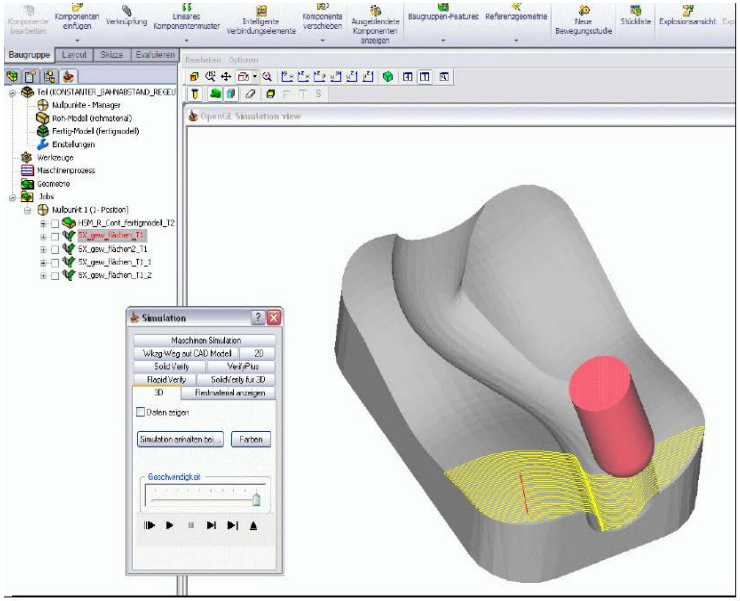
HEIDENHAIN



CAD – CAM – CNC

From 3-D Model to NC Program

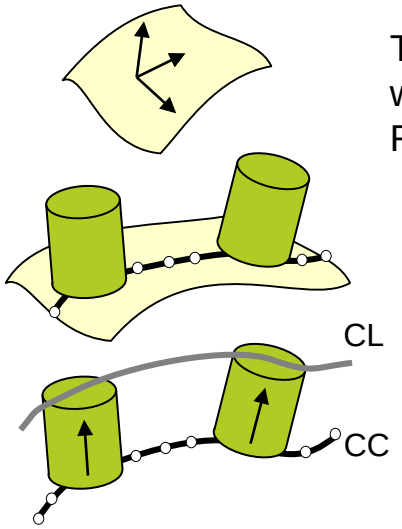
CAD – CAM – CNC

CAD
- Design

CAM
- Path generation
- Tool compensation

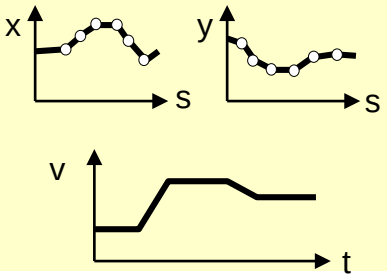
CNC
- NC program interpreter
- Motion control
- Tolerance monitoring
- Velocity profiles

Mecha-tronics
- Feed rate control
- Machine & Drives



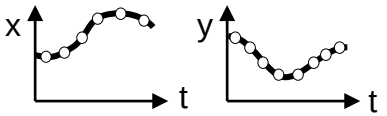
The workpiece contour is modeled with NURBS (Non-Uniform Rational B-Splines).

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



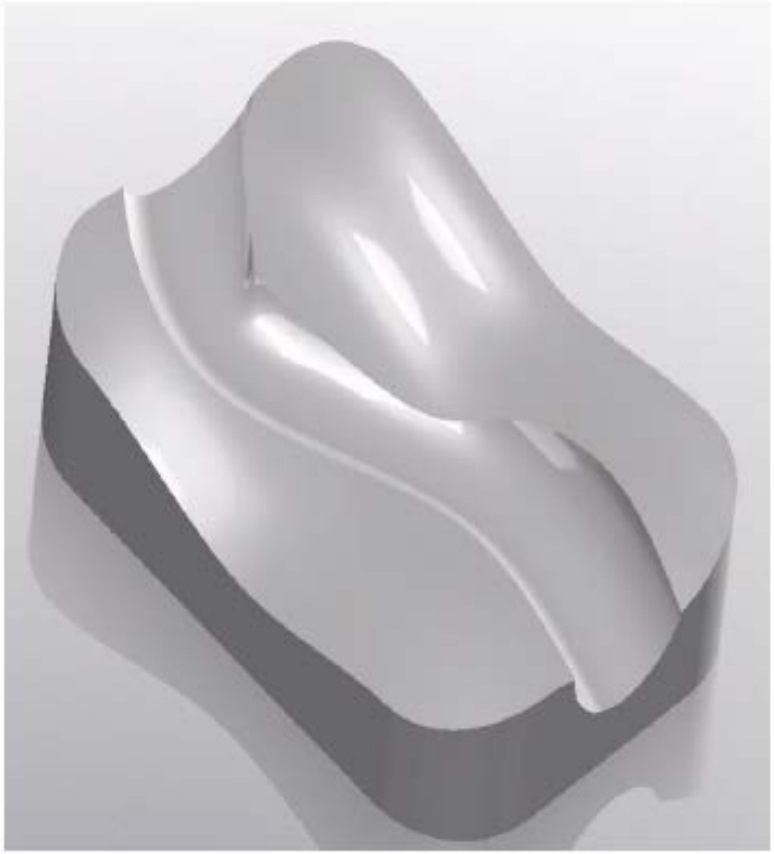
Convert path into axis movement and velocity profile

