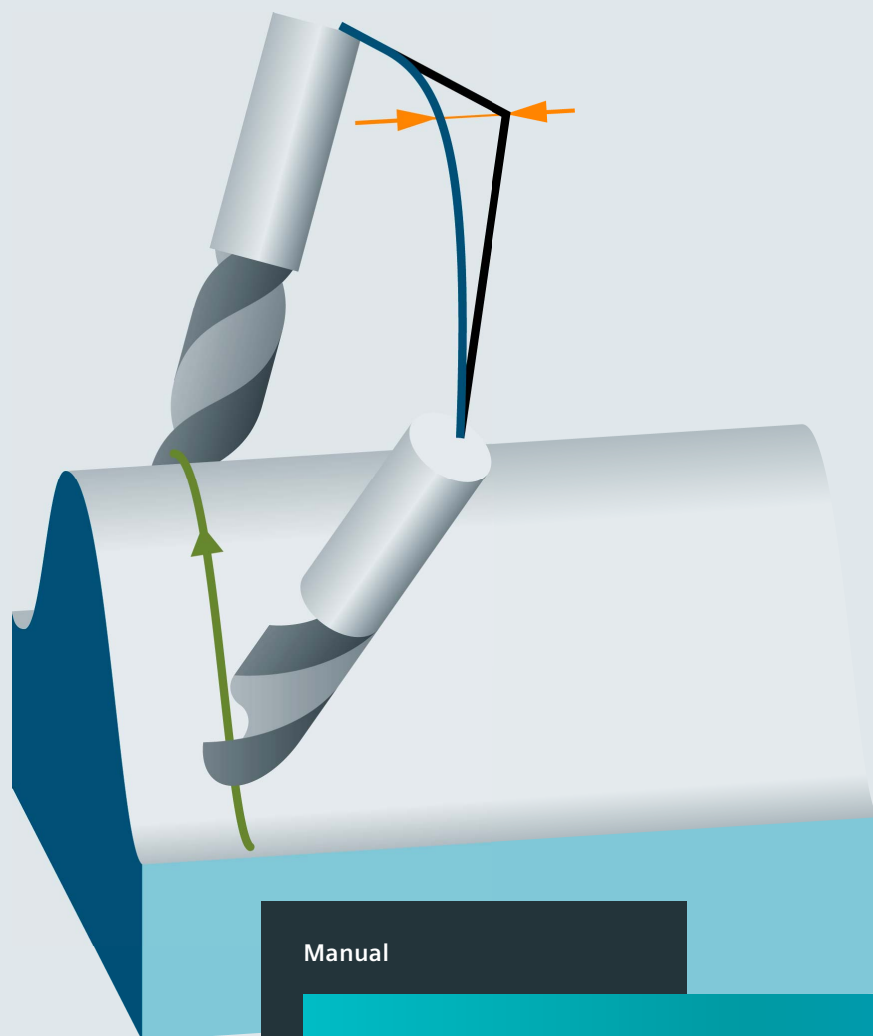


SIEMENS



Manual

SINUMERIK

Milling with SINUMERIK

Mold making with 3- to 5-axis simultaneous milling

Edition

09/2018

www.siemens.com/sinumerik

Milling with SINUMERIK

Mold-making with 3- to 5-axis simultaneous milling

Manual

Valid for:

Control

SINUMERIK 828D
SINUMERIK 840D sl

Edition 09.2018

Document Order No. 6FC5095-0AB10-0BP5

Basic information	1
General information on workpiece production	2
Important functions for 3- to 5-axis machining	3
Aerospace, structural parts	4
Engine and turbine components	5
Complex free-form surfaces	6
Difference between 3, 3+2 and 5 axes	7
Reference section	8

SINUMERIK® documentation**Edition history, trademarks**

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

Status code in the "Remarks" column:

- A** New documentation.
- B** Unrevised reprint with new order number.
- C** Revised version with new edition status.

Edition	Order no.	Remark
09.2018	6FC5095-0AB10-0BP5	C
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.

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

This document was created using a number of layout and graphic tools.

The reproduction, transmission or use of this document or its contents is not permitted without express written authority. Offenders will be liable for damages. All rights, including rights created by patent grant or registration of a utility model or design, are reserved.

© Siemens AG 2009 - 2018. All rights reserved.

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. 6FC5095-0AB10-0BP5

Siemens AG

	Page
1 Basic information	9
1.1 Introduction	10
1.2 SINUMERIK MDynamics milling technology packages	11
1.3 SINUMERIK Operate user interface	12
1.3.1 Operating and setup functions	12
1.3.2 SINUMERIK Operate - Multitouch	13
1.4 Requirements for milling	16
1.5 Linear axes, rotary axes and kinematics	17
1.5.1 Axes and programming	17
1.5.2 Kinematics of 5-axis machining centers	19
1.6 Surface quality, velocity, accuracy	20
1.7 Additional software	22
1.7.1 SinuTrain	22
1.7.2 VNC Viewer - remote control on an external PC	23
1.7.3 Analyze MyWorkpiece /Toolpath additional software	24
2 General information on workpiece production.....	25
2.1 Process chain for producing 3- to 5-axis workpieces	26
2.2 CAM systems	27
2.2.1 Tolerance and accuracy	28
2.3 Program structure for milling	29
2.4 Programs, data transfer and editor	33
2.4.1 Program manager	33
2.4.2 External storage media - data transfer	34
2.4.3 EES - Execution from External Storage (NC Extend)	35
2.4.4 Graphic and file formats in the program manager	36
2.4.5 Program editor - useful functions	37
2.5 JOG mode, zeros, tools	38
2.5.1 Zero offsets	38
2.5.2 Tool management	39
2.5.3 Manual retraction - return	40
2.6 Fundamentals, measuring in JOG and AUTOMATIC	41
2.6.1 Measurement logs in JOG	42
2.6.2 Measurement logs in AUTOMATIC	44
2.7 Measuring workpiece/tool in JOG	46
2.7.1 Workpiece measuring in JOG	46
2.7.2 Workpiece measuring functions in JOG	46
2.7.3 Example for measuring a workpiece in JOG	47
2.7.4 Measure tool in JOG	48

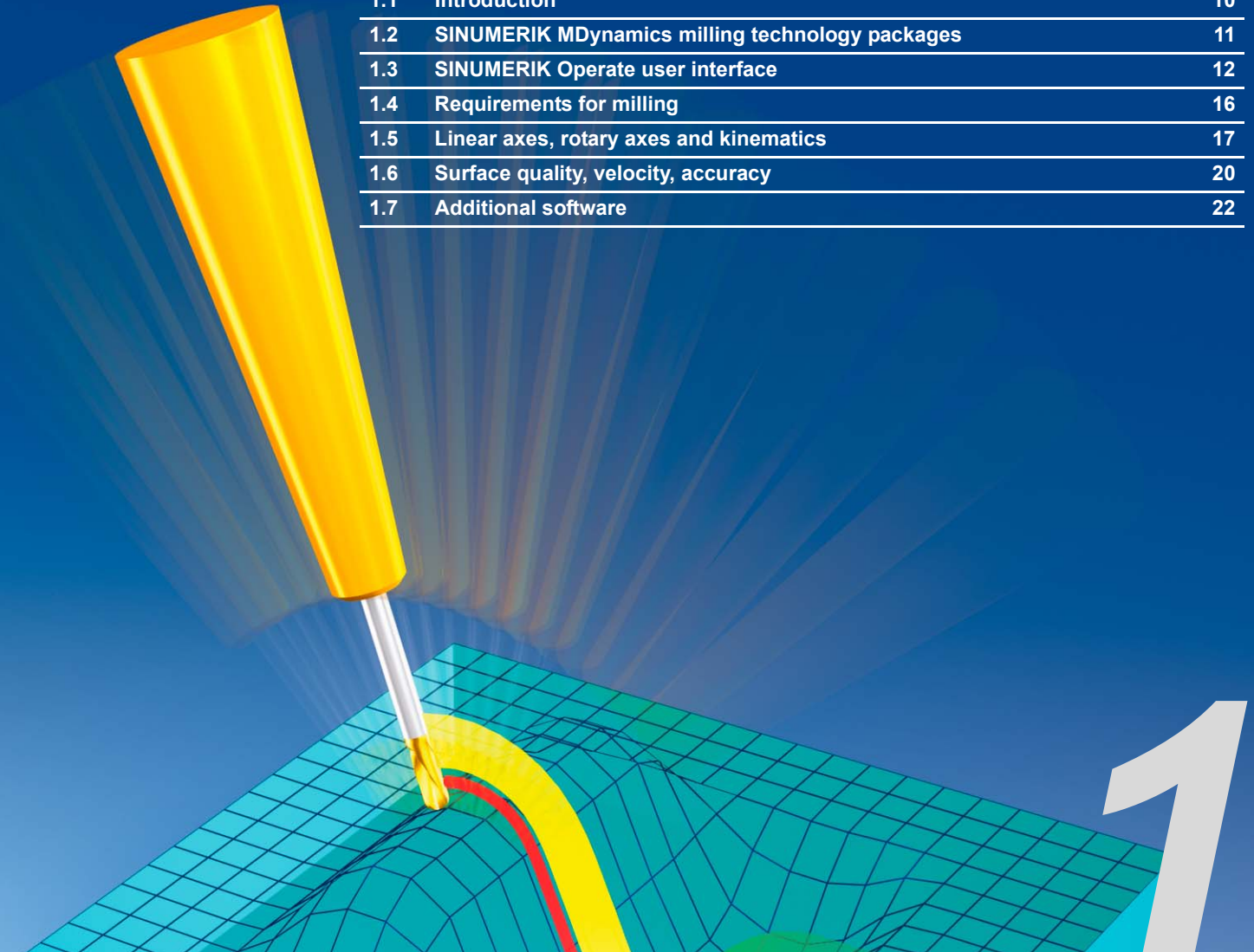
	Page
2.7.5	Tool reference point 48
2.7.6	Example: Measure tool in JOG 50
2.8	Measure workpiece/tool in AUTOMATIC - process measuring 52
2.8.1	Measure workpiece in AUTOMATIC 52
2.8.2	Workpiece measuring cycles in AUTOMATIC 52
2.8.3	Example for measuring a workpiece in AUTOMATIC 53
2.8.4	Tool measuring in AUTOMATIC - process measuring 54
2.8.5	Example for measuring tools in the AUTOMATIC mode 54
2.9	Checking and measuring machine/spindle 56
2.9.1	Spindle 3D angular deviation CYCLE995 56
2.9.2	Checking/measuring the machine with the kinematics measuring cycle CYCLE996 57
2.9.3	CYCLE9960 measure kinematics completely 59
2.10	Workpiece visualization 60
2.10.1	Simulation 60
2.10.2	Quick viewer for mold-making 61
2.10.3	Simultaneous recording 63
2.11	Executing programs (AUTOMATIC) 64
2.11.1	Display all relevant mold-making functions 64
2.11.2	Program running time 65
2.11.3	Handwheel override 66
2.11.4	Block search 67
2.12	Collision avoidance 68
3	Important functions for 3- to 5-axis machining 71
3.1	Introduction 72
3.2	Explanation of the terms: swivel, frames and TRAORI 73
3.3	Transforming coordinate systems - Frames 74
3.4	Swivel - CYCLE800 75
3.5	TRAORI 5-axis transformation 81
3.5.1	Basics of simultaneous 5-axis machining 81
3.5.2	The TRAORI command 85
3.5.3	Programming the tool orientation 86
3.5.4	Orientation interpolation and orientation reference 92
3.5.5	Example application involving TRAORI and frames 100
3.5.6	Useful NC commands for active TRAORI 101
3.6	High Speed Settings - CYCLE832 102
3.6.1	Parameters for CYCLE832 103
3.6.2	CYCLE832 and CUST_832 109

3.7	Advanced Surface and Top Surface	110
3.7.1	Compressor (COMPCAD, COMPSURF)	112
3.7.2	Continuous-path mode, Look Ahead – G64, G645	113
3.7.3	Feedforward control and jerk limitation – FFWON, SOFT,	115
3.7.4	Smoothing tolerance CTOL, OTOL, ATOL	116
3.7.5	Path reference UPATH/SPATH	116
3.7.6	Smoothing the orientation characteristic (ORISON, ORISOF)	117
3.7.7	Technology G groups	118
3.7.8	Pre-processing memory control FIFOCTRL	119
3.8	Milling radius compensation 2D/3D-WRK	120
3.8.1	3D-WRK cutter radius compensation for circumferential and face milling	120
3.8.2	CUT3DC/CUT3DCD circumferential milling	121
3.8.3	CUT3DF/CUT3DFD face milling	122
3.8.4	Circumferential milling with limiting surfaces (CUT3DCC/CUT3DCCD)	126
3.8.5	Overview of the tool types for the 3D compensations.	127
3.8.6	Explanation of the commands	127
3.8.7	Programming example for circumferential milling	129
3.9	Volumetric Compensation System (VCS)	130
3.10	VNCK - virtual machine	131
4	Aerospace, structural parts	133
4.1	Special functions for structural parts	134
4.2	Programming example for the pocket on a structural part	135
5	Engine and turbine components	139
5.1	Special functions for engine and turbine components	140
5.2	Example: Turbine blade	141
6	Complex free-form surfaces	145
6.1	Special functions for free-form surfaces	146
6.2	Example: Milling a manta ray	147
7	Difference between 3, 3+2 and 5 axes	151
7.1	5-axis simultaneous or 3+2-axes	152
7.2	Example workpiece milled by 3 to 3+2 axes	153
7.3	Example with 3 axes milled	155
7.4	Example with 3+1 axes milled	156
7.5	Example with 3+2 axes milled	157
7.6	Comparison of the three variants	158

8 Reference section	159
8.1 Overview of higher-order functions	160
8.1.1 Motion commands	160
8.1.2 Dynamic response	162
8.1.3 5-axis functionality	164
8.2 Further information/documentation	170
8.3 Index	172

Basic information

Content	Page
1.1 Introduction	10
1.2 SINUMERIK MDynamics milling technology packages	11
1.3 SINUMERIK Operate user interface	12
1.4 Requirements for milling	16
1.5 Linear axes, rotary axes and kinematics	17
1.6 Surface quality, velocity, accuracy	20
1.7 Additional software	22



1.1 Introduction

With mold-making, the main objectives are to achieve perfect surface quality, precision, and velocity 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, Siemens can offer an integrated solution for these requirements in the form of its SINUMERIK products.

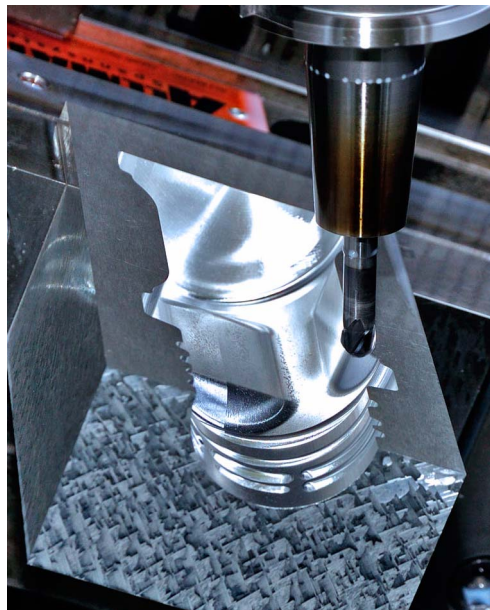
SINUMERIK controls have powerful, highly-developed functions which, if intelligently used, significantly simplify the complete operation involving programming and milling, and at the same time the production result is improved.

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 aspects cannot be covered in depth by this manual, additional information can be found in the appropriate SINUMERIK documentation and relevant literature, See "Further information/documentation" on page 170.

Range of milling



Blow mold for plastic bottles



Turbine impeller

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 via 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 identification
- 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
- Measure kinematics CYCLE996/9960

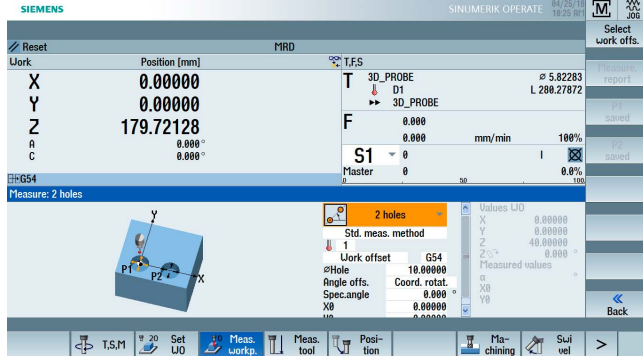
1.3 SINUMERIK Operate user interface

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 a 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.

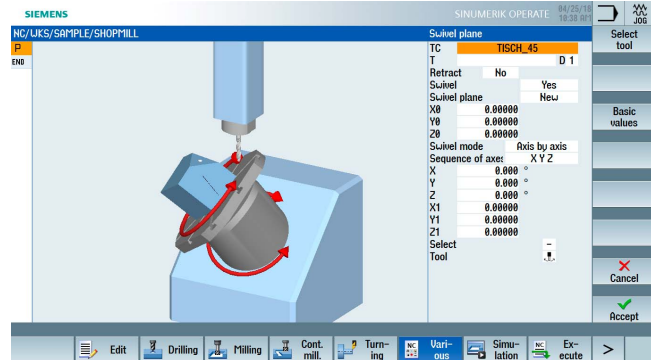
1.3.1 Operating and setup functions

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



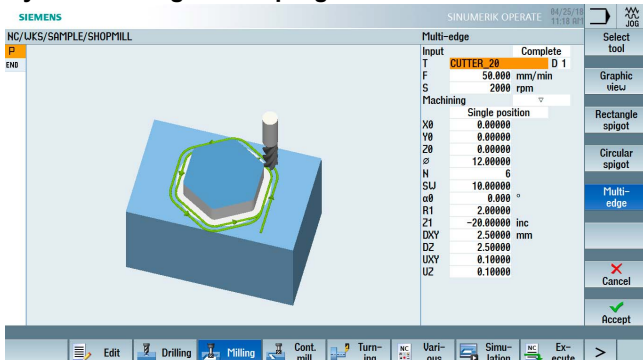
Support for many kinematics



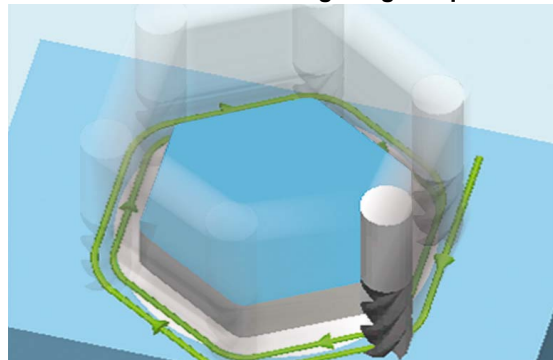
Animated elements clarify the parameters, for example during programming

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

Cycle multi-edge in the programGUIDE



Animated elements - Moving image sequence



1.3.2 SINUMERIK Operate - Multitouch

SmartOperation permits an intelligent CNC operation in SINUMERIK Operate using current technologies, such as multitouch and widescreen displays. The Display Manager permits you to see a lot of information at a glance.

Multitouch operation

SINUMERIK Operate provides additional operator functions for multitouch panels.

- Extended control elements via the function key block **1**
- Gesture operation (e.g. wiping, moving and zoom functions)

The selected functions can be activated directly via the function key block. These are, for example, undo changes or restore and activate the virtual keyboard **2** or of the calculator **3**.

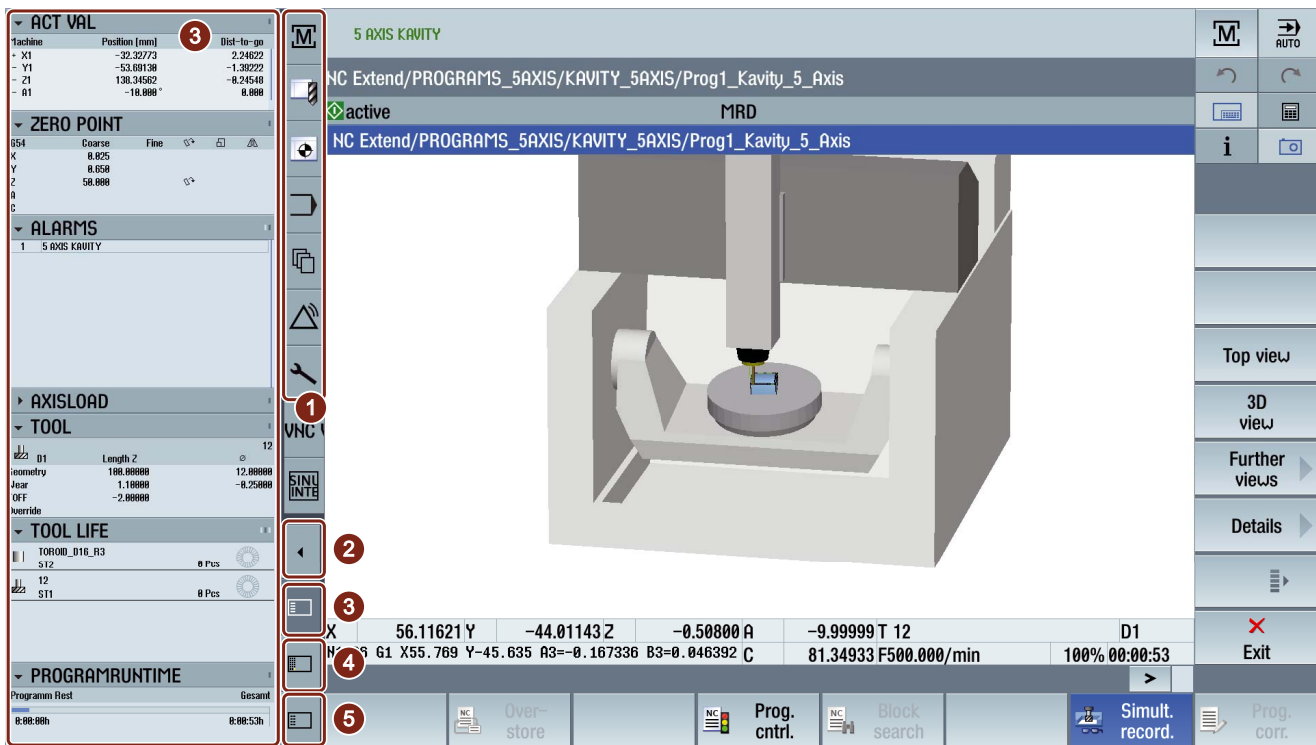
The screenshot displays the SINUMERIK Operate interface. At the top, the status bar shows 'SIEMENS' and 'SINUMERIK OPERATE' along with the date and time. The main display area shows a G-code program with lines N10 through N200. A 'Pocket calculator' window is open, showing a grid of mathematical functions and a display area with the input '8, 3, 15, 0, 1.25, 0, 1, 2, 11100, 11, 111)'. A virtual keyboard is also visible, with a red circle 2 highlighting it. A function key block is located on the right side of the screen, with a red circle 1 highlighting it. A red circle 3 highlights the calculator window. The bottom toolbar contains various operation modes: Edit, Drilling, Milling, Cont. mill., Turning, Vari-ous, Simulation, and Ex-ecute.

SINUMERIK Operate expansion using a sidescreen

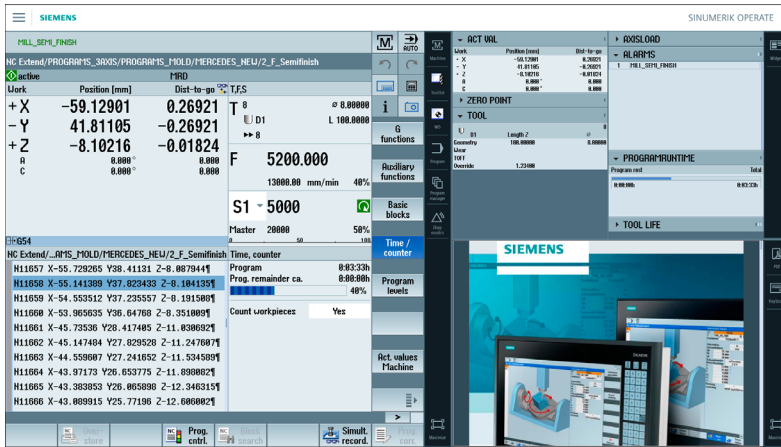
Widescreen format panels offer the option of using the additional area for the faster display of additional elements and operating. Using the navigation bar **1** the so-called sidescreen is shown and hidden **2**.

You can display the following elements via the navigation bar:

- Machine function direct selection **1** e.g. machine, tool list, zero offsets or program.
- Displaying (widgets) **3**
Standard widgets are for example, zeros, alarms, tool service life and program run time.
- Virtual keys (pages)
Activate the ABC keyboard **4** or of the machine control panel **5**



Display Manager (at least 22 inch Full HD SINUMERIK blackline plus panel)



With the Display Manager, the screen area is divided into several display areas.

In addition to SINUMERIK Operate, widgets, keyboards, a machine control panel and various applications are provided in the various areas.

1.4 Requirements for milling

Free-form surfaces
Mold-making



Design standards in all application areas are becoming increasingly demanding. Expectations in terms of ergonomics, the air drag coefficient (CW value) or simply aesthetic appeal are creating a need for more complex surface geometries to be achieved quicker and with greatest precision. The design primarily comes from CAD systems, the machining programs from CAM systems.

Engine and turbine components
e.g. impellers



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
Aviation 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:

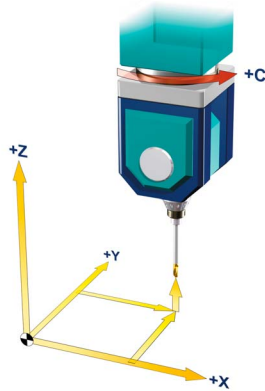
- Free-form surfaces (mold-making)
- Turbine and engine components (impellers, blisks)
- Structural parts (aviation industry)

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



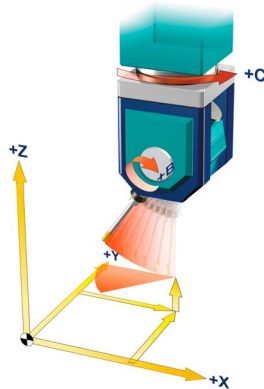
3-axis area

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.

Setting the tool at an angle



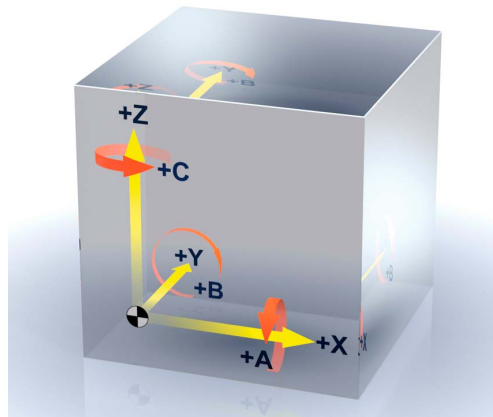
5(3+2)-axis area

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.

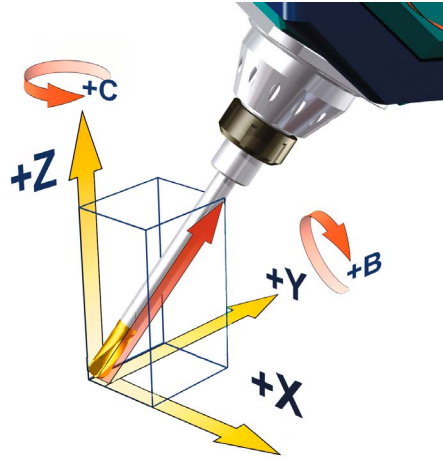
Linear and rotary axes



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

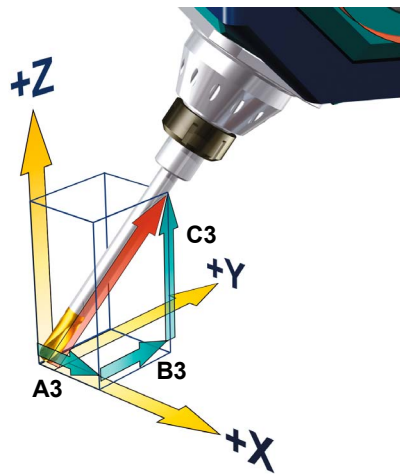
Programming rotary axes

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 defined in a unique manner.

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

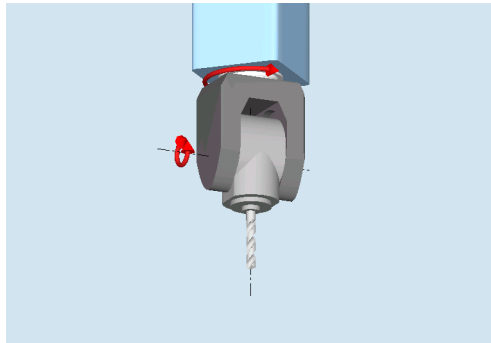
In addition to programming based on the direction vector and rotary axis positions, there are also other forms of angle programming. These include, for example, Euler or RPY angles. Further information regarding this topic can be found in Section "Programming the tool orientation" on page 86.

1.5.2 Kinematics of 5-axis machining centers

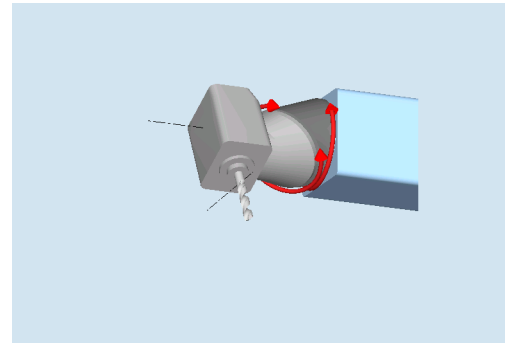
A 5-axis machine can control tool motion in five axes. These are the three linear axes (with which you will already be familiar) and two additional rotary axes. There are different kinematic solutions for the two rotary axes. We will present the most common of these schematically. With SINUMERIK controls, 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

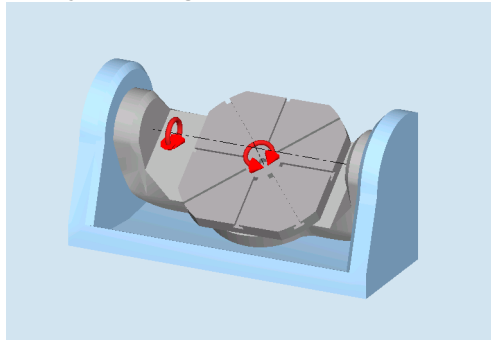


Head kinematics nutated *

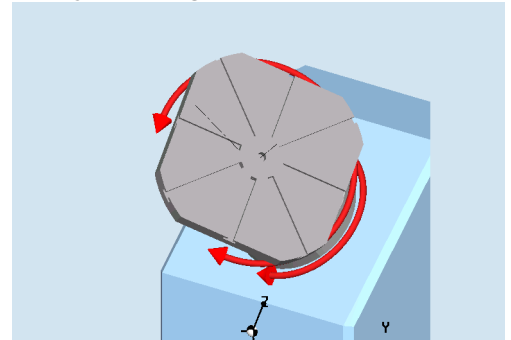


Two rotary axes in the table

Rotary/swiveling table

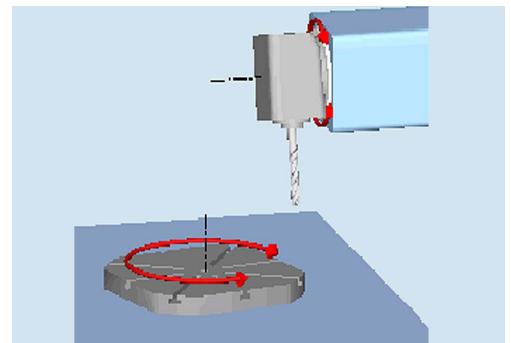
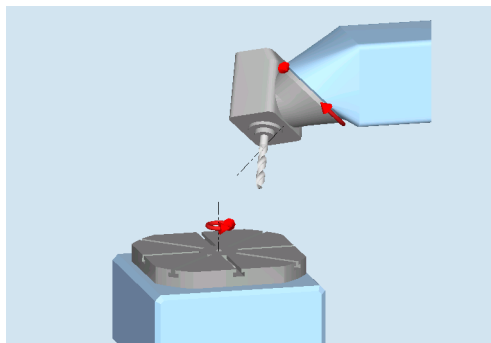


Rotary/swiveling table nutated *



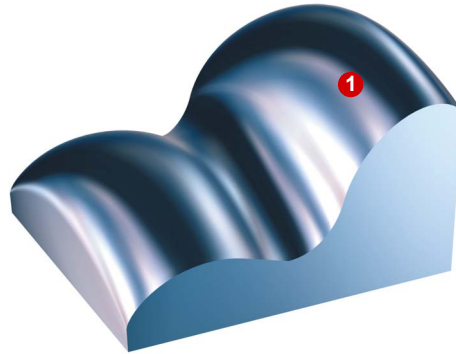
*: If the axis of rotation is not perpendicular to a linear axis, then this is known as a "nutated" axis.

Mixed kinematics



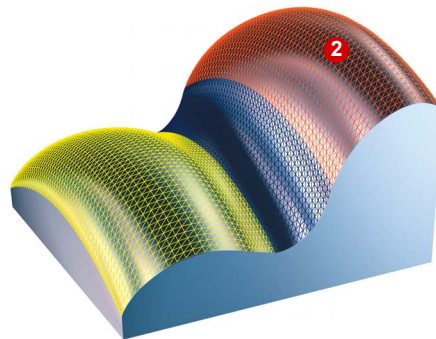
1.6 Surface quality, velocity, accuracy

Special attention must be paid to the **CAD -> CAM -> (postprocessor) -> 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 movements.



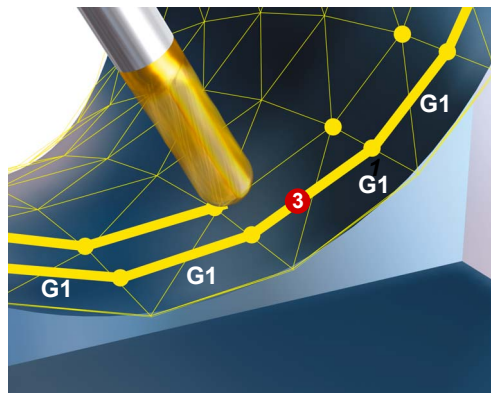
In CAD systems, surfaces **1** of higher orders are constructed (free-form surfaces).

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 **2** by a number of individual small planes.

This produces deviations from the original free-form surface.

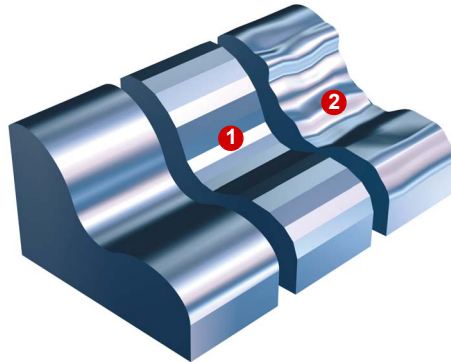


The CAM programmer overlays this polyhedron with tool paths. From these, the postprocessor generates NC blocks within the specified error tolerances. These usually comprise many short straight line elements, G1 X Y Z. **3**

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.

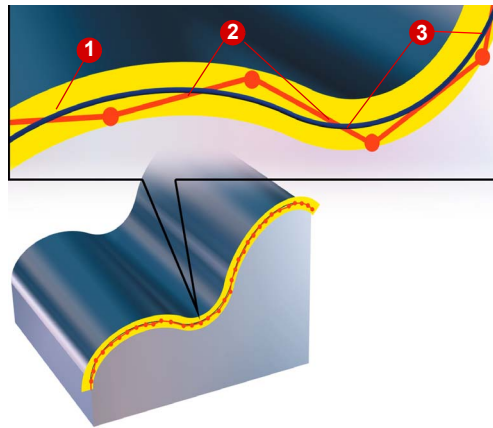
This can result in undesirable remachining.

