotation, relative to a frame that is already active such as angle such as ROTS
RPL=…	Rotation in the plane
MIRROR X… Y… Z…	Absolute mirroring
AMIRROR X… Y… Z…	Incremental mirroring, relative to a frame that is already active
SCALE X… Y… Z…	Absolute scaling
ASCALE X… Y… Z…	Incremental scaling, relative to a frame that is already active

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…)

Special frames

TOFRAME	Tool frame, coordinate system with Z axis in tool direction, zero point is the tool tip
TOFRAMEX	Tool frame, coordinate system with X axis in tool direction, zero point is the tool tip
TOFRAMEY	Tool frame, coordinate system with Y axis in tool direction, zero point is the tool tip
TOFRAMEZ	Tool frame, coordinate system with Z axis in tool direction, zero point is the tool tip, identical to TOFRAME
TOROT	Tool frame, coordinate system with Z axis in tool direction, only contains the rotation component from TOFRAME. The zero point remains unchanged.

TOROTX	Tool frame, coordinate system with X axis in tool direction, only contains the rotation component from TOFRAME. The zero point remains unchanged.
TOROTY	Tool frame, coordinate system with Y axis in tool direction, only contains the rotation component from TOFRAME. The zero point remains unchanged.
TOROTZ	Tool frame, coordinate system with Z axis in tool direction, only contains the rotation component from TOFRAME. The zero point remains unchanged.

8.2 Further information/documentation

Information about the SINUMERIK system can be found in a number of sources. User and manufacturer documentation is supplemented by user forums and information on the Internet. An overview of this additional information is provided below.

SINUMERIK technical documentation

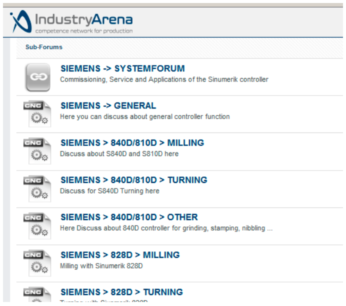
The complete SINUMERIK documentation, application examples and FAQs can be downloaded from the from the Internet.

https://support.industry.siemens.com/cs/ww/en/view/108464614

CNC4you - user portal

This portal provides up-to-date information about SINUMERIK controls and real-life examples.

www.siemens.com/cnc4you



SINUMERIK - user forum

The SINUMERIK user forum is a platform that allows you to discuss technical issues with other SINUMERIK users. The forum is moderated by experienced Siemens technicians and users.

https://en.industryarena.com/siemens

8.3 Index

Further information

For further information on SINUMERIK, visit
www.siemens.com/sinumerik

In-depth technical documentation is available from our Service&Support Portal:
www.siemens.com/automation/support

For a personal discussion, you can locate your nearest contact at:
www.siemens.com/automation/partner

You can place your order directly online at the Mall:
www.siemens.com/automation/mall

Siemens AG
Industry Sector
Drive Technologies
Motion Control
Postfach 3180
91050 Erlangen
GERMANY

Subject to change without prior notice
6FC5095-0AB10-0BP4