Point for point TNC processing



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

HEIDENHAIN



CAD

3-D Model

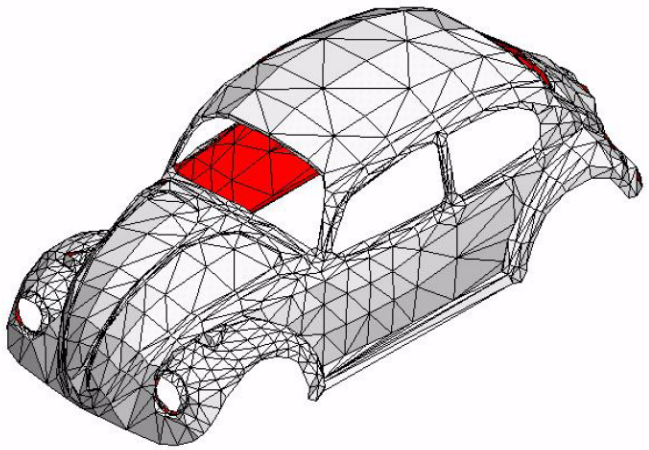
3-D Model Creation (CAD)

Design

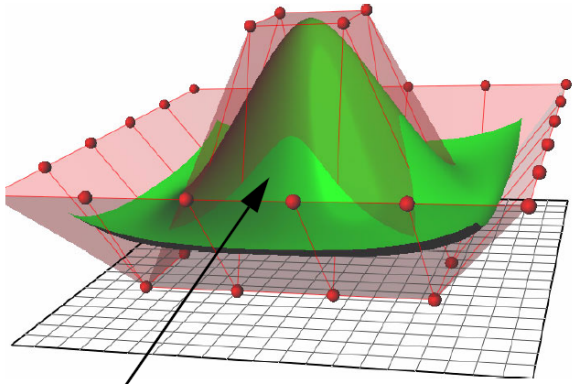
- CAD (*computer-aided design*) →
- Creation of virtual component in 3-D

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

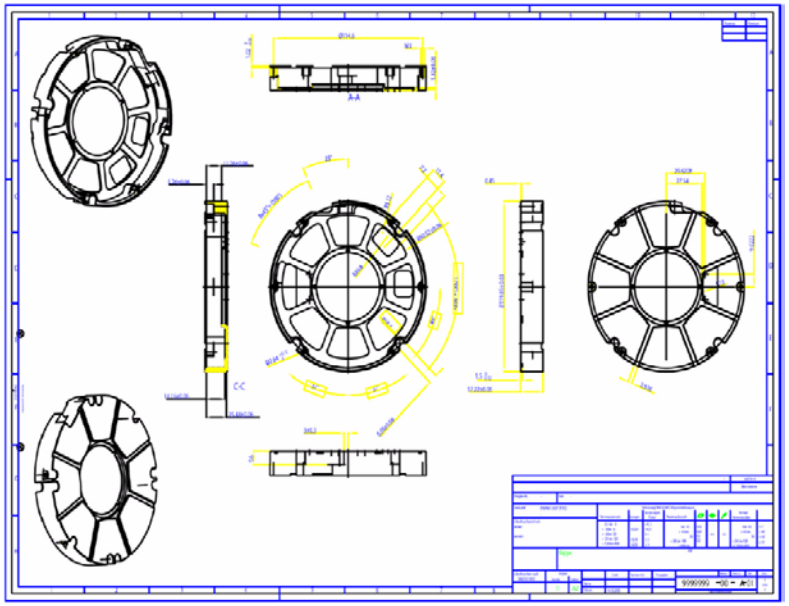
- DXF (established memory format for 2-D drawings)
- IGES (data exchange format for 2-D drawings and 3-D data)
- STEP (standardized file exchange format for geometry data)

