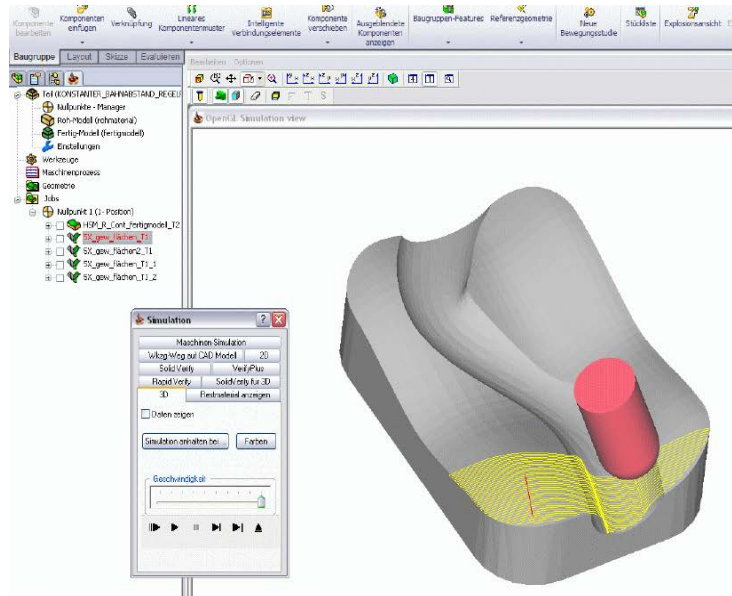


HEIDENHAIN



CAD – CAM – CNC

From 3-D Model to NC Program



CAD – CAM – CNC

CAD

Design

CAM

Path generation

Tool compensation

CNC

NC program interpreter

Motion control

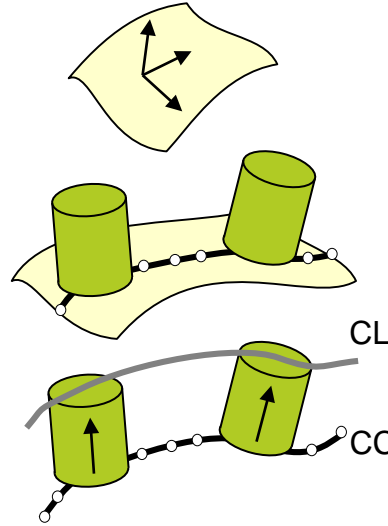
Tolerance monitoring

Velocity profiles

Mechanics

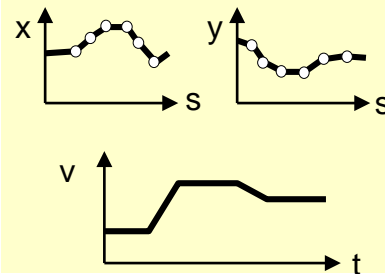
Feed rate control

Machine & Drives



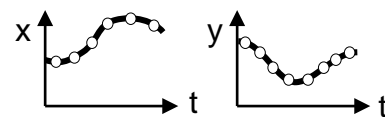
The workpiece contour is modeled with NURBS (Non-Uniform Rational B-Splines).

Surfaces of the CAD contour are reproduced **point for point** by paths

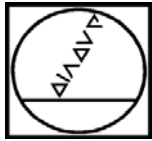


Convert path into axis movement and velocity profile

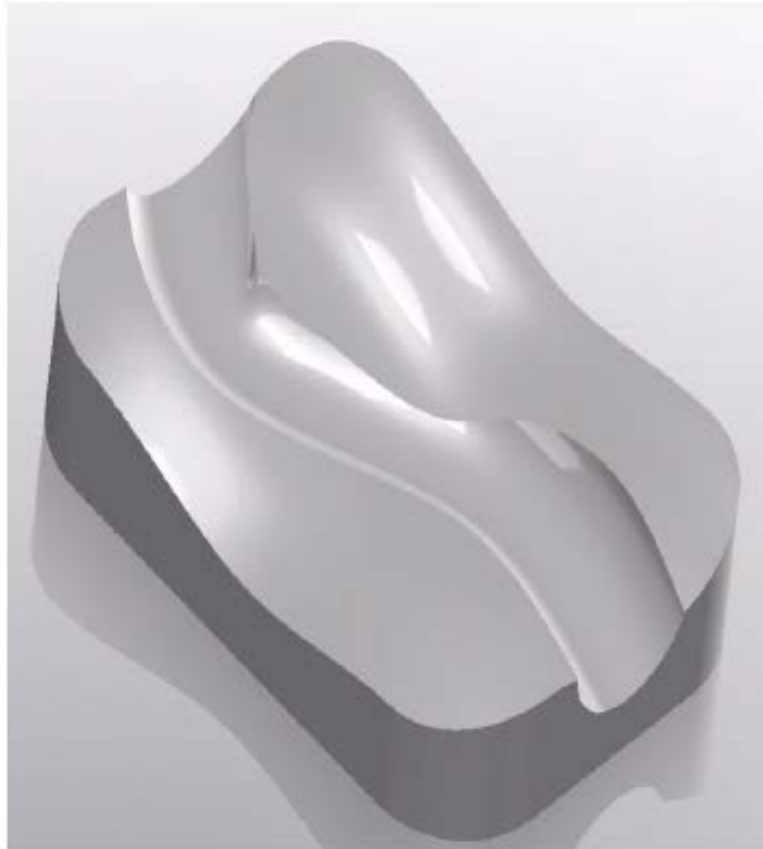
Point for point TNC processing



Axis movements are available in a fixed time grid as nominal and actual movements



HEIDENHAIN



CAD
3-D Model



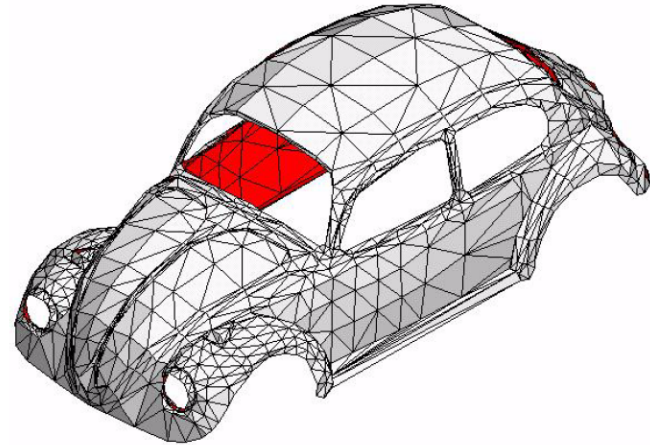
3-D Model Creation (CAD)

Design

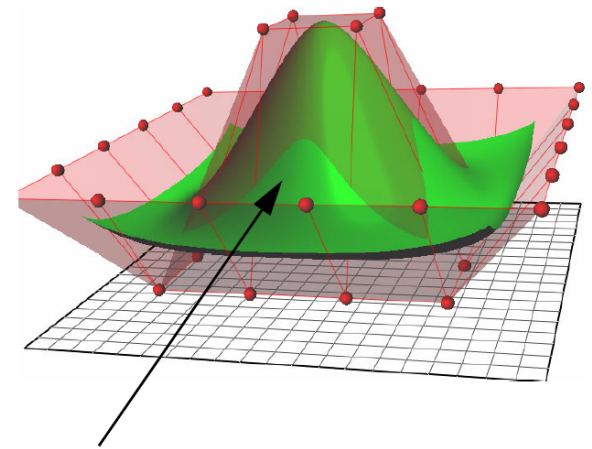
- CAD (*computer-aided design*) →
- Creation of virtual component in 3-D

Various 3-D model descriptions exist

- Polygon modeling
- NURBS (Non Uniform Rational B-Splines)
- Splines



Polygon model



A fourth degree NURBS surface (green), defined by 36 control points (red) over a two-dimensional parameter area (lower grid)



3-D Model Creation (CAD)