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



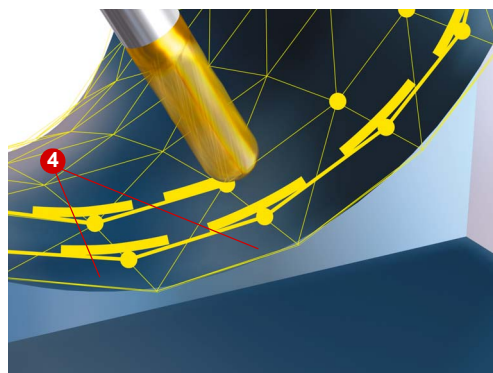
Compressor function (COMPCAD/COMPSURF)

At the block transitions, the interpolation leads to step changes in velocity in the machine axes, which in turn can cause resonance in the machine elements and can ultimately be detected as a beveled pattern **1** or as the effects of vibrations **2** on the workpiece surface.



In accordance with the specified tolerance band, the compressor **1** takes a sequence of NC commands, **2** combines them and compresses them into a spline **3**, which can be directly executed by the control.

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 velocities, thereby increasing productivity.



Programmable smoothing (G645)

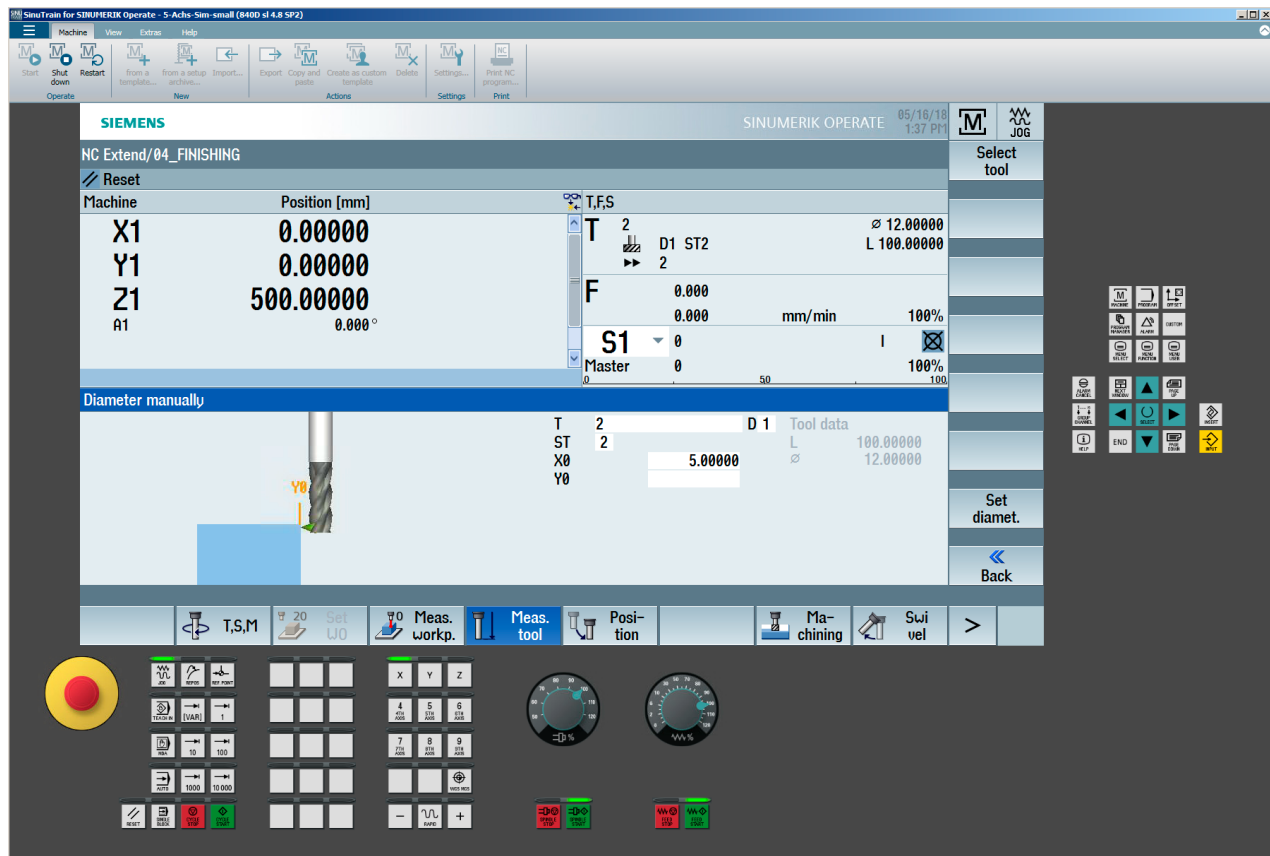
Discontinuous block transitions can be blended using the smoothing function to create steady characteristics. This involves inserting geometrical elements **4** at the corners (block transitions).

The tolerance of these geometrical elements can be adjusted (see also 3.6).

1.7 Additional software

1.7.1 SinuTrain

SinuTrain provides for SINUMERIK controls an intuitive PC-based solution that consists of control-identical software. SinuTrain allows practical offline programming at a PC; these NC programs can be transferred directly to the CNC. Using SINUMERIK Operate and the original SINUMERIK CNC kernel, all operating and programming operations can be used and CNC programs executed, without any restrictions.



Advantages at a glance

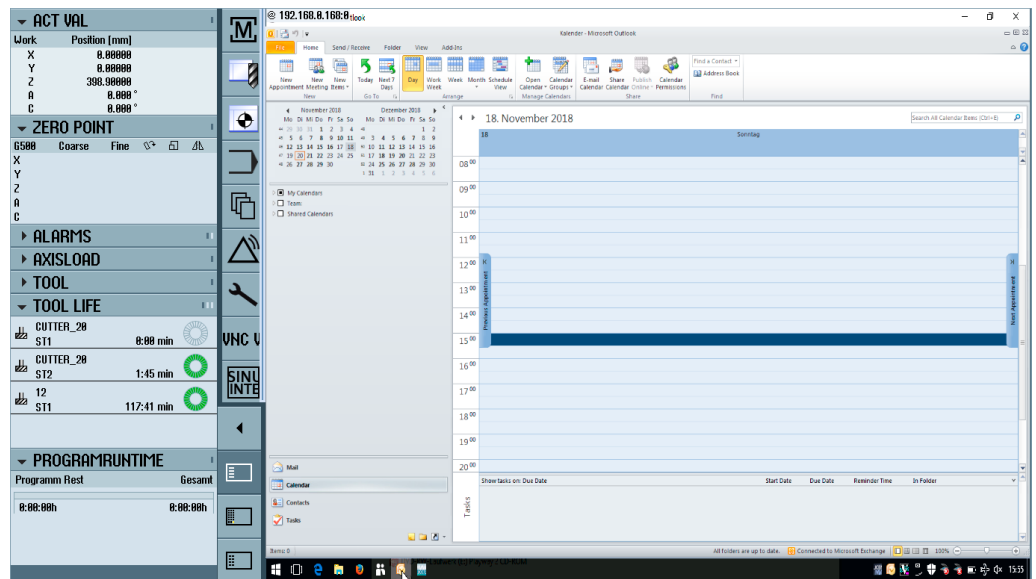
- Almost 100-percent offline verification/evaluation of the NC programs thanks to the original CNC kernel.
- Use as optimum programming station by adaptation to the real machine (e. g. by the machine manufacturer).
- Perfectly tailored packages for training and production planning.
- Operating, programming and simulating exactly as for the real SINUMERIK.

1.7.2 VNC Viewer - remote control on an external PC

In SINUMERIK Operate, a VNC (Virtual Network Computing) Viewer can be integrated in the operating area to access a remote PC located in the network of the control. This access allows applications on the PC to be operated directly from the control. For example, missing information can be fetched from a CAD/CAM system or from the production planning, or process participants informed about changed production parameters.

After a one-off configuration of the connection, a switch can be made to the remote connected PC by pressing a key.

Access to an application of the external computer

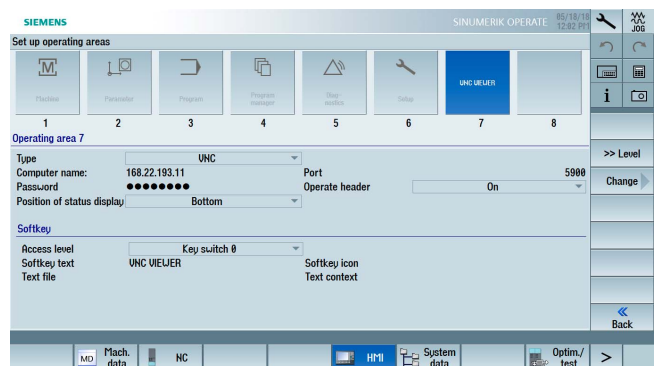


VNC area configuration on the user interface

Remote access to a PC in the network is possible via the interfaces, e.g. X130.

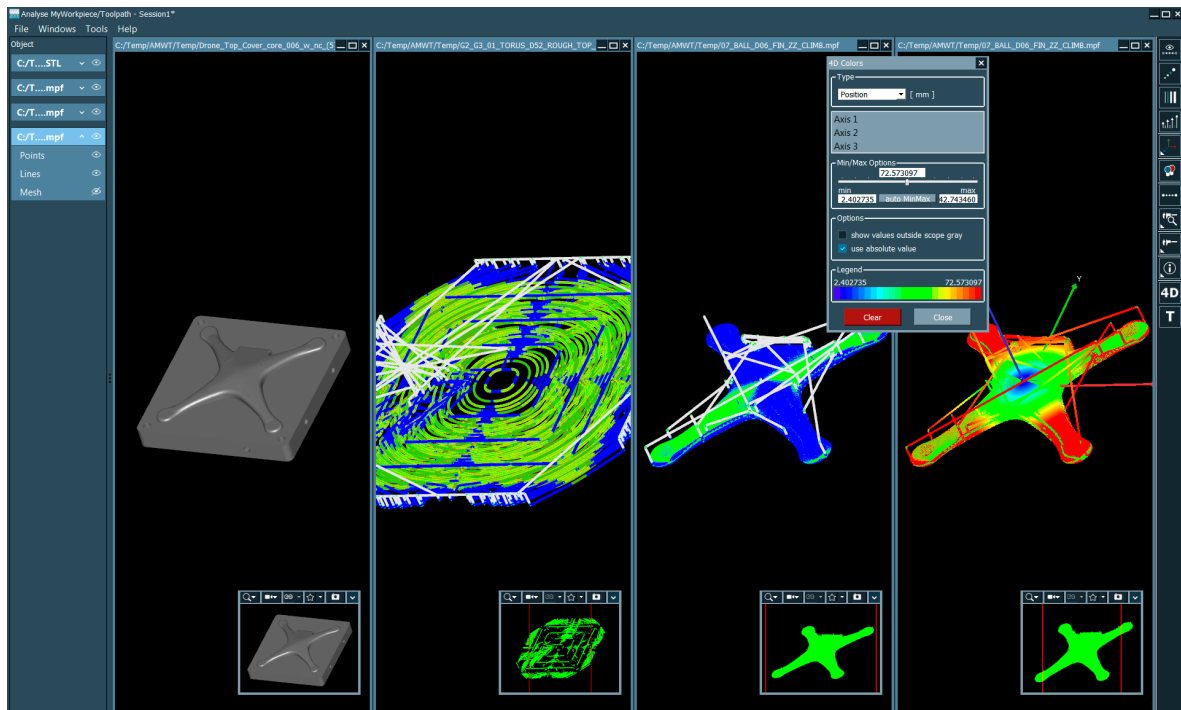
The configuration is performed in SINUMERIK Operate.

Further information can be found in the manuals with the keyword VNC or obtained from the machine manufacturer.



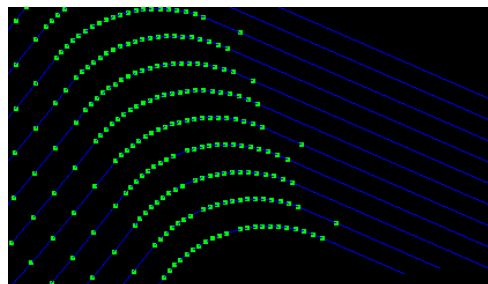
1.7.3 Analyze MyWorkpiece /Toolpath additional software

After the processing, the Analyze MyWorkpiece /Toolpath 3D visualization tool allows the completed geometry from the CAD system, the NC program as well as the recording of the tool path to be represented concurrently in three dimensions and compared with each other. Filters and color assignments highlight any potentially problematic areas. They show immediately whether the errors are in the NC program or were caused by the machine kinematics or the clamping.

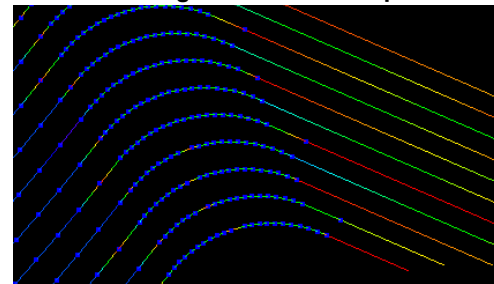


Optimizations and compensations are possible even before production of a workpiece. For example, any discrepancies in the NC program caused by an irregular distribution of points, differences for the travel velocities of neighboring milling paths or tool paths not appropriate for fine machining are shown.

Missing points in the curvature path

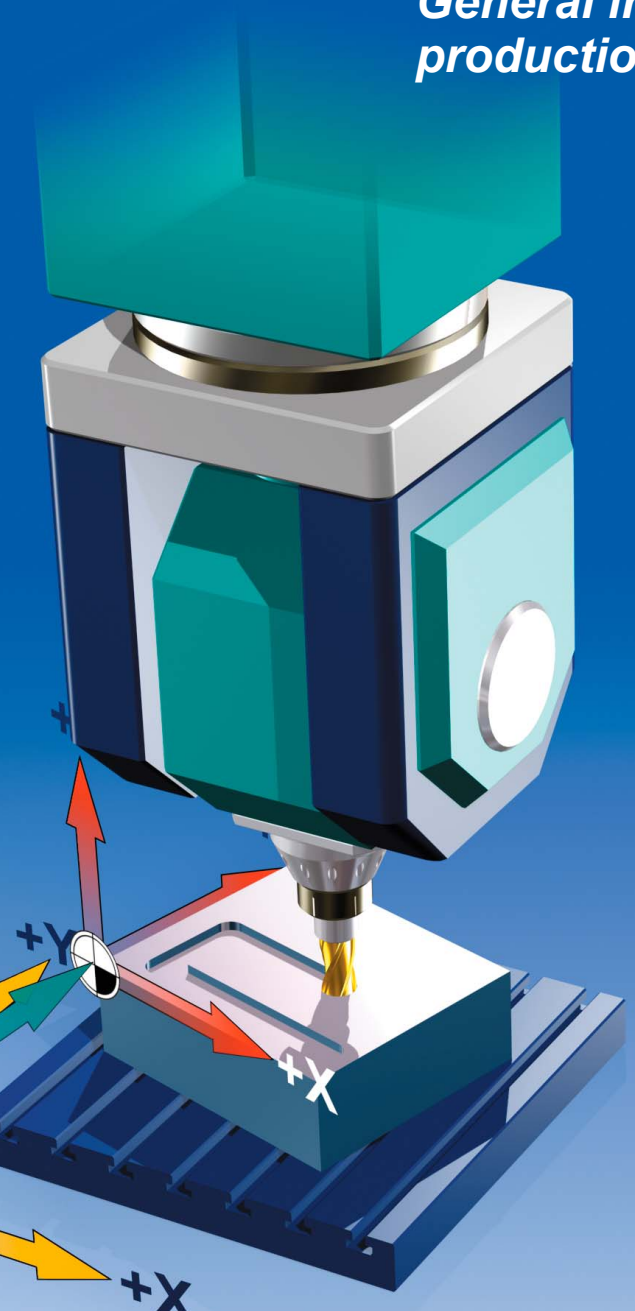


4D evaluation: Color switching in the curvature path



The capability to measure points with micrometer accuracy makes it possible to evaluate whether errors are surface-relevant. This saves valuable machine time for the production of faulty parts because corrective work is reduced to a minimum.

General information on workpiece production



Content	Page
2.1 Process chain for producing 3- to 5-axis workpieces	26
2.2 CAM systems	27
2.3 Program structure for milling	29
2.4 Programs, data transfer and editor	33
2.5 JOG mode, zeros, tools	38
2.6 Fundamentals, measuring in JOG and AUTOMATIC	41
2.7 Measuring workpiece/tool in JOG	46
2.8 Measure workpiece/tool in AUTOMATIC - process measuring	52
2.9 Checking and measuring machine/spindle	56
2.10 Workpiece visualization	60
2.11 Executing programs (AUTOMATIC)	64
2.12 Collision avoidance	68

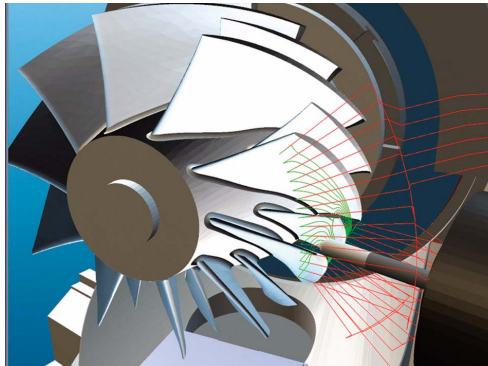
2



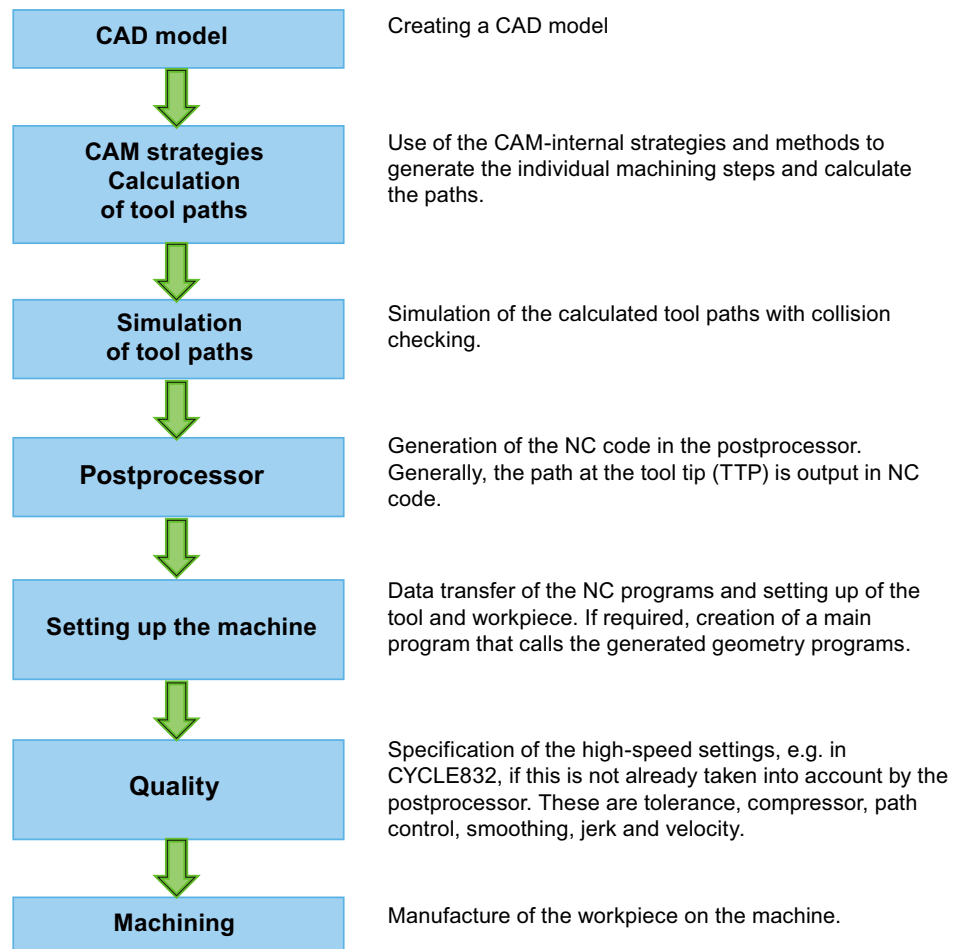
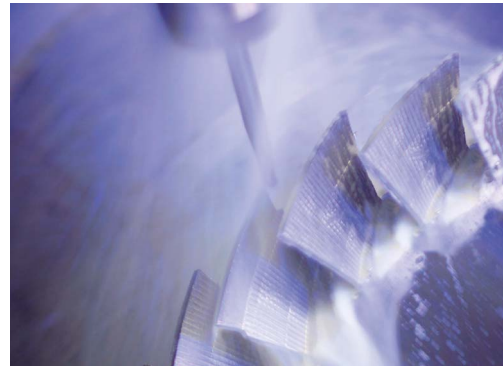
2.1 Process chain for producing 3- to 5-axis workpieces

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

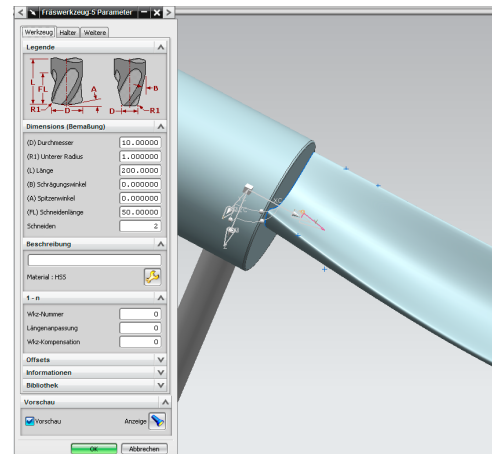


2.2 CAM systems

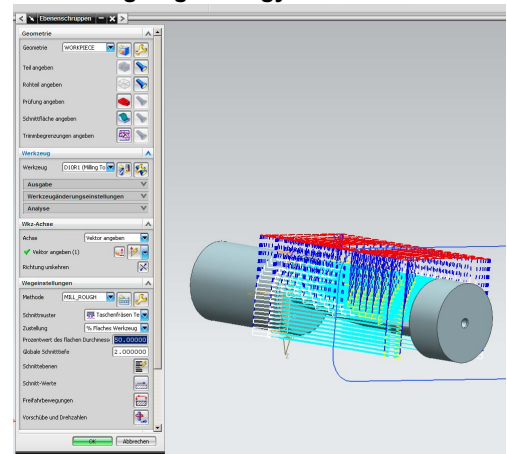
Within the context of the process chain, it is the CAM system that is responsible for the key task of generating the NC programs. The quality of this data plays a decisive role in determining the results of workpiece production.

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

Fetch 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 a clamping device. Define geometries such as a zero.

► Specify 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.

► Specify 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 with the postprocessor**

The postprocessor converts the sequences into NC programs taking into account the control-specific syntax and the control's special 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.

2.2.1 Tolerance and accuracy

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 and on the workpiece. The tolerance for roughing strategies is greater than for 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.

Tolerance guide values

Technology	Recommended values
Roughing	Tolerance 0.05 - 0.1 mm - with 5-axes OTOL = 0.3
Semi-finishing	Tolerance 0.01 - 0.05 mm - with 5 axes OTOL = 0.15
Finishing	Tolerance 0.002 mm - max. point separation 0.3 - 0.5 mm with 5-axes OTOL = 0.01 - 0.1

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.

Accuracy guide values

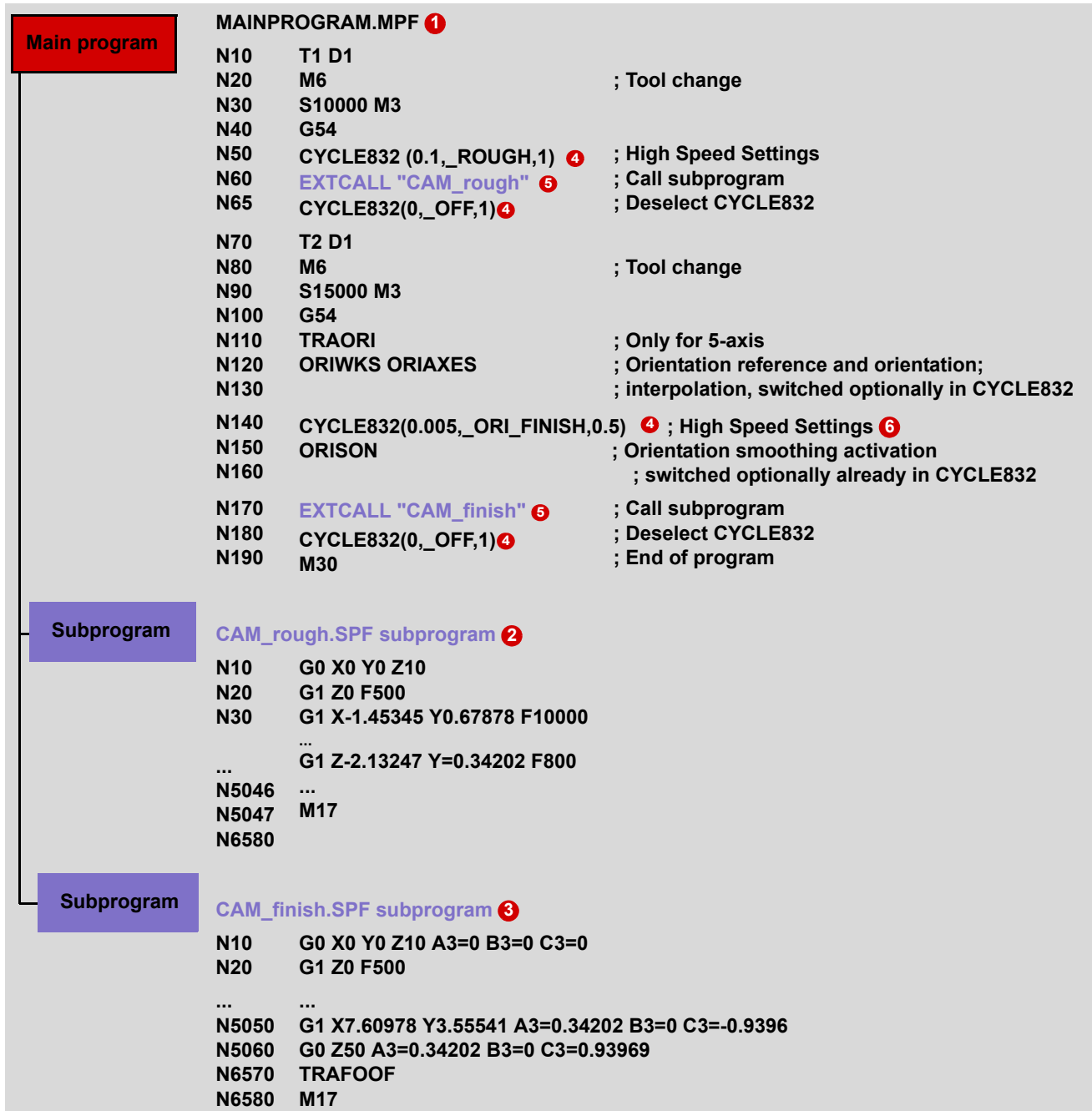
Axes interpolation	Recommended values
Linear axes (X Y Z)	for 3-axis programs, at least 4 decimal places
Rotary axis positions (A B C)	for 5-axis programs, at least 6 decimal places in linear and rotary axes for optimum surface quality
Direction vector (A3 B3 C3)	4 decimal places in linear axes and at least 7 decimal places for 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 purposes, a main program is generated **1** that includes all technology data. The main program calls one or more subprograms **2**, **3** that contain only the pure geometric data of the workpiece. The tool change defines how the content is divided into subprograms.



Main program: The main program includes the two key functions for milling, CYCLE832 ④ and subprogram call (e.g. with EXTCALL) ⑤.

CYCLE832 ④ : 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 accuracy. 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 102.)

TIP

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

EXTCALL ⑤ : 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 35.)

ORISON (only for Advanced Surface COMPCAD) ⑥ : 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: The subprogram contains 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 ⑦ milling. These are designated A3, B3, and C3.

CYCLE832 in the subprogram

For machining purposes, a main program is generated **1**, which only serves as call program for the individual subprograms. The main program calls one or more subprograms **2**, **3** 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 **4** to **7** are identical (see page 30).

Main program	MAINPROGRAM.MPF 1		
	N10	CYCLE800(4,"TABLE",200000,57,0,0,0,0,0,0,0,0,1,,1)	; Swivel basic position
	N20	CYCLE800()	; Deselect swivel
	N30	WORKPIECE(,,,"CYLINDER",0,0,-32,-80,46)	; Define blank
	N40	EXTCALL "CF_CARD:/PROG/CAM_ROUGH.SPF" 5	; Call subprogram
	N50	STOPRE	; Stop preprocessing
	N60	EXTCALL "CF_CARD:/PROG/CAM_FINISH.SPF" 5	; Call subprogram
	N65	STOPRE	; Deselect CYCLE832
	N70	...	; Further subprograms
	N200	M30	; End of program
Subprogram	CAM_ROUGH.SPF subprogram 2		
	N10	T1 D1	; Tool call
	N20	M6	; Tool change
	N30	S10000 M3	; Technological data
	N40	G54	; Zero offset
	N50	CYCLE832 (0.1,_ROUGH,1) 4	; CYCLE832 roughing
	N60	G0 X0 Y0 Z10	; 3-axis
	N70	G1 Z0 F500	
	N80	G1 X-1.45345 Y0.67878 F10000	
	N90	...	
	N6000	CYCLE832(0,_OFF,1)	; Deselect CYCLE832
	N6010	M17	; End of subprogram
	Subprogram	CAM_FINISH.SPF subprogram 3	
N10		T2 D1	; Tool call
N20		M6	; tool replacement
N30		S15000 M3	; technological data
N40		TRAORI	; TRAORI for 5-axis On
N50		ORIWKS	; Orientation reference
N60		ORIXES	; Orientation interpolation, switched optionally already in CYCLE832
N70		G54	; Zero offset
N80		CYCLE832(0.005,_ORI_FINISH,0.5) 4	; CYCLE832 5-axis finishing
N90		ORISON 6	; Orientation smoothing activation
N95			; switched optionally already in CYCLE832
N100		G0 X0 Y0 Z10 A3=0 B3=0 C3=0	
N110		G1 Z0 F500	
N120		G1 X7.60978 Y3.55541 A3=0.34202 B3=0 C3=-0.9396	
N130		G0 Z50 A3=0.34202 B3=0 C3=0.93969 7	
N140		...	
N6000		TRAFOOF	; TRAORI Off
N6010		CYCLE832(0,_OFF,1)	; Deselect CYCLE832
N6020		M17	; End of subprogram

CYCLE832 guidelines When CYCLE832 is used, the sequence of the NC commands in the program must be observed.

Whereby:

- CYLCE832 must be called **immediately** before the mold-making program.
- After selection of CYCLE832, only motion commands (G0, G1, G2, G3) should be programmed.
- Manufacturer-specific subprograms, e.g. for tool change or swivel, can overwrite settings of the CYCLE832 and must be programmed **outside** the CYCLE832 selection and deselection.

Programming overview	Description
T1 D1 M6	Tool change
...	
CYCLE800 (4, "HEAD", 200000, 57, 0, 0, 0, 0, 0, 0, 0, 0, 0, 1, , 1)	Settings, e.g. swivel
...	
CYCLE832 (0.1, _ROUGH, 1)	Select CYCLE832
...G1 X-1.45345 Y0.67878 F10000...	Mold-making program
CYCLE832 (0, _OFF, 1)	Deselect CYCLE832
...	
CYCLE800 ()	Deselect CYCLE800
...	

Example, NC commands in the 5-axis program sequence

```

...
CYCLE800 ()
TRAFOOF
T="CUTTER_10"
M6
S15061 M3
CYCLE800 (0, "TABLE", 100000, 27, 0, 0, 0, 288.079, 0, -10.196, 0, 0, 0, -1, 0, 1)
G00 X-13.532 Y-24.856 M03
TRAORI
ORIWKS ORIAXES
G54
TOFFL=0
TOFFR=0
CYCLE800 ()
CYCLE832 (0.05, _TOP_SURFACE_SMOOTH_ON+_ORI_SEMIFIN, 1)
...

```


2.4 Programs, data transfer and editor

2.4.1 Program manager

The program manager offers 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.

Name	Type	Length	Date	Time
Part programs	DIR		02/12/18	8:19:27 PM
Subprograms	DIR		02/12/18	8:19:28 PM
Workpieces	DIR		04/20/18	11:31:40 AM
EXAMPLE1	WPD		02/12/18	8:19:27 PM
EXAMPLE2	WPD		02/12/18	8:19:27 PM
EXAMPLE3	WPD		02/12/18	8:19:27 PM
EXAMPLE4	WPD		02/12/18	8:19:27 PM
EXAMPLE5	WPD		02/12/18	8:19:27 PM
MOLD_SEMIFINISH	WPD		02/12/18	8:19:27 PM
TEMP	WPD		04/20/18	11:31:40 AM

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 interface, hard drive, etc.

Procedure when calling the geometry program with 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 on the HMI user memory (NC)
CALL "SAMPLE"
 - Subprogram is in the directory on the CompactFlash card
EXTCALL "CF_CARD:/PROGRAMS/SAMPLE.SPF"
 - Subprogram is on a local drive
EXTCALL "LOCAL_DRIVE:/PROGRAMS/SAMPLE.SPF"
 - Subprogram is on a USB stick
EXTCALL "USB:/MOLD_DIE/CAM_ROUGH.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"

Processing of 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.

- 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 velocity (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.

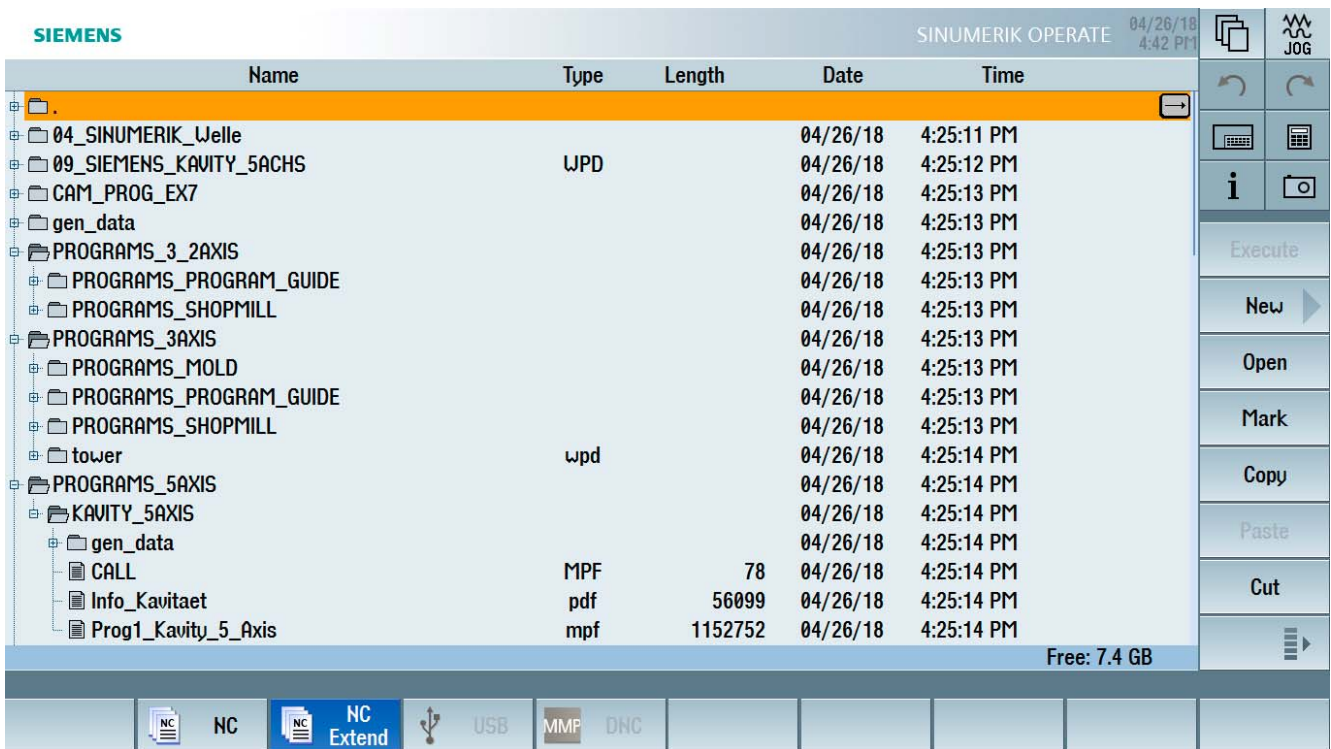


Direct execution from a USB FlashDrive is not recommended. Disconnecting during operation will stop machining and, under certain circumstances, cause damage to the workpiece.