CAD system-neutral formats

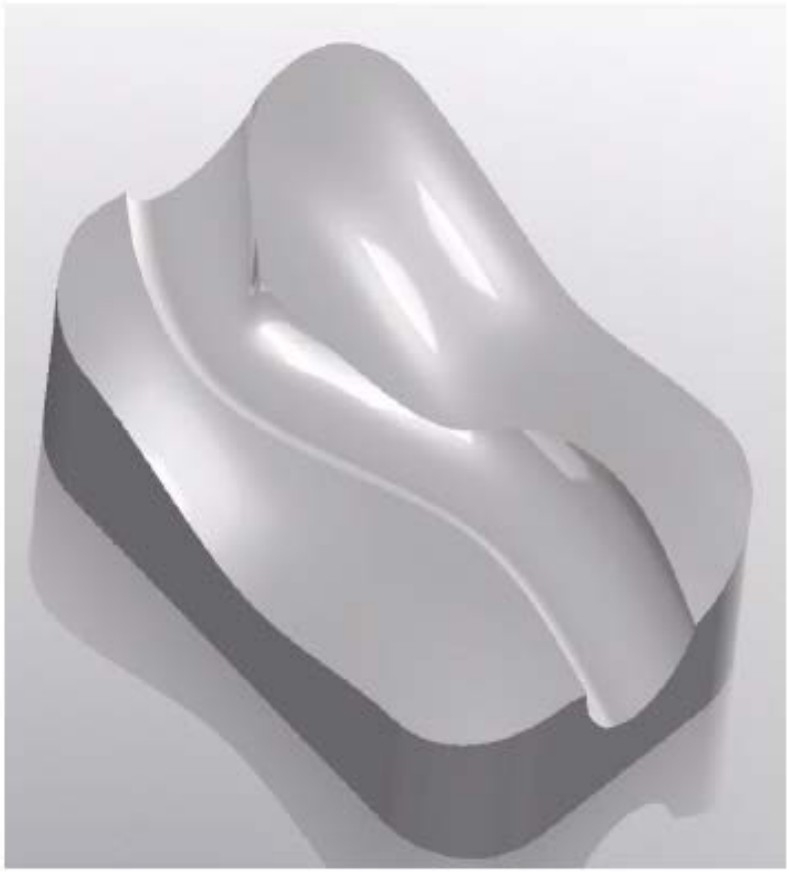
- Usually only for edge, surface and volume model transfer
- Design history is usually lost
- Data transferred usually has only limited suitability for further processing