Memory formats for 2-D/3-D models

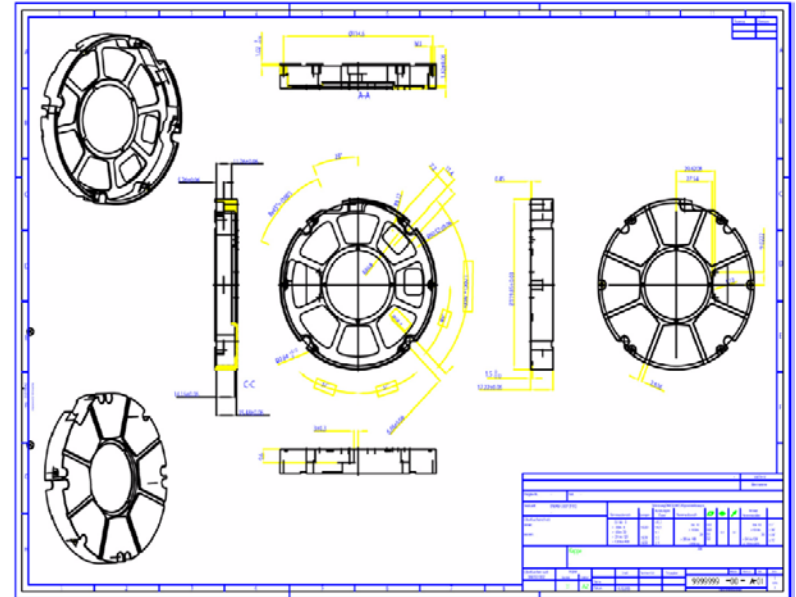
- DXF (established memory format for 2-D drawings)
- IGES (data exchange format for 2-D drawings and 3-D data)
- STEP (standardized file exchange format for geometry data)

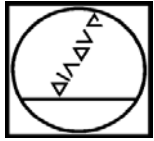
CAD system-neutral formats

- Usually only for edge, surface and volume model transfer
- Design history is usually lost
- Data transferred usually has only limited suitability for further processing

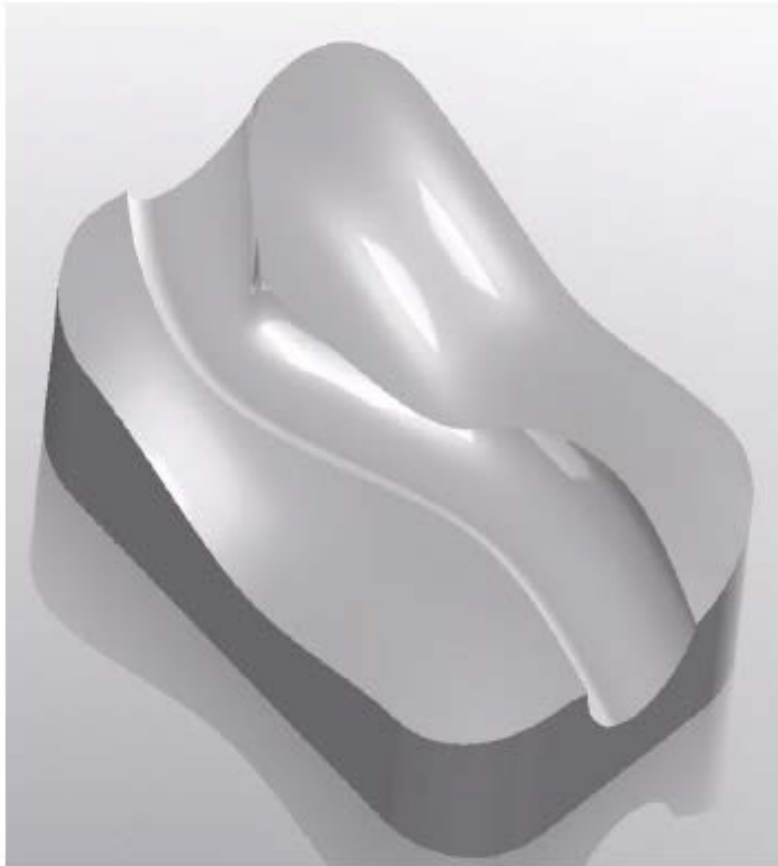
CAD system-specific formats

- Transfer of complete CAD models
- Only available for a few systems





HEIDENHAIN



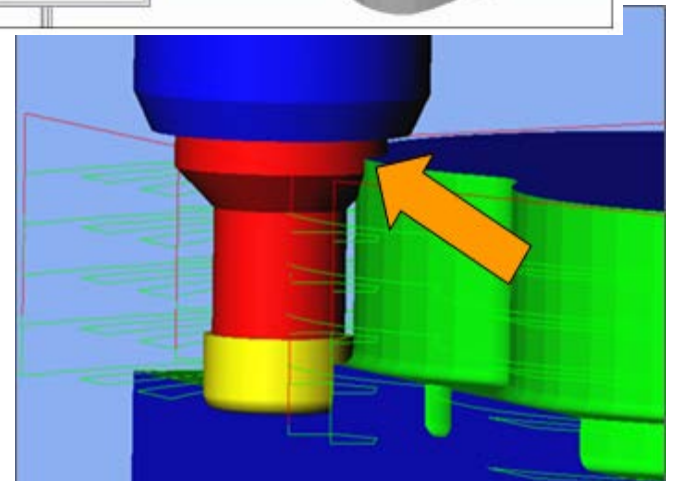
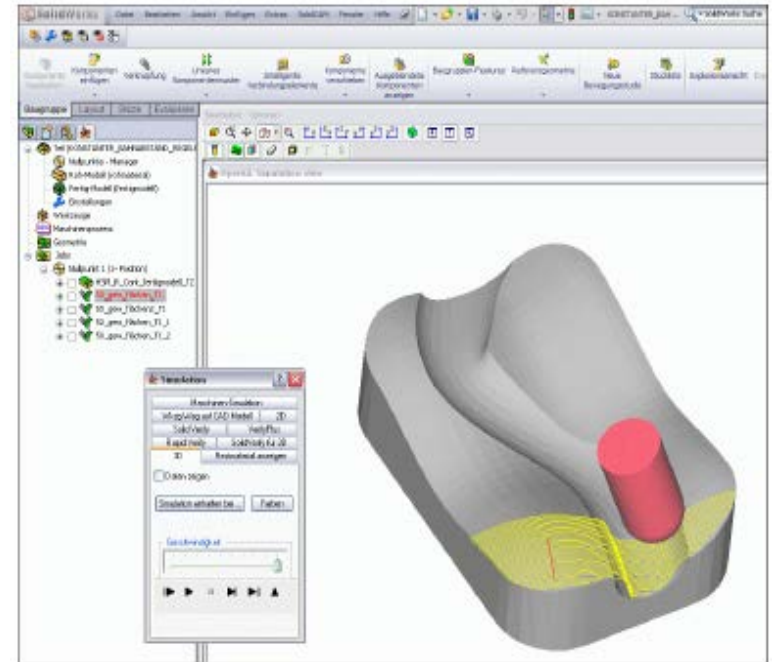
CAM
Programming



NC Program Creation (CAM)

Programming

- CAM (*computer-aided manufacturing*) →
- Select the areas to be machined
- Specify the **production process** with requisite **technology data** and **machining strategies**
- The CAM system calculates the tool paths and creates a neutral NC program
- The CAM system features a simulation function for testing the program





NC Program Creation (CAM)

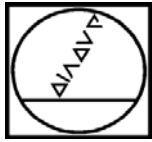
NC program generation

- A machine-specific NC program is generated from the neutral CLDATA program (created by the CAM when the program is saved)
- The generation is with the postprocessor
- The postprocessor is the connection between CAM and the machine control
- Output of a structured program with
 - Structure
 - Comments
 - Subprograms (retracting, resetting contours and positions)
 - Q parameters for feed rate assignment

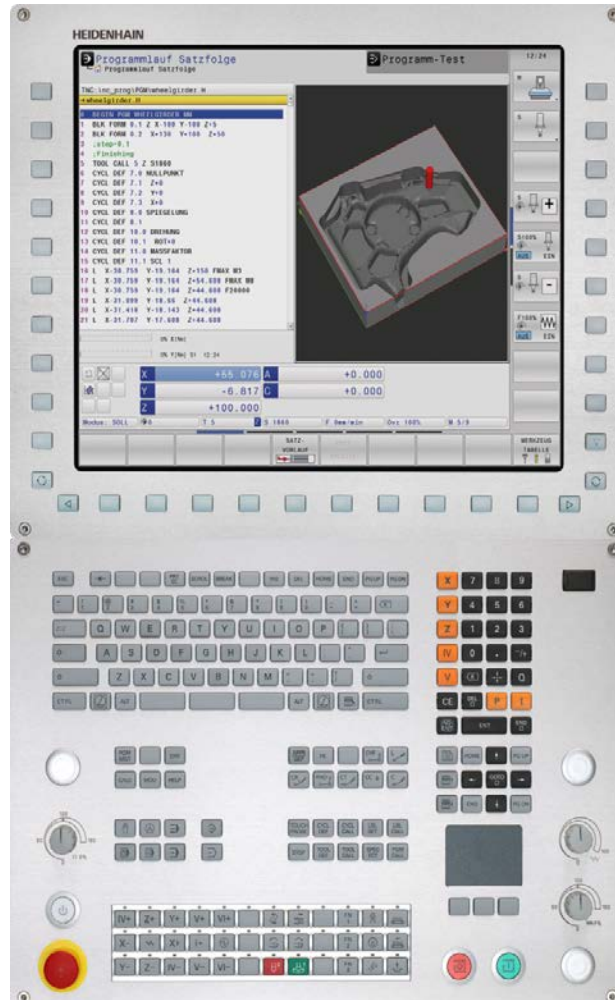
```
01110101000000
01110101000000
01010000010101
01110101000000
01110101000000
01010000010101
```



```
;oversize +0.5
;Z-oversize +0
;HSS end mill cutter dm 2
TOOL CALL 219 Z S5095
FN 0: Q10 =+764 ;FXY
FN 0: Q11 =+382 ;FZ-
FN 0: Q12 =+600 ;FZ+
M3
L X-15.531 Y+35.968 R0 I
L Z+25 R0 FMAX
M9
```