2.4.3 EES - Execution from External Storage (NC Extend)

Using function **EES - Execution from External Storage** (as of 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 compensation for NC stop is possible.



The EES function makes EXTCALL calls superfluous. If required, existing EXTCALL calls can be converted.

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

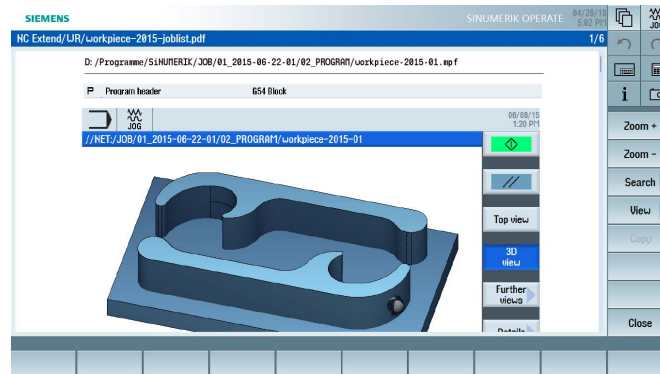
2.4.4 Graphic and file formats in the program manager

In addition to the standard NC formats (*.mpf, *.spf), the program manager also directly displays additional graphic and file formats at the control. 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 - a paperless production environment.

TIP

An existing DXF file can be read into the contour editor as contour (DXF reader option).

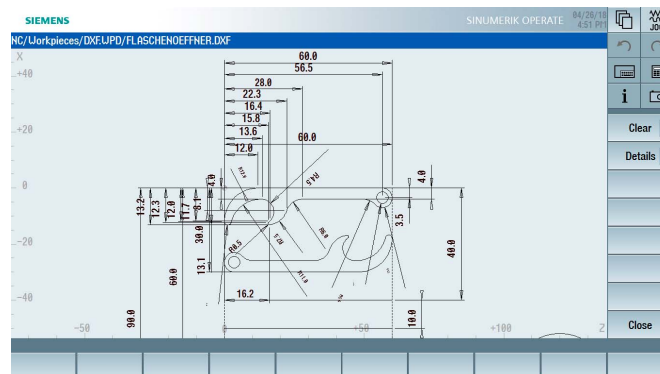
Syntax documents TXT, PDF



Pixel files BMP, PNG, GIF, JPG



Vector graphic DXF



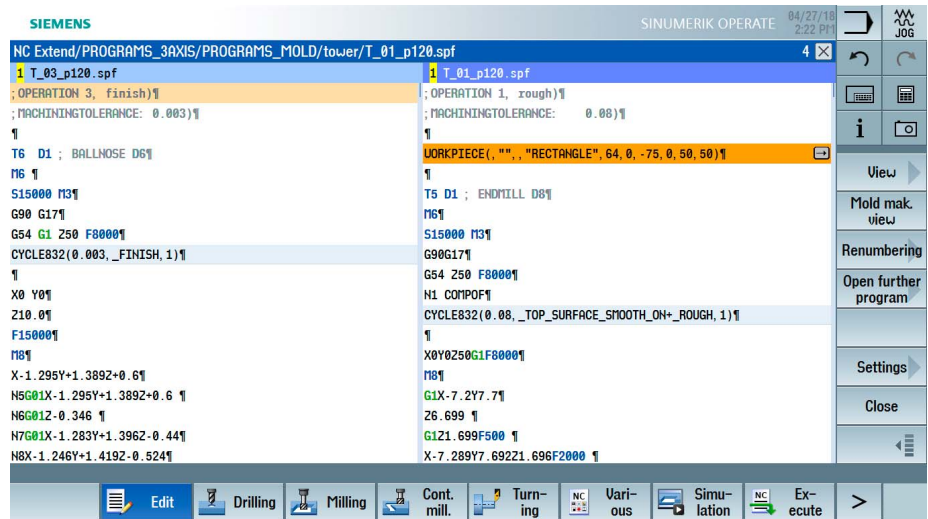
2.4.5 Program editor - useful functions

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 cut out, copy, paste and renumber for effectively programming and displaying large programs.

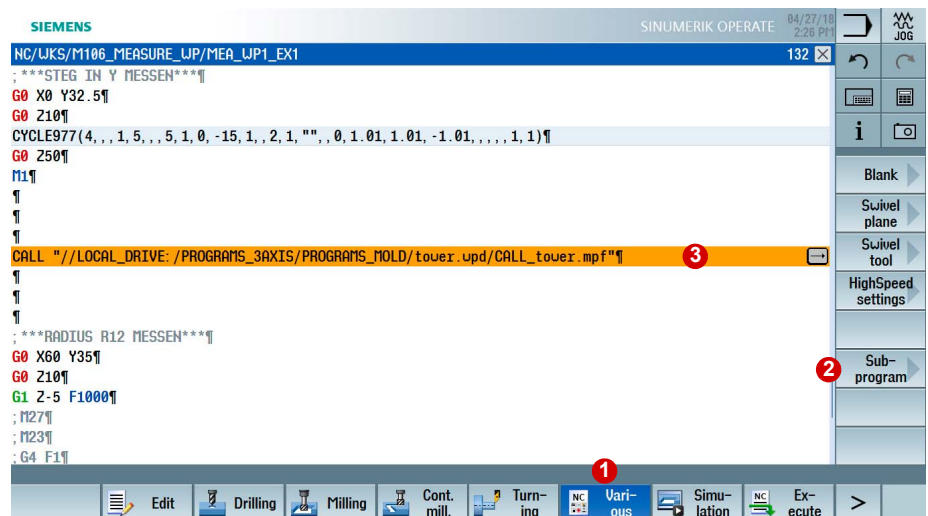
Program editor supplementary functions

- 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 the **Various** softkey

Dual editor and syntax highlighting



Subprogram call with transfer of the complete path



2.5 JOG mode, zeros, tools

2.5.1 Zero 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 the program execution and can be a combination of different offsets.

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

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

SIEMENS		SINUMERIK OPERATE							04/27/18 2:49 PM			JOG
Work offset - Overview [mm]												
			X	Y	Z	A	C	SP1				
Act.value Machine			0.000	0.000	500.000	0.000	0.000	88.095				
DRF			0.000	0.000	0.000	0.000	0.000	0.000	Active			
Rotary table ref.			0.000	0.000	0.000	0.000	0.000	0.000				
Basic reference			0.000	0.000	0.000	0.000	0.000	0.000				
Total basic WJ0			0.000	0.000	0.000	0.000	0.000	0.000	Overview			
G54			85.120	125.230	803.130	0.000	0.000	0.000				
Tool reference			0.000	0.000	0.000	0.000	0.000	0.000				
Workpiece ref.			0.000	0.000	0.000	0.000	0.000	0.000	Base			
Programmed WJ0			0.000	0.000	0.000	0.000	0.000	0.000				
Cycle reference			0.000	0.000	0.000	0.000	0.000	0.000				
Total WJ0			85.120	125.230	803.130	0.000	0.000	0.000	G54 ...			
Tool: FACEMILL 63			0.000	0.000	120.000				G519			
TOFF			0.000	0.000	0.000							
Work actual value			-85.120	-125.230	-423.130	0.000	0.000	138.495				
											Details	
	Tool list		Tool wear		Magazine		Work offset		User variable		Setting data	

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** displays. All of the relevant tool data is displayed in the tool list. Using softkey **4** you can create, delete, load or unload tools and directly load them into the program. With the tool catalog **5**, which is displayed when new tools are created, you can quickly create tools with the aid of pre-defined tool types.

For the milling technology, schematic diagrams of the tools are displayed above the tool list. The display can be configured. You can switch this on and off via the >>> **Settings** expanded softkey bar (not for multitasking machines).

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. For example, you can disable locations or assign tools to locations.

Loc.	Type	Tool name	ST	D	Length	∅				
9		CENTERDRILL 12	1	1	120.000	12.000	90.0	↻	✓	
10		DRILL 8.5	1	1	120.000	8.500	118.0	↻	✓	
11		DRILL 10	1	1	120.000	10.000	118.0	↻	✓	
12		PREDRILL 30	1	1	120.000	30.000	180.0	↻	✓	
13		DRILL_Tool	1	1	110.000	25.000		↻	✓	
14		THREAD CUTTER	1	1	110.000	20.000		1 ↻	✓	
15		THREADCUTTER M10	1	1	130.000	10.000	1.500	↻	✓	
Magazine / location number										
17		3D_TASTER	1	1	100.000	8.000		2 ↻	✗	
18										

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	

2.5.3 Manual retraction - return

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 perpendicular.

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

The screenshot shows the Siemens SINUMERIK OPERATE interface. At the top, it displays 'SIEMENS' and 'SINUMERIK OPERATE' with a date and time of 04/27/18 3:45 PM. The main area is divided into several sections:

- Work:** Shows the current position in millimeters: X: -65.000, Y: -40.000, Z: 1.000. The A and C axes are at 0.000°.
- T,F,S:** Shows tool parameters: T: THREADCUTTER M10 (D1), F: 0.000 mm/min, S1: 0.
- Retract tool:** A 3D model of a tool is shown with a green arrow pointing up along the Z-axis, indicating the retraction direction.

The bottom bar contains a 'Retract' button with a red '2' above it, and a 'Back' button with a red '1' above it.

Procedure

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

2.6 Fundamentals, measuring in JOG and AUTOMATIC

Measuring in JOG

When **measuring in JOG** (setup), the machine is prepared for 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 compensations of the coordinate system that have been determined.

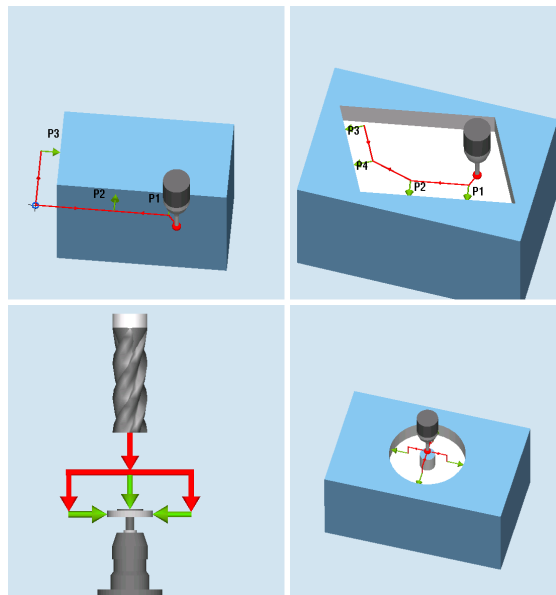
Measuring in AUTOMATIC (process measuring)

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.

Measuring cycles/functions for all measuring tasks

Examples of measuring cycles/functions



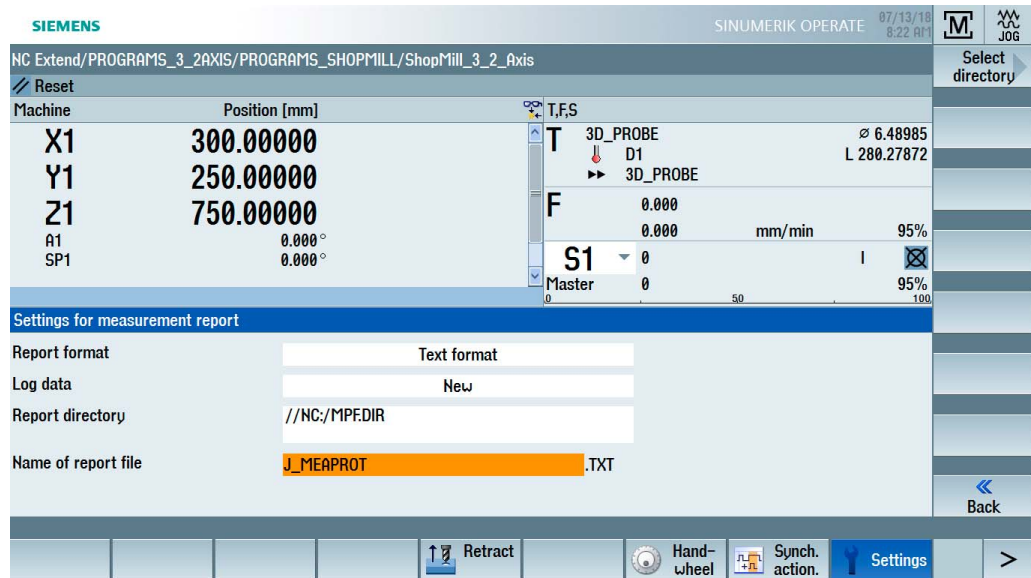
The SINUMERIK features an extensive pool of practical measuring cycles/functions for measuring tasks in JOG and AUTOMATIC modes. These measuring cycles/functions enable you to measure workpieces and tools using a graphically supported process.

The measuring tasks are performed with switching probes and dynamometers or laser measuring systems.

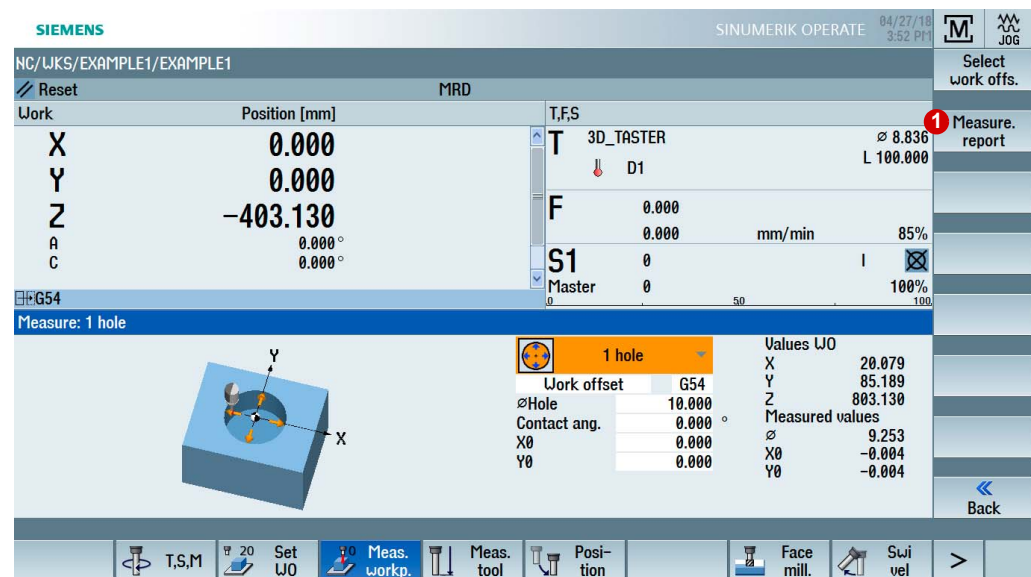
When measuring in JOG, the measuring functions can also be used with edge probes or tools with known dimensions. During the measurement, the measuring points are manually approached and the current position is saved in the measuring function via a softkey.

2.6.1 Measurement logs in JOG

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



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



The following data is determined and logged:

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

SIEMENS SINUMERIK OPERATE 04/27/18 3:54 PM

NC/WKS/TEMP/J_MEAPROT.TXT 2

Date : 2018-04-27 Time: 15:52:58

Workpiece No: [blank]

Results measure: 1 Hole ②

Measuring plane: G17

Probe no. : 1

Correction into: Work offset, Coarse

G54

	Coarse [mm]	Fine [mm]	Rot [deg]	
X	20.079	0.000	0.000	
Y	85.189	0.000	0.000	
Z	803.130	0.000	0.000	

Results:	Setpoint	Measured	Difference	
X0	0.000	-0.004	-0.004	mm
Y0	0.000	-0.004	-0.004	mm

②

Edit Drilling Milling Cont. mill. Vari-ous Simulation Ex-ecute >

Select tool Build group Search Mark Copy Paste Cut

Example of a log file for workpiece measurement using measuring variant 1 hole. ② .

2.6.2 Measurement logs in AUTOMATIC

As of SW 4.7 SP2, standard logs 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 log in JOG, additional data is captured:

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

Procedure

- ▶ Select the display and display mode of the measurement result screen **1**.
- ▶ Switch on logging **2** and select the standard log **3**.
- ▶ Select the format (TXT, CSV) **4**.
- ▶ Configure whether the log should be recreated after each measurement or the measurements should be saved consecutively in the same file. **5**.
- ▶ Select the storage 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, you must program CYCLE160 so that the user data is written to the log file.

G code program example with call and a measurement function

```

...
N50 CYCLE150(10,1001,"MEAS_PROTOCOL.TXT")
N51 S_PROTTEXT[0]=REP("") ; field, delete all data
N52 S_PROTTEXT[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 [mm]		
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:
*****
**
-----
--
: 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
-----
--
    
```

The standard logs can be saved as CSV or text file.



You can use CYCLE150, also without measuring functions, for example, to display texts for the operator on the complete screen.

2.7 Measuring workpiece/tool in JOG

2.7.1 Workpiece measuring in JOG

Once the machine has been powered up and the reference point approached, the axis positions relate to the machine coordinate system. The zero 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** permit 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 compensation is activated.

Practical measuring functions are provided to facilitate measurements

Measuring functions in JOG

The screenshot shows the SINUMERIK JOG control interface. The main screen displays machine status (SINUMERIK OPERATE, 04/27/18, 4:00 PM) and a list of measuring functions with corresponding icons and descriptions:

- Calibrate probe → Calibrate probe
- Measure edge → Measure edge
- Aligning an edge → Aligning an edge
- Measure corner → Measure corner
- Measure pocket/hole → Measure pocket/hole
- Measure spigot/hole → Measure spigot/hole
- Align plane → Align plane
- Back → Back (exit measuring in JOG)

A secondary window titled "Align edge" is open, showing a list of options:

- Align edge
- Spacing 2 edges
- Rectangular corner
- Any corner
- Rectang. pocket
 - 1 hole
 - 2 holes
 - 3 holes
 - 4 holes
- Rectang.spigot
 - 1 circ. spigot
 - 2 circ. spigots
 - 3 circ. spigots
 - 4 circ. spigots
- Align plane

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 should be setup with the **Right-angled corner** measuring function. The compensation should be made in zero offset G54.

- ▶ Select the **Right-angled corner** softkey. You can also select the cycle from the selection list **1**.
- ▶ Traverse the probe to measuring point **P1**.
- ▶ Select the zero offset, e.g. **G54** **2**.
- ▶ When you press "NC Start", the associated measuring points P1, P2, and P3 **3** are automatically approached starting from the manually assumed preliminary position. This means that the probe approaches the workpiece, is triggered and then returns to the start position.
- ▶ After each measuring process, the measured values can be saved in a measurement log **5** the measurement log can be configured in advance.
- ▶ Once all measuring points have been approached, press **Set zero offset** **4**.

The screenshot displays the SINUMERIK OPERATE interface with the following data:

Machine	Position [mm]
X1	-0.02778
Y1	-0.02868
Z1	500.00000
A1	0.000°
SP1	0.000°

Parameter	Value
Work offset	G54
Corner	Outs.corner
Pos. 1	
X0	0.00000
Y0	0.00000

Measured values	Values W0
α0	0.00000
X0	0.00000
X0	40.00000
Y0	0.00000

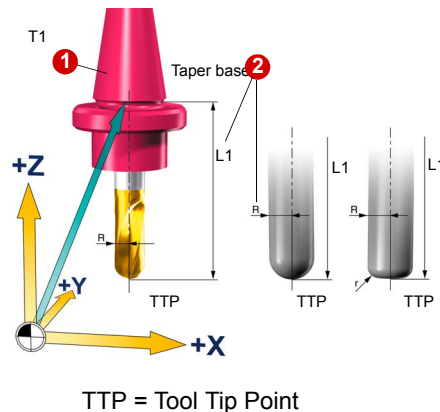
- ▶ The corner point now corresponds to the setpoint position. The calculated offset is stored in the zero offset.

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 compensation data in the tool list. When the tool is called, the control considers the tool compensation data.

You can determine the tool compensation 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



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.

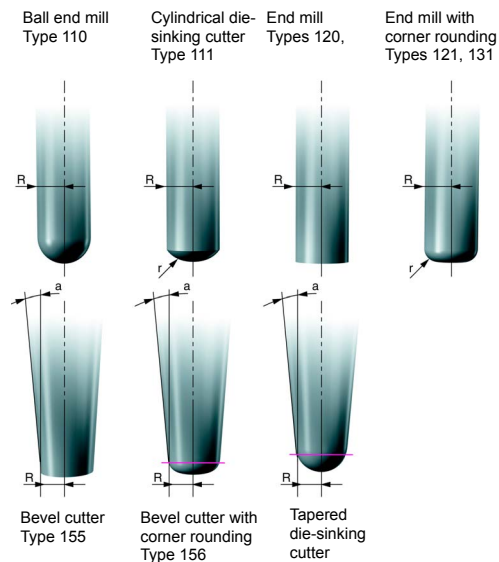
New tool - favorites		
Type	Identifier	Tool position
120	- End mill	
140	- Facing tool	
200	- Twist drill	
220	- Center drill	
240	- Tap	
710	- 3D probe	
711	- Edge finder	
110	- Ball nose end mill	
111	- Conical ball end	
121	- End mill corner rounding	
155	- Bevelled cutter	
156	- Bevelled cutter corner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

The tool magazine is equipped as usual. The tool numbers (T1...) or the plain text name (MILLING_12...) of the tools ① are entered into the tool table and the tools are assigned a tool compensation D ②, consisting of 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 compensation 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, via the TOFFL command in the NC program.



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

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

Programmable tool compensation (TOFFL, TOFFR)

The TOFFL/TOFF or TOFFR commands can be used to change the effective tool length (TOFFL/TOFF) or the effective radius (TOFFR) in the NC program without changing the tool compensations. 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: Measure tool in JOG

Function

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

- Calibrate a dynamometer
- Determine the tool length or the radius of milling tools or the tool length of drills, and enter this data into the tool compensation memory.

Requirements for using measuring functions

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

Procedure

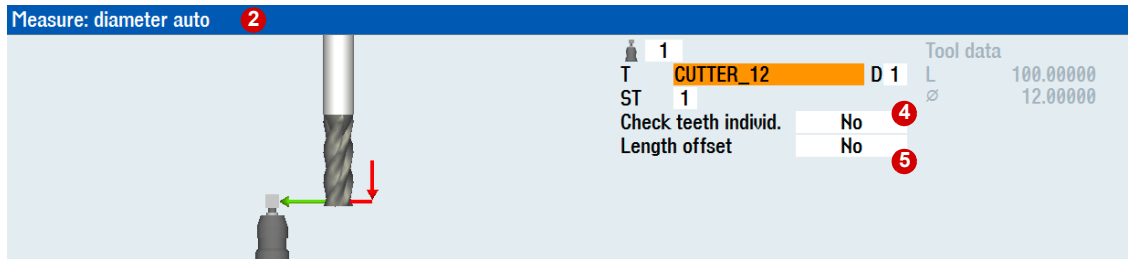
- ▶ In JOG mode, select the **Measure tool** softkey **1**. In the horizontal softkey bar, select whether you want to measure automatically or manually.
- ▶ Click the associated **Radius/diameter auto** softkey **2** or **Length auto** **3** and enter the offset, especially for tools with rounded cuts, for example.
- ▶ As of SW 4.5, 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.
- ▶ As of SW 4.7, a tool offset **5** can be specified in the +/- direction. When measuring the tool in automatic mode, a measurement log **6** can be created after each measuring process.
- ▶ Click **NC Start** to initiate the measuring process; the tool compensations for radius and length 1 will be entered in the active tool compensation data.

Measuring function: Measure tool in JOG

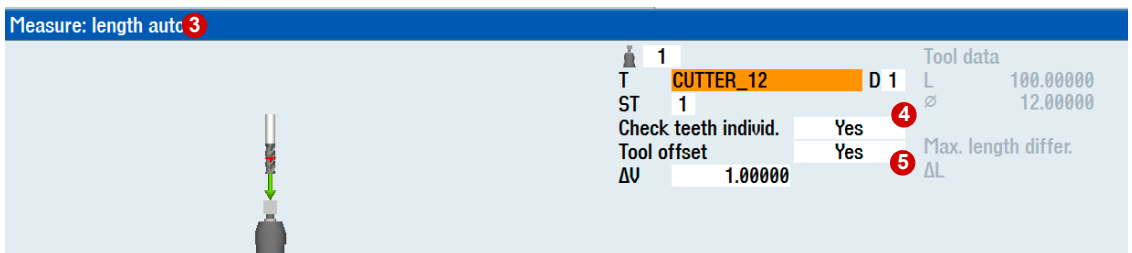
The screenshot shows the Siemens SINUMERIK OPERATE interface in JOG mode. The top bar displays 'SIEMENS', 'SINUMERIK OPERATE', the date '04/27/18', and the time '4:31 PM'. The 'M' (Measure) softkey is active. The main display area is divided into several sections:

- Work Position [mm]:** X: -0.02778, Y: -0.02868, Z: 179.72128, A: 0.000°, C: 0.000°.
- MRD (Machine Read Data):** T: 3D_PROBE, D1, L 280.27872; F: 0.000 mm/min, 100%; S1: 0, 100%; Master: 0, 100%.
- Right-hand Softkey Bar:** Length manually, Diameter manually, Length auto (3), Diameter auto (2), Calibrate probe, Calibrate fixed pt., Back.
- Bottom Softkey Bar:** T,S,M, Set W0, Meas. workp., Meas. tool (1), Position, Ma-chining, Swi vel, >

Measure diameter/radius



Measure length



Measurement log

SIEMENS SINUMERIK OPERATE 04/27/18 4:39 PM M JOG

Reset	MRD	T,F,S	Select tool
Work	Position [mm]		Measure report 6
X	11.86039	T CUTTER_12 D1	∅ 12.00000
Y	0.00000	CUTTER_12	L 100.00000
Z	650.00000	F 0.000	
A	0.000°	0.000 mm/min	100%
C	0.000°	S1 0	I [X]
		Master 0	100%

Measure: length auto

1
T CUTTER_12 D 1 Tool data L 100.00000
ST 1 L 12.00000
Check teeth individ. No
Tool offset Yes
ΔU 1.00000

Back

T,S,M 20 Set W0 Meas. workp. Meas. tool Position Ma-chining Swi vel >

2.8 Measure workpiece/tool in AUTOMATIC - process measuring

2.8.1 Measure workpiece in AUTOMATIC

For process measuring in Automatic mode, measuring cycles are specifically parameterized for the measuring 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 compensations (actual value is measured)
- Zero offset compensation (setpoint - actual value deviation)
- Tool data compensation (setpoint - actual value deviation)

2.8.2 Workpiece measuring cycles in AUTOMATIC

Practical measuring cycles are provided to facilitate process measuring.

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

Calibrate probe	Calibrate probe	Length						
Edge, measure	Edge distance	Radius in ring						
Measure corner	Corner	Radius at edge						
Measure hole	Hole	Comp. to sphere						
Measure spigot	Spigot							
Measure plane, angle deviation kinematics	3D							
Meas. result	Meas. result	Back	Back	Back	Back	Back	Back	Back
						Meas. workp.		Meas. tool

2.8.3 Example for measuring a workpiece in AUTOMATIC

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 zero offset.

Requirements for using cycles

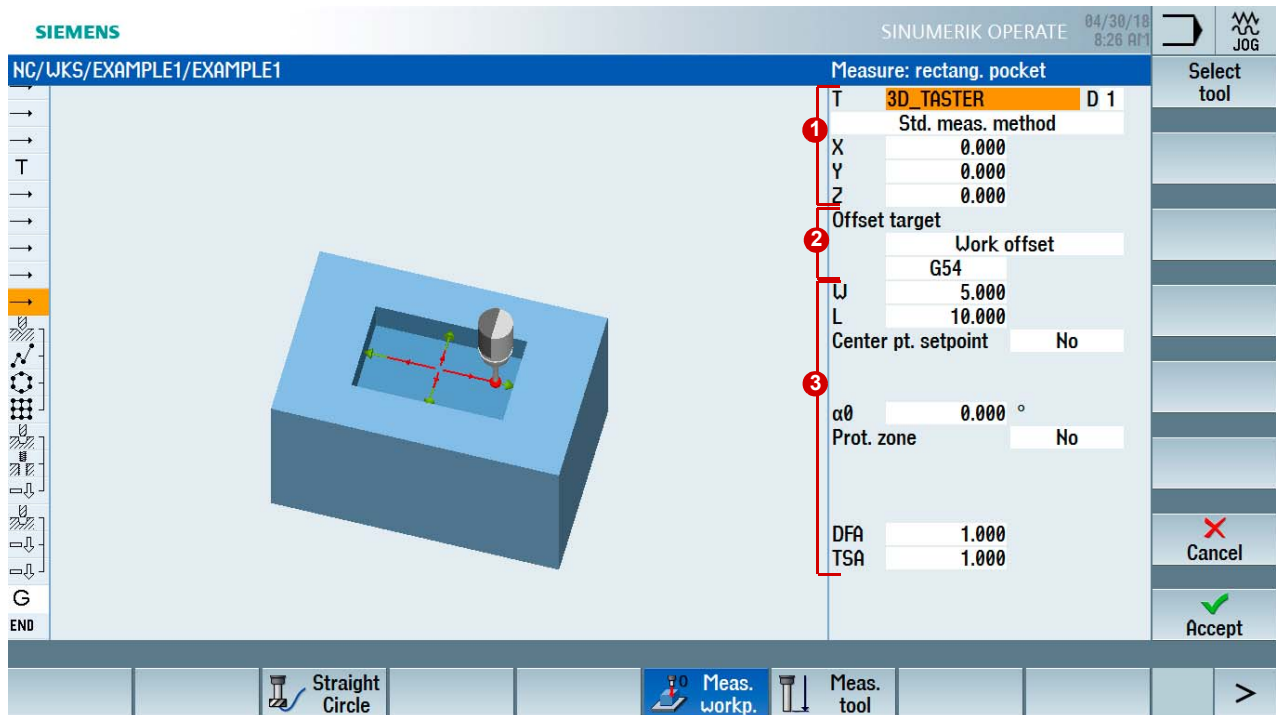
- The probe is calibrated and active; the tool compensation is activated.

Determining zero 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 the **Measure workpiece > Hole > Right-angled pocket** softkeys.
- ▶ Select the tool (3D probe), specify the measuring technique and enter the starting point of the measurement. ①.
- ▶ You can define whether the result of the measurement should be a compensation or merely a measuring process ②.
 - Compensation in the zero offset with specification of the zero offset
 - Compensation in the tool compensation data
 - Measurement only

As you are setting up the workpiece here, the compensation is made in the zero offset.

- ▶ Enter the setpoints of the right-angled pocket and parameterize additional cycle parameters ③.
After the measurement, the measured values are compensated in the active zero offset frame.



2.8.4 Tool measuring in AUTOMATIC - process measuring

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

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

Requirements

- The dynamometer is calibrated
- The tool is clamped

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

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 compensation.

Determining the tool length:

- ▶ Create a new program for measuring the tool.
- ▶ Select **Measure tool** measuring cycle.
- ▶ The measuring process is performed with stationary spindle and the measured values are entered into the tool geometry component ①.
- ▶ Select the length as measured value ②. Each tooth can be individually measured to check for tool breakage.
- ▶ Assign parameters to the measuring process ③.

SIEMENS SINUMERIK OPERATE 04/30/18 8:38 AM

NC/WKS/SAMPLE/SHOPMILL

P
END

Measure: tool

T CUTTER_32 D 1

Calibrat. data set 1

Measure Length ②

Check teeth individ. No

Meas. with stationary spindle ①

Tool offset Yes

ΔV 1.00000 in X

DFA 1.00000

TSA 1.00000

TZL 0.00000

TDIF 1.01000 ③

Select tool

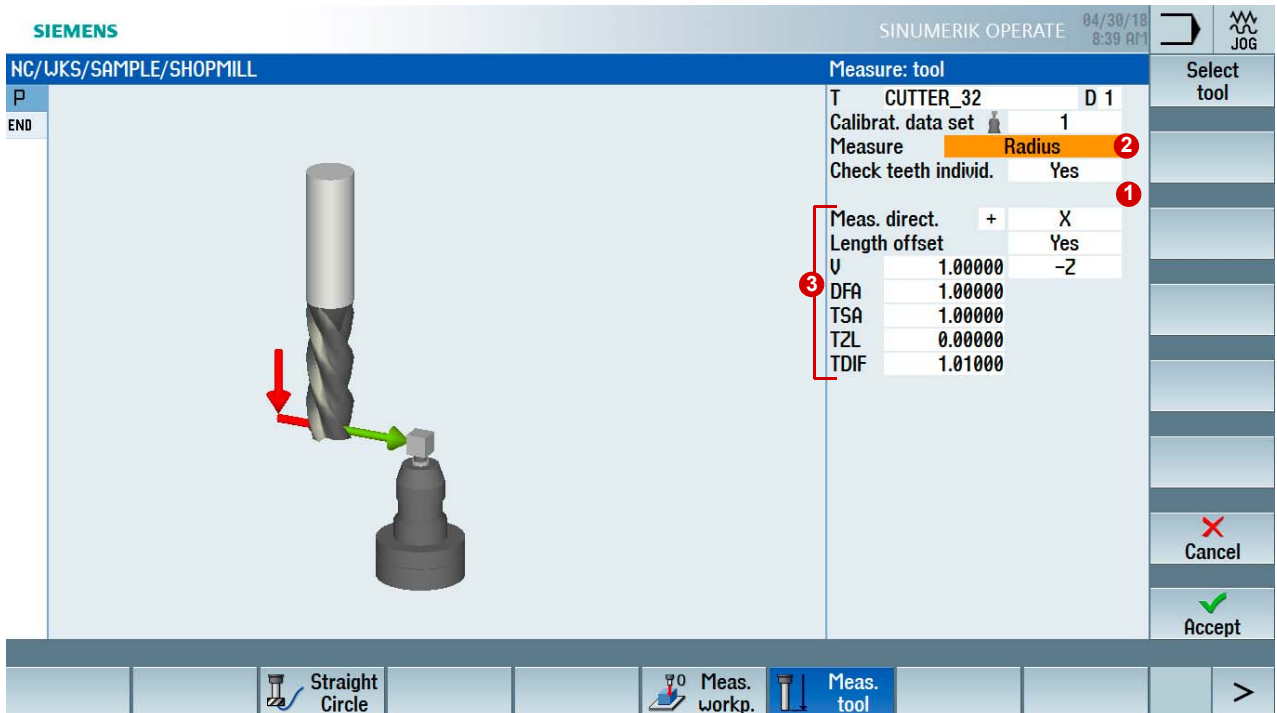
Cancel

Accept

Straight Circle Meas. workp. Meas. tool

Determining the tool radius:

- ▶ The measuring process is performed with rotating spindle and the setpoint - actual value deviation is entered optionally into the radius wear **1**.
- ▶ Select the radius as measured value **2**. Each tooth can be individually measured to check for tool breakage.
- ▶ Assign parameters to the measuring process **3**.

**Single cutting edge measurement (as of SW 4.5)**

T	CUTTER_32	D 1
Calibrat. data set	1	
Measure	Radius	
Check teeth individ.	Yes	4
Meas. direct.	+ X	
Length offset	Yes	
U	1.00000	-Z
DFA	1.00000	
TSA	1.00000	
TZL	0.00000	
TDIF	1.01000	

The single cutting edge measurement **4** can be used for remeasuring (compensation in the wear) and initial measuring (compensation 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.