CAD system-specific formats

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



HEIDENHAIN



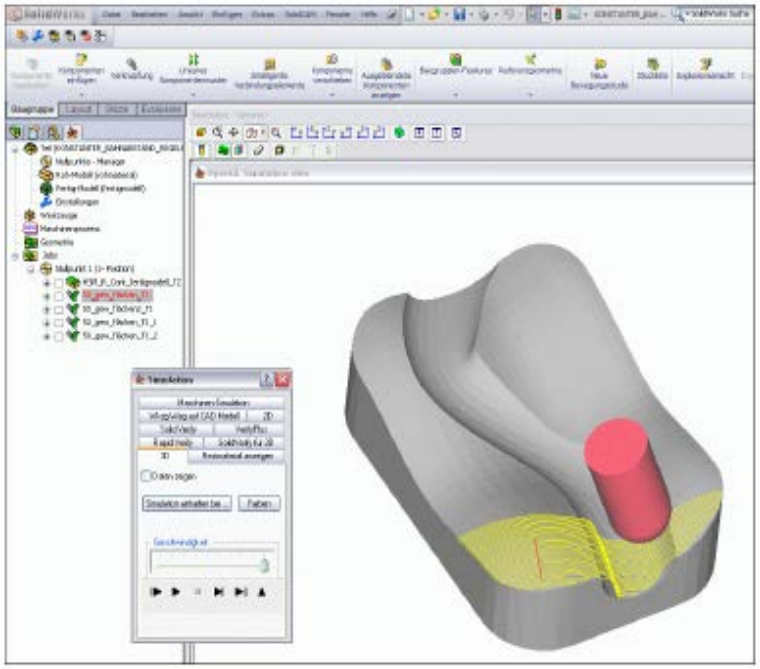
CAM

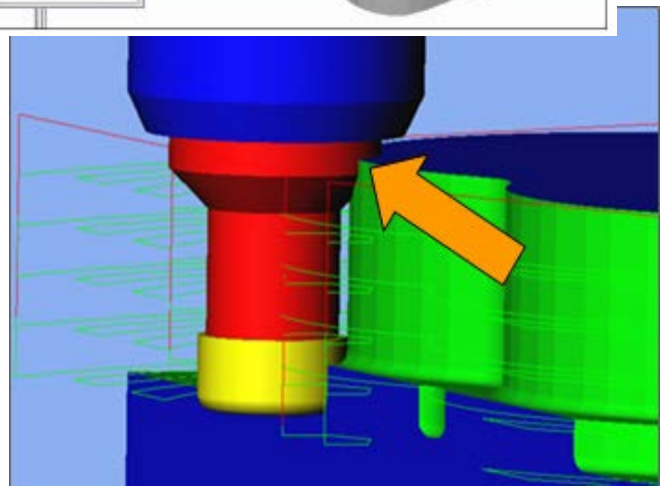
Programming

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

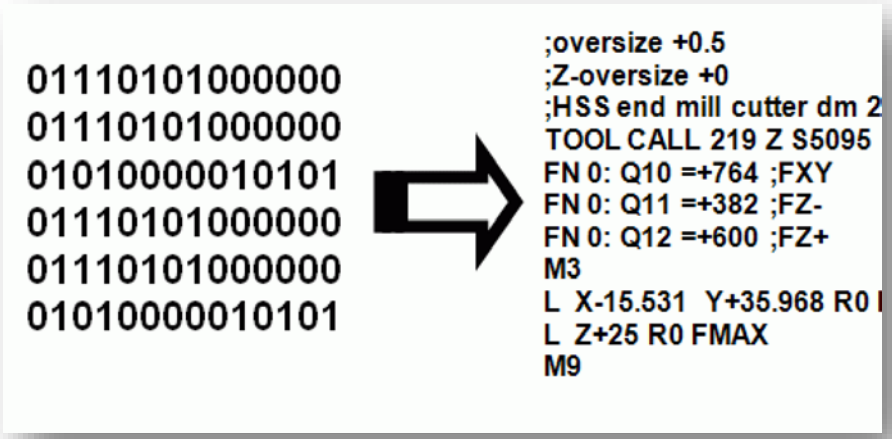




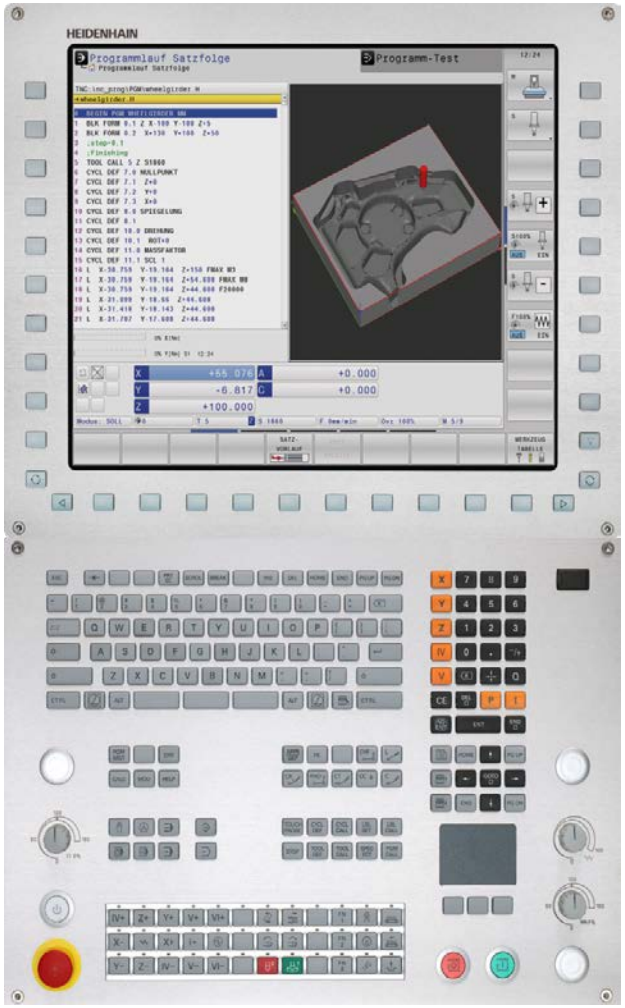
NC Program Creation (CAM)