HEIDENHAIN



CNC Control



NC Program (CNC Control)

Production

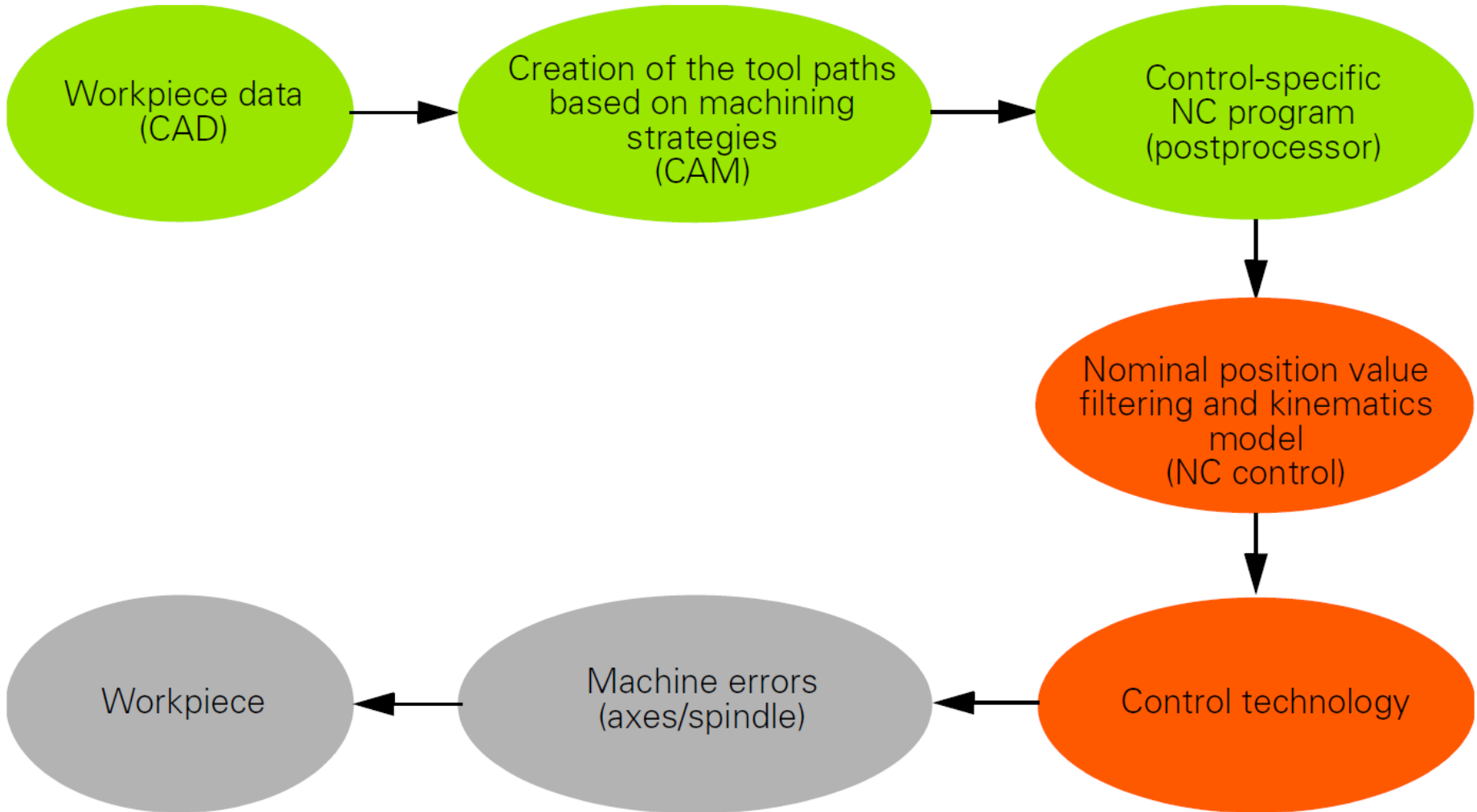
- CNC (*computer numerical control*) →
- Automated machining with several simultaneously controlled axes
- Classification according to the number of simultaneously interpolated axes
- Differentiation between point, linear and path control

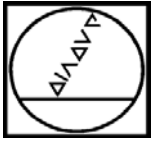




NC Program (CNC Control)

Overview of production process





HEIDENHAIN



Output Formats of NC Programs



Output Formats of NC Programs

Plain language

Introduced in 1976, plain language dialog is still the standard programming language for all TNC controls. When you hear people talking about shop-floor programming, they mean plain-language conversational programming from HEIDENHAIN.

- Block format for **three axes**

```
L X+80 Y+0 Z-10 R0 F9000
```

- Block format for **up to five axes**

```
L X+20 Y+10 Z+2 A+15 C+6 R0 F9000  
M128 F5000
```

The screenshot displays a CNC control interface with a 3D model of a part on the right and a list of NC program blocks on the left. The interface is titled "Manual operation" and "Programming and editing". The 3D model shows a complex, curved part with a blue wireframe overlay. The list of blocks includes coordinates for X, Y, Z, and R, along with feed rates (F) and other parameters. The interface also features a toolbar with various icons for manual operation, such as M, S, T, and S100% (OFF/ON), and a bottom row of buttons for BEGIN, END, PAGE, FIND, START, START SINGLE, and RESET + START.

Line	Block	X	Y	Z	R	F
17364	L	X-23.929	Y-51.59	Z-42.816		
17365	L	X-23.929	Y-51.49	Z-42.142		
17366	L	X-23.653	Y-51.254	Z-41.029		
17367	L	X-23.472	Y-51.072	Z-39.919		
17368	L	X-23.234	Y-50.884	Z-38.81		
17369	L	X-23.008	Y-50.698	Z-37.724		
17370	L	X-22.721	Y-50.522	Z-36.674		
17371	L	X-22.423	Y-50.324	Z-35.111		
17372	L	X-22.201	Y-49.902	Z-32.995		
17373	L	X-21.971	Y-49.571	Z-31.882		
17374	L	X-21.732	Y-49.333	Z-30.772		
17375	L	X-21.504	Y-49.104	Z-29.796		
17376	L	X-21.27	Y-48.871	Z-28.823		
17377	L	X-21.13	Y-48.79	Z-28.27		
17378	L	X-20.984	Y-48.585	Z-27.721		
17379	L	X-20.833	Y-48.424	Z-27.174		
17380	L	X-20.74	Y-48.341	Z-26.85		
17381	L	X-20.439	Y-48.04	Z-25.885		
17382	L	X-20.294	Y-47.895	Z-25.445		
17383	L	X-20.139	Y-47.739	Z-25.011		
17384	L	X-19.974	Y-47.575	Z-24.585		
17385	L	X-19.849	Y-47.449	Z-24.277		
17386	L	X-19.717	Y-47.318	Z-23.973		
17387	L	X-19.58	Y-47.181	Z-23.675		
17388	L	X-19.293	Y-46.894	Z-23.101		
17389	L	X-18.984	Y-46.585	Z-22.549		
17390	L	X-18.652	Y-46.252	Z-22.022		
17391	L	X-18.306	Y-45.907	Z-21.527		
17392	L	X-17.948	Y-45.549	Z-21.083		
17393	L	X-17.561	Y-45.161	Z-20.642		
17394	L	X-17.173	Y-44.774	Z-20.239		
17395	L	X-16.771	Y-44.372	Z-19.871		



Output Formats of NC Programs

To be observed with program output (plain language):

- 4 decimal places (optionally 5 decimal places)
- Output circles with CC/C – CT – CR. Avoidance of linearization (control traverses circle blocks more quickly/fluidly)
- Output of radius-compensated traversing blocks with precision fits

The screenshot displays a CNC control interface with the following components:

- Manual operation** (left tab) and **Programming and editing** (right tab).
- NC Program Output:** A list of blocks with coordinates (X, Y, Z) and tool identifiers (L, S, T).