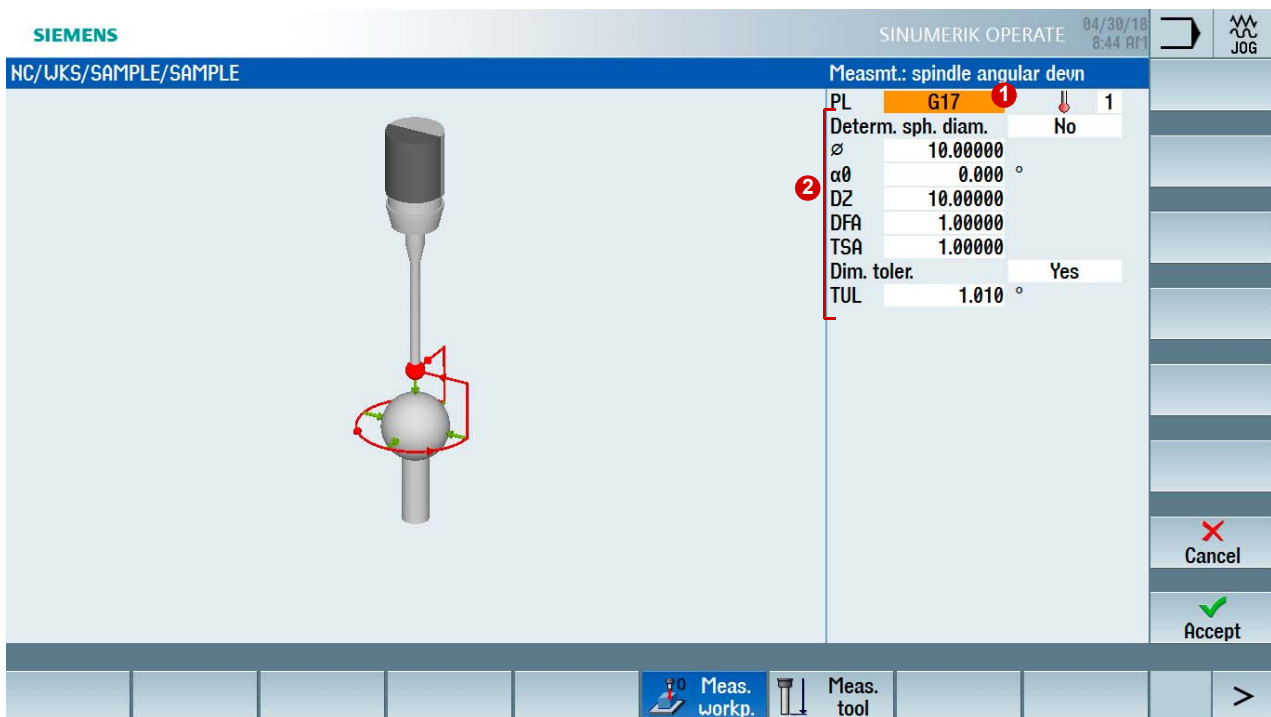
2.9 Checking and measuring machine/spindle

2.9.1 Spindle 3D angular deviation CYCLE995

With this measuring variant, the angularity (parallelism) of a spindle to the machine tool is measured on a calibration ball. The measurement is performed by combining the **sphere CYCLE997** and **outer circle segment CYCLE979** measuring variants.

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.



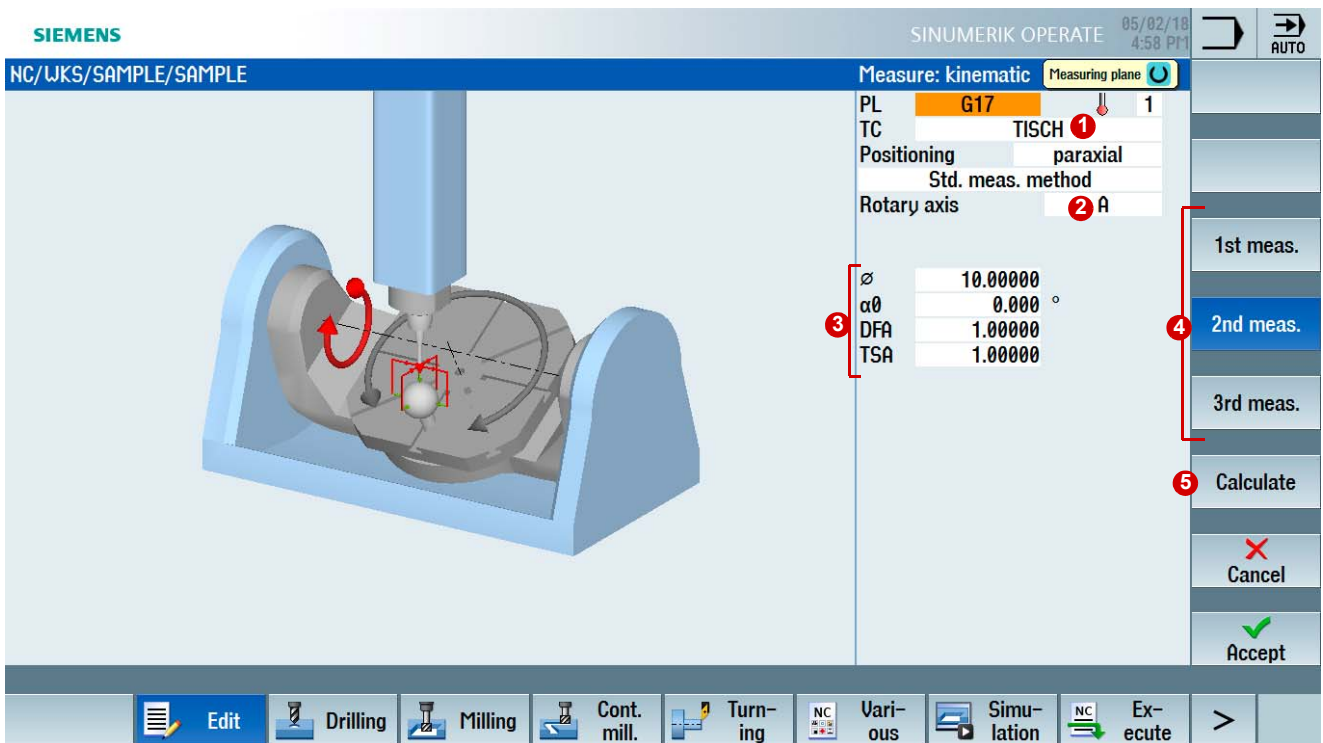
Checking the 3D angular deviation with CYCLE995

- ▶ Select the appropriate machining plane ① and the measuring parameters ②.
- The 1st measurement of the calibration ball is performed with CYCLE997 and the measurement is repeated. From two 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 second measurement is performed with CYCLE979 at 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 performed in full once all the rotary axes have been measured.

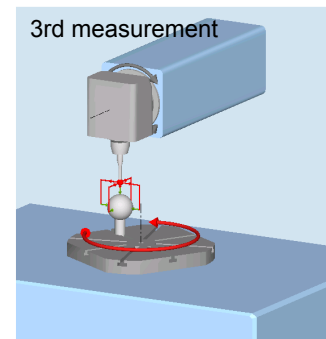
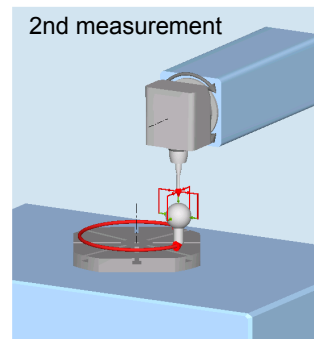
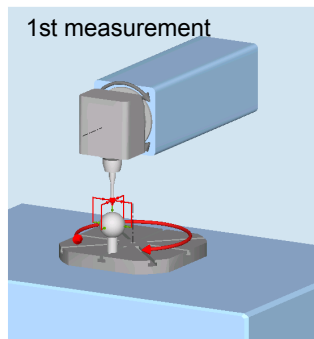


Checking the machine with CYCLE996:

- ▶ Create a new program to measure the kinematics.
- ▶ In the part program editor, select the >> **MEASURE MILL** > **MEASURE WORKPIECE** > **3D** > **KINEMATICS** softkeys in the Programs operating area.
- ▶ Create 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 to be measured **2**.
- ▶ Assign parameters to the measuring process **3** for measurements 1 to 3 **4**.
Press the **Calculate** softkey once all rotary axes have been measured **5**.

After pressing the softkey, the **Calculate kinematics** dialog opens. 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 compensation purposes)



Please take care when modifying the swivel data. This affects the kinematics directly and if an error is made with regard to the compensation value, this can result in damage to the machine during operation.

2.9.3 CYCLE9960 measure kinematics completely

CYCLE9960 corrects or checks the geometric vectors for defining the kinematic 5-axis transformation **1** based on kinematic chains and the tool carrier (classically via \$TC_CARR). With only one call of the cycle, the kinematics are measured, and the determined values and deviations are represented in the measurement result screen.

For the measurement, up to twelve positions of a measuring sphere on each rotary axis are sensed using workpiece probes. The sphere positions are defined in a specified rotary axis area in accordance with the geometric ratios on the machine **2**. The sphere position is set via the automatic repositioning of each rotary axis to be measured.

With CYCLE9960, it is also possible to measure the deviation on the tool tip (TCP) with active transformation for various rotary axis positions, after the measurement and compensation of the kinematics. These deviations can be compensated with VCS (Volumetric Compensation System).

Possible fields of application

- Check and correct the machine kinematics, e.g. during the machining process or for collision.
- Measure and adapt the reference, e.g. for kinematics with changeable heads.
- Measure and correct interpolation points, e.g. for compensation of the TCP via VCS Rotary.

SIEMENS SINUMERIK OPERATE 05/09/18 4:20 PM

NC/WKS/SAMPLE/SAMPLE

Measure: kinematic

PL	G17	1
TC	TISCH	
Kinematic	Meas. only	
Calibrate	Yes	
Positioning	paraxial	
∅	10.00000	
DFA	1.00000	
TSA	1.00000	
Rotary axis		A
AS	0.000 °	
AE	360.000 °	
AN	12	
CP	0.000 °	
αA	0.000 °	
Rotary axis		C
CS	-100.000 °	
CE	100.000 °	
CN	12	
AP	0.000 °	
αC	0.000 °	
Tolerance	No	

Cancel Accept

Edit Drilling Milling Cont. mill. Turning Vari-ous Simu-lation Ex-ecute >

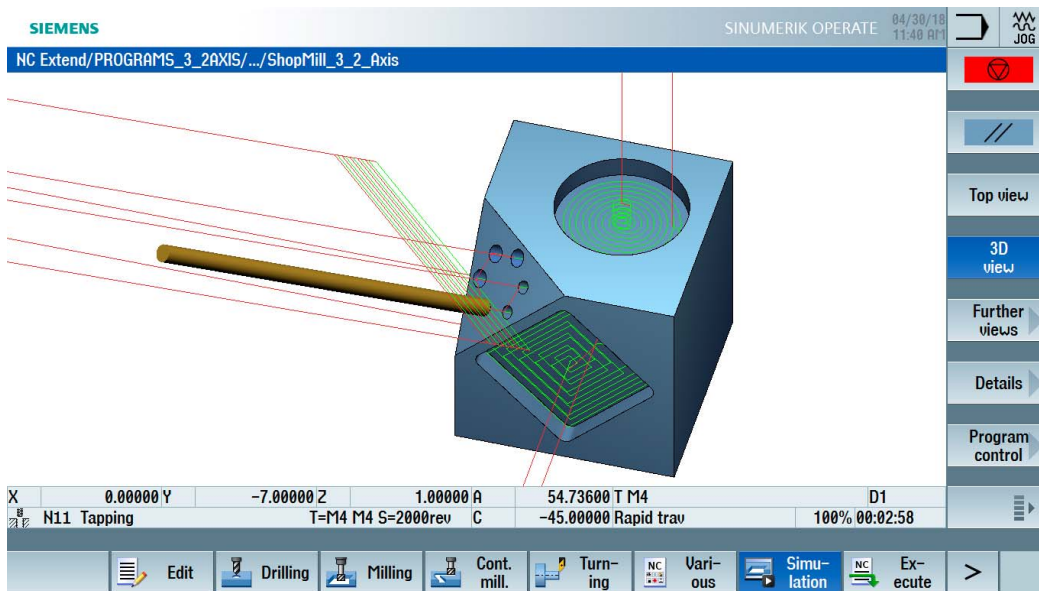
2.10 Workpiece visualization

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.

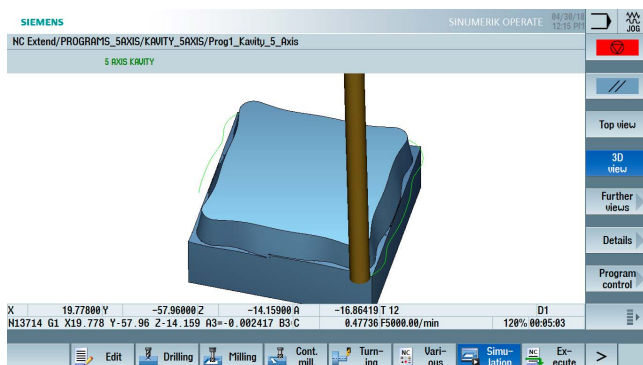
Opening a simulation:

- ▶ Open the NC program in the program editor.
- ▶ Press the **Simulation** softkey. 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].

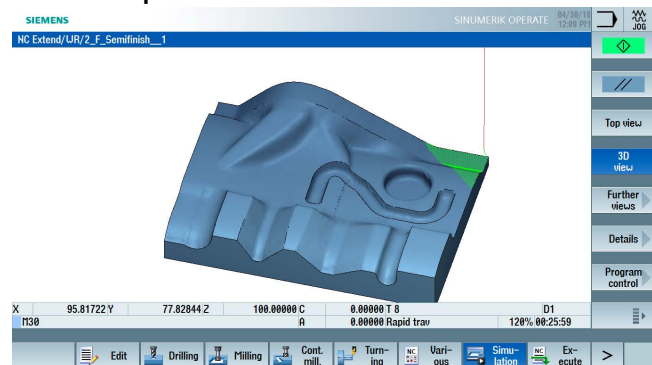


- Start simulation
- Reset
- Top view
- 3D view
- Side views
- Zoom, magnify, turn, cut
- Override, single block
- Display tool paths

5-axis simultaneous simulation



3-axis workpiece simulation



2.10.2 Quick viewer for mold-making

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 vectors as 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 the >> softkeys and then **modal-making view**. The NC program and the graphic are displayed in parallel.
- ▶ By pressing the **NC blocks** or **Graphic** softkeys ①, you can display and hide the views.

Zoom and cutout:

- ▶ Press the **Details** softkey ② and then **Zoom+** or **Zoom-** to zoom the views.
- ▶ Press the **Details** softkey ② and then **Magnify+** or **Magnify-** to enlarge or shrink the cutout. Using the cursor keys, you can move the cutout.

Select erroneous NC block:

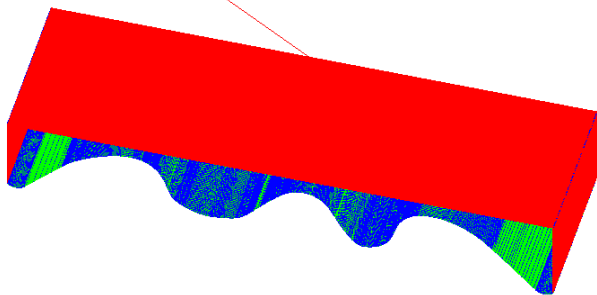
If there are errors in the graphic, you can directly select the NC block in the program.

- ▶ Press the >> softkey ③ 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.

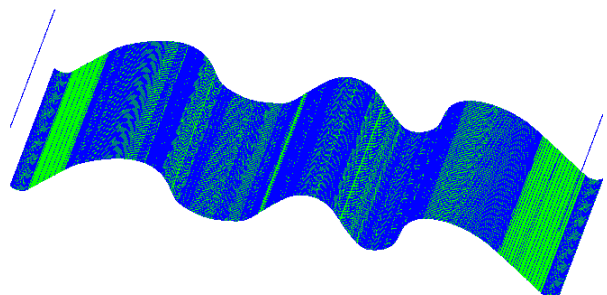
Hide G0, G1, G2, G3 lines:

- ▶ With the softkeys **4** you can hide G0, G1, G2, G3 or points so that the graphic lines are shown clearly. In the example, the rapid traverse movements G0 are hidden.

Lines G0, G1, G2, G3 shown



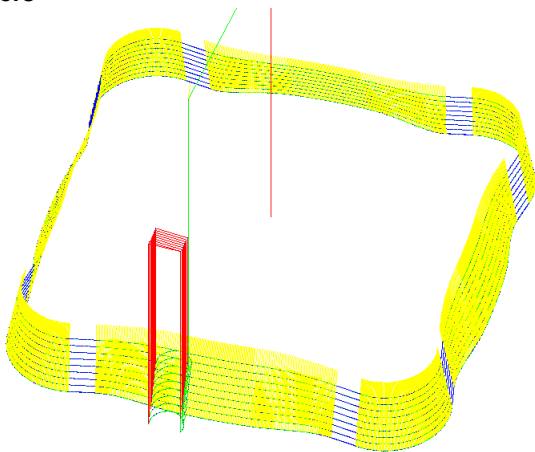
Hide G0 lines

**Display vectors, surfaces and curvature:**

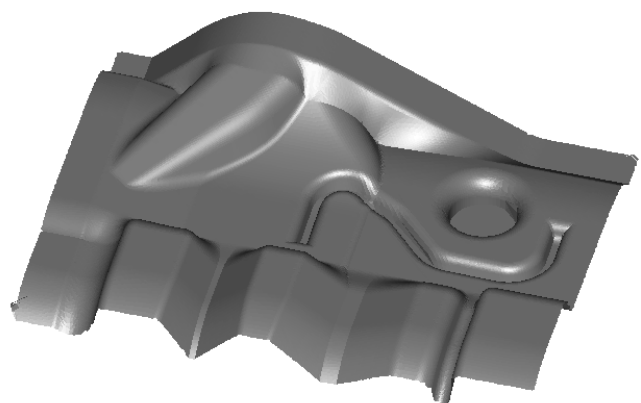
In addition to the classic view, for mold-making programs, you can also display the rotary axis vectors and grid mesh (surface, mesh), for example.

- ▶ Press the >> softkey **3** and then **Vectors** or **Surface** or curvature. The appropriate view is displayed.

Vectors



Surface (mesh)

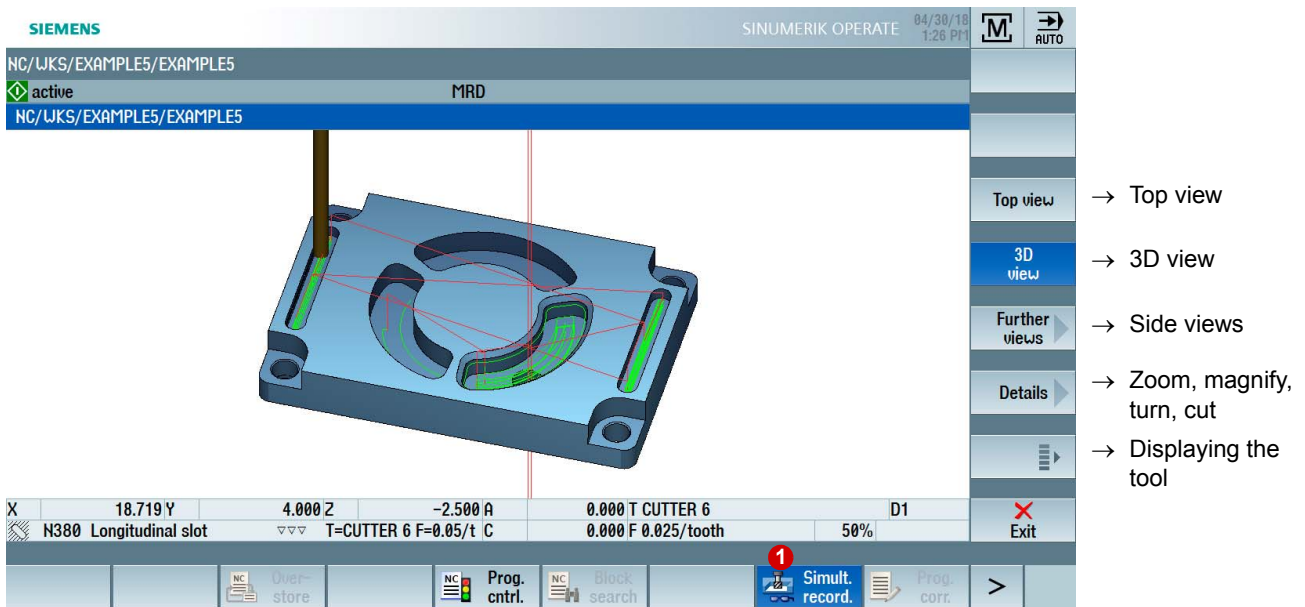


2.10.3 Simultaneous recording

In Automatic mode, i.e. during the program execution on the machine, the tool movements can be simultaneously recorded in the top 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.

Activating simultaneous recording:

- ▶ Before starting the NC program, press the **Sim. rec.** softkey **1**.
- ▶ 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.



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

The screenshot shows the SINUMERIK Operate interface with the following data and softkey bar:

Machine	Position [mm]	Dist-to-go	T,F,S
X1	-101.28447	0.00000	T 8
Y1	-73.13830	0.00000	D1 8
- Z1	471.48213	-231.48213	F 8
A1	0.000°	0.000	

Additional data shown includes: S1 2860 mm/min, Master 20000, and a list of G functions (1: G0 to 30: COMPOF). The softkey bar at the bottom includes: Over-store, Prog. cntrl., Block search, Simult. record., Prog. corr., and a navigation arrow.

Selection of the various displays during the execution:

- Active G and M functions of up to 16 G groups.
- Basic blocks, running time and workpiece counters
- 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 color coded, e.g. CTOL, OTOL for the tolerances.

The screenshot shows the 'All G functions' display with the following data:

G Function	Parameter	Value
1: G1	17: NORM	33: FTOCOF
2:	18: G450	34: OSOF
3:	19: BNAT	35: SPOF
4: FIFCTRL	20: ENAT	36: PDELAYON
5:	21: SOFT	37: FNORM
6: G17	22: CUT2DF	38: SPIF1
7: G40	23: CDOF	39: CPRECOF
8: G54	24: FFWJON	40: CUTCONOF
9:	25: ORIJKS	41: LFOF
10: G645	26: RM1	42: TCOFR
11:	27: ORIC	43: G140
12: G601	28: WJALIMON	44: G340
13: G710	29: DIAMOF	45: UPATH
14: G90	30: COMPSURF	46: LFTXT
15: G94	31: G810	47: G290
16: CFIN	32: G820	48: G462
		49: CP
		50: ORIEULER
		51: ORIAXES
		52: PAROTOF
		53: TOROTOF
		54: ORIROTA
		55: RTLIION
		56: TOLJUCS
		57: FENDNORM
		58: RELIEVEOF
		59: DYNFINISH
		60: WJALCS0
		61: ORISOF
		62:
		63:
		64: GFRAME0

Additional data shown includes: CTOL: 0.00400, OTOL: 0.057, STOLF: 1.000. The softkey bar at the bottom includes: Over-store, Prog. cntrl., Block search, Simult. record., Prog. corr., and a navigation arrow.

2.11.2 Program running time

To gain an overview of the program running time and the number of machined workpieces, use the **Times/counters** functions **1**.

Pressing the softkey the first time shows how long the program has already been running **2**. At every further start of the program, the time required to run the entire program the first time and the remaining time are displayed **3**. In addition, you can follow how much of the current program has been completed in percent on a progress bar **4**.

The running times can be displayed block-by-block/non-modally and saved **5**. The running times determined by the simulation are shown with a blue background. After the first processing, the actually determined times are shown with a green background. The **Settings** softkey is used to configure whether the running times should be displayed and saved **6**.

When a program is executed for the first time, its progress is estimated based on the program size and the actual program offset; this means it can differ from the actual running time.

The screenshot displays the Siemens SINUMERIK OPERATE interface. At the top, it shows 'SIEMENS' and 'SINUMERIK OPERATE' with a date and time '08/06/18 11:47 AM'. The main area is divided into several sections:

- Coordinate Table:** Shows 'Work' (X, Y, Z, A, C) and 'MRD' (Dist-to-go) values. X is 3.133, Y is 24.521, Z is -3.000.
- Program Block List:** Lists blocks N140 to N210 with descriptions like 'Mill pocket', 'Drilling', 'RAPID', 'Boring', and 'Thread milling'. Running times are shown in green circles.
- Time, counter Section:**
 - 'Program' running time: 0:13:15h (marked with 2)
 - 'Prog. remainder ca.': 0:04:05h (marked with 3)
 - Progress bar: 76% (marked with 4)
 - 'Count workpieces': Yes
- Bottom Bar:** Includes 'Synchron. action.' and 'Settings' (marked with 6).

2.11.3 Handwheel override

In the AUTOMATIC mode, while executing a program, small compensations 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 compensation acts as override to the traversing motions 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 AUTOMATIC mode, press the **Program control** softkey **1** and activate **DRF handwheel offset** **2**.

Displaying override

- Open the expanded softkey bar >> **3** and then the **Overrides** softkey **4**.

The left screenshot shows the SINUMERIK OPERATE interface in AUTOMATIC mode. The top bar displays 'SINUMERIK OPERATE' and '05/02/18 4:28 PM'. The main screen is titled 'DRF' and shows the following data:

Machine	Position [mm]
X1	0.00000
Y1	0.00000
Z1	500.00000
A1	0.000°

The right screenshot shows the 'Override' screen. The top bar displays 'SINUMERIK OPERATE' and '05/02/18 4:31 PM'. The main screen is titled 'Override' and shows the following data:

Function	Value
Tool	0.00000
Max	100.00000
DRF	0.00000
X	0.00000
Y	0.00000
Z	0.00000
A	0.00000
C	0.00000

2.11.4 Block search

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

The screenshot displays the Siemens SINUMERIK OPERATE interface in the 'Block search mode'. The top status bar shows 'SIEMENS' and 'SINUMERIK OPERATE' with the date '09/12/18' and time '3:45 PM'. The 'Reset' button is visible in the top left. The 'Machine' table shows the following coordinates:

Machine	Position [mm]
X1	-39.93574
Y1	-4.30145
Z1	129.67212
A1	0.000°

The 'Block search mode' panel is active, showing options for 'With calculation' and 'Without approach'. The 'Search for text' panel is also visible, with options for 'Interrupt point' and 'Search pointer'. The 'Start search' button is located in the top right corner. The bottom navigation bar includes icons for 'Over-store', 'Prog. cntrl.', 'Block search', 'Simult. record.', and 'Prog. corr.'.

Possible search variants:

- 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

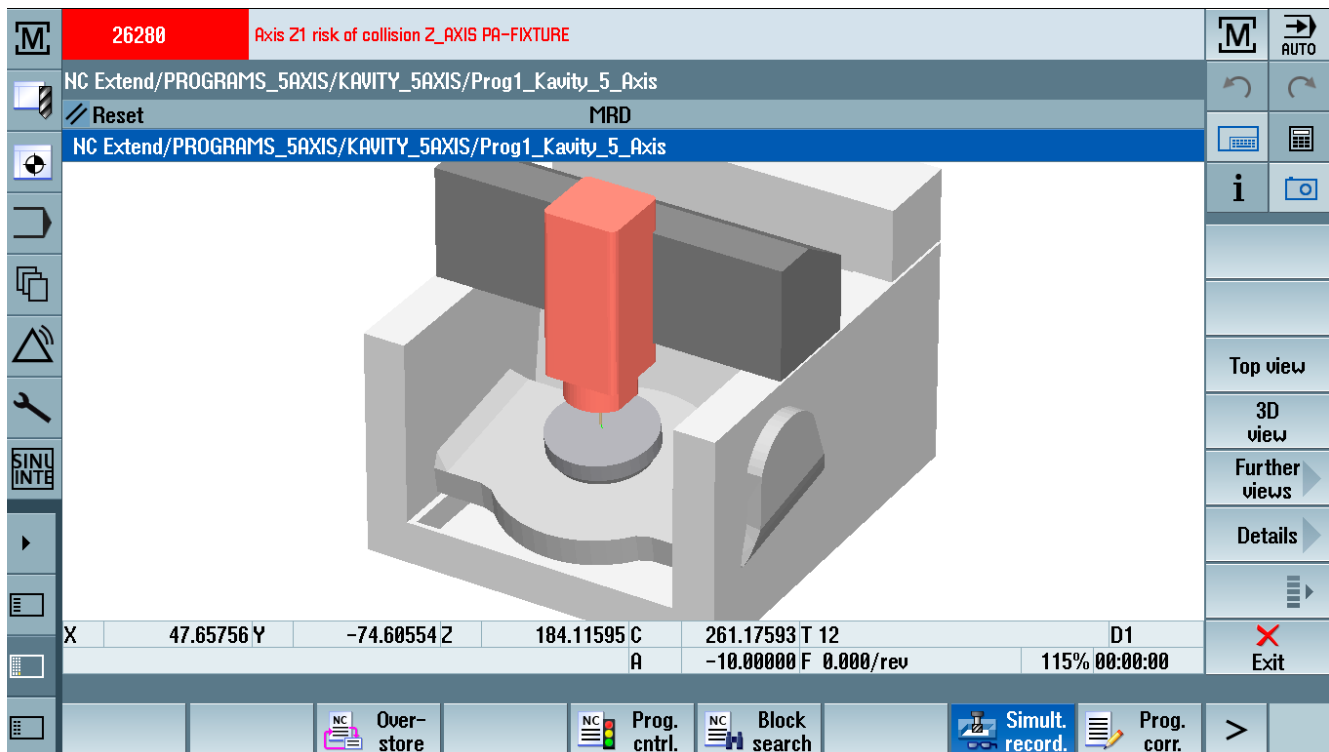
To use the block search:

- ▶ The block search can be configured individually using the **Search mode** softkey **1** e.g. with and without calculation or with/without approach.
- ▶ Using the **Interruption point** softkey **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.
- ▶ **Start search** **4** performs the search; CYCLE START executes the program at the search location.

2.12 Collision avoidance

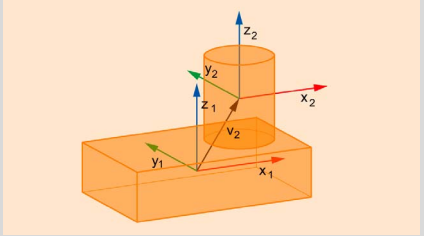
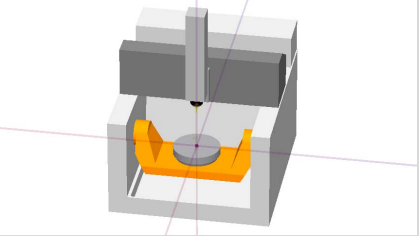
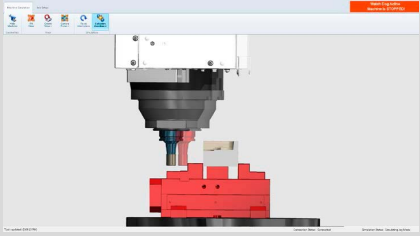
The improved collision avoidance available in three variants on the control prevents the collision of moving machine components with static components; it is active in setup operation and in automatic operation.

Collision avoidance is based on a machine model. The kinematics of the machine are described as a kinematic chain. Protection areas are attached to these chains for the protected machine parts. The geometry of the protection areas is defined using protection area elements. Protection areas are, for example, spindles, tools, tool table and machine covers.



The "Collision avoidance" function regularly calculates the clearance from these protection areas. When two protection areas approach one another and a specific safety clearance is reached, an alarm is displayed and the program is stopped before the corresponding traversing block and/or the traversing motion is stopped.

Overview of the Collision Avoidance variants

Collision Avoidance ECO	Collision Avoidance	Collision Avoidance ADVANCED
		
<ul style="list-style-type: none"> ■ Components / protection areas are described with simple geometric bodies. ■ The collision locations are visualized directly from the CNC user interface. ▶ The collision of moving machine components with static components is prevented, e.g. spindle with the machine covers. 	<ul style="list-style-type: none"> ■ Includes Collision Avoidance ECO ■ More protection area elements ■ Components/elements are modeled as STL files. ▶ The collision of moving machine components with static components is prevented thanks to the more exact geometry description of the components. 	<ul style="list-style-type: none"> ■ In addition, tools and workpiece material removal for collision monitoring are considered. ■ Further software from third-parties on an external PC in real-time is used to match position values, setup data and tool geometries (e.g. ModulWorks Corp.). ▶ Machine, tool and workpiece are protected dynamically.



Important functions for 3- to 5-axis machining

Content	Page
3.1 Introduction	72
3.2 Explanation of the terms: swivel, frames and TRAORI	73
3.3 Transforming coordinate systems - Frames	74
3.4 Swivel - CYCLE800	75
3.5 TRAORI 5-axis transformation	81
3.6 High Speed Settings - CYCLE832	102
3.7 Advanced Surface and Top Surface	110
3.8 Milling radius compensation 2D/3D-WRK	120
3.9 Volumetric Compensation System (VCS)	130
3.10 VNCK - virtual machine	131

3.1 Introduction

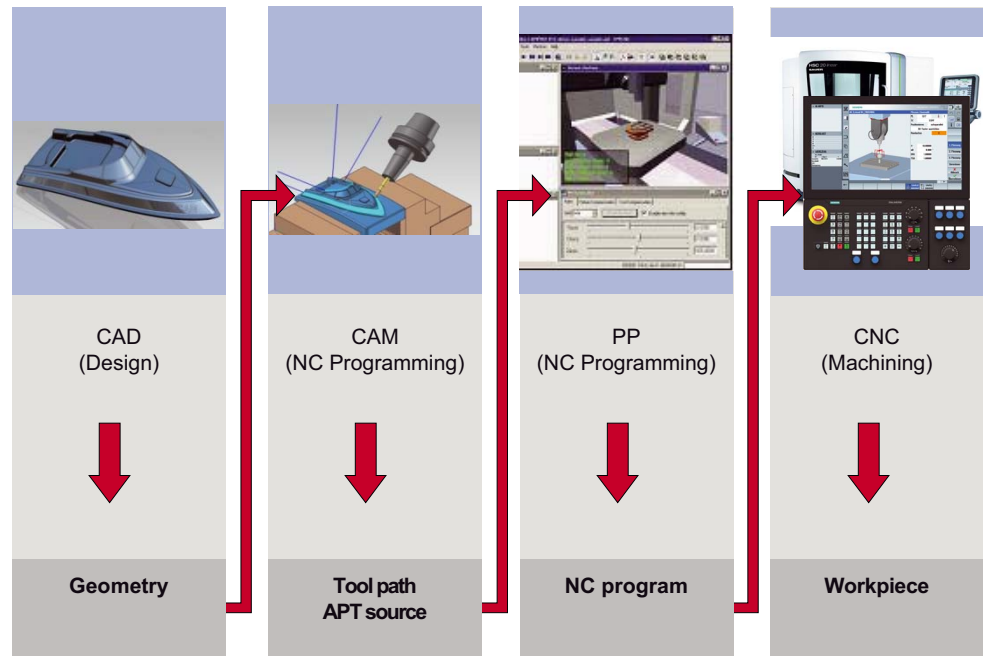
Especially within the context of mold-making machining, the entire CAD/CAM/CNC process chain plays a major role in terms of ensuring optimum results on the machine.

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.



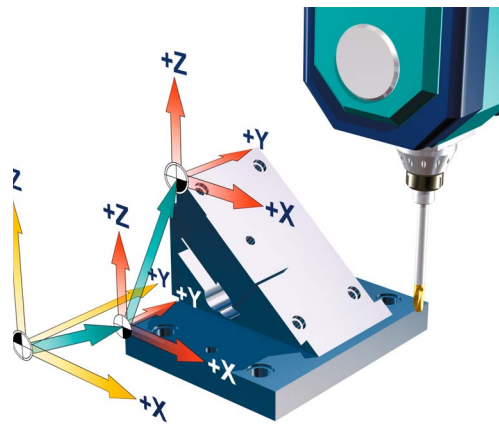
3.2 Explanation of the terms: swivel, frames and TRAORI

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

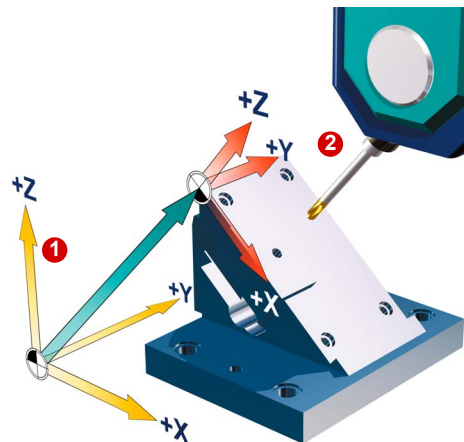
Frames



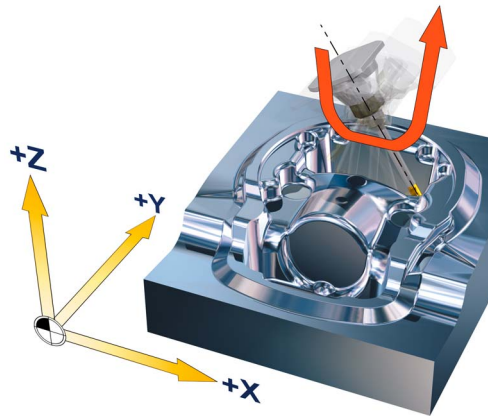
The frame is a self-contained arithmetic rule that transforms one Cartesian coordinate system into another Cartesian coordinate system.