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



HEIDENHAIN



CNC Control

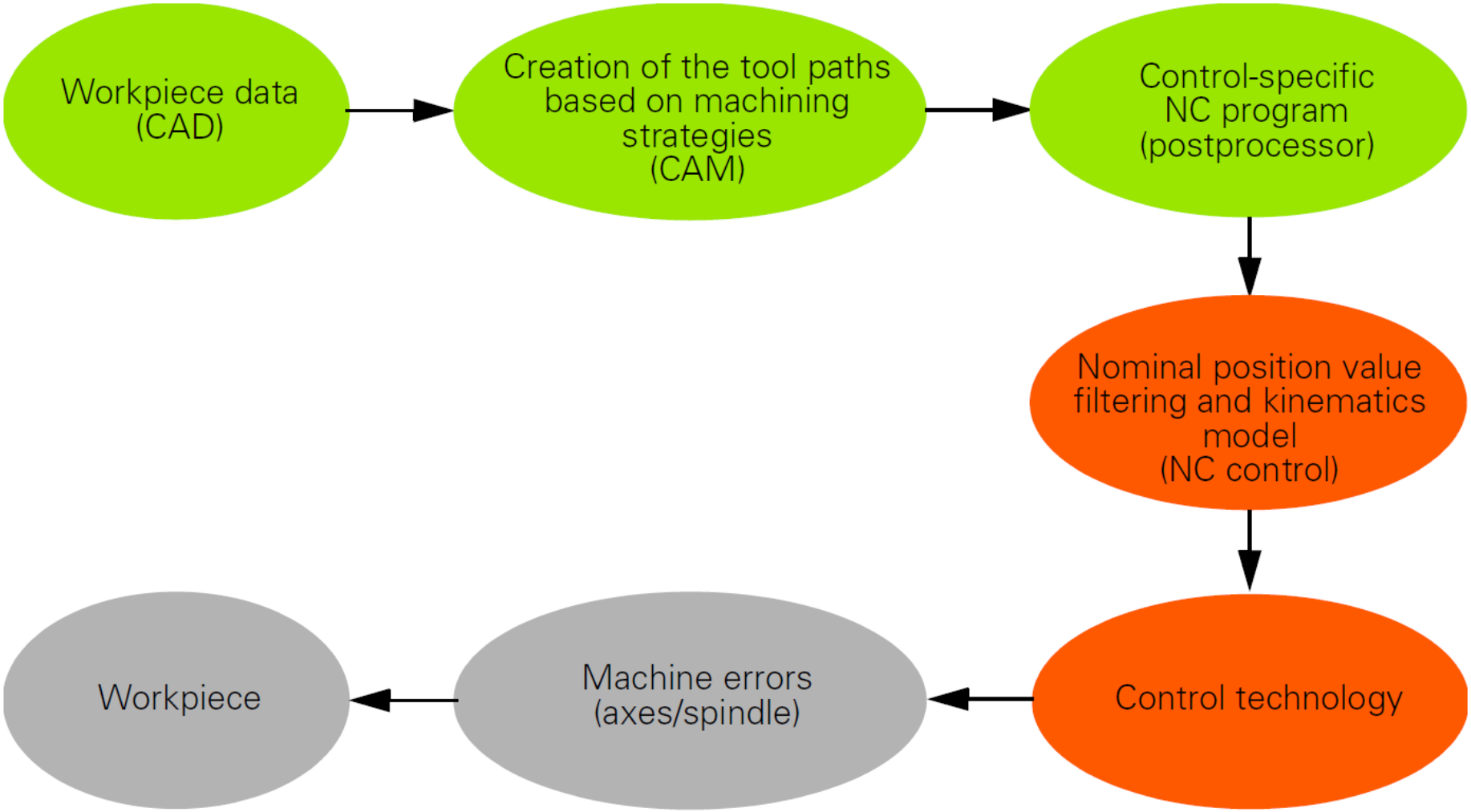
NC Program (CNC Control)

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

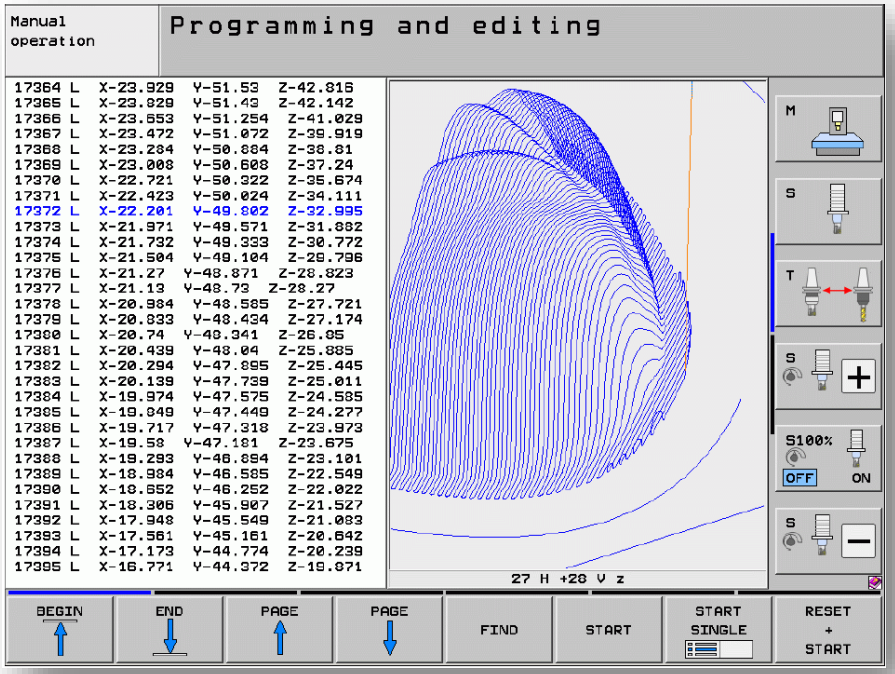
Introduced in 1976, plain language dialog is still the standard programming language for all TNC controls. When you hear people talking about shop-floor programming, they mean plain-language conversational programming from HEIDENHAIN.