17364	L	X-23.929	Y-51.59	Z-42.816
17365	L	X-23.929	Y-51.49	Z-42.142
17366	L	X-23.653	Y-51.254	Z-41.029
17367	L	X-23.472	Y-51.072	Z-39.919
17368	L	X-23.234	Y-50.884	Z-38.81
17369	L	X-23.008	Y-50.608	Z-37.24
17370	L	X-22.721	Y-50.322	Z-35.674
17371	L	X-22.423	Y-50.024	Z-34.111
17372	L	X-22.201	Y-49.902	Z-32.995
17373	L	X-21.971	Y-49.571	Z-31.882
17374	L	X-21.732	Y-49.333	Z-30.772
17375	L	X-21.504	Y-49.104	Z-29.796
17376	L	X-21.27	Y-48.871	Z-28.823
17377	L	X-21.13	Y-48.79	Z-28.27
17378	L	X-20.984	Y-48.585	Z-27.721
17379	L	X-20.833	Y-48.424	Z-27.174
17380	L	X-20.74	Y-48.341	Z-26.85
17381	L	X-20.439	Y-48.04	Z-25.885
17382	L	X-20.294	Y-47.895	Z-25.445
17383	L	X-20.139	Y-47.739	Z-25.011
17384	L	X-19.974	Y-47.575	Z-24.585
17385	L	X-19.849	Y-47.449	Z-24.277
17386	L	X-19.717	Y-47.318	Z-23.973
17387	L	X-19.58	Y-47.181	Z-23.675
17388	L	X-19.293	Y-46.894	Z-23.101
17389	L	X-18.984	Y-46.585	Z-22.549
17390	L	X-18.652	Y-46.252	Z-22.022
17391	L	X-18.306	Y-45.907	Z-21.527
17392	L	X-17.948	Y-45.549	Z-21.083
17393	L	X-17.561	Y-45.161	Z-20.642
17394	L	X-17.173	Y-44.774	Z-20.239
17395	L	X-16.771	Y-44.372	Z-19.871
- 3D Model:** A 3D wireframe model of a part, likely a turbine vane, with a coordinate system showing 27 H, +28 V, and z.
- Control Panel:** Includes buttons for M (Machine), S (Spindle), T (Tool), S100% (Spindle Speed), OFF/ON (Spindle Speed), and S (Spindle) with a minus sign.
- Navigation Buttons:** BEGIN, END, PAGE, PAGE, FIND, START, START SINGLE, and RESET + START.



Output Formats of NC Programs

Vector processing

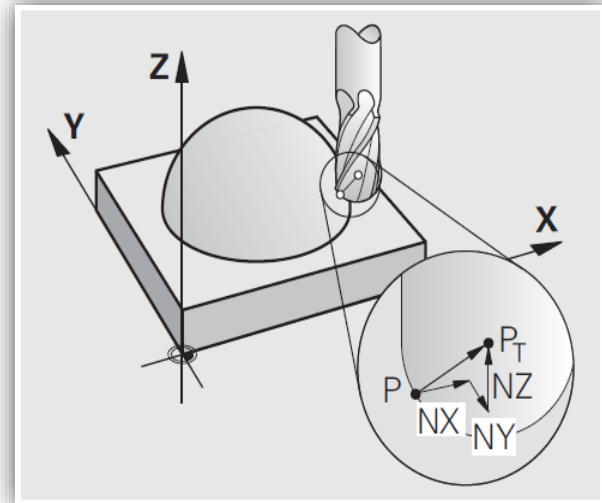
A normalized vector is a mathematical quantity with a value of 1 and any direction.

The TNC requires up to two normalized vectors for LN blocks, one to determine the direction of the surface-normal vector (N vector), and another (optional) to determine the tool orientation (T vector).

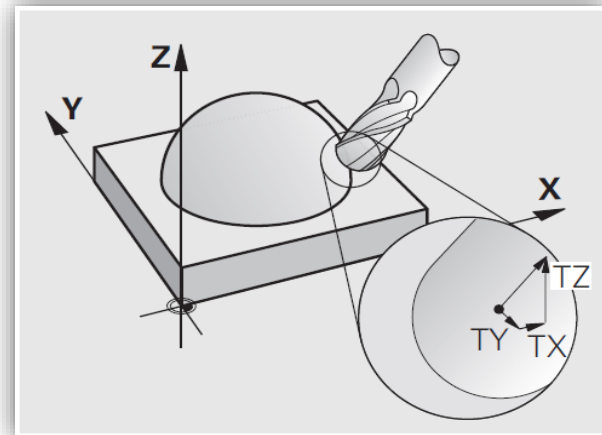
- Block format for five axes

```
7 LN X+31,737 Y+21,954 Z+33,165  
NX+0,2637581 NY+0,0078922 NZ-0,8764339  
TX+0,0078922 TY-0,8764339 TZ+0,2590319 F1000 M128
```

N vector



T vector





Output Formats of NC Programs

To be observed with program output (vector):

- 7 decimal places
- N vector (surface-normal vector) is always vertical to the surface

Benefits of vector output:

- Machine-neutral program output
- Use of 3-D radius compensation to adapt the program to real tool data (compensation via DL, DR and DR2).
- Use of the 3D-TOOLCOMP option (Option #92 – available on the iTNC 530)

The screenshot displays the iTNC 530 programming environment. On the left, a list of NC program lines is shown, each containing a line number (LN) and a series of vector coordinates (X, Y, Z, V, N, F, R, S, T, U, V, W, X, Y, Z). The coordinates are formatted with 7 decimal places. On the right, a 3D surface model is visible, showing a curved surface with a grid of points. The surface is colored in shades of blue and purple. The interface includes a top menu bar with 'Manual operation' and 'Programming and editing', and a right sidebar with various tool and function icons. The bottom of the screen shows a control panel with buttons for 'BEGIN', 'END', 'PAGE', 'PAGE', 'FIND', 'START', 'START SINGLE', and 'RESET + START'.



Output Formats of NC Programs

To be observed with program output / postprocessor adaptation:

- Use of structure
- Feed rates via Q parameters, e.g.
 - Q50 = 5000 ; F XY
 - Q51 = 2000 ; F Z
- Use of subprograms (labels) for retracting, resetting, machining positions etc.
- Use of HEIDENHAIN cycles where possible (drilling, bore milling)
- Tilted-axis machining via PLANE functions
- Output fits with radius compensation (RR/RL)
- Compensate inclinations via M128 / TCPM

The screenshot displays a CNC programming software interface. The top bar shows 'Manual operation' and 'Programming and editing'. The main window is divided into two sections: a 3D model of a part on the right and a list of NC program lines on the left. The 3D model shows a blue part with a red line indicating a specific feature. The NC program lines are as follows:

```
3629 LN X+70.5278 V+19.0015 Z-10.7404
NX+0.5304027 NV+0 NZ+0.0476959
3630 LN X+89.3268 V+19.0015 Z-10.0454
NX+0.5113009 NV+0 NZ+0.0594010
3631 LN X+36.7646 V+19.0015 Z+0.7962 NX+0.5
NV+0 NZ+0.0560254
3632 LN X+36.7646 V+19.0015 Z+10 NX+0 NV+0
NZ+0 F04
3633 LN X+107.2359 V+10.5022 Z+10 NX+0 NV+0
NZ+0 FMAX
3634 LN X+107.2359 V+10.5022 Z-9.6701 NX+0
NV+0 NZ+0 FMAX
3635 LN X+107.2359 V+10.5022 Z-27.6701 NX+0
NV+0 NZ+0 FMAX
3636 LN X+107.2359 V+10.5022 Z-29.6701
NX+0.0062906 NV+0 NZ+0.9999002 F03
3637 LN X+99.3263 V+10.5022 Z-29.4909
NX+0.0390054 NV+0 NZ+0.999239 F02
3638 LN X+89.6111 V+10.5022 Z-29.4512
NX+0.0710912 NV+0 NZ+0.9974125
3639 LN X+97.9329 V+10.5022 Z-29.301
NX+0.1076746 NV+0 NZ+0.9941082
3640 LN X+87.0575 V+10.5022 Z-29.2029
NX+0.1433911 NV+0 NZ+0.9896740
3641 LN X+96.2059 V+10.5022 Z-29.1569
NX+0.1708139 NV+0 NZ+0.9830029
3642 LN X+95.519 V+10.5022 Z-29.0093
NX+0.214077 NV+0 NZ+0.9760159
3643 LN X+94.7776 V+10.5022 Z-28.8271
NX+0.2481866 NV+0 NZ+0.9687173
3644 LN X+94.0406 V+10.5022 Z-28.6244
NX+0.29289 NV+0 NZ+0.959974
3645 LN X+93.2936 V+10.5022 Z-28.3901
NX+0.3104067 NV+0 NZ+0.9486236
3646 LN X+92.5553 V+10.5022 Z-28.129
NX+0.3503994 NV+0 NZ+0.9366165
3647 LN X+91.8266 V+10.5022 Z-27.8414
NX+0.3930626 NV+0 NZ+0.9239002
3648 LN X+91.1066 V+10.5022 Z-27.5276
NX+0.4168813 NV+0 NZ+0.9099069
3649 LN X+90.423 V+10.5022 Z-27.1906
NX+0.4404036 NV+0 NZ+0.8930312
3650 LN X+89.7498 V+10.5022 Z-26.846
NX+0.470359 NV+0 NZ+0.8776109
3651 LN X+89.0663 V+10.5022 Z-26.4576
NX+0.5106963 NV+0 NZ+0.8597612
3652 LN X+88.4011 V+10.5022 Z-26.0440
```