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 zero offset.

Swivel



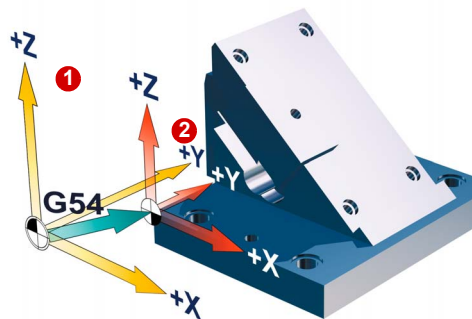
The tool is aligned with the machining surface by moving the rotary axes. This example involves rotation of rotary axis B **1** and the tool is positioned in relation to the XY plane **2**. Machining then takes place in this plane.

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 compensatory movements are initiated by the TRAORI function when the rotary axes are rotated.

3.3 Transforming coordinate systems - Frames

Coordinate systems



Machine coordinate system **1** with reference point and zero offset (G54, G55, etc.) are familiar 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.

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

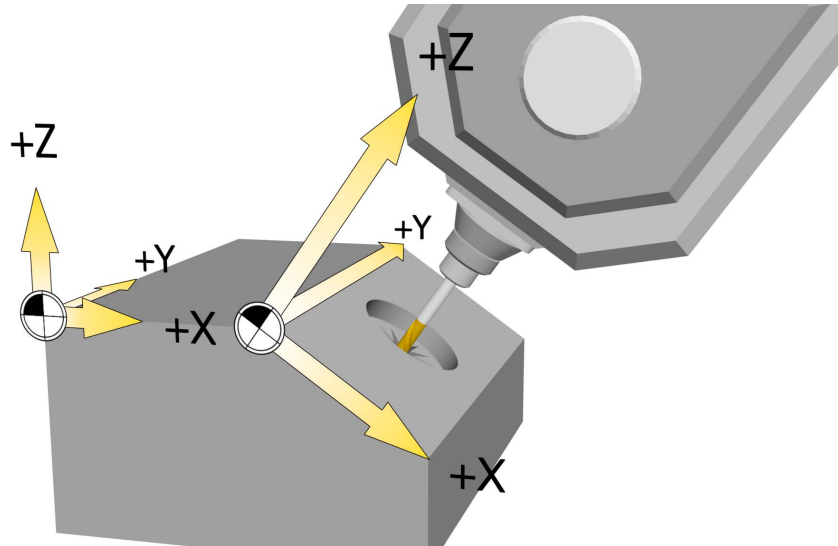
Possible frames include

- Basic frame (basic offset, G500)
- Settable frames (G54, G55, 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 Swivel - CYCLE800

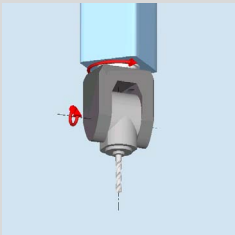
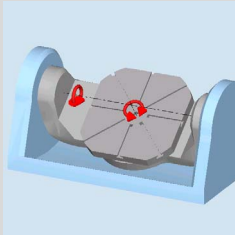
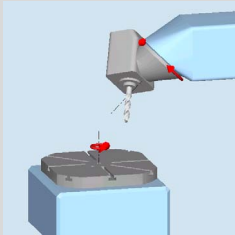
Function



You can use swivel heads or swivel tables to set up and machine inclined planes. Swiveling is possible in JOG mode and AUTOMATIC mode. 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. The machining plane then remains fixed during machining. When the axes are swiveled, the active zero points and tool compensations are automatically converted for the swiveled state; this results 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 movement 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 record **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**.

SIEMENS SINUMERIK OPERATE 05/02/18 7:11 PM

NC/WKS/SAMPLE/SAMPLE

Swivel plane

PL G17 (XY) **1**

TC KOPF_TISCH_45

Retract No

Swivel Yes **2**

Swivel plane New

X0 0.00000 **3**

Y0 0.00000 **3**

Z0 0.00000

Swivel mode Axis by axis **4**

Sequence of axes X Y Z **4**

X 0.000 ° **5**

Y 0.000 ° **5**

Z 0.000 °

X1 0.00000

Y1 0.00000 **6**

Z1 0.00000

Select Tool

Basic values

Cancel

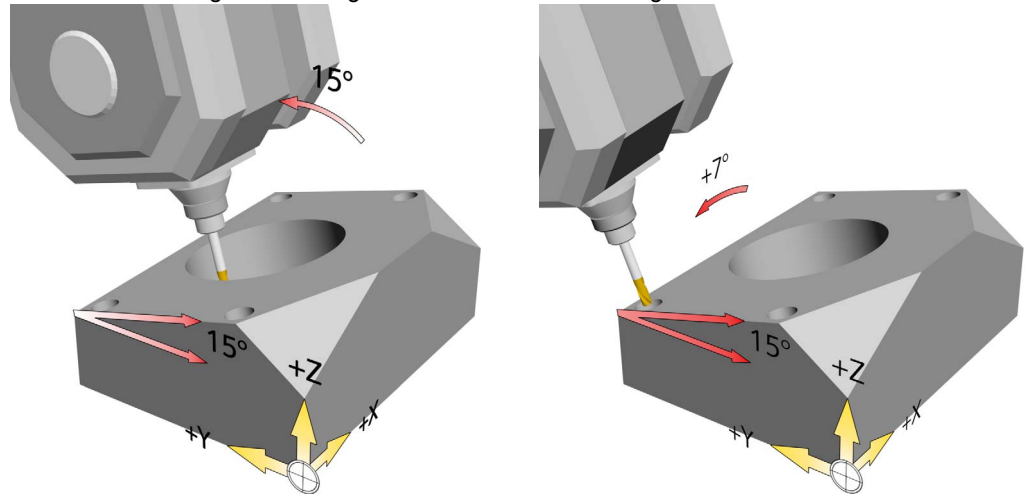
Accept

Edit Drilling Milling Cont. mill. Turning Vari-ous Simulation Ex-ecute

Example program 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 by Y=-7 or +7 degrees, face grooving with milling cutter for the holes and creating two holes at angles of +7 degrees and two holes at -7 degrees.



```

N100 ; Swivel the plane to the basic position
N110 CYCLE800(4,"TABLE",200000,57,0,0,0,0,0,0,0,0,1,,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="CUTTER_D32"
N150 M06
N160 S5000 M03
N170 G0 G54 X0 Y0
N180 TRANS Z50 ;Zero 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,-15,0,0,0,0,1,,1)
    
```

NC/LKS/SAMPLE/CYCLE800

Swivel plane	
PL	G17 (XY)
TC	TISCH
Retract	Max
Swivel	Yes
Swivel plane	New
X0	0.00000
Y0	0.00000
Z0	50.00000
Swivel mode Axis by axis	
Sequence of axes: X Y Z	
X	-15.000 °
Y	0.000 °
Z	0.000 °
X1	0.00000
Y1	0.00000
Z1	0.00000
Select	+
Tool	.

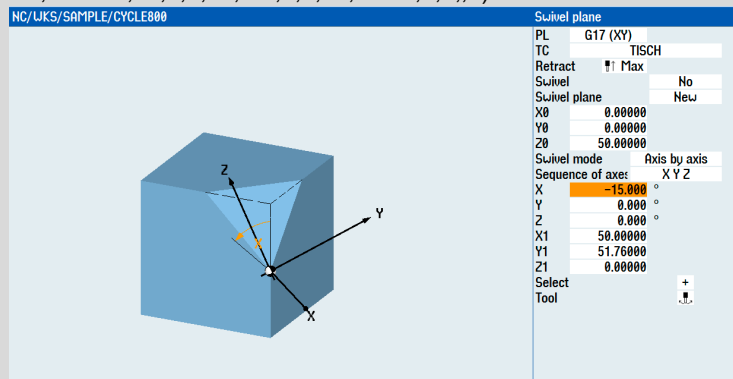
```

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)

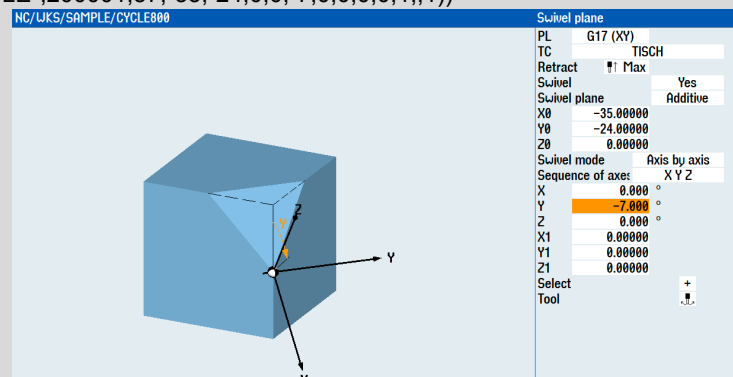
```



```

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

```



```

N400 ; Area -7 degrees with milling cutter face grooving so that the drill inserts vertically later
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)

NC/WKS/SAMPLE/CYCLE800		Swivel plane	
PL	G17 (XY)	Retract	Max
TC	TISCH	Swivel	No
Swivel plane	New	X0	0.00000
X0	0.00000	Y0	0.00000
Y0	0.00000	Z0	50.00000
Z0	50.00000	Swivel mode	Axis by axis
Sequence of axes	X Y Z	X	-15.000 °
X	-15.000 °	Y	0.000 °
Y	0.000 °	Z	0.000 °
Z	0.000 °	X1	50.00000
X1	50.00000	Y1	51.76000
Y1	51.76000	Z1	0.00000
Z1	0.00000	Select	+
Tool		Tool	

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)

NC/WKS/SAMPLE/CYCLE800		Swivel plane	
PL	G17 (XY)	Retract	Max
TC	TISCH	Swivel	Yes
Swivel plane	Additive	X0	35.00000
X0	35.00000	Y0	-24.00000
Y0	-24.00000	Z0	0.00000
Z0	0.00000	Swivel mode	Axis by axis
Sequence of axes	Y Z X	Y	7.000 °
Y	7.000 °	Z	0.000 °
Z	0.000 °	X	0.000 °
X	0.000 °	X1	0.00000
X1	0.00000	Y1	0.00000
Y1	0.00000	Z1	0.00000
Z1	0.00000	Select	+
Tool		Tool	

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)

NC/LKS/SAMPLE/CYCLE800		Swivel plane	
PL	G17 (XY)		
TC	TISCH		
Retract	Max		
Swivel	No		
Swivel plane	New		
X0	0.00000		
Y0	0.00000		
Z0	50.00000		
Swivel mode	Axis by axis		
Sequence of axes	X Y Z		
X	-15.000 °		
Y	0.000 °		
Z	0.000 °		
X1	50.00000		
Y1	51.76000		
Z1	0.00000		
Select	+		
Tool			

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)

NC/LKS/SAMPLE/CYCLE800		Swivel plane	
PL	G17 (XY)		
TC	TISCH		
Retract	Max		
Swivel	Yes		
Swivel plane	Additive		
X0	-35.00000		
Y0	-24.00000		
Z0	0.00000		
Swivel mode	Axis by axis		
Sequence of axes	Y X Z		
Y	-7.000 °		
X	0.000 °		
Z	0.000 °		
X1	0.00000		
Y1	0.00000		
Z1	0.00000		
Select	+		
Tool			

N640 ; Deep-hole drilling D8.5 with -7 degrees

N650 MCALL CYCLE83(10,0,2,-20,-,5,054,0,100,0,0,4,1,0,0,1,11121112)

N660 G0 X0 Y0

N670 X0 Y60

N680 MCALL

N690 ; Swivel the plane to the basic position

N700 CYCLE800(2,"TABLE",200000,57,0,0,0,0,0,0,0,0,0,0,1,,1)

NC/LKS/SAMPLE/CYCLE800		Swivel plane	
PL	G17 (XY)		
TC	TISCH		
Retract	2		
Swivel	Yes		
Swivel plane	New		
X0	0.00000		
Y0	0.00000		
Z0	0.00000		
Swivel mode	Axis by axis		
Sequence of axes	X Y Z		
X	0.000 °		
Y	0.000 °		
Z	0.000 °		
X1	0.00000		
Y1	0.00000		
Z1	0.00000		
Select	+		
Tool			

N710 ; Deselection CYCLE800

N720 CYCLE800()

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 cases 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.

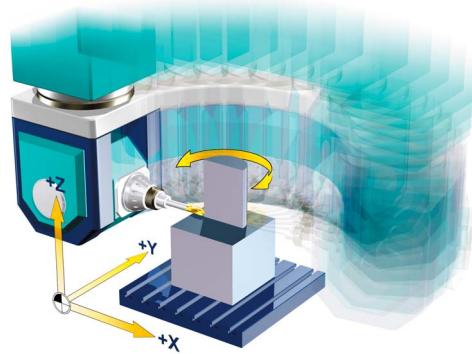
3.5.1 Basics of simultaneous 5-axis machining

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.

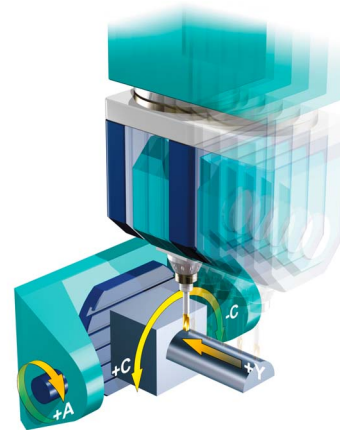
Machine kinematics and machine movement

Motion sequence for head/head kinematics **Motion sequence for table/table 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.

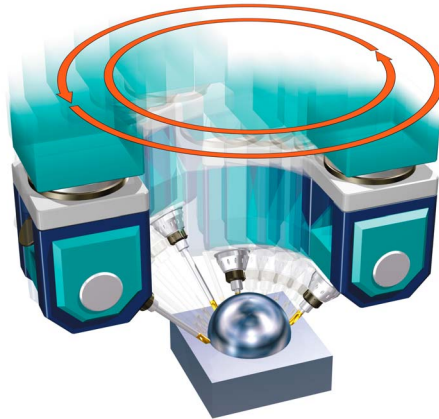
It is clear from the example that totally different machine movements produce the same machining result. If we look at the movements more exactly, we can see that in both cases the movement of the tool tip on the workpiece and the tool orientation relative to the surface are identical.



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.

Effect of the tool length on the 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.



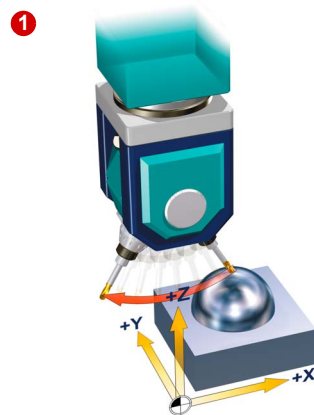
In the graphic, you can see that different tool lengths result in different machine movements. To machine the circle on the lower part of the half sphere, traversing must be performed in a circle in the X-Y plane and also the C axis must be rotated through 360° during the circular movement. Whereas the 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.

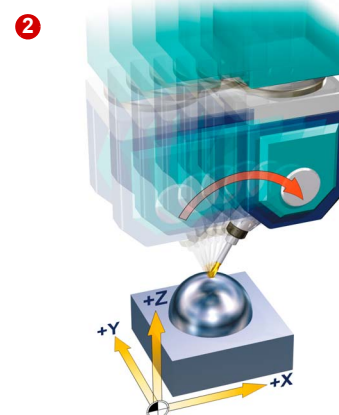
Rotary axis movement and tool orientation

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

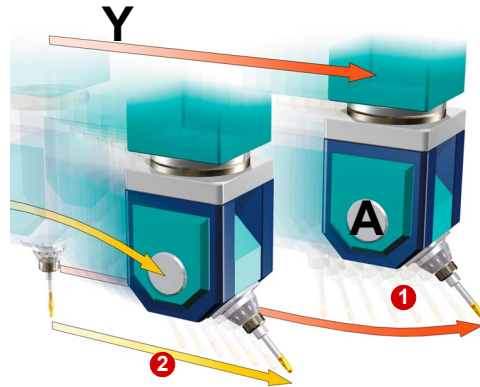


Example **1** shows that the movement of the A axis not only changes the tool orientation to the workpiece. At the same time, the tool tip moves in space, a circular path in the Z-X plane in our example. This is an unwanted side effect.

Generally, the movement shown in **2** is desirable, in which compensatory movements of the linear axes ensure that the tool tip is fixed in space during rotary axis movements.

Simultaneous movement of linear/rotary axes

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

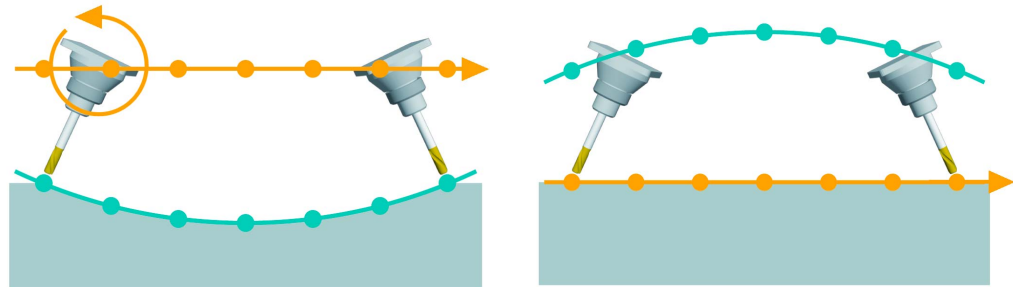


The graphic shows the simultaneous movement of the A axis and the Y axis. In situation **1**, both axes move synchronously; they are interpolated linearly with one another. A straight path of machine axes Y and A has been programmed. Such a traversal command **G1 Y... A...** produces the red, curved path for the tool tip.

As a rule however, the behavior shown in situation **2** is wanted. 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. The command must be programmed alone in an NC block.
TRAORI(n)	Activates the orientation transformation configured with n.
n	The number of the transformation (n = 1 or 2), TRAORI(1) corresponds to TRAORI.
TRAFOON(<transformatio n name>)	If TRAORI is defined via a kinematic chain, TRAORI can also be activated with TRAFOON(<transformation name>).
TRAFOOF	Deactivate transformation



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

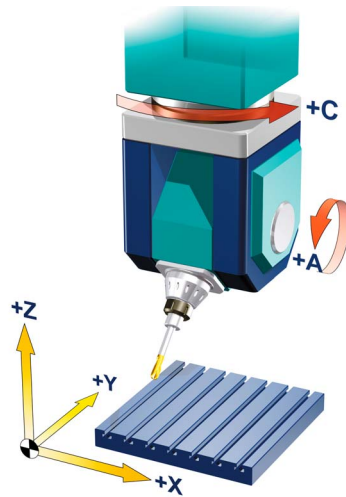
3.5.3 Programming the tool orientation

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

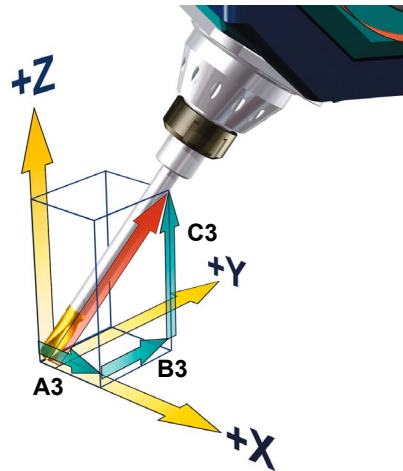
...

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 enables 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.

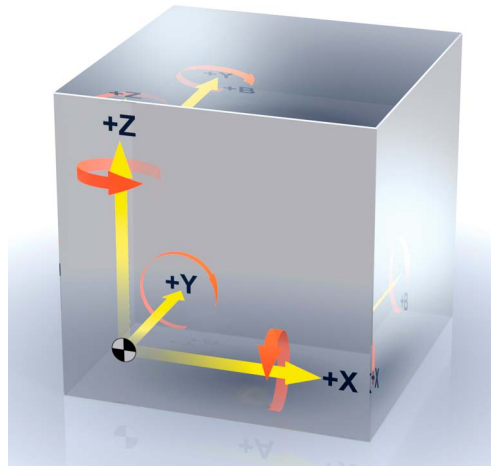
TIP

The use of the direction vector is recommended. The accuracy level selected should be 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

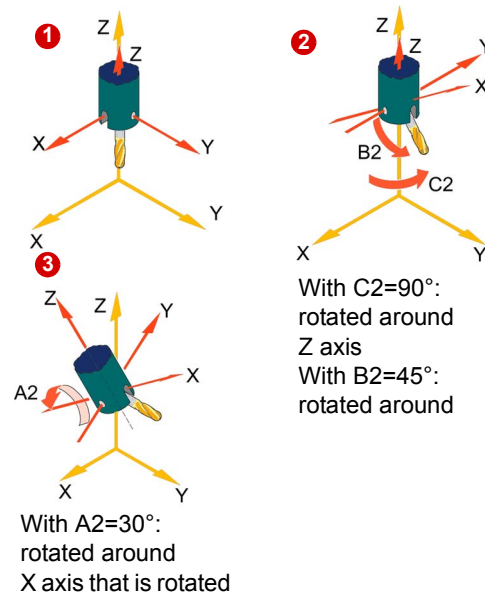


With Euler angles, rotation is first around the Z axis, then around the rotated X axis and finally around the resulting Z axis. There are different definitions for RPY angles.

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 angles



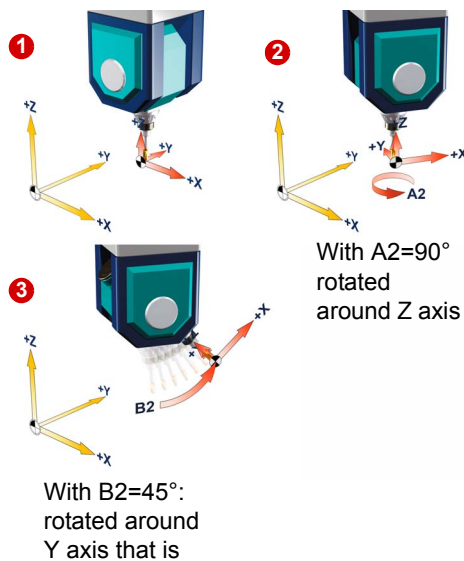
The values programmed with A2, B2, and C2 during orientation programming are interpreted as RPY angles (in degrees).

Starting from the basic position ① :

The orientation vector is obtained by first rotating a vector in the Z direction around the Z axis with C2 ②, then rotating it around the new Y axis with B2, ③ and finally rotating it around the new X axis with A2 (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 angles

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

Starting from the basic position 1 :

The orientation vector is obtained by first rotating a vector in the Z direction around the Z axis with A2 2, then rotating it around the new X axis with B2, 3 and finally, rotating it around the new Z axis with C2 (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 ...

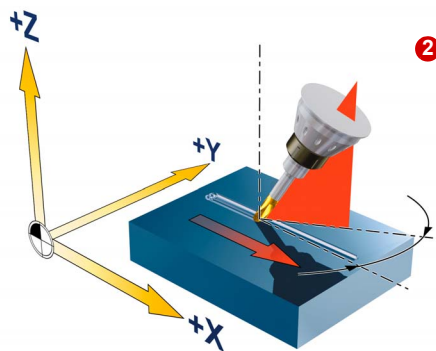
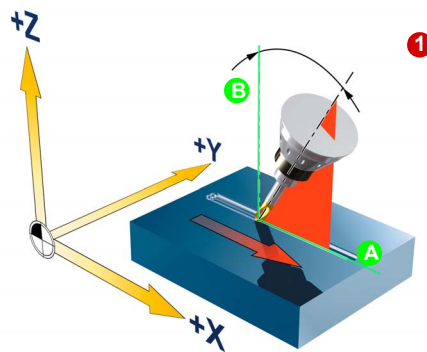
```

In this case, the value of C2 (rotation around the Z axis) is meaningless and does not have 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	<p>Lead angle for programming tool orientation.</p> <p>Angle relative to the surface normal vector in the plane put up by the path tangent and the surface normal vector.</p>
TILT	<p>Tilt angle for programming tool orientation.</p> <p>The TILT angle defines the rotation of the lead angle around the surface normal vector</p>

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.

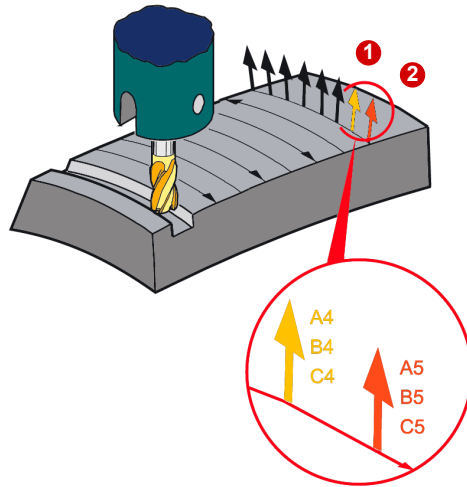
N10 TRAORI

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.

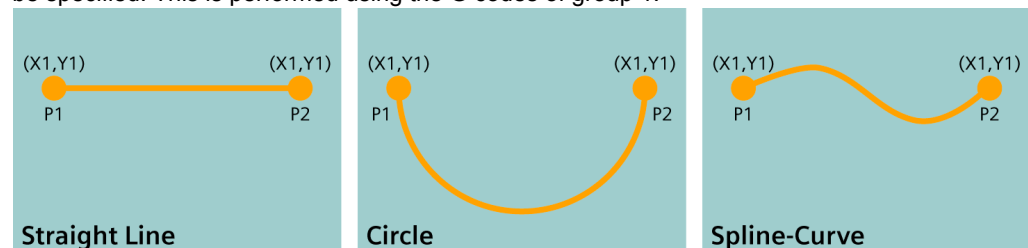
The start vector at the **start of the block** is programmed with A4, B4, C4. The vector at the **end of the block** is programmed with A5, B5, C5.

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.

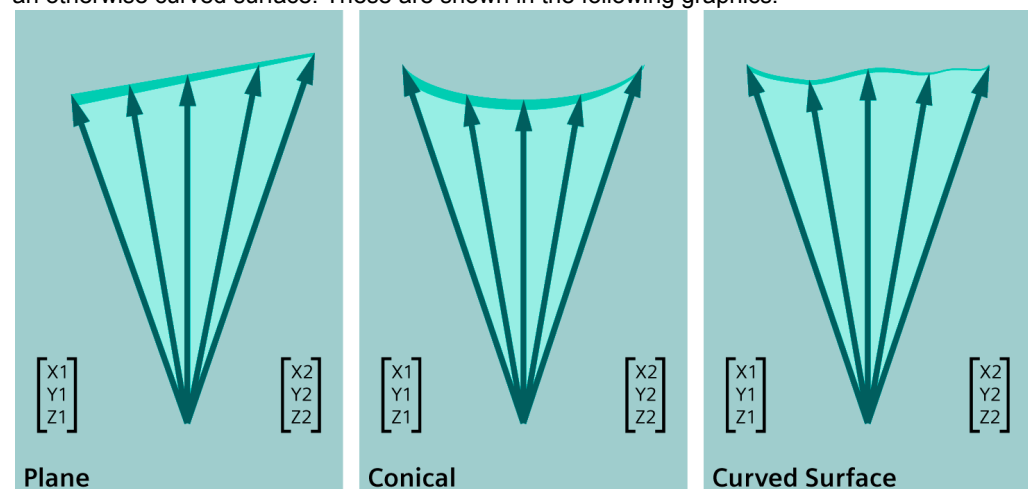
Motion interpolation in the 3-axis area

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.



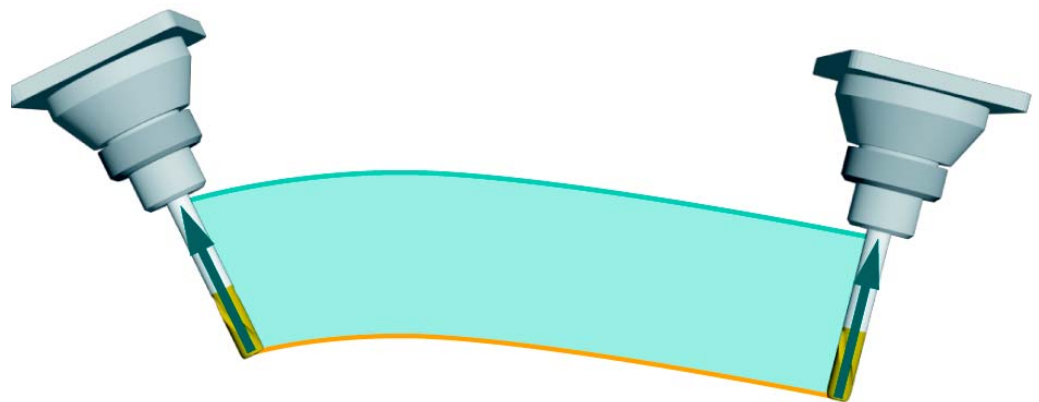
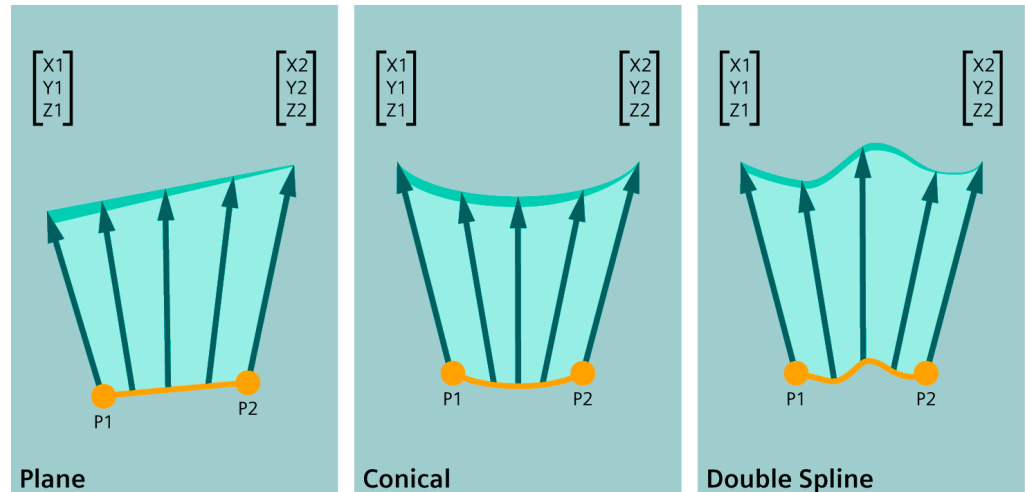
Interpolation of the tool orientation in a point

In the same way, there are numerous ways to move from 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 on 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.

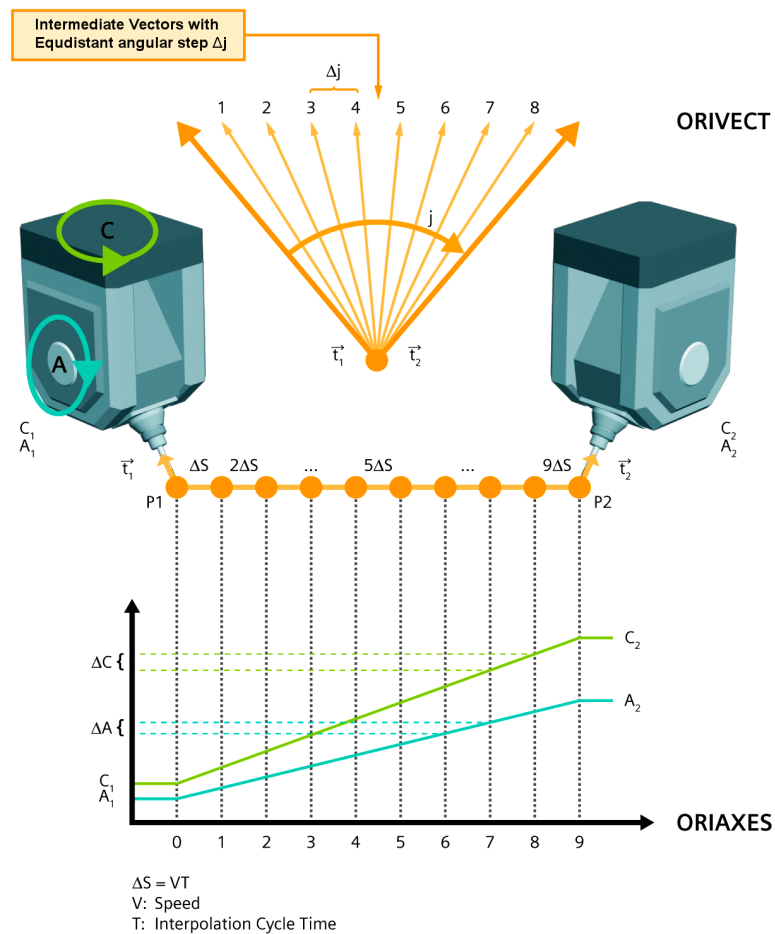


This is best seen when the position of the tool tip and the tool orientation are changed simultaneously. This can generally be described by two curves. The orange line describes the movement of the tool tip while the green line defines the movement of a second point on the tool. These two curves clearly 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 t_1 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, t_2 , C2 and A2. For the interpolation of the tool tip with the programmed velocity, 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 velocity, the entire traversing distance D between P1 and P2 is divided into nine equidistant interpolation increments.

In the same way with ORIAXES, the difference of the rotary axis positions $C_2 - C_1$ and $A_2 - A_1$ 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 ΔJ 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 to 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.

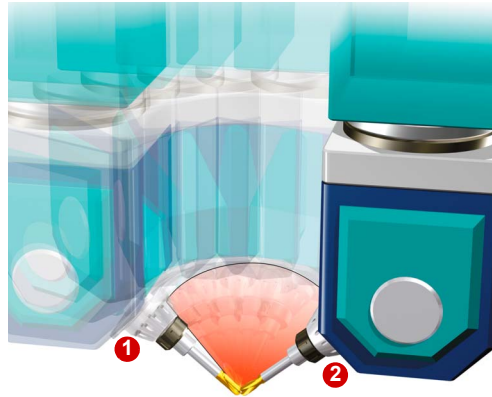
With **ORIWKS**, the programmed orientation relates to the workpiece coordinate system that may be rotated in relation to the machine coordinate system using a frame.

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 (ORIXES)

**Linear interpolation
(ORIXES)**



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 mold-making when ball mills are used.