- Block format for **three** axes

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

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

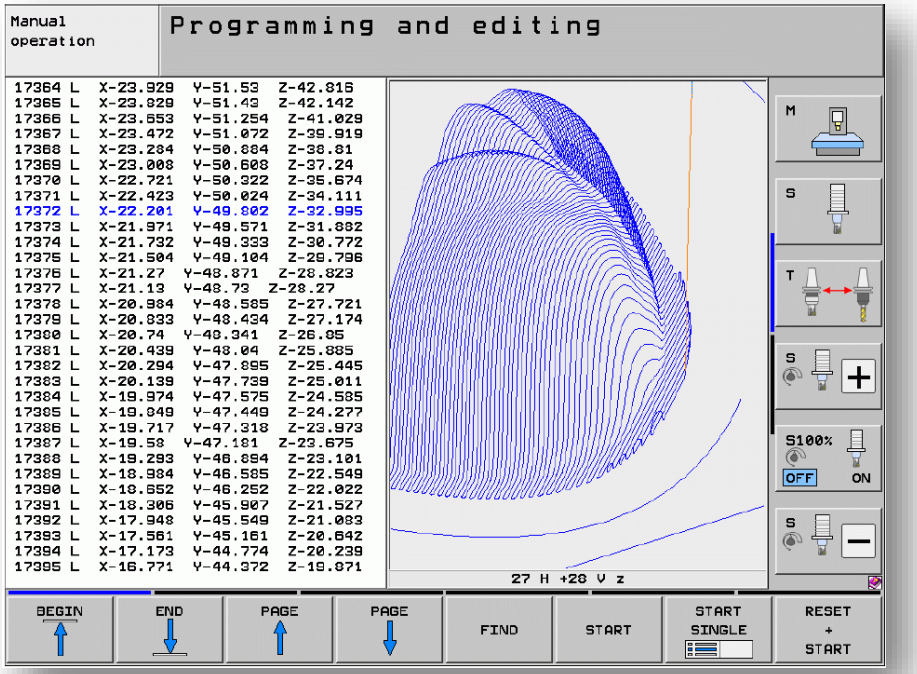
```
L X+20 Y+10 Z+2 A+15 C+6 R0 F9000
M128 F5000
```



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



Output Formats of NC Programs

Vector processing

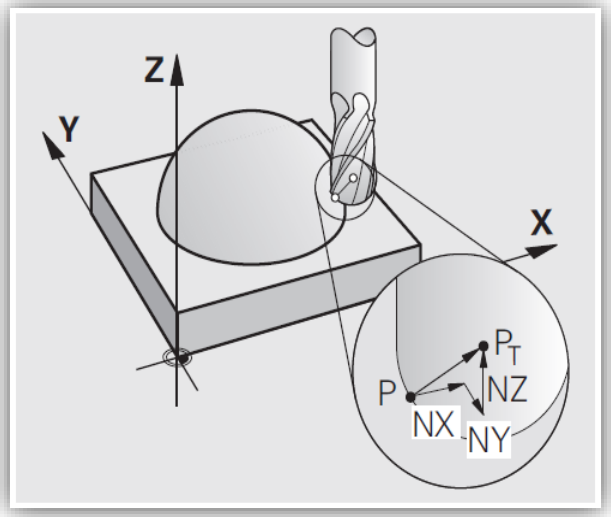
A normalized vector is a mathematical quantity with a value of 1 and any direction.

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).