The bottom bar contains control buttons: BEGIN, END, PAGE, PAGE, FIND, START, START SINGLE, and RESET + START. The right side of the interface shows a vertical toolbar with icons for various functions, including a 3D model icon, a 'T' icon, and a 'S100%' icon with 'OFF' and 'ON' buttons. The status bar at the bottom right shows '340 H +12 U z'.



Output Formats of NC Programs

Simulation (test run) on the TNC

Simulation using a CAM system does not substitute the simulation of the TNC control.

The following problems can be intercepted with the test run:

- Limit switch error message due to erroneously set datum (workpiece blank in work space)
- Erroneous values in HEIDENHAIN cycles (e.g. positive depth with drilling)
- Circle end point error messages (end point tolerance with TNC 0.001 – 0.016 mm)
- Identifying erroneous point outputs from the CAM (TNC 620 / TNC 640 graphics)

The top screenshot shows the TNC control interface with the following NC program code in the left pane:

```
TNC:\1_TNC_DEMOS\3_exhibite\6_bli\1_start_Blisk.h
0 BEGIN PGM 1_START_BLK MM
1 CALL PGM
TNC:\1_TNC_DEMOS\4_extras_for_experts\Options\res
ot.H
2 BLK FORM 0.1 Z X-100 Y+45 Z+5
3 BLK FORM 0.2 X+105 Y+262 Z+135
4 FUNCTION MODE MILL "ACTable_mill"
5 CALL PGM vorfransen.h
6 CALL PGM Blisk.h
7 END PGM 1_START_BLK MM
```

The bottom screenshot shows the same interface with a detailed 3D model of a part. The data table on the left is as follows:

TRAVELSD LONGO	
X	-1027.285
Y	1972.715
Y	-807.755
Z	2192.244
Z	-1372.131
Z	1827.869

BLK FORM	
X	-100.000
Y	105.000
Z	45.000
Z	262.000
Z	5.000
Z	135.000

Datum	
X	-124.757
Y	-608.110
Z	358.173

REF POINT	
X	-347.958
Y	92.134
Z	-486.042



HEIDENHAIN

Motion Control



Motion Control

LOOK AHEAD

Interpolator memory

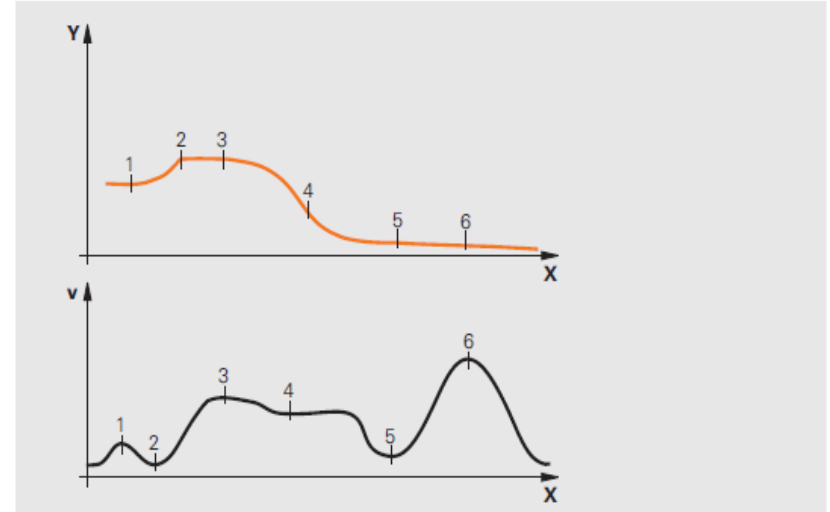
- Adapting the speed to the contour:
- The geometry saves up to 1024 blocks in a memory (stack). The interpolator reads this memory and adapts the feed rate accordingly. A LOOK AHEAD of 512 blocks is sufficient.
- Set by machine parameter (7400)

iTNC 530

- Value is adjustable via parameter

TNC 640

- Cyclic advance calculation (max. of 5000 blocks, no setting possible)





ADP—Advanced Dynamic Prediction

Improved surface quality with more rapid machining

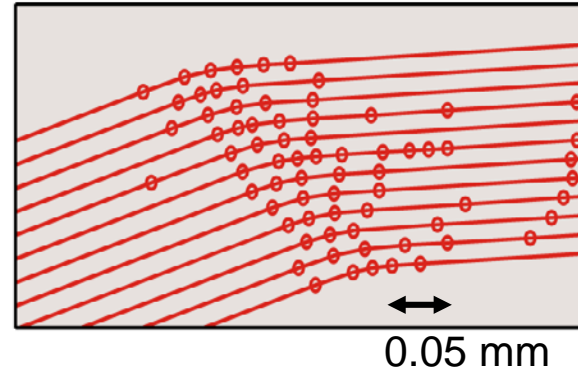
NC data with an insufficient resolution and variable point density in adjacent paths can lead to

- feed rate fluctuations and
- errors on the workpiece surface

ADP improves the calculation of the optimum speed for a precise and smooth movement of the TCP

Benefits:

- Higher contour speed and shorter machining times
- Superior surface quality



Without ADP



With ADP





dynamic + precision

- TNC controls support fast and exact machining for producing high-quality surfaces
 - Reliable management of contour tolerances
 - High repeatability of adjacent paths
 - Fast and vibration-free machine operation
- Dynamic Precision with high-performance control functions such as
 - Cross Talk Compensation,
 - Active Vibration Damping,
 - Load, Position and Motion Adaptive Control
- Achieves significantly improved precision on the Tool Center Point



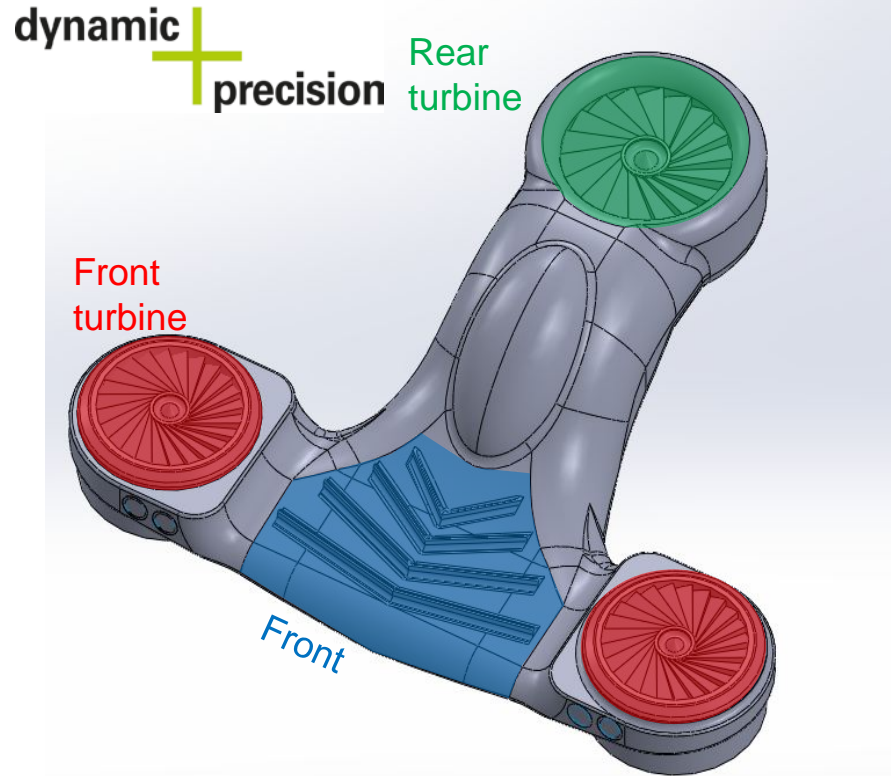
Motion Control – Application Example

Futuristic hovercraft

- Material: Aluminum EN AW 5083 (3.3547)
- Tool: Ball cutter, D = 3 mm

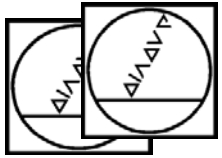
Productivity improvements with finishing:

- Total: 8%
- Front turbine: 10%
- Rear turbine: 7%
- Front: 8%



The HEIDENHAIN solution for high production rates of precisely machined workpieces

- Increases accuracy during dynamic contouring movements
- Less rejects and reworking
- Saves time and costs



HEIDENHAIN



Functions for 5-Axis Machining



Functions for 5-Axis Machining

TCPM (Tool Center Point Management)

- Compensation of offset caused by shifting of the rotary axes.
- (Functions: M128 / TCPM)

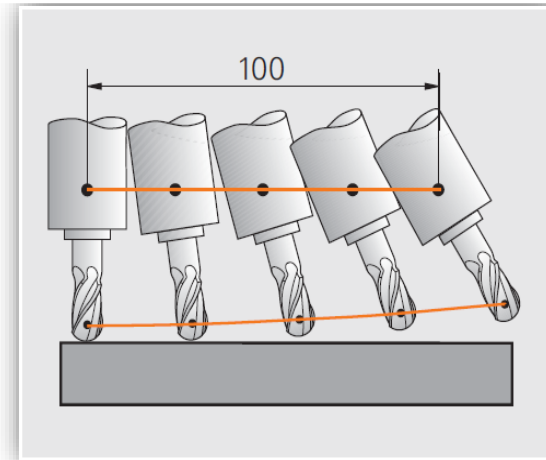
Inclination without M128:

- The leading point of the swivel head moves along the programmed path:
- Contour: Incorrect
- Feed rate: Incorrect

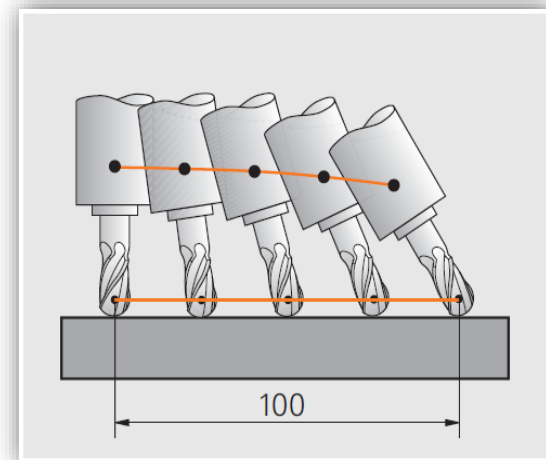
Inclination with M128:

- The tool center point moves along a straight line:
- Starting point and target point are correct
- Contour: Correct
- Feed rate: Correct

L X+100 B-30 R0 F1000



L X+100 B-30 R0 F1000 M128



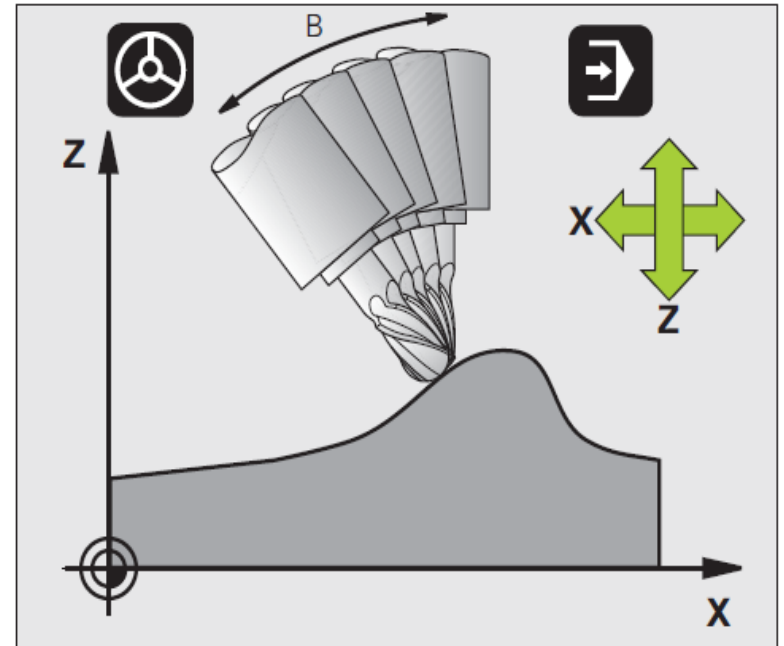


Functions for 5-Axis Machining

M128

During a rotary axis movement, the TNC controls the basic axes through an update so that the original tool contact point on the workpiece is also maintained during the movement.

- Feed rate after M128 limits the feed rate for the compensation movement
- M129 resets M128
- The feed rate calculation can be selected via machine parameters



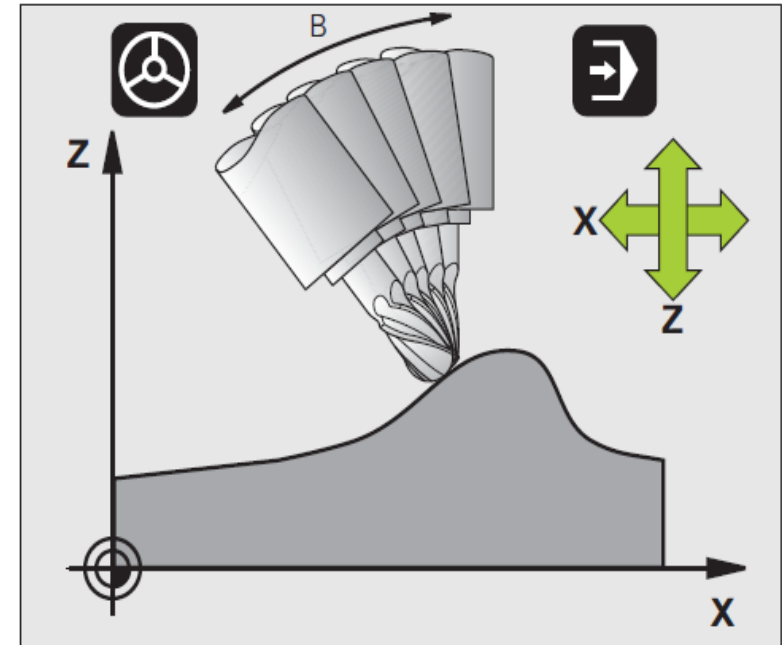


Functions for 5-Axis Machining

TCPM

A further development of M128. Programming via three input parameters:

- Type of programmed feed rate
 - F TCP (feed rate related to tool tip)
 - F CONT (sectionalize feed rate across axes)
- Interpretation of the programmed rotary axis coordinates
 - AXIS POS (programmed rotary access coordinates are axis angles)
 - AXIS SPAT (programmed rotary axis coordinates are spatial angles)



SPEC
FCT

PROGRAMM
FUNKTIONEN

FUNCTION
TCPM

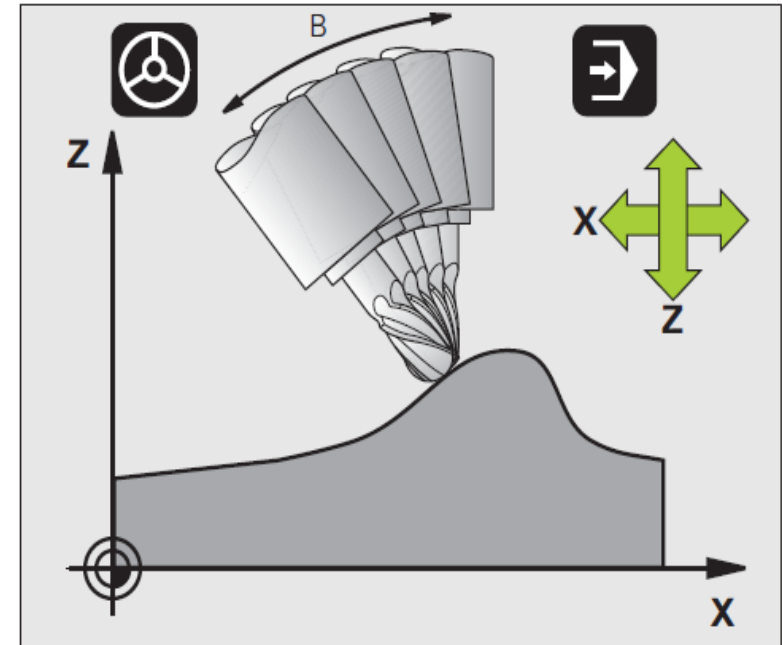


Functions for 5-Axis Machining

TCPM

A further development of M128. Programming via three input parameters:

- Interpolation type between starting position and target position
 - PATHCTRL AXIS (setting for face milling → ball cutter/milling)
 - PATHCTRL AXIS (setting for peripheral milling → end mill/contour milling)
- RESET TCPM resets TCPM



SPEC
FCT

PROGRAMM
FUNKTIONEN

FUNCTION
TCPM

RESET
TCPM



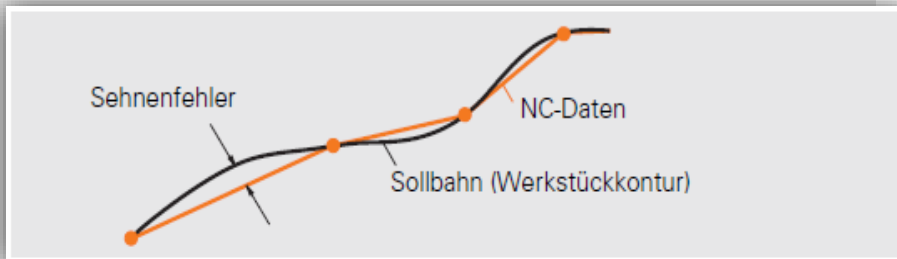
Functions for 5-Axis Machining

Cycle 32

Modification of machining result with regard to precision, surface quality and speed

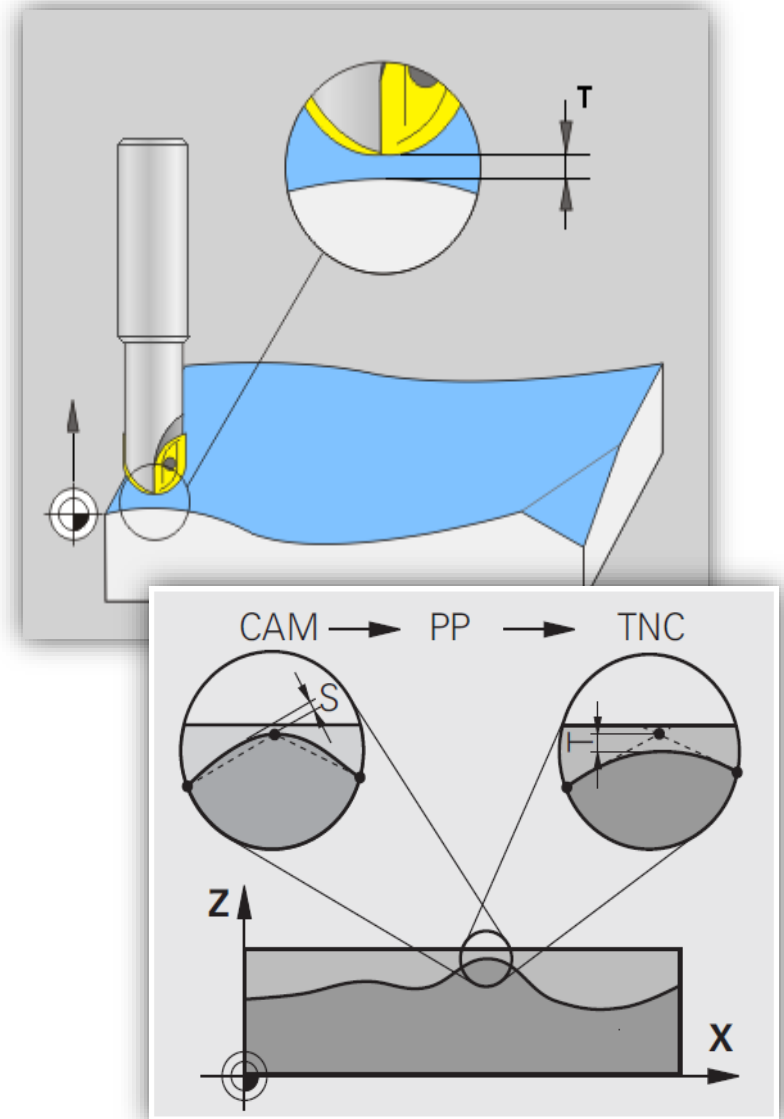
Tolerance value T

- Permissible contour deviation (1.1 to 2x chord tolerance from the CAM)



HSC MODE

- HSC MODE:0
- HSC MODE:1
→ Only effective with active HSC filter or Advanced HSC filter





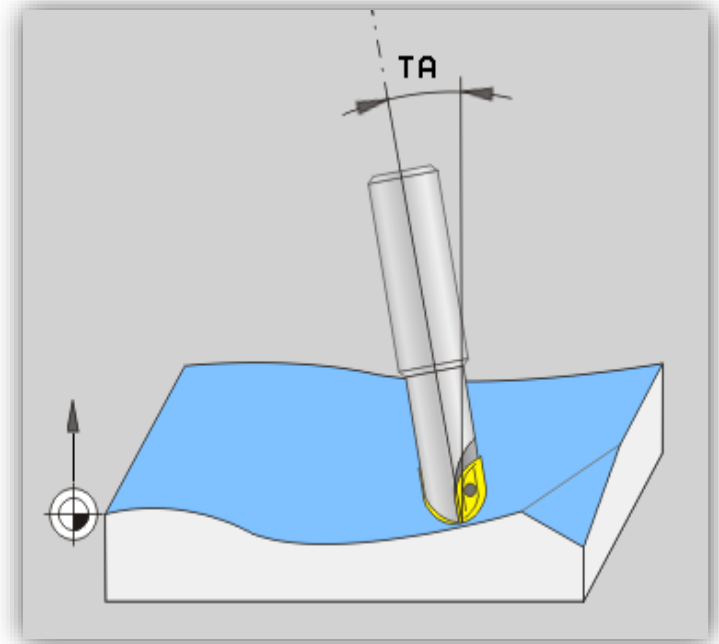
Functions for 5-Axis Machining

Cycle 32

Modification of machining result with regard to precision, surface quality and speed

Tolerance for rotary axes

- Tolerance for the rotary axis positions with five-axis machining (TCPM/M128 is active)





Functions for 5-Axis Machining

■ Preference: **Rapid machining / roughing**

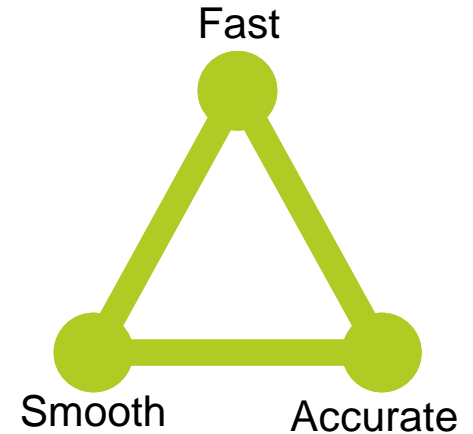
- Large Cycle 32 tolerance
 - Select MP settings with higher jerk/accelerations
 - NC data with less data density is possible
- Poorer surface/precision → **oversize**

■ Preference: **Precise machining**

- Small Cycle 32 tolerance
- Select MP settings with lower jerk/accelerations
- NC data must be fine enough to precisely approach transitions/corners.