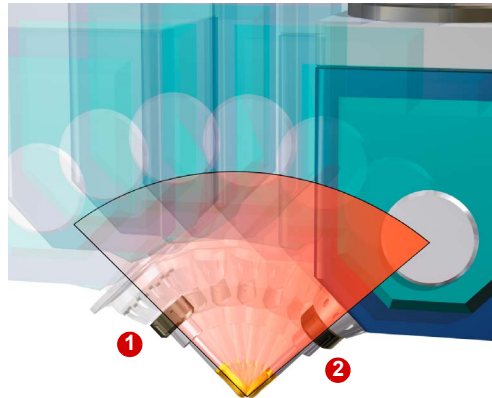
Axis/linear interpolation

**ORIXES
(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

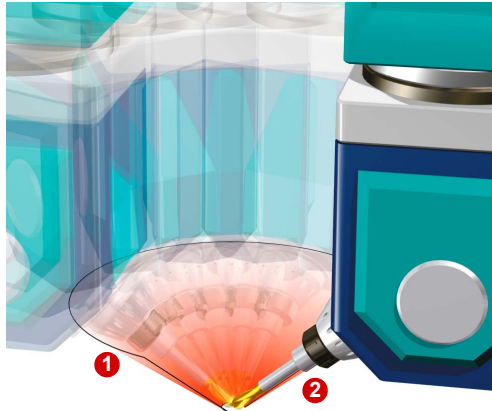
Interpolation of the orientation vector in a plane (large circle interpolation)

ORIPLANE

Interpolation in a plane (large circle interpolation), identical to ORIVECT

Cone surface interpolation (ORICONxx)

Cone surface interpolation (ORICONCW)

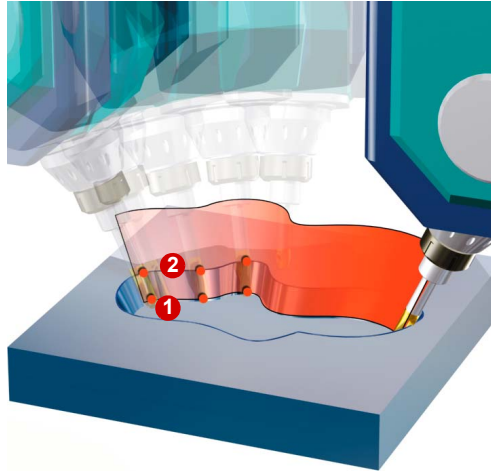


With cone surface interpolation, during reorientation the tool moves on a programmable peripheral surface of a cone located anywhere in space.

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

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 the path of a second point on the tool **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)

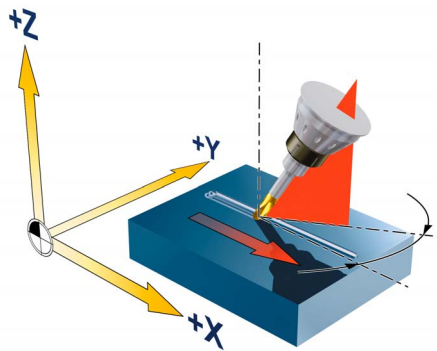
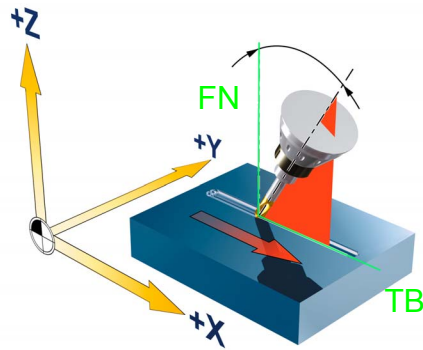
PO[YH] = (ye, y2, y3, y4, y5)

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.

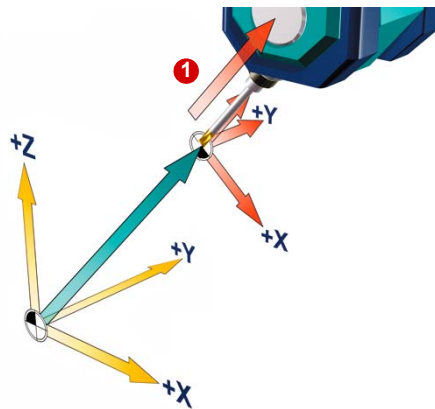
3.5.5 Example application involving TRAORI and frames

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 returning 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 return frame
N130	G1 G91 Z50 F500	; Return movement 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 This means that in JOG mode, the tool can be returned 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 return in the tool direction.

Notice:

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

TOROT to machining plane

TOROT for G17

TOROTY for G18 -> tool axis Y

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 the 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.

3.5.6 Useful NC commands for active TRAORI

In conjunction with the programming of TRAORI, the following NC commands can help to simplify programming.

ORIRESET Approach basic position of the tool orientation (ORIRESET)

Through ORIRESET(...), the orientation axes of the relevant machine kinematics are traversed linearly and synchronously from their current positions to the programmed basic positions. If a basic position is not programmed for an axis, the position from the associated machine data \$MC_TRAFO5_ROT_AX_OFFSET_1/2 is used.

Programming:

Example of machine kinematics CA (channel axis names C, A).

- ORIRESET(90, 45)
; C to 90 degrees, A to 45 degrees
- ORIRESET(, 30)
; C from \$MC_TRAFO5_ROT_AX_OFFSET_1/2[0], A to 30 degrees
- ORIRESET()
; C from \$MC_TRAFO5_ROT_AX_OFFSET_1/2[0], A to \$MC_TRAFO5_ROT_AX_OFFSET_1/2[1]

STAT Defining the preferred direction for active TRAORI with STAT

For active TRAORI, the control automatically executes compensatory movements of the rotary axes to achieve specific positions of the tool positioning. Because these compensatory movements are made by the control in negative and positive rotary axis positions, the preferred direction can be hard coded with the STAT command. The rotary axes are then moved only in the positive or negative direction to reach the target position. For example, STAT can be used instead of prepositioning with CYCLE800.

Programming:

STAT is programmed for ORIWKS and ORIAXES

- STAT=-1; (axis remains at - position. Only for ORIAXES)
- STAT=1; (axis remains at + position. Only for ORIAXES)

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.

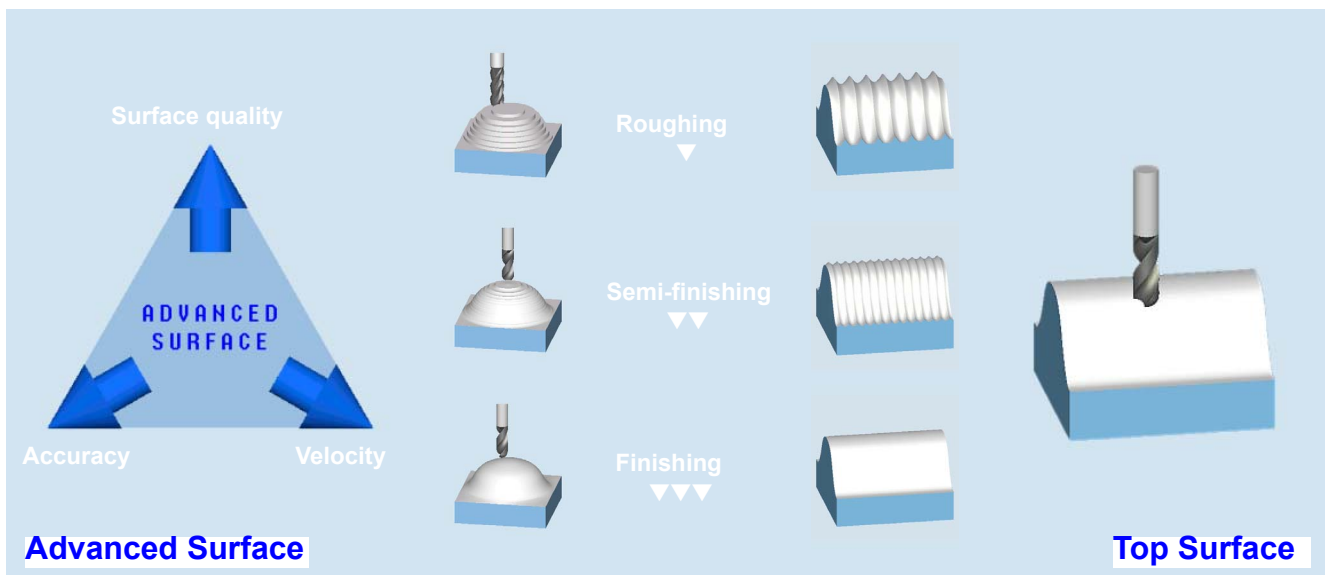
- When **roughing**, the emphasis is on velocity due to the smoothing of the contour.
- When **finishing**, the emphasis is on surface quality and accuracy.

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, as of 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.



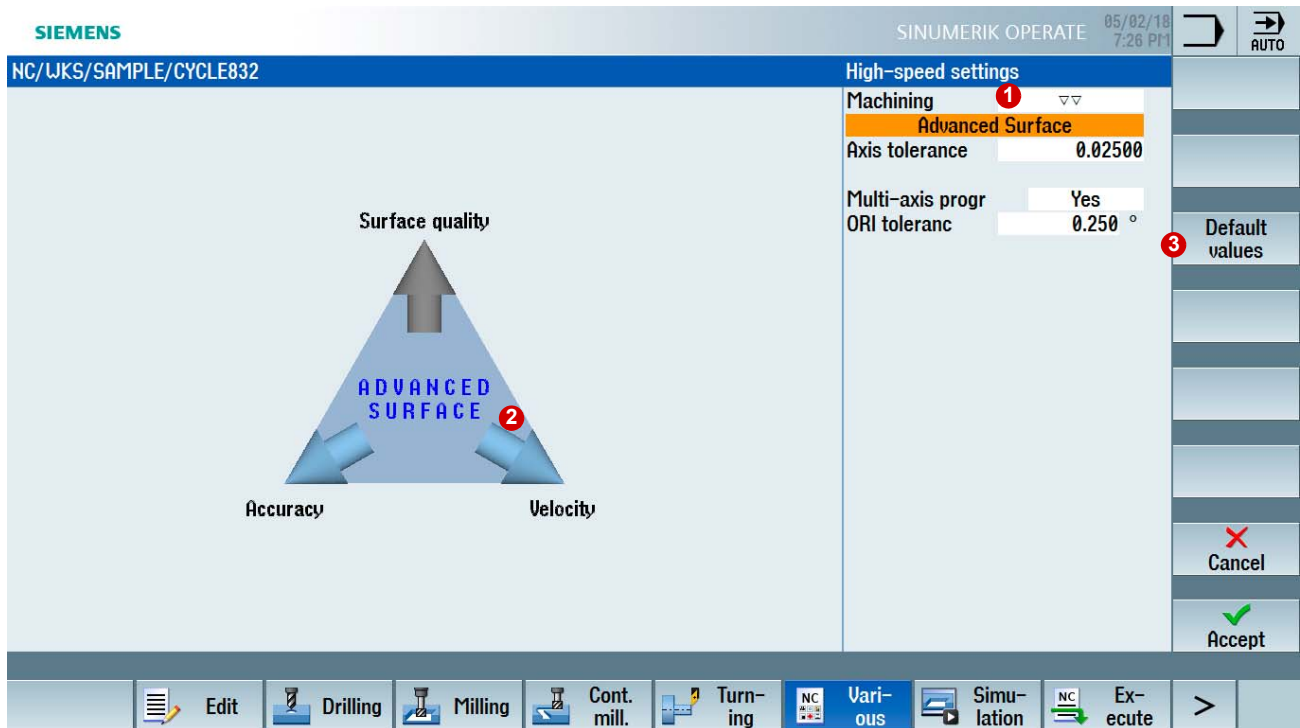
3.6.1 Parameters for CYCLE832

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 manufacturer).

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.

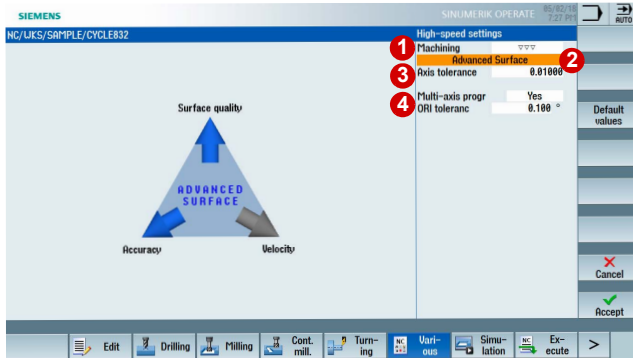


By selecting the machining strategy, the CAM program execution is optimized for the machine.

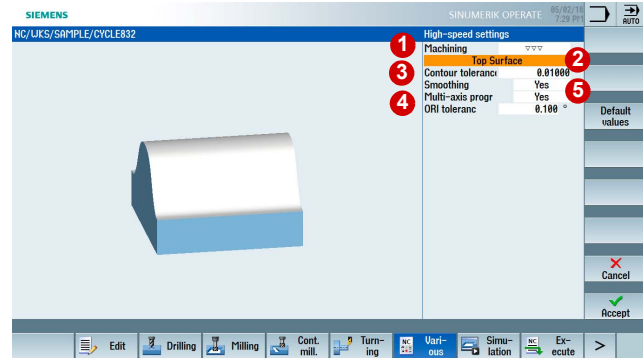
Depending on the selected machining **1** the blue arrows **2** point in the **Velocity**, **Surface quality** or **Accuracy** direction. Using the **Default values** softkey **3** preassigns the cycle parameters, depending on the selected machining.

Parameters for the High Speed Setting cycle

Advanced Surface



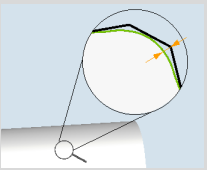
Top Surface

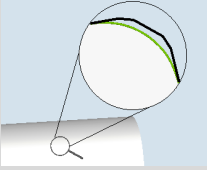


Parameterizing CYCLE832

- ▶ At Machining, select one of the deselection, finishing, semi-finishing and roughing parameters.
- ▶ If the Top Surface option is activated, then you can select between the two motion controls. Advanced Surface is the default setting.
- ▶ Enter a value in the **Axis/contour tolerance** field.
- ▶ For **multi-axis programs**, as of SW 4.5, you can smooth the orientation characteristic in the cycle and apply an **ORI tolerance**.
- ▶ For Top Surface, **smoothing** can also be activated via a setting data item. When smoothing, inhomogeneous areas along the path are smoothed.

CYCLE832 parameters (as of SW 4.7 SP2)

Machining	<ul style="list-style-type: none"> ■ Finishing (<code>_FINISH</code>) ■ Semi-finishing (<code>_SEMIFIN</code>) ■ Roughing (<code>_ROUGH</code>) ■ Deselection (<code>_OFF</code>) 	<p>Selection for 3-axis programs with plain text.</p> <p>Choosing "Deselection" resets the values to the default values.</p>
	<ul style="list-style-type: none"> ■ Finishing (<code>_ORI_FINISH</code>) ■ Semi-finishing (<code>_ORI_SEMIFIN</code>) ■ Roughing (<code>_ORI_ROUGH</code>) 	<p>Selection for multi-axis programs with orientation tolerance in plain text.</p>
Tolerance_tol. 	<ul style="list-style-type: none"> ■ Chord tolerance, contour tolerance, axis tolerance <p>(Chord tolerance must be imported from the CAM system 1:1)</p>	<p>Tolerance of the linear axes</p>

<p>Smoothing</p> 	<ul style="list-style-type: none"> ■ Yes (<code>_TOP_SURFACE_SMOOTH_ON</code>) (can be set via setting data item) ■ No (<code>_TOP_SURFACE_SMOOTH_OFF</code>) 	<p>Yes: Optimized path within the contour tolerance Inhomogeneous areas in the path in the standard tolerance range of 10 µm are reliably smoothed. The workpiece assumes a shiny surface.</p> <p>No: Close-contour path The specified contour is precisely followed. Engraving and geometries with a depth ≤ 0.005 mm = 5 µm are visible on the workpiece surface.</p>
<p>Multi-axis program</p>	<ul style="list-style-type: none"> ■ No (can be set via setting data item) ■ Yes 	<p>No: For 3-axis programs without rotary axes. The value 1 is entered automatically</p> <p>Yes: For multi-axis programs with rotary axes. The orientation smoothing is activated; ORI tolerance > 0° must be entered.</p>
<p>ORI tolerance</p>	<ul style="list-style-type: none"> ■ OTOL tolerance (approx. factor 10 of the linear tolerance) 	<p>Has two meanings:</p> <ul style="list-style-type: none"> ■ 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. ORISON is deactivated automatically for Top Surface (COMPSURF).

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_SEMIFIN**)▼▼
 - 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

CYCLE832 Advanced Surface programming example

```

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 value 0.1
N50      ; [3] = Roughing, [1] = Standard without ORI
N60      EXTCALL "CAM_ROUGH_0" ; Call subprogram CAM_ROUGH_0
N65      CYCLE832(0,_OFF,1)   ; Deselection CYCLE832
N70      CYCLE832(0.1,_ROUGH,1) ; 3-axis program, tolerance value 0.1
N80      ; [_ROUGH] = Roughing as plain text, [1] = Standard without ORI
N90      EXTCALL "CAM_ROUGH_1" ; Call subprogram CAM_ROUGH_1
N95      CYCLE832(0,_OFF,1)   ; Deselection CYCLE832
N100     CYCLE832(0.01,11,0.1) ; 5-axis program (with orientation), tolerance value 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     CYCLE832(0,_OFF,1)   ; Deselection CYCLE832
N140     CYCLE832(0.005,_ORI_FINISH,0.05) ; 5-axis program (with orientation), tolerance value 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     CYCLE832(0,_OFF,1)   ; Deselection CYCLE832
N200     M30                  ; End of program

```

CYCLE832 Top Surface programming example

```

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 CYCLE832(0,_OFF,1) ; Deselection 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] = semi-finishing, [1] = standard without ORI
N90 EXTCALL "CAM_SEMIF_1" ; Call subprogram CAM_SEMIF_1
N95 CYCLE832(0,_OFF,1) ; Deselection CYCLE832
N100 CYCLE832(0.001,_TOP_SURFACE_SMOOTH_ON+_ORI_FINISH,0.01)
N110 ; 5-axis program, tolerance value 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 CYCLE832(0,_OFF,1) ; Deselection CYCLE832
N200 M30 ; End of program

```



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 Section 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 feedforward control (speed or acceleration feedforward control). 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).



You can see which functions of CYCLE832 are currently active during processing in the "All G functions" display. See "Executing programs (AUTOMATIC)" on page 64.

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.

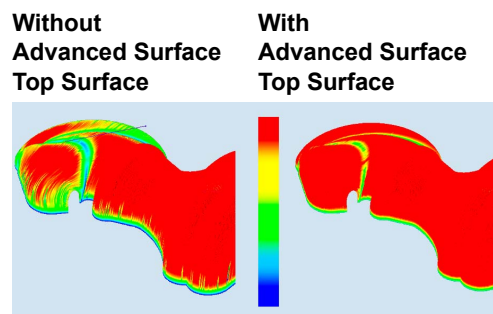
3.7 Advanced Surface and Top Surface

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

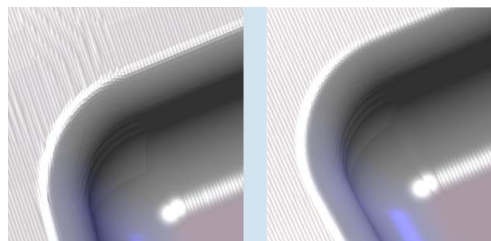
When the new motion control is used, an optimized **Look Ahead** function contributes to perfect surface quality through reproducible results in adjacent milling paths, accuracy and increased velocity. The compressor ensures precise contour accuracy and maximum machining velocity. Intelligent jerk limitation reduces the stress on the machine's 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.

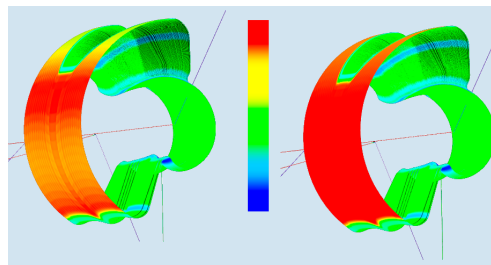
Top Surface is based on the COMPSURF compressor. This has been improved for machining demanding CAD/CAM programs.



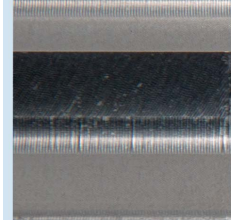
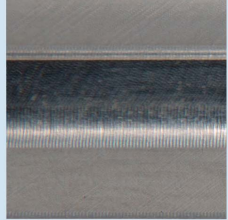
Increased machining velocity due to an improved velocity profile. The red areas show the maximum velocity.



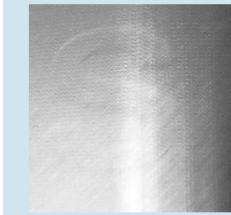
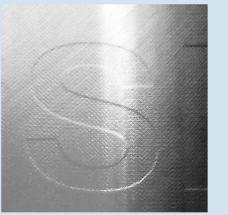
Perfect surface quality thanks to reproducible results in adjacent milling paths. The surface is much more homogeneous.



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

Without
Top SurfaceWith
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 onTop 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.

Advanced Surface validation / Top Surface machine data

A check program is available to validate the correctly set machine data. This program provides support to set the machine data for Advanced Surface and Top Surface as recommended. It checks the machine data and writes an overview of the machine data with details of the values that differ from the recommended values.



The check program and the associated documentation can be downloaded from the Industry Online Support Portal.

>> <https://support.industry.siemens.com/cs/ww/en/view/109738423>

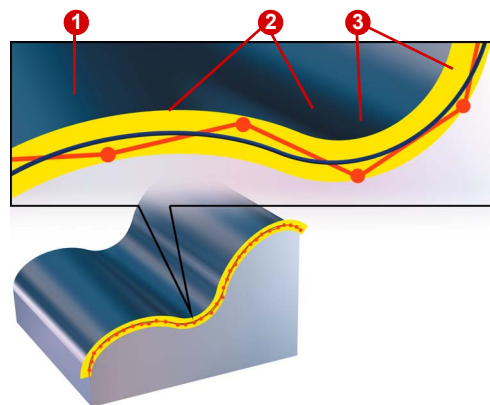
3.7.1 Compressor (COMPCAD, COMPSURF)

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 velocity 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 (default for Advanced Surface)	Compressor on - surface quality and velocity are further optimized. COMPCAD smooths the points along the 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. Preferably used to mill free-form surfaces (recommended) .
COMPCURV (do not use anymore)	Compressor on. Blocks are approached using a polynomial. Block transitions are jerk-free. Preferably used for circumferential milling .
COMPSURF (default 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



In accordance with the specified tolerance band, the compressor ① takes a sequence of motion commands ② combines them and compresses them into a spline ③, which can be directly executed by the control. 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 velocities, thereby increasing productivity.

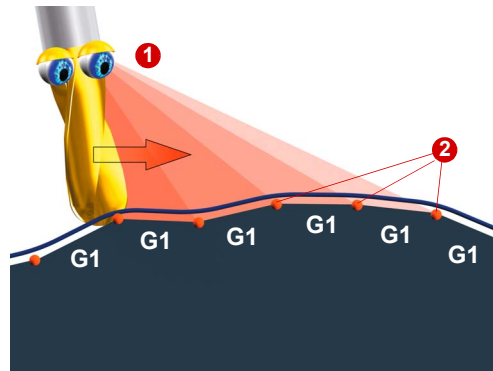
3.7.2 Continuous-path mode, Look Ahead – G64, G645

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 corner smoothing and tangential block transitions within the defined tolerances. With G645, the smoothing movement 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 connection with Advanced Surface, it is recommended that you only work with G645.

Application of G64, G645



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

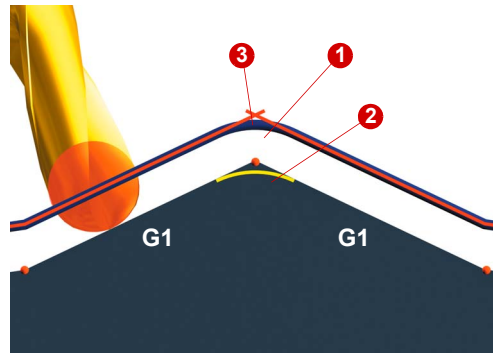
Look Ahead – anticipatory velocity control ①

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.

Corner rounding ②

The Look Ahead function also means that the control 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 sharp corners, **3** for example, the continuous-path command **G645** forms transition elements **1**, **2** at the block boundaries. 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.

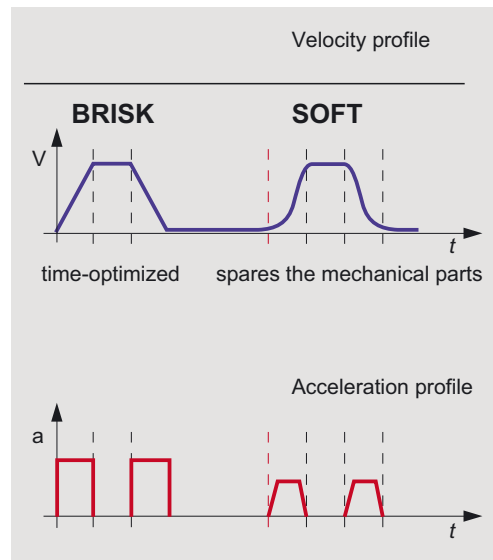
3.7.3 Feedforward control and jerk limitation – FFWON, SOFT, ...

Feedforward control 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)	Feedforward control "on"
FFWOF	Feedforward control "off"
BRISK (not recommended)	Without jerk limitation Path axes accelerate abruptly
SOFT (recommended)	With jerk limitation Jerk-limited acceleration of the path axes Axial jerk limitation

Jerk limitation function



To make acceleration as smooth on the machine 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 protect the machine and improves the surface quality of workpieces, as much less machine resonance is generated.

BRISK:

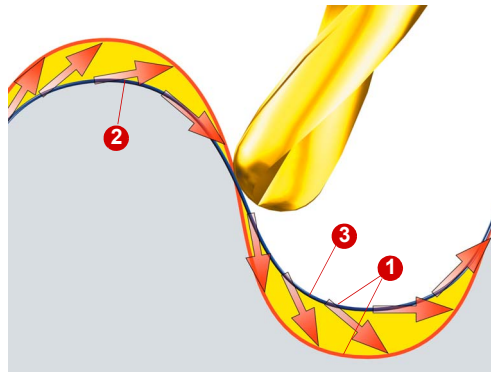
Acceleration behavior: Path axes accelerate abruptly in accordance with the set machine data.

The axis slides accelerate at the maximum rate until the feedrate is achieved. **BRISK** enables time-optimized machining, but with jumps in the acceleration curve..

SOFT:

Acceleration behavior: Jerk-limited acceleration of the path axes.

The axis slides accelerate at a constant rate until the feedrate is achieved. As a result of the jerk-free acceleration characteristic, **SOFT** permits a higher path accuracy and less stress on the machine.



Feedforward control.

In cases where axes are not feedforward-controlled, 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** on curved contours. The following error depends on the servo gain factor that is set (dependent on mechanics) and the axis velocity.

The **FFWON** feedforward control function brings the velocity-dependent following error down toward zero during path traversal. Traversing with feedforward control 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 FGROU axes and the other path axes. The path axes that are not included in the FGROU should either be routed synchronously to the path of the FGROU axes or to the curve parameters. Therefore, for the axes that are not contained in FGROU, 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)

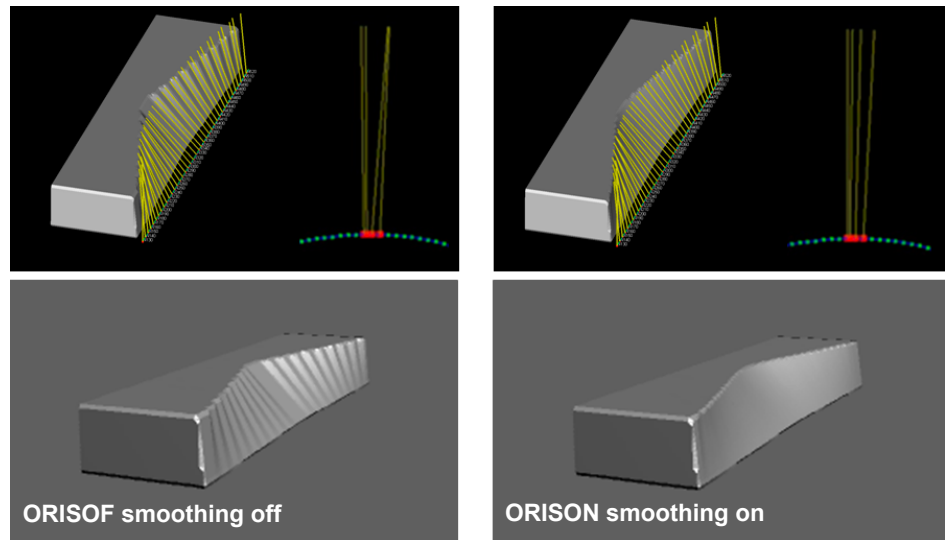
The ORISON function can be used to smooth fluctuating orientation across several blocks. The aim is to achieve a smooth characteristic for the orientation and therefore a more harmonious movement of the axes.

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 velocities 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 activated automatically 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.



NOTE

ORISON is deactivated automatically 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

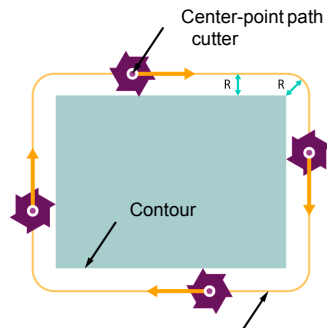
The G groups are switched and activated automatically when the machining method (e.g. finishing or roughing with CYCLE832 or CUST_832) is selected.

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.

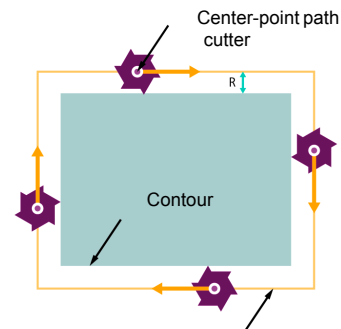
3.8 Milling radius compensation 2D/3D-WRK

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



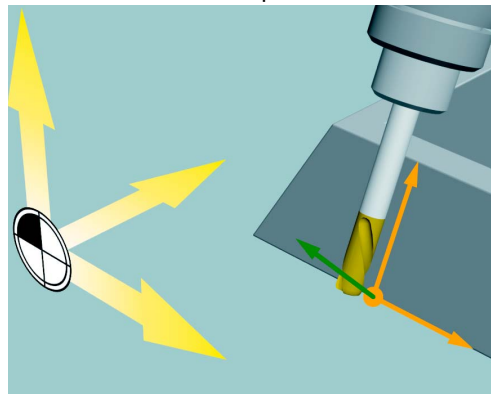
G41/G451

3.8.1 3D-WRK cutter radius compensation for circumferential and face milling

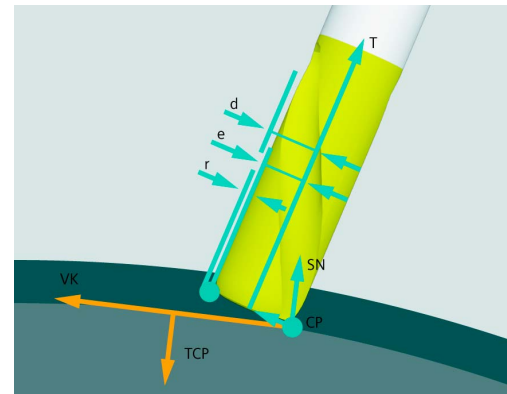
The 3-dimensional case is more complicated. In two dimensions, it is implicitly clear that the tool (standard tool) is aligned perpendicular to the compensation plane and therefore the compensated path is always perpendicular to the path tangent in the X-Y plane. In three dimensions, the tool orientation 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 for structural parts, and on the other hand, the face milling for free-form surfaces that only requires a constant compensation value for the Z axis in the 2-dimensional case.

3-dimensional cutter radius compensation

CUT3DC/CUT3DCD circumferential milling
Green: Direction of compensation



CUT3DF/CUT3DFD face milling



Because in practice, suitable standard tools are often not available, a tool that does not deviate too much from a standard tool (up to 10%) can be used.

3.8.2 CUT3DC/CUT3DCD 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/CUT3DCD and activated with G41/G42 (left/right).

The properties of the compensations for circumferential milling are compared in the table.

CUT3DC (SINUMERIK 840D sl and powerline)	CUT3DCD (SINUMERIK 840D sl as of V4.7 SP3 HF1)
<ul style="list-style-type: none"> ■ Calculation of the tool compensation in the control/CNC 	<ul style="list-style-type: none"> ■ Calculation of the tool compensation / standard tool in CAD/CAM and calculation of the difference by entering the wear in the control/CNC.
<p>Supplementary conditions</p> <ul style="list-style-type: none"> ■ Can be used only to a limited extent ■ Can be used only to a limited extent with compressor: COMPCAD - Advanced Surface COMPSURF - Top Surface 	<p>Supplementary conditions</p> <ul style="list-style-type: none"> ■ Can be used with compressor: COMPCAD - Advanced Surface COMPSURF - Top Surface ■ The tools should have maximum 5-10% negative wear of the standard tool.

NOTE

We recommend the use of the CUT3DCD function for circumferential milling.

Programming circumferential milling with CUT3DCD

- 3D TRC for circumferential milling referred to a differential tool is activated via the CUT3DCD command. It should be applied if the programmed contour refers to the center-point path of a differential tool, and a tool other than a differential tool is used for machining.
- When calculating the 3D tool radius compensation, only the wear value of the radius of the active tool (\$TC_DP15,\$TC_DP16) and any programmed tool compensation offsets OFFN and TOFFR/TOFFL are taken into account. The basic radius (\$TC_DP6) of the active tool is not taken into account. Additional information, basic length = \$TC_DP3.
- CUT3DCD can be used with various types of **tool orientation (TRAORI active!)**
 - Direction vector
CUT3DCD G41 G01 X145.983 Y99.000 Z-20.876 A3=0.5 B3=0 C3=1
 - Round axis values
CUT3DCD G41 G01 X145.983 Y99.000 Z-20.876 B-26.56 C0
- The CUT3DCD 3D radius compensation is restricted to cylindrical tools.

CUT3DCD programming example

Finishing NC program, 5-axis program

```

N1 G17 G40 G90
N2 CYCLE800()
N3 TRAF00F
N4 T="CUTTER_10"
N5 M6
N6 S15061 M3
N7 CYCLE800(0,"TISCH",100000,27,0,0,0,288.079,0,-10.196,0,0,0,-1,0,1)
N8 G00 X-13.532 Y-24.856 M03
N9 TRAORI
N10 ORIWKS ORIAXES
N11 G54
N12 TOFFL=0
N13 TOFFR=0
N14 CYCLE800()
N15 CYCLE832(0.05,_TOP_SURFACE_SMOOTH_ON+_ORI_SEMIFIN,0.1)
N16 G0 X0 Y0 Z100.0
N17 CUT3DCD
N18 G41
N19 G1 X-5.945 Y-60.546 A3=0.021144 B3=0.341366 C3=0.939693 F2500
N20 Z5.502 A3=0.021144 B3=0.341366 C3=0.939693
N21 G1 Z0.502 A3=0.021144 B3=0.341366 C3=0.939693 F2500
N22 G1 X-5.743 Y-59.698 A3=0.021144 B3=0.341366 C3=0.939693 F500
.....
N17525 g40 X0 Y0
N17526 CYCLE832(0,_OFF,1)
N17528 TRAF00F
N17529 M17

```

3.8.3 CUT3DF/CUT3DFD 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 by programming the normal vectors A4, B4, C4 (start of block) and A5, B5, C5 (end of block - recommended programming).

The compensation is defined with CUT3DF/CUT3DFD and activated with G41/G42, whereby there is no difference between G41 and G42 in this case.

The activation must be performed in a linear block (G0/G1). Switching off with G40 can be made in a linear block or in a separate NC block. A change of the 3D TRC variant with active tool radius compensation is ignored without alarm.

Before switching on CUT3DFD with G41/G42, Top Surface / COMPSURF / CYCLE832 must be activated, not immediately before tooling, but rather approximately **1000 x CTOL = path** before the tooling or G41/G42 programmed, e.g. 1000 x 0.01 mm = 10 mm.

For traversing motions in the orientation direction of the tool, alarm 10769 is triggered and machining stopped. As workaround, a G40 and then a G41/G42 can be programmed again for the machining.

The properties of the compensations for face milling are compared in the table.

CUT3DF (SINUMERIK 840D sl and powerline)	CUT3DFD (SINUMERIK 840D sl as of V4.8 SP2)
<ul style="list-style-type: none"> ■ Calculation of the tool compensation in the control/CNC 	<ul style="list-style-type: none"> ■ Calculation of the tool compensation / standard tool in CAD/CAM and calculation of the difference by entering the wear in the control/CNC.
<p>Supplementary conditions</p> <ul style="list-style-type: none"> ■ Can be used with G64/G642/G645 ■ Can be used only to a limited extent with compressor: COMPCAD - Advanced Surface COMPSURF - Top Surface 	<p>Supplementary conditions</p> <ul style="list-style-type: none"> ■ Can be used only with compressor: COMPSURF - Top Surface ■ The tools should have maximum 5-10% negative wear of the standard tool.