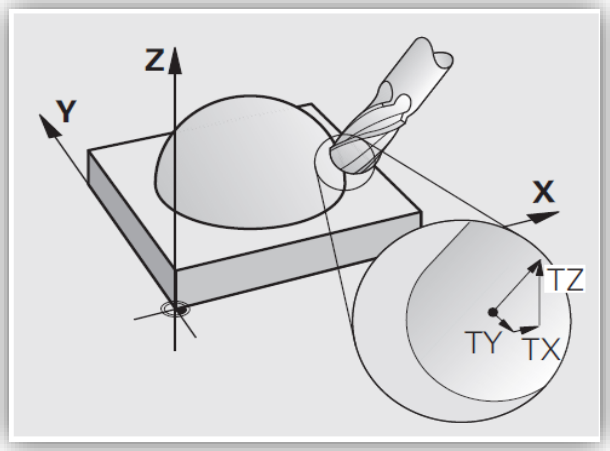
- Block format for five axes

```
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



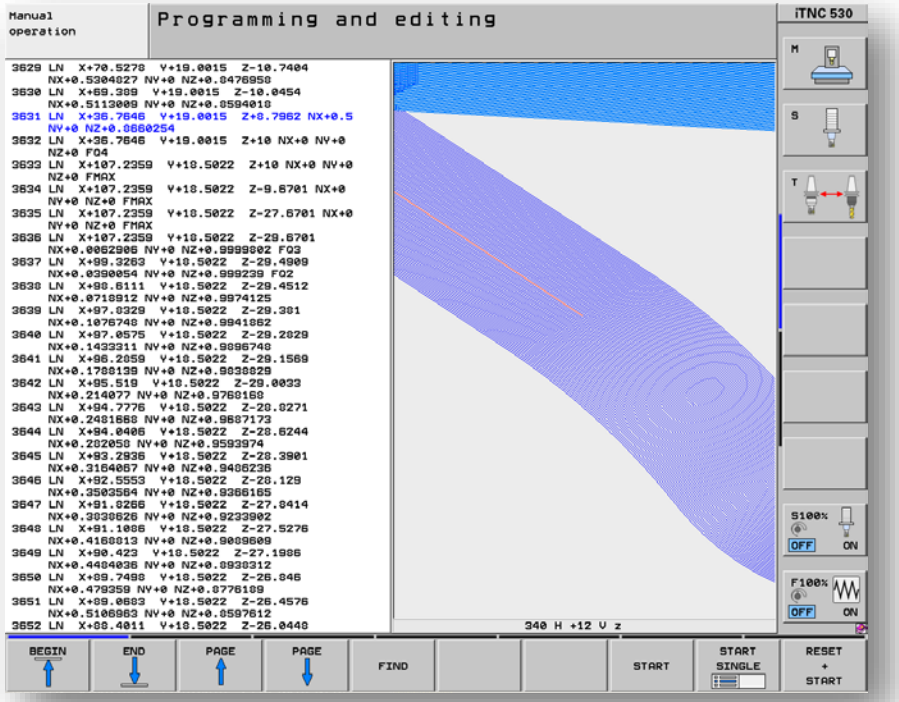
Output Formats of NC Programs

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:

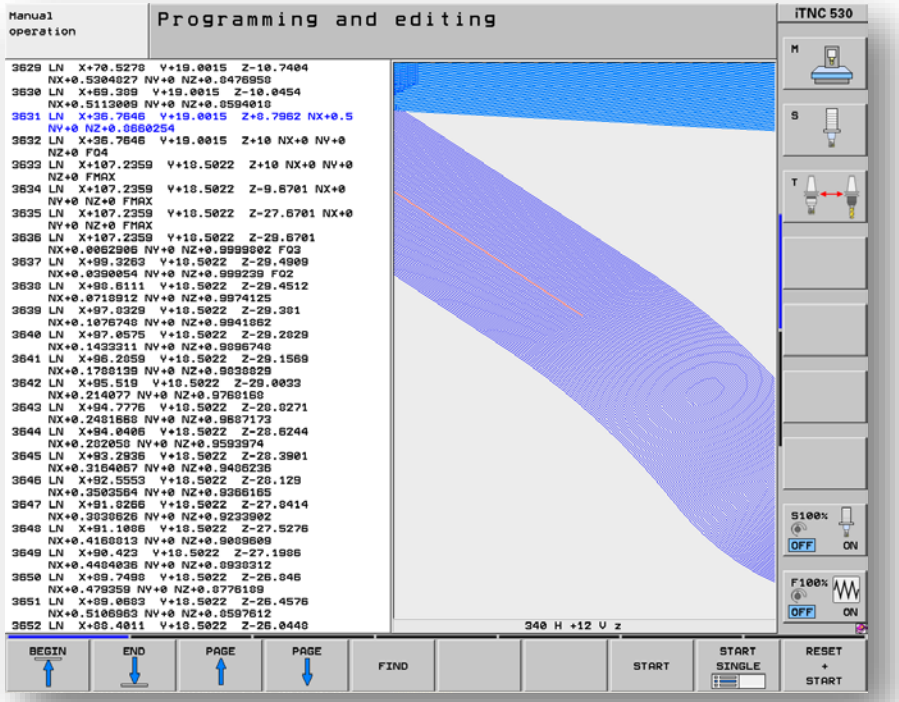
- Machine-neutral program output
- Use of 3-D radius compensation to adapt the program to real tool data (compensation via DL, DR and DR2).