■ Preference: **Surface quality—attractive surface**

- Average Cycle 32 tolerance
- Select MP settings with lower jerk/accelerations
- NC data with small chord error generated from CAD model



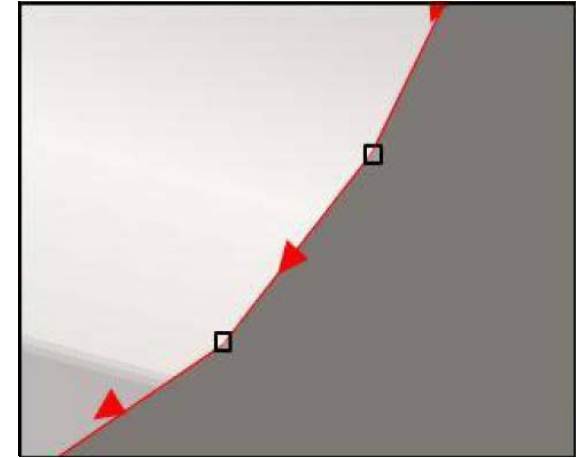


Functions for 5-Axis Machining

M124 T:

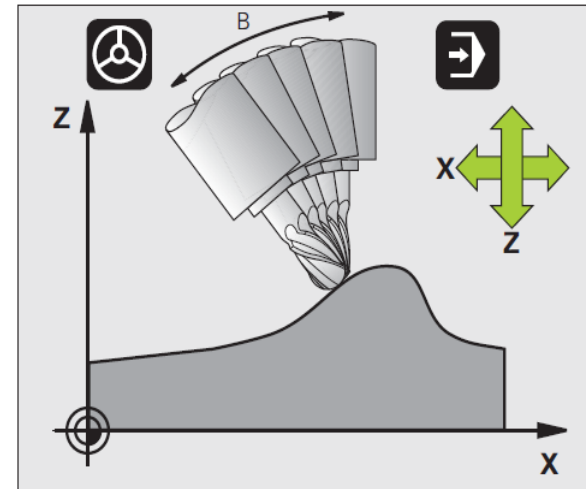
In Parameter T you define a minimal point distance, up to which the TNC does not consider points during processing

Points spaced very closely together cause additional changes in direction, and have a negative influence on the surface quality and the maximum possible feed rate.



M118 X.. Y.. Z.. A.. B.. C..:

M118 handwheel superimposition enables you to traverse axes with the handwheel—superimposed to program running—in operating modes Single Block and Full Sequence.





Functions for 5-Axis Machining

Block processing time:

The maximum possible feed rate depends on

- the block machining time (0.5 ms with Option #2)
- the distance between points in the NC program

Inclined-tool machining:

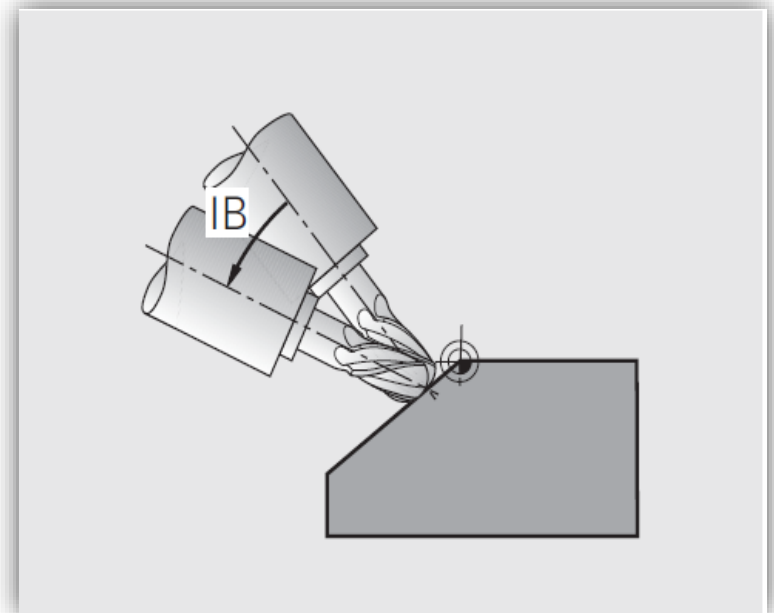
To improve cutting conditions, it often makes sense to modify the tool inclination from its perpendicular position to the surface.

Two definition possibilities are available for this:

- Inclined-tool machining via incremental traverse of a rotary axis
- Inclined-tool machining via normal vectors

$$\frac{\text{point distance [mm]} * 1000 * 60}{\text{block processing [ms]}} = F_{max} \left[\frac{\text{mm}}{\text{min}} \right]$$

$$\frac{0.2\text{mm} * 1000 * 60}{0.5\text{ms}} = 24000 \frac{\text{mm}}{\text{min}}$$





Functions for 5-Axis Machining

M120 LA.. (Look Ahead for contours):

The TNC checks radius-compensated paths for contour undercuts and tool path intersections, and calculates the tool path in advance from the current block.

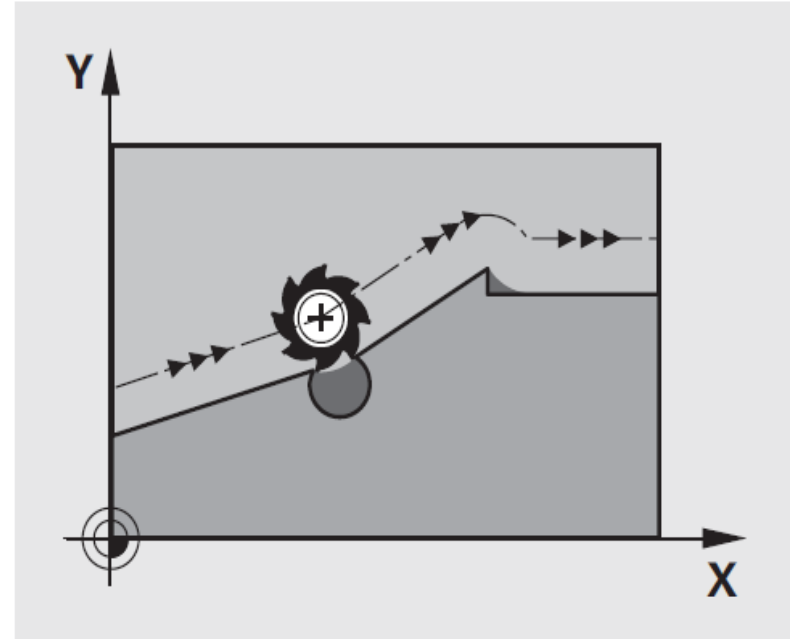
The LA parameter defines the number of blocks to be precalculated.

- Max. 99
- Usual input LA5- LA10

M120 must be in an NC block that also contains the radius compensation RL or RR. M120 is effective from this block.

Resetting/canceling:

- M120 LA0 or M120 (without LA)
- R0
- CALL PGM
- Tilting the working plane





Functions for 5-Axis Machining

"Tuning" cycle

Activation of various MP subfiles via user cycles.

The cycle contains (according to machine tool builder) various settings for jerk, acceleration and Cycle 32 tolerance, thus influencing

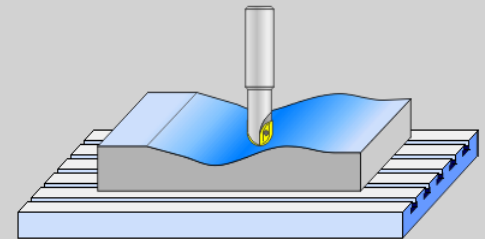
- Accuracy
- Surface definition
- Velocity

Function	Filters	Cycle 32 settings			Time
		T	Mode	TA	
Default	A-HSC	0.02	0	0.1	2:37
Accurate	HSC	0.005	0	0.1	4:43 AM
Smooth	A-HSC	0.02	0	0.1	2:58 AM
Roughing	Simple	0.1	1	0.1	2:29 AM

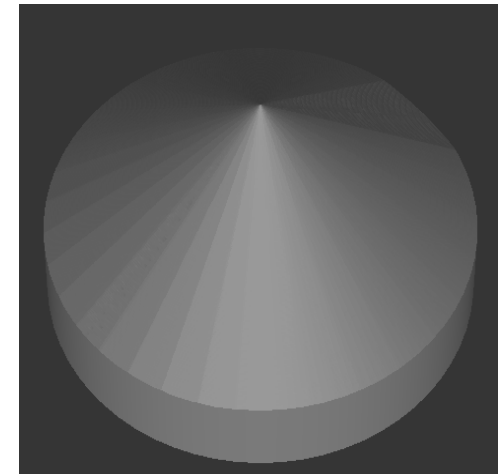
332 - Tuning

Tuning mode

- 0 = standard
- 1 = exact
- 2 = smooth
- 3 = rough



Q395 = 0..3





Functions for 5-Axis Machining

Global program settings (GPS)

Option #48

Definition of various coordinate transformations and settings globally effective in the background for the selected NC program, without the NC program needing to be modified.

- Swapping axes
- Additional, adaptive NP shifting
- Superimposed mirroring
- Axis locking
- Handwheel superimpositioning (with virtual axis)
- Superimposed basic rotation
- Superimposed rotation
- Globally valid feed-rate factor
- Limit level (definition of machining





Functions for 5-Axis Machining

KinematicsOpt

Option #44



KinematicsOpt comprises touch probe cycles for automatic testing and optimization of the machine kinematics

- Simple programming of user cycles
- Simple, quick kinematics post-optimization
- Optimization is possible during program run
- Simple adjustment of changer heads
- Saving and restoring kinematics transformations

KinematicsOpt enables you to improve the tilt and inclination precision of your machine