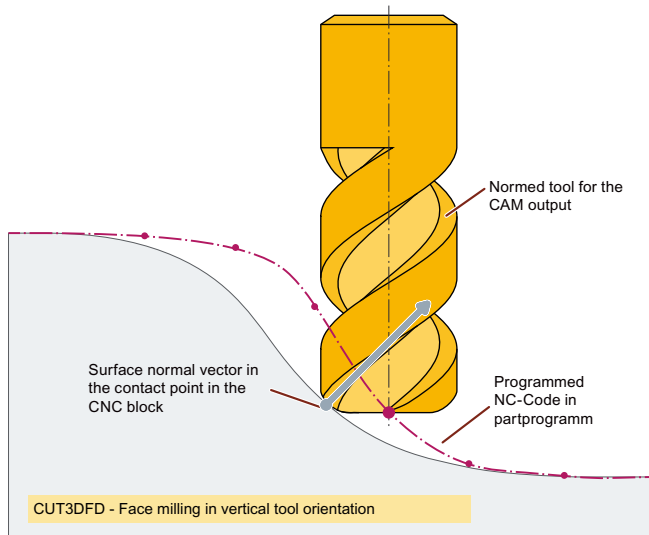
NOTE

We recommend the use of the CUT3DFD function for face milling.

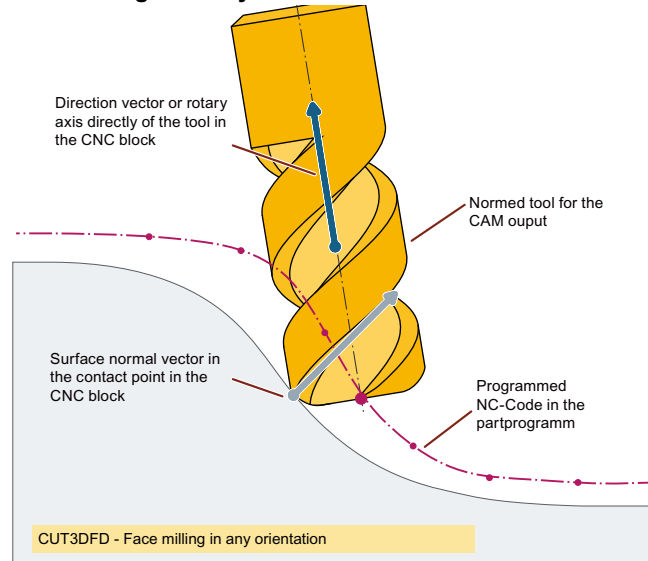
Programming face milling with CUT3DFD

- CUT3DFD can be used without additional tool orientation as **3-axis program**, purely based on the **surface normal vector**.
 - CUT3DFD G41 G01 X145.983 Y99.000 Z-20.876 A5=0.643 B5=0 C5=0.357
- CUT3DFD can be used as 5-axis program with various types of tool orientation (**TRAORI active!**)
 - **Direction vector**
CUT3DFD G41 G01 X145.983 Y99.000 Z-20.876 A3=0.5 B3=0 C3=1 A5=0.643 B5=0 C5=0.357
 - **Round axis values**
CUT3DFD G41 G01 X145.983 Y99.000 Z-20.876 B-26.56 C0 A5=0.643 B5=0 C5=0.357
- The CUT3DFD 3D radius compensation acts for all tool types for machining free-form surfaces.

Face milling with perpendicular tool orientation



Face milling with any tool orientation

**CUT3DFD programming example****Finishing NC program, 3-axis program**

```

N1 G17 G40 G90
N2 T="BALL_D16" D1
N3 M6
N4 S15061 M3
N5 G54
N6 TOFFL=0
N7 TOFFR=0
N8 CYCLE832(0.01, _TOP_SURFACE_SMOOTH_ON+_FINISH,1)
N9 R2=5000
N10 R3=5000
N11 R4=35000
N12 G0 X-11.0000 Y0.0000 Z74.0000
N13 M3
N14 G0 X-11.0000 Y0.0000 Z-26.0000
N15 G41 CUT3DFD
N16 G1 X-10.2235 Z-25.8978 A5=0.0000000 B5=0.0000000 C5=1.0000000 F=R2
N17 G1 X-9.5000 Z-25.5981 A5=0.0000000 B5=0.0000000 C5=1.0000000
N18 G1 X-8.8787 Z-25.1213 A5=0.0000000 B5=0.0000000 C5=1.0000000
N19 G1 X-8.4019 Z-24.5000 A5=0.0000000 B5=0.0000000 C5=1.0000000
N20 G1 X-8.1022 Z-23.7765 A5=0.0000000 B5=0.0000000 C5=1.0000000
N21 G1 X-8.0000 Z-23.0000 A5=-1.0000000 B5=0.0000000 C5=0.0000000
N22 G1 X-8.0000 Z-18.0000 A5=-1.0000000 B5=0.0000000 C5=0.0000000 F=R3
N23 G1 X-7.9904 Z-17.6075 A5=-0.9987955 B5=0.0000000 C5=0.0490677
.....
N38 G01 G40 X-20.168 Y7.651 Z23.355
N39 G01 X-17.453 Y8.537 Z39.237
N40 G00 X-14.929 Y9.361 Z54.001
N41 CYCLE832(0, _OFF,1)
N42 TRAFOOF
N43 M30

```

CUT3DFD programming example **Finishing NC program, 5-axis program**

```

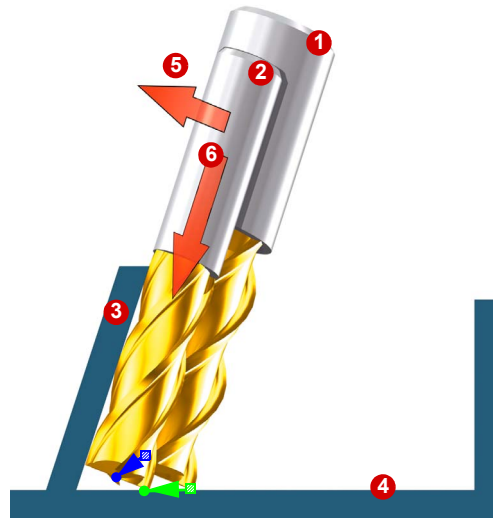
N1 G17 G40 G90
N2 CYCLE800()
N3 TRAFOOF
N4 T="BALL_D6" D1
N5 M6
N6 S15061 M3
N7 CYCLE800(0,"TABLE",100000,27,0,0,0,288.079,0,-10.196,0,0,0,-1,0,1)
N8 G00 X-13.532 Y-24.856 M03
N9 TRAORI
N10 ORIWKS ORIAXES
N11 G54
N12 TOFFL=0
N13 TOFFR=0
N14 CYCLE800()
N15 CYCLE832(0.05,_TOP_SURFACE_SMOOTH_ON+_ORI_SEMIFIN,1)
N16 G00 X-18.974 Y8.041 Z54.001 A-10.196 C=DC(+288.079)
N17 G00 X-19.647 Y7.821 Z50.064 A-10.196 C=DC(+288.079)
N18 G01 X-20.168 Y7.651 Z47.016 F1485.4
N19 G41 CUT3DFD
N20 G01 X-20.505 Y7.541 Z45.047 A3=+.168277208 B3=+.054933772 C3=+.984207835
A5=+1. B5=-.000003008 C5=+.000000718
N21 G01 X-20.506 Y7.365 Z45.047 A3=+.168631651 B3=+.053835903 C3=+.984207835
A5=+1. B5=+.000004199 C5=+.000000719
N22 G01 X-20.507 Y7.194 Z45.047 A3=+.168973327 B3=+.052753699 C3=+.984207835
A5=+1. B5=-.000007571 C5=+.000000721
N23 G01 X-20.508 Y7.025 Z45.047 A3=+.169308954 B3=+.051666392 C3=+.984207835
A5=+1. B5=-.000002005 C5=+.000000722
.....
N38 G01 G40 X-20.168 Y7.651 Z23.355
N39 G01 X-17.453 Y8.537 Z39.237
N40 G00 X-14.929 Y9.361 Z54.001
N41 CYCLE832(0,_OFF,1)
N42 TRAFOOF
N43 M30

```

3.8.4 Circumferential milling with limiting surfaces (CUT3DCC/CUT3DCCD)

A combination of these two methods is used when machining special structural components in the aviation industry. Pockets are machined where the wall inclination forms an acute angle to the floor. The wall is machined with 5-axis 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 recognizes the fact that it is not just a question of compensating in the machining surface direction ⑤, but also of making an adjustment 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.

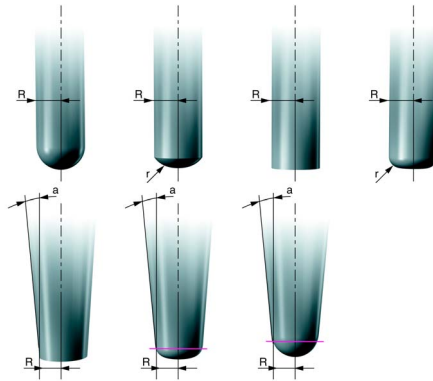
This compensation variant is called **Circumferential milling with limiting surfaces**. It is selected via the code CUT3DCC or CUT3DCCD and activated with G41/42 as for CUT3DC.

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.

3.8.5 Overview of the tool types for the 3D compensations.



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.

3.8.6 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 1/2D 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.
------------	---

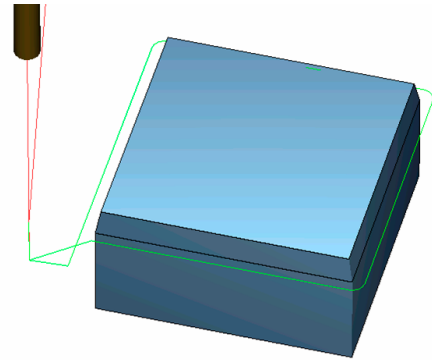
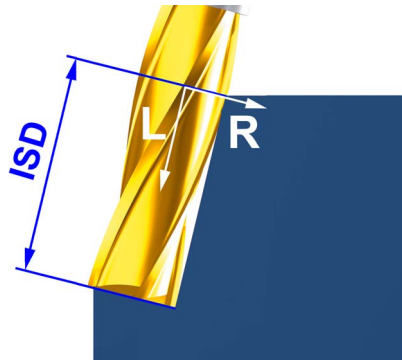
ORID	No changes in orientation in inserted circular blocks at external corners. Orientation motion is performed in the linear blocks.
-------------	--

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.
CUT3DFD	Activating the 3D radius compensation based on a differential tool for face milling.
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.

3.8.7 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.

CUT3DC example



```

N100 CYCLE800(2,"TABLE",200000,57,0,0,0,0,0,0,0,0,0,0,1,,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

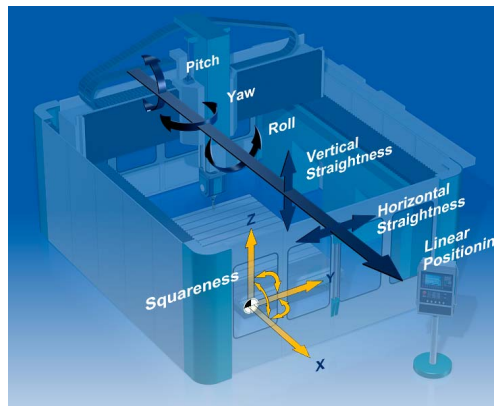
```

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.

Types of VCS compensation



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

As regards the VCS commissioning process and machine calibration, please contact your machine manufacturer.



3.10 VNCK - virtual machine

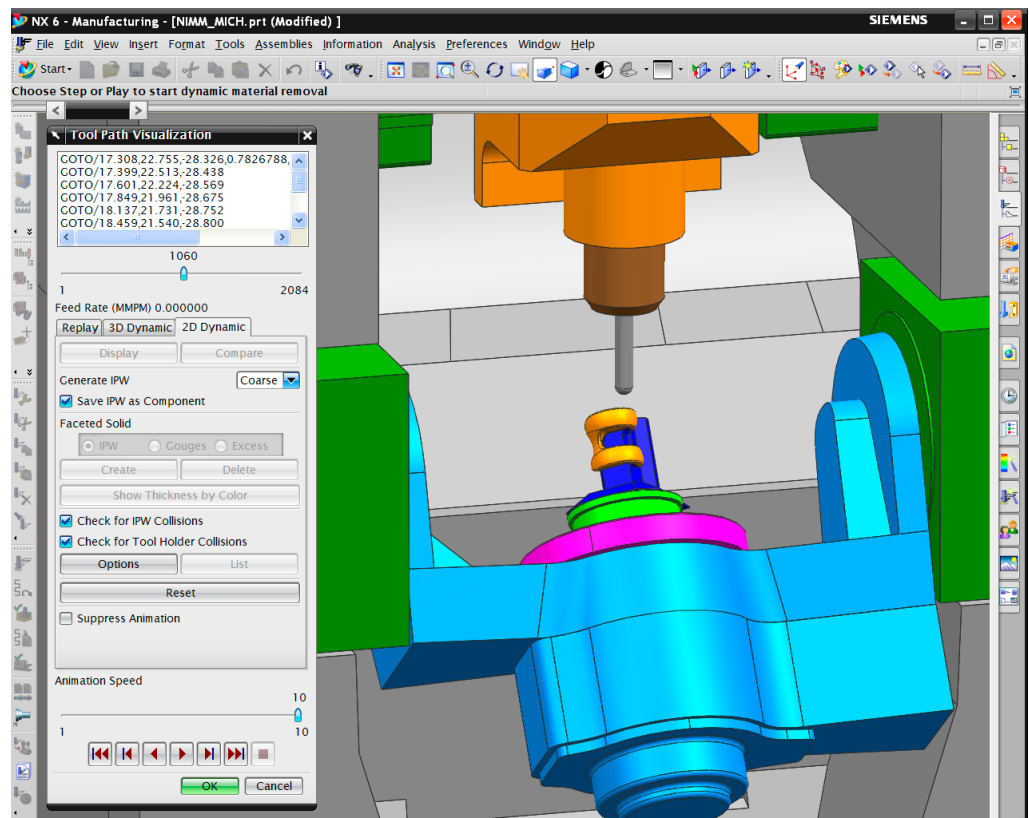
Due to the level of complexity and machining time associated with 5-axis milled workpieces, a deliberate effort is made to check that the programs are free of errors before actual production commences. To ensure that the data obtained is as realistic as possible, virtual models of the machine and control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

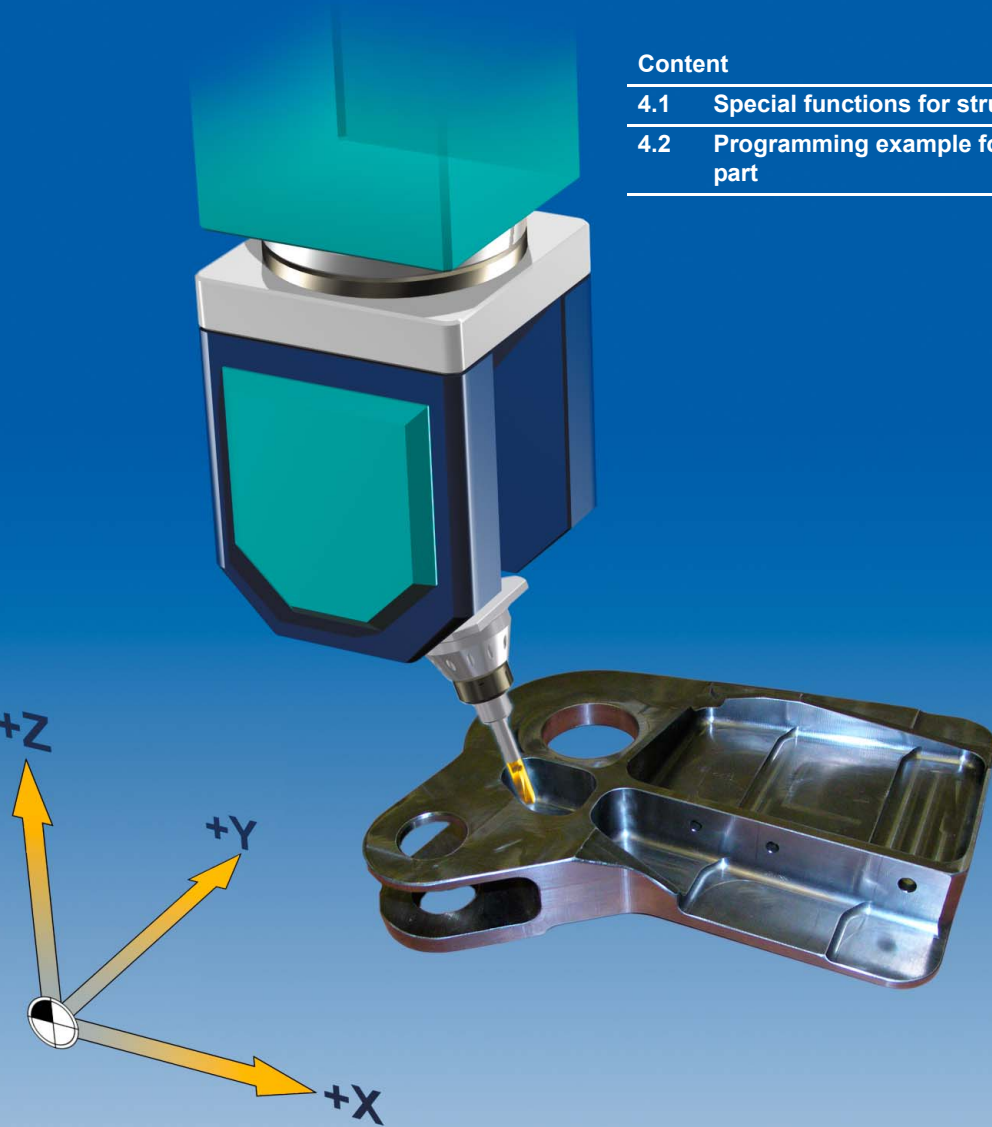
Using the virtual machine in conjunction with the SIEMENS VNCK offers many advantages:

- 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, clamping 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

Content	Page
4.1 Special functions for structural parts	134
4.2 Programming example for the pocket on a structural part	135



4.1 Special functions for structural parts

Structural parts are frequently used within the aviation 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.

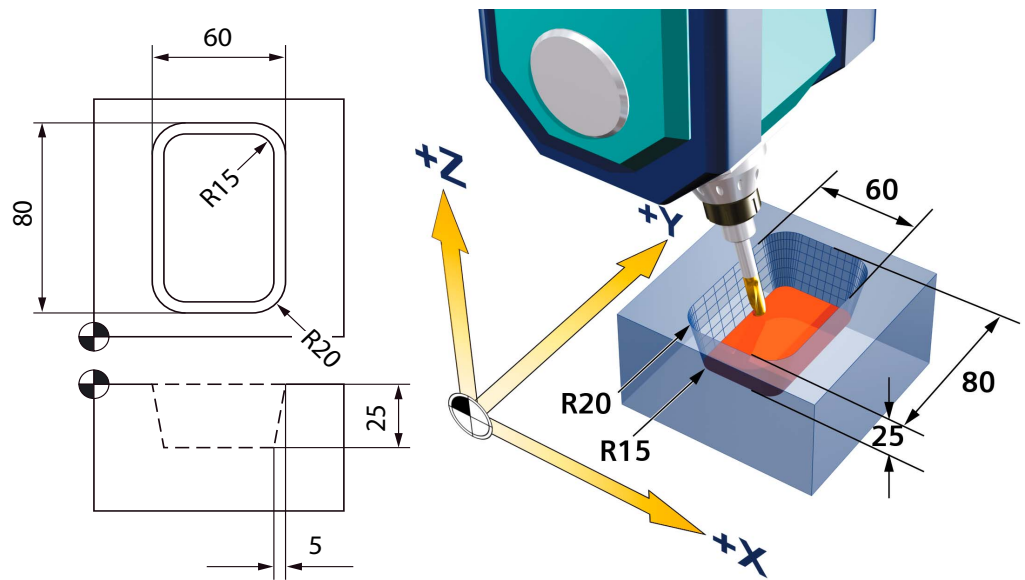
Key functions for machining structural parts:

- VNCK,
because the high level of material usage requires a form of simulation that resembles the control exactly.
- VCS,
because maximum accuracy is required in conjunction with extremely large workpieces.
- CYCLE832
Easy preselection of the key machine settings for roughing and finishing.
- ORIVECT,
because this is the only kind of orientation interpolation that ensures the necessary precision when creating inclined walls.
- 3D tool radius compensation,
because this also 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.

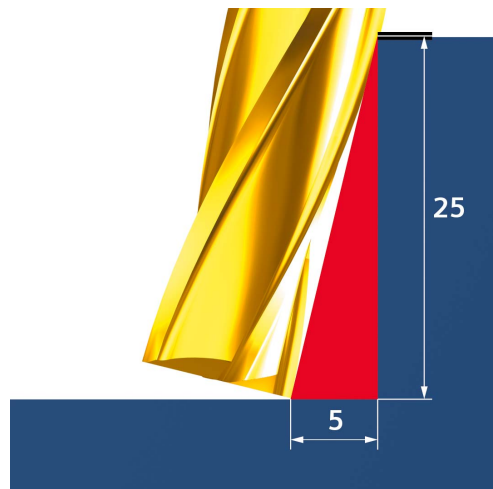


4.2 Programming example for the pocket on a structural part

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 based on 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

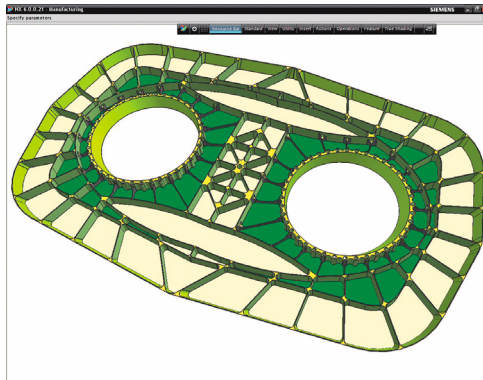
In the example, the pocket is rough machined and the pocket walls machined via circumferential milling with large circle interpolation ORIVECT and ORICONxx. Programming is with RPY angles A2=,B2=,C2= (upper profile) with automatic angle calculation and insertion depth ISD

```
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",200000,57,0,0,0,0,0,0,0,0,0,0,1,,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 ; zero offset 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
deactivated
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.

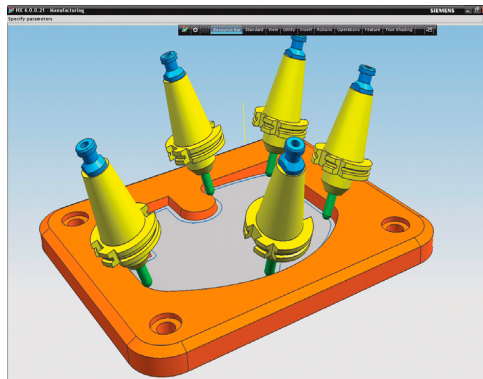
Support for ribs and pockets



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

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

Tools set at an angle during profile milling

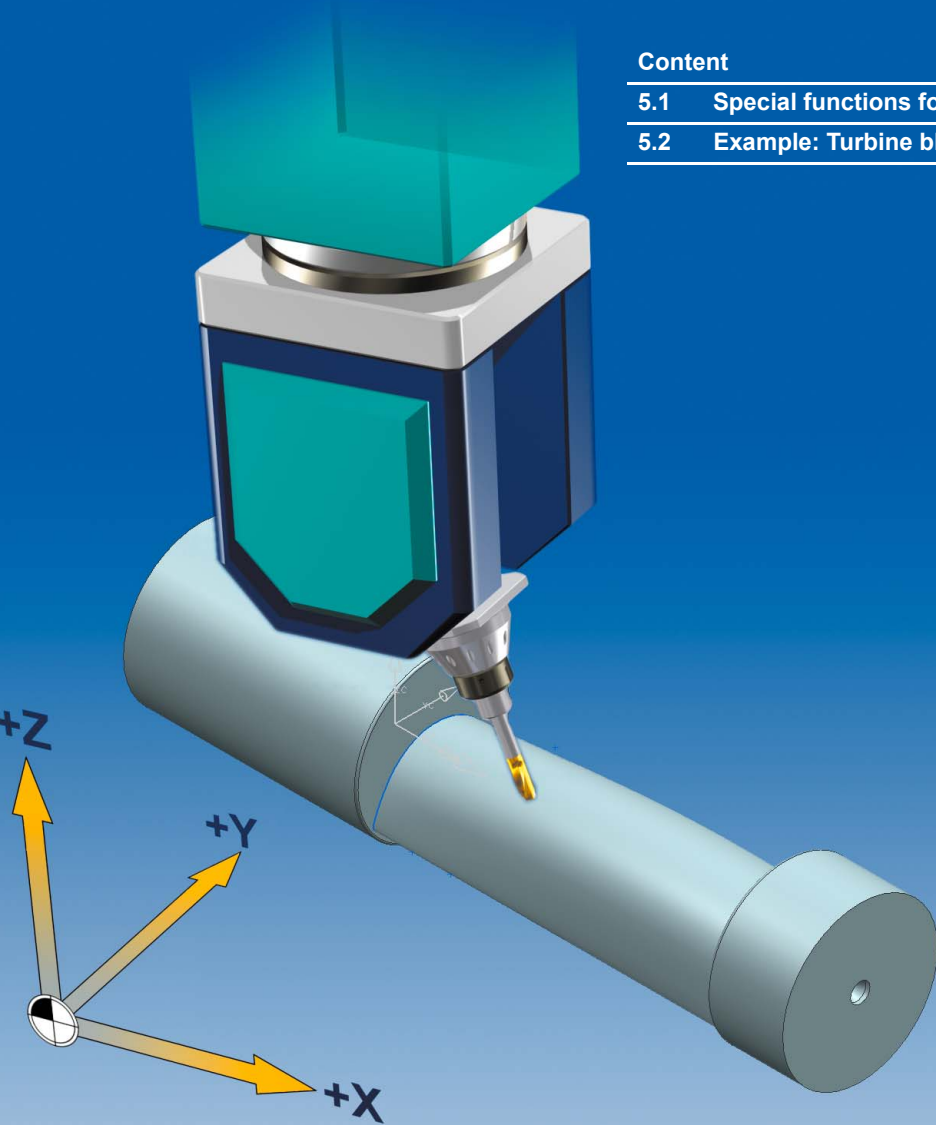


Automatic profile milling with variable axes makes programming faster. CAM systems offer a range of automatic tool position and tool axis settings for 5-axis milling on walls and other profiles.

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

Engine and turbine components

Content	Page
5.1 Special functions for engine and turbine components	140
5.2 Example: Turbine blade	141



5.1 Special functions for engine and turbine components

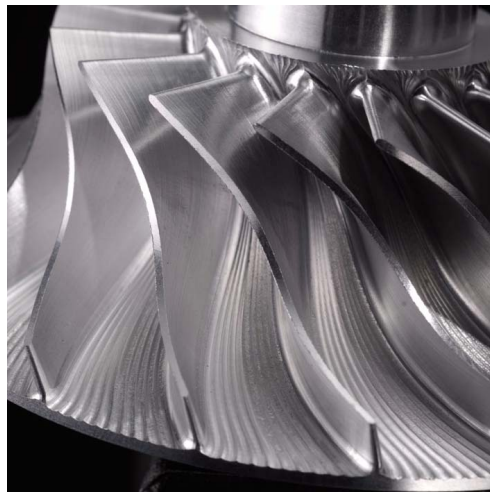
Turbine impellers or blades are subject to the toughest of requirements in terms of surface quality and contour accuracy. As a result, the NC is expected to process large data volumes within a very short period. Even the smallest jumps in deceleration and acceleration can result in surface defects (e.g.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

Key functions for machining parts for engine and turbine technology:

- High Speed Settings CYCLE832, because optimum data compression within the tolerance band combined with feedforward control 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.

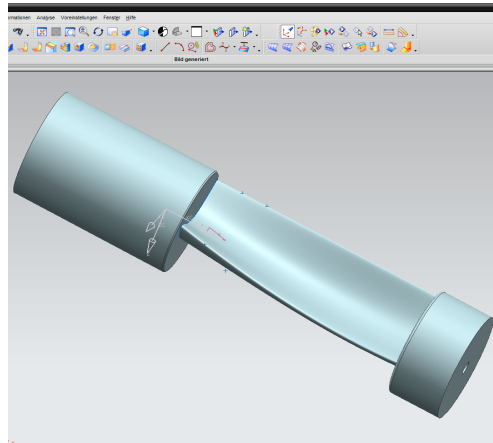
Impeller



5.2 Example: Turbine blade

This example relates to the milling of a turbine blade. Modeled is performed in a CAD/CAM system.

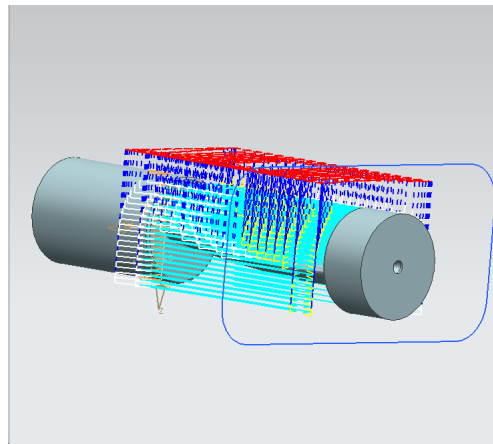
Turbine blade



At the modeling stage, it is essential to ensure that the machining strategies take account of the clamping 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 clamping device.

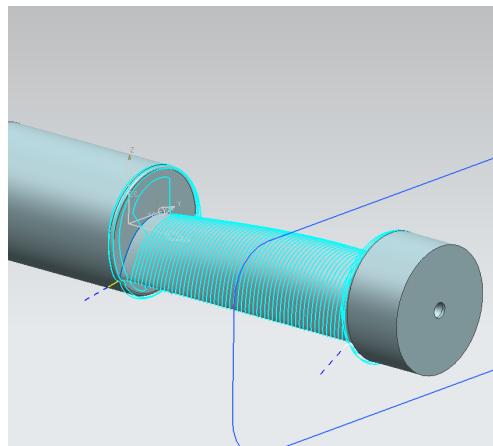
Plane roughing



Roughing was performed 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.

Example program code

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

Example start program

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 GOTO 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 TRAORI ;Transformation On
N150 ORIWKS ORIAXES
N155 CYCLE832(0.05,3,1) ; High Speed Settings (DYNROUGH)
N160 EXTCALL "ROUGHING_1"
N165 CYCLE832() ; High Speed Settings deselection
N170. TRAFOFF ; Transformation Off
..

```

```

N175 OP_5:
N180 T="SPHERE_D6"
N185 M6
N190 M25 ;Release C-axis clamping (OEM-specific)
N195 M27 ;Release B-axis clamping (OEM-specific)
N200 S15000 M3
N205 R1=2000 ;Milling feed
N210 R2=1000 ;Approach feed
N215 G54 G0 X0 Y0 C0 A0 D1
N220 G0 Z100
N225 TRAORI ;Transformation On
N230 ORIWKS ORIAXES
N220 CYCLE832(0.01,1,1) ;High Speed Settings (DYNFINISH)
N225 ORISON
N230 OTOL=0.5
N235 EXTCALL "FINISH_1"
N240 CYCLE832() ; High Speed Settings deselection
N245 TRAFOOF
N250 SUPA G0 Z0 D0 ; Max. retraction in Z (MCS)
N255 SUPA G0 X0 Y0 D1 ; Max. retraction in X and Y (MCS)
N260 M30

```

Example subprogram FINISH_1

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

```

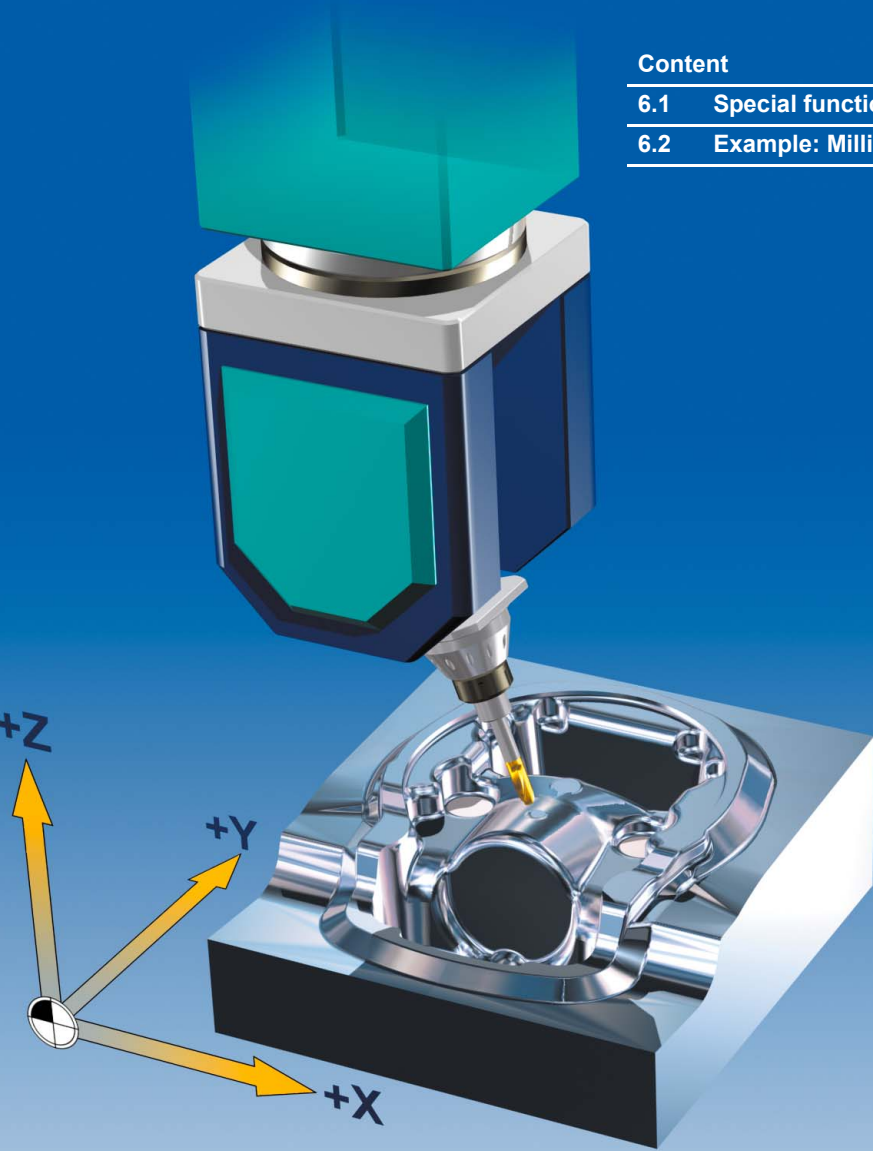
N100 MSG("FINISHING")
N105 ;Positioning movement
N110 G0 X-52.73538 Y-17.80536 Z31.9 A3=-.39485858 B3=.49800333 C3=.77206177
N115 M3
N120 ; Approach movement
N125 G1 Z13.37361 A3=-.39485858 B3=.49800333 C3=.77206177 F=R2
N130 X-47.99708 Y-23.7814 Z4.10887 A3=-.39485858 B3=.49800333 C3=.77206177
N135 ; Start motion
N140 G1 X-47.84399 Y-23.76942 Z3.80953 A3=-.39485858 B3=.49800333
N145 C3=.77206177 M8 F=R1
N150 X-47.69248 Y-23.71986 Z3.51328 A3=-.39485858 B3=.49800333 C3=.77206177
... ..

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 ;Return 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 SUPA G0 Z0.0 D0 ; Max. return in Z (MCS)
N6600 SUPA G0 X0.0 Y0.0 A0.0 C0.0 D1 ; Max. return in XY (MCS)
N6605 ; and move rotary axes to basic position
N6610 M17 ; End of subprogram

```


Complex free-form surfaces

Content	Page
6.1 Special functions for free-form surfaces	146
6.2 Example: Milling a manta ray	147



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 smoothing and jerk limitation, which anticipates a configurable number of traversing blocks so that the machining velocity can be optimized.

In addition, feedforward control 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.

Key functions for machining free-form surfaces:

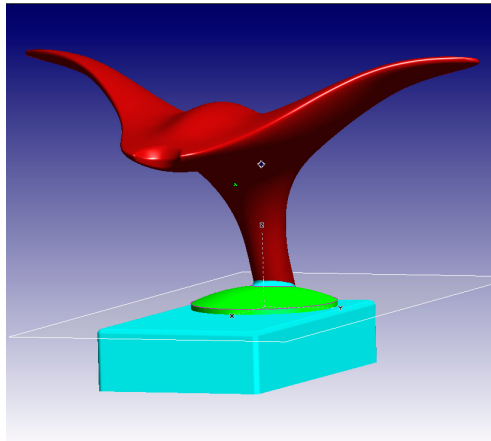
- High Speed Settings CYCLE832,
as optimum data compression within the tolerance band, combined with feedforward control and jerk limitation, can be set for the required surface finish without contour deviation.
- TRAORI,
as it has an integrated 5-axis transformation feature for all kinds of machine kinematics, enabling the tool to be perfectly oriented in relation to the surface so that no contour or surface defects can occur.
- VCS,
because 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. Modeled is performed in a CAD/CAM system.

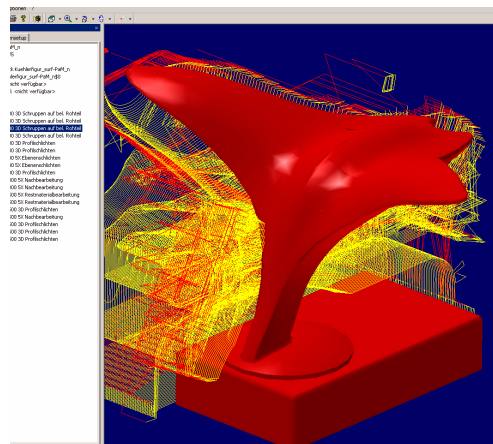
Manta ray in the CAM system



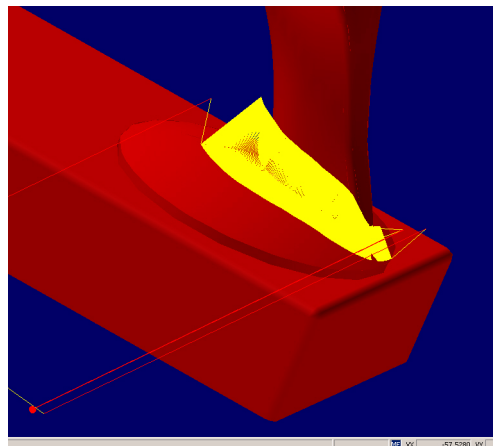
The manta ray is modeled as a free-form surface in the CAM system using a digitized scatterplot. The machining strategies included 3D plane roughing and several 3-axis and 5-axis semi-finishing and finishing operations.

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.

Example program code

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.

Example start program

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 zero offset
N105  ORIWKS ORIAXES       ;Workpiece coordinate system, axis
                                interpolation
N110  GOTOF _ROUGH_01      ;Subprogram jump label for calling roughing with
                                ;the ROUGH_01.MPF program.
                                ;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.
                                ;Program stop
N250  M00
N280  ...
N360  _FINISH_05:         ;
N370  EXTCALL "FINISH_05" ;Subprogram call for the last finishing program
N380  STOPRE              ;
N390  M00                 ;
N400  M30                 ;End of program

```

Example roughing subprogram: 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.

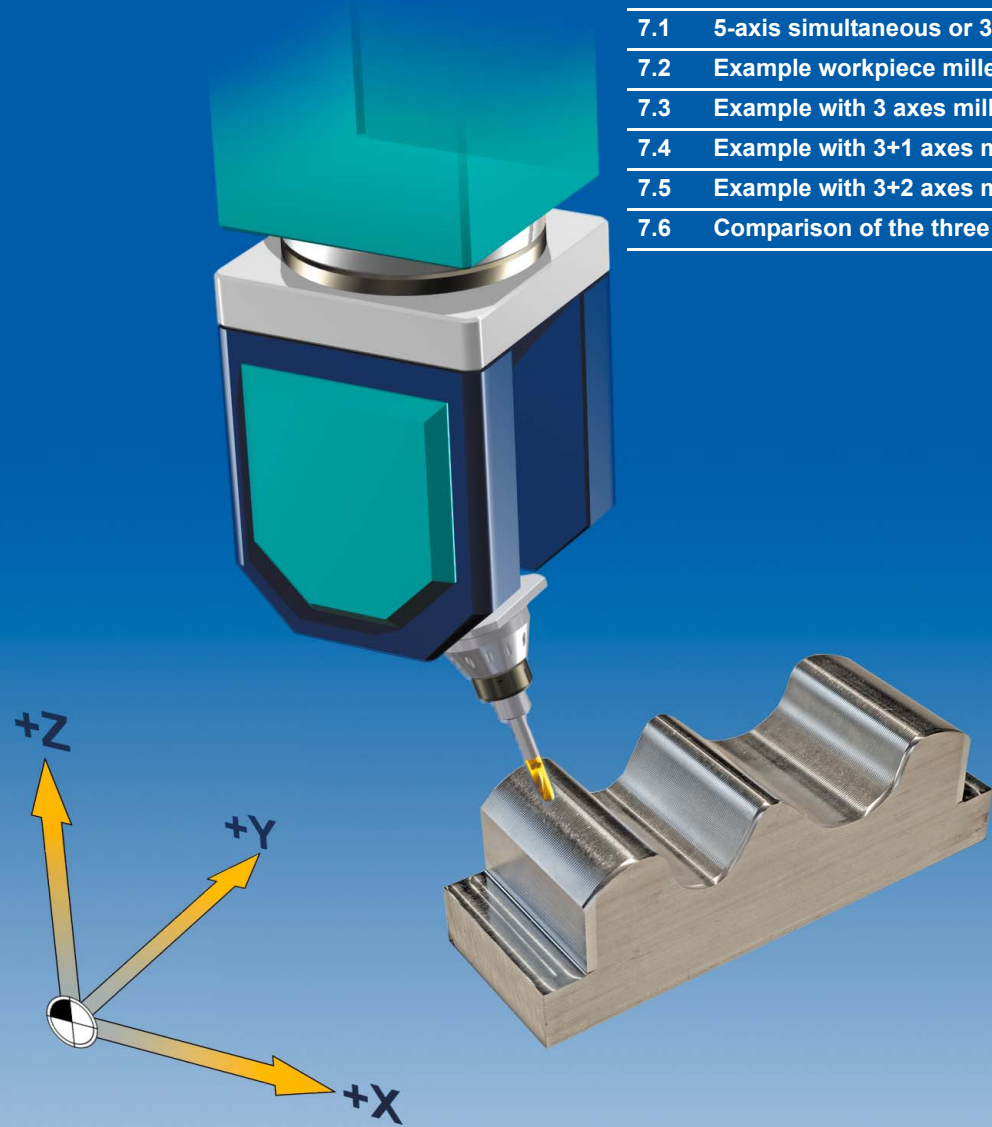
```

N100 ; TOOL ;Tool specification in the form of a comment
N110 ; T1 radius milling ;Tool dimensions
      tool D32 R2
N120 G90 G17 G54 ;Absolute dimension specification, select
      ;working plane and zero offset
N130 TRAFOOF ;Deactivate all active transformations and
      frames
N140 CYCLE800(1,"TC1",0,57,0,0,0,0,0,0,0,-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 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,-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 original 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 as of 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 ;Return in Z
N4590 CYCLE800(1,"TC1",0,57,0,0,0,0,0,0,0,-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 between 3, 3+2 and 5 axes

Content	Page
7.1 5-axis simultaneous or 3+2-axes	153
7.2 Example workpiece milled by 3 to 3+2 axes	153
7.3 Example with 3 axes milled	155
7.4 Example with 3+1 axes milled	156
7.5 Example with 3+2 axes milled	157
7.6 Comparison of the three variants	158



7

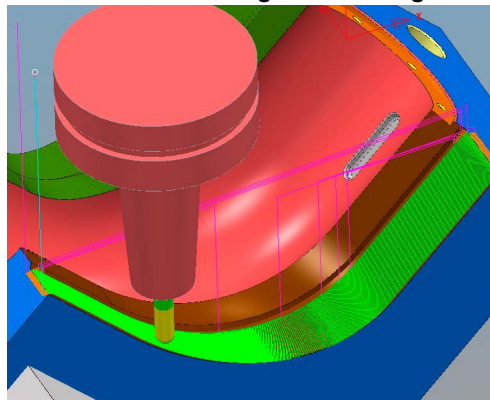
7.1 5-axis simultaneous or 3+2-axes

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:

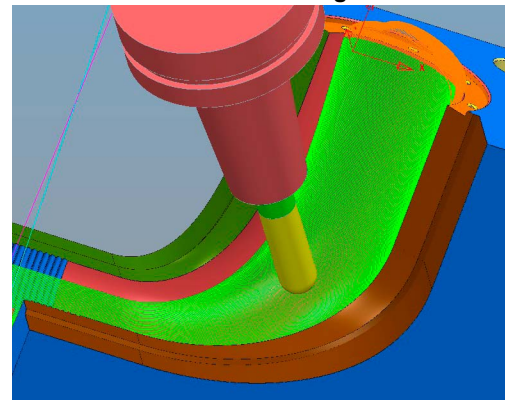
- ▶ As much as possible 3-, 3+1- and 3+2-axis (set at an angle) roughing/semi-finishing
- ▶ 5-axis simultaneous milling only for the finishing

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

3+2-axis semi-finishing set at an angle



5-axis simultaneous finishing



3+2-axis machining

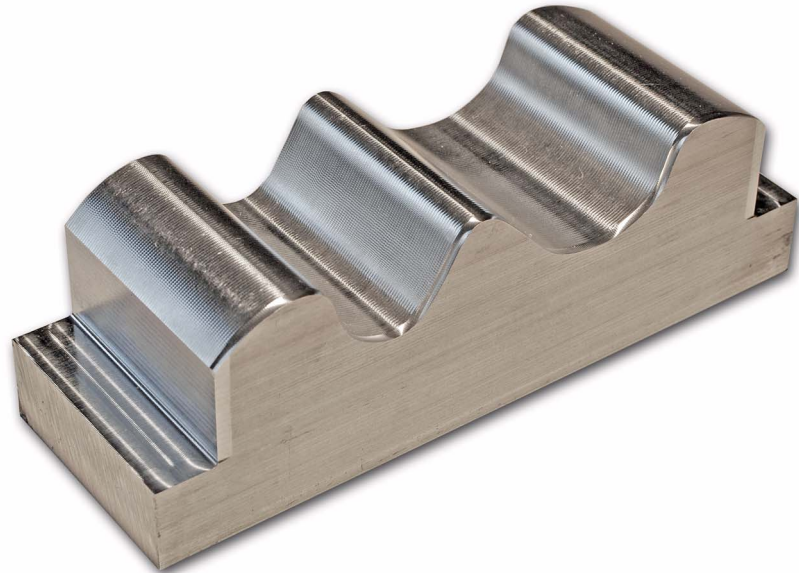
Advantages	Disadvantages
<ul style="list-style-type: none"> ■ Less programming work ■ No limitation of the dynamic response as only linear axes are used ■ Greater stiffness of the moving axes with 3+2-axis machining (depending on the machine concept). As a result, longer tool life and higher surface quality 	<ul style="list-style-type: none"> ■ Limitation of the workpiece 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
<ul style="list-style-type: none"> ■ 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 	<ul style="list-style-type: none"> ■ 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 Example workpiece milled by 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.

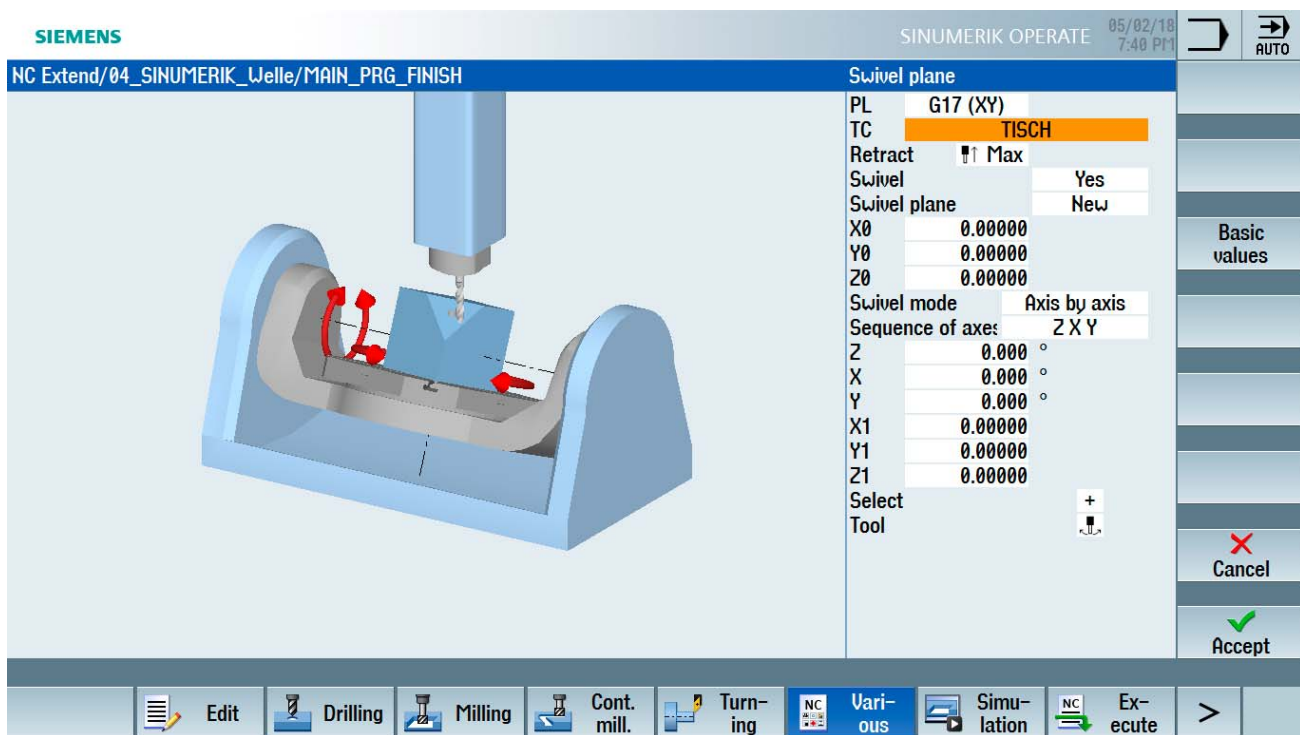
The distance between the individual milling paths in the finishing programs has been set to be relatively large so that the uniformity of the individual paths can be evaluated more accurately and errors can be seen.

Pre-positioning of the milling cutter 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 ; Deselect CYCLE800, move to basic
      ,0,0,0,0,1,,1) ; position
N105 ... ;
```



Basic position of 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 with 3 axes milled

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

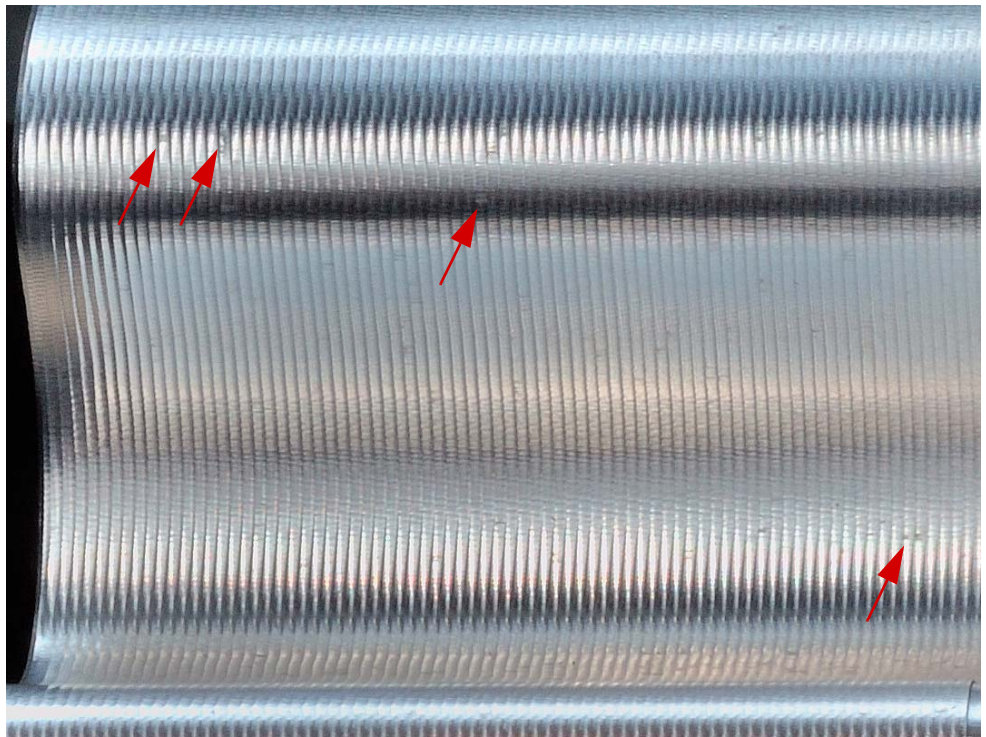
3-axis finishing

```

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 ; Begin machining
F=R1
N210 G01 X-3.9224 Y-5.503 Z-11.7235 F=R2 ;
N290 ... ;
N4580 G00 X102.5039 Y46.3472 Z10 ;
N4600 CYCLE832() ; Deselect High Speed Settings
N4610 M5 ;
N4620 M30 ; End of program

```

Image excerpt of the upper section of the shaft



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 with 3+1 axes milled

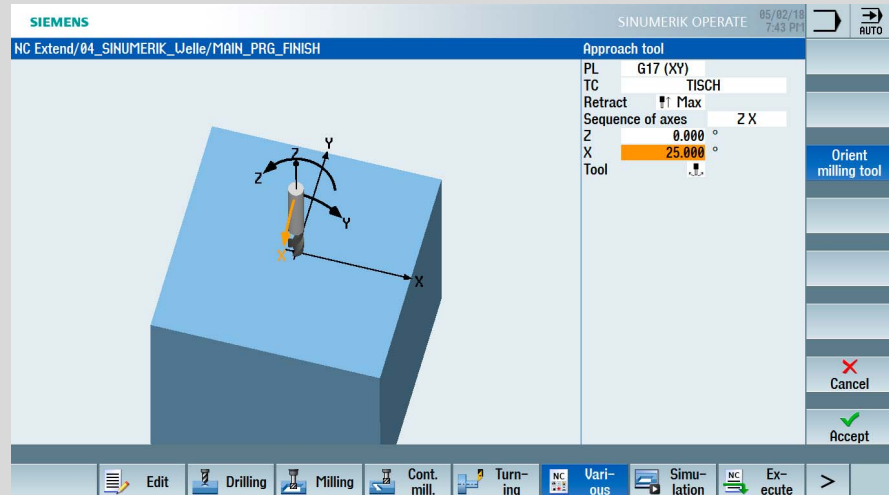
In the second example, the X axis has been at an angle of 25°.

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 about the X axis by 25°
-1,100,1)

```



```

N190 CYCLE832(0.005,1,1) ; High Speed Settings finishing with
; tolerance 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() ; Deselect High Speed Settings
N4610 CYCLE800(4,"TABLE",110000,39,0,0,0, ; Swivel to basic position
0,0,0,0,0,0,.,1)
N4620 CYCLE800() ; Deselect swivel
N4630 M5 ;
N4640 M30 ; End of program

```

7.5 Example with 3+2 axes milled

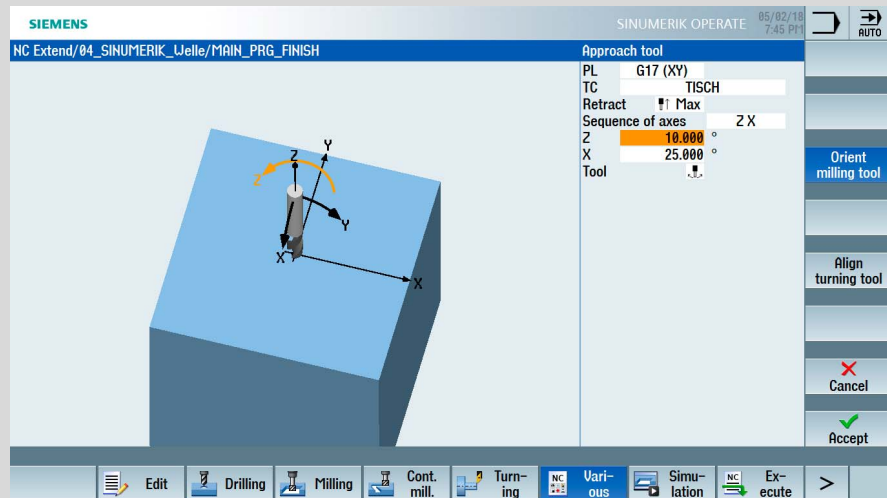
In the third example, the X axis has been set at an angle of 25° and the Z axis has been set at an angle of 10°.

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, ; Swivel to basic position
0,0,0,0,0,0,1,,1)
N180 CYCLE800(4,"TABLE",101,39,,,,10,25,,, ; Swiveled about the X axis by 25° and
,,-1,100,1) about the Z axis by 10°

```



```

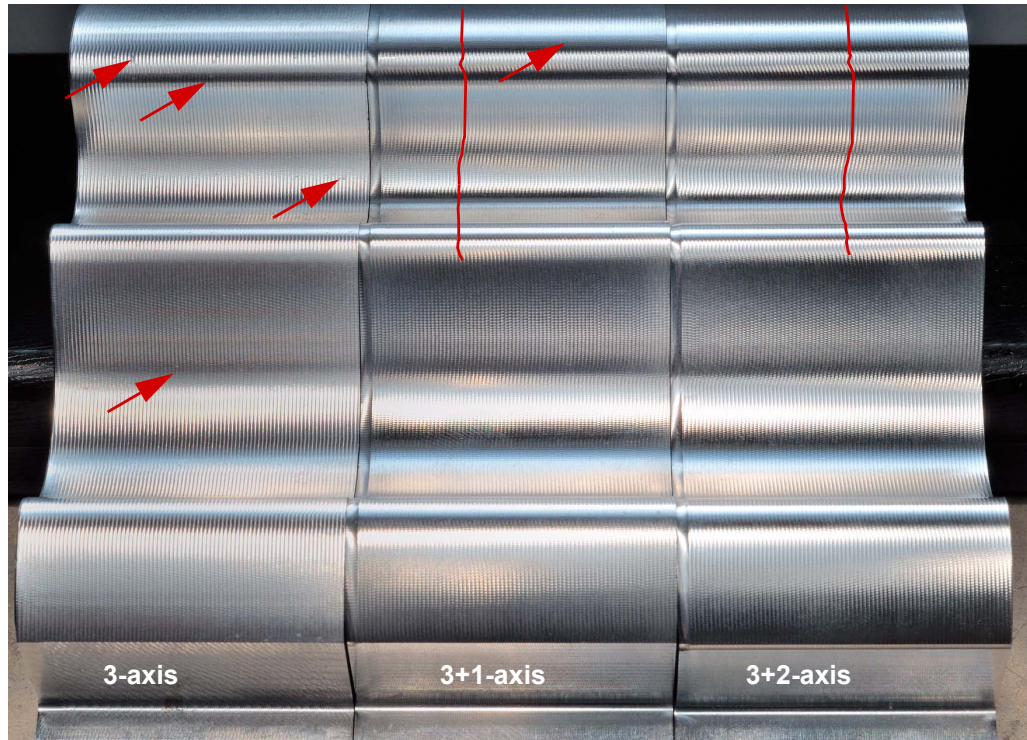
N190 CYCLE832(0.005,1,1) ; High Speed Settings finishing with
; tolerance 0.005
N200 G00 X-3.9247 Y-5.5063 Z10 ; Approach with G0
N210 G00 X-3.9247 Y-5.5063 Z-6.7226 ;
N220 G01 X-3.9247 Y-5.5063 Z-11.7226 ; Begin machining
F=R1
N230 G01 X-3.9224 Y-5.503 Z-11.7235 F=R2 ;
N290 ... ;
N4580 G00 X102.5039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

```

7.6 Comparison of the three variants

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 variants



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

Content	Page
8.1 Overview of higher-order functions	160
8.2 Further information/documentation	170
8.3 Index	172

8

8.1 Overview of higher-order functions

Higher-order functions of the SINUMERIK control 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.

8.1.1 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=	
AR=	End point in polar coordinates, polar radius, also in the case of linear interpolation
	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 Activation of B spline
SD=... B spline order (max. 3)
PL=... Interval length (node vector), "Non-uniformity"
PW=... Weightings, i.e. denominator of rational B splines with polynomial representation

Example

N20 BSPLINE X... Y... SD=... PL=... PW=...

POLY Activation of polynomial interpolation, B spline representation in polynomial format
SD=... B spline order (max. 5!! -> different from BSPLINE)
PL= ... Interval length (node vector), "Non-uniformity"

Syntax

PO[axis] = (block end position, a2 (quadratic coefficient), a3 (cubic coefficient), a4, a5) -> numerator polynomials
 PO[] = (Nblock end, 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.

8.1.2 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)
	Smoothing
G641	ADIS = ... smoothing distance ADISPOS =... smoothing distance for G0, constant velocity
G642	Smoothing with single-axis tolerances or ADIS, ADISPOS via intermediate blocks, constant acceleration
G645	Path control mode with smoothing of corners and tangential block transitions while adhering to defined tolerances
G60, G64, G645	G code group 10

Velocity programming

	Conventional block-by-block (non-modal) velocity programming in
G94	inches/min or mm/min
G93	Inverse time
G95	inches, mm per spindle revolution
G96	Constant cutting rate
	Programming of velocity/feedrate profiles
	To permit flexible definition of the feedrate characteristic, feedrate programming according to DIN 66025 has been extended by means of 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 feedrate value is defined via the block's path and subsequently applies as a modal value.
FLIN	Linear path velocity profile: The feedrate value is approached linearly using the path, from the current value at the start of the block right through to the end of the block; it then applies as a modal value.
FCUB	Cubic path velocity profile: The F values that have been programmed block-by-block are combined (in relation to the end of the block) by means of 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 most recently programmed F value will be used for this purpose.
F=FPO(..)	Path velocity profile via polynomial: The F address describes the feedrate characteristic on the basis of a polynomial, from the current value right through to the end of the block. The end value is valid thereafter as a modal value.
endfeed	Feedrate at end of block
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 limitation
BRISK	Acceleration limitation

Feedforward control

FFWON	Feedforward control on
FWOF	Feedforward control off

8.1.3 5-axis functionality

Transformation

TRAORI	Activation of transformation 1
TRAORI(1)	Activation of transformation 1
TRAORI(2)	Activation of transformation 2
TRAORI(1, ..., ..., ...)	Activation of transformation 1, generic transformation, additional 3 parameters for basic orientation vector
TRAORI(2, ..., ..., ...)	Activation of transformation 2, generic transformation, additional 3 parameters for basic orientation vector
TRAFOON(<Name>)	Activation of TRAORI when set via kinematic chain
TRAFOOF	Deactivation of transformation

Orientation programming

ORIEULER	Orientation programming on the basis of Euler angles (default)
ORIRPY	Orientation programming on the basis of RPY angles
	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 have the following significance:

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 cone 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: GROOVE=.

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	<p>Interpolation on a peripheral surface of the cone with tangential transition Also required: A3=... B3=... C3=... or XH=..., YH=..., ZH=... End orientation</p> <p>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.</p>
ORICURVE	<p>Orientation interpolation with specification of the tool tip motion and that of a second point on the tool.</p> <p>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)</p> <p>If the BSPLINE or POLY additional information is omitted, straightforward linear interpolation will be performed accordingly between the start and the end orientation.</p>
ORISON	<p>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.</p>
ORISOF	<p>Smoothing of the orientation characteristic OFF.</p>

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 1/2D 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	3D TRC for circumferential milling referred to a differential tool (only when 5-axis transformation is active)
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 the Z direction of the coordinate system currently defined via the frame
CUT3DFD	3D TRC for face milling with orientation change referred to a differential tool (only with active 5-axis transformation)
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 the currently active frame
ROT X... Y... Z...	Absolute rotation
AROT X... Y... Z...	Incremental rotation, relative to the currently active frame
ROTS X... Y...	Absolute rotation, defined by two angles. The angles are the angles of the lines of intersection between the inclined plane and the main planes with respect to the axes.
AROTS X... Y...	Incremental rotation, relative to the currently active frame, angles as for ROTs
RPL=...	Rotation in the plane
MIRROR X... Y... Z...	Absolute mirroring
AMIRROR X... Y... Z...	Incremental mirroring, relative to the currently active frame
SCALE X... Y... Z...	Absolute scaling
ASCALE X... Y... Z...	Incremental scaling, relative to the currently active frame

Frame operators

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

Special frames

TOFRAME	Tool frame, coordinate system with Z axis in tool direction, zero is the tool tip
TOFRAMEX	Tool frame, coordinate system with X axis in tool direction, zero is the tool tip
TOFRAMEY	Tool frame, coordinate system with Y axis in tool direction, zero is the tool tip

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

All information about SINUMERIK, news, products, service products, downloads, contacts and service can be found on the SINUMERIK homepage.

<https://www.siemens.com/sinumerik>

■ SINUMERIK technical documentation

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

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

■ CNC4you - portal for users

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

<https://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>

■ SinuTrain

Learn more about the control-identical NC programming station.

<https://www.siemens.com/sinustrain>

8.3 Index

Number

3- to 3+2-axis 153
 3D angular deviation 56
 5-axis transformation 81

A

ATOL 116
 AUTO mode 52
 Accuracy 20, 28, 102
 Advanced Surface 110
 Check program 111
 Programming example 106
 Align plane 46, 52
 Example 47
 Analyze MyWorkpiece /Toolpath 24
 Animated elements 12
 Automatic mode 52, 53
 Determining the tool radius 55
 Measuring the workpiece 53

B

BRISK 115
 Basic frame 74
 Basics of simultaneous machining 81
 Block search 67

C

CAD-CAM-CNC 10
 CAM system 27
 COMPCAD 112
 Compressor function 21
 COMPOF 112
 COMPSURF 112
 CTOL 116
 CUST_832 109
 CUT2D 127
 CUT2DF 127
 CUT3DC 120, 121, 127
 CUT3DC example 129
 CUT3DCC 126, 128
 CUT3DCCD 128
 CUT3DCD 121, 127

CUT3DF 120, 122, 128
 CUT3DFD 122, 128
 CUT3DFF 128
 CUT3DFS 128
 CYCLE150 44
 CYCLE800 75
 CYCLE832 30, 102
 Advanced Surface 102
 Parameter 104
 Top Surface 102
 CYCLE995 56
 CYCLE996 57
 CYCLE9960 59
 CYCLE997
 Example 53
 Calculate kinematics 58
 Calibrate probe 46
 Check program
 Advanced Surface 111
 Top Surface 111
 Chord error 84
 Compressor 21, 23, 112, 161
 Compressor function 21
 Cone surface interpolation (ORICONxx)
 97
 Continuous-path mode 113
 Coordinate systems 74

D

DYNFINISH 118
 DYNORM 118
 DYNPOS 118
 DYNROUGH 118
 DYNSEMIFIN 118
 Determining the tool length 54, 61
 Direction vector 18
 Dynamic response 162

E

EES
 Call examples 35
 EES (Execution from External Storage)
 35
 EXTCALL 30, 34

Engine and turbine components 140
 Example
 Frames 74
 Execution from external storage 35
 Explanation of terms
 Frames 73
 Swivel 73
 TRAORI 73
 External storage media 34

F

FFWOF 115
 FFWON 115
 FIFOCTRL 119
 Feedforward control 115, 163
 Fork 19
 Frame, frames 168
 Frame operators 168
 Programmable frames 168
 Free-form machining 20
 Free-form surfaces 146
 Example 147

G

G code group 59 109
 G645 113

H

Handwheel override 66
 DRF 66
 TOFFON 66
 High Speed Settings 102

I

ISD 127

J

JOG 50
 Measure tool 50
 JOG mode 46
 Jerk 163
 Jerk limitation 115

K

Kinematics 19
 Kinematics measuring cycle 57

L

Large circle interpolation 96
 Linear axes 17
 Linear interpolation (ORIAXES) 96
 Look Ahead 113, 162

M

MCS 38
 Machine kinematics 75
 Manual retraction 40
 Measure corner 46
 Measure edge 46, 52
 Measure kinematics completely 59
 Measure pocket 53
 Measure pocket/hole 46
 Measure spigot 52
 Measure spigot/hole 46
 Measure tool 54
 Measure tool in AUTOMATIC 54
 Measure tool in JOG 48
 Measure workpiece in AUTOMATIC 52
 Measurement log
 AUTOMATIC 44
 JOG 42
 Measuring cycles 41, 46, 54
 selecting 46
 Measuring cycles in AUTOMATIC 52
 Measuring cycles in JOG 46
 Measuring in JOG and AUTOMATIC 41
 Measuring tools 48
 Memory expansion 35
 Milling technology packages 11
 Mold-making view 61
 Motion commands 160
 Multitouch 13

N

Nutated axis 12, 19, 23

O

ORIAXES 96
 ORICONxx 97
 ORICURVE 98
 ORIMKS 95
 ORIPATH 99
 ORIRESET 101
 ORISON 30, 117
 ORIVECT/ORIPLANE 96
 ORIWKS 95
 OTOL 116
 Orientation interpolation 92
 Orientation interpolation and orientation reference 92

P

Prepositioning with CYCLE800 154
 Probe 46, 50, 53
 Process chain 10, 26
 CAD-CAM-CNC 16
 Process measuring 52
 Program editor 37
 Dual editor 37
 Subprogram call 37
 Program manager 33, 36
 BMP, PNG, GIF, JPG 36
 DXF 36
 TXT, PDF 36
 Program run 65
 Program structure
 CYCLE832 in the main program 29
 CYCLE832 in the subprogram 31
 Programmable smoothing
 Spline interpolation 21
 Programming direction vector 18
 Programming rotary axes 18

R

Return 40
 Right-angled corner 47
 Rotary axes 17, 19
 Rotary axis positions 18
 Running time 65

S

SINUMERIK 828D 11
 SINUMERIK 840D sl 11
 SINUMERIK MDynamics 11
 SINUMERIK Operate 12
 SOFT 115
 SPATH 116
 STAT 101
 Sample workpiece 153
 Sidescreen 14
 Simulation 60
 Simultaneous recording 63
 Smoothing of the orientation characteristic 117
 Spline 21
 Spline compressor 112
 Spline interpolation 98
 Structural parts
 Example 135
 Functions 134
 Surface quality 10, 20, 102
 Swivel
 CYCLE800 75
 Swivel head 75
 Swivel table 75
 Syntax highlighting 37

T

TOFFL 49
 TOFFR 49
 TOROT 100
 TRAORI 81
 Programming 85
 Technology G groups 118, 162
 Tolerance 28
 Tolerance_tol. 104
 Tool
 Measuring in JOG 50
 Tool geometries 48
 Tool list 39
 Tool management 39
 Tool orientation 86
 Direction vector 87
 LEAD/TILT 89
 ORIEULER/ORIRPY 88
 Rotary axis positions 86, 88
 Surface normal vector 89
 Tool radius compensation 167

- Tool reference point 48
- Tool table 48
- Tool type
 - Milling tool types 49
- Tool wear list 39
- Top Surface
 - Check program 111
 - Programming example 107
- Transforming coordinate systems
 - Frames 74

U

- UPATH 116

V

- VCS 130
- VNCK - virtual machine 131
- Vector interpolation (ORIVECT/ORIPLANE) 96
- Velocity 20, 102
- Velocity programming 162
- Volumetric Compensation System 130

W

- WCS 38
- Workpiece
 - Setting up 46
- Workpiece visualization 60

Z

- Zero offsets 38

Further information

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

Support and technical documentation
<https://support.industry.siemens.com>

Local partners worldwide
www.siemens.com/automation-contact

Online catalogue and mall
www.siemens.com/automation/mall

Siemens AG
Digital Factory
Motion Control
Postfach 3180
91050 ERLANGEN
Germany