- Use of the 3D-TOOLCOMP option (Option #92 – available on the iTNC 530)



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



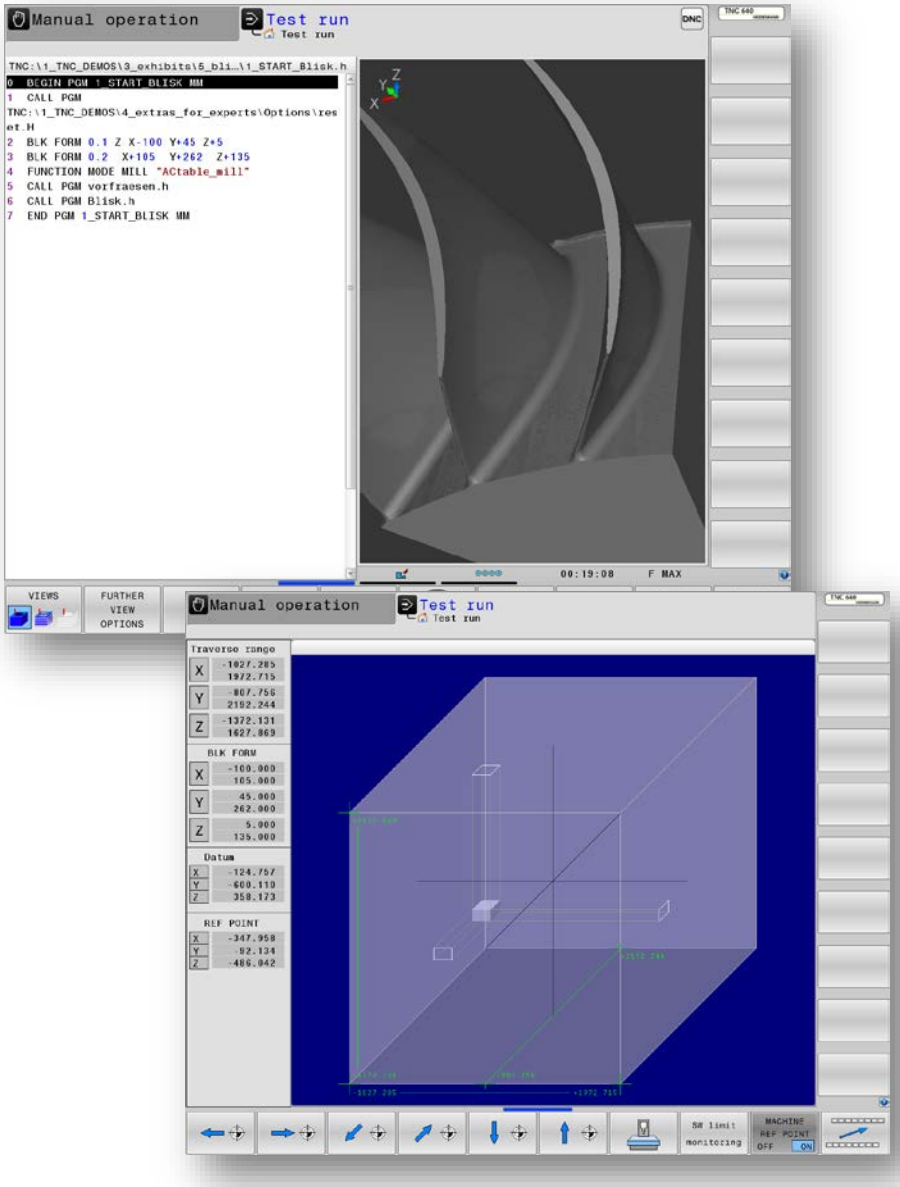
Output Formats of NC Programs

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)



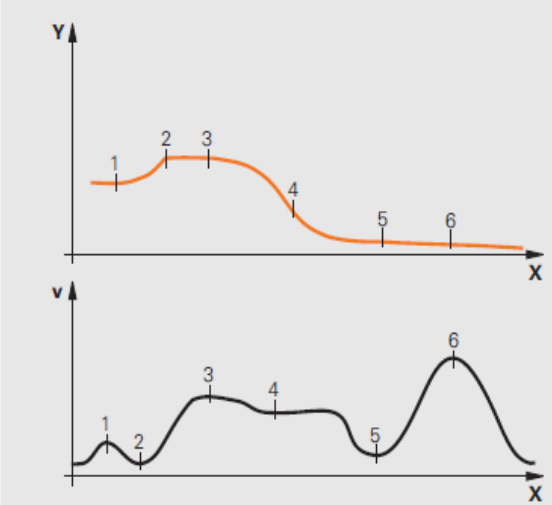
HEIDENHAIN

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

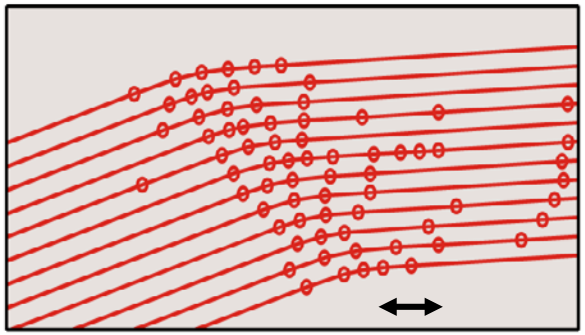
NC data with an insufficient resolution and variable point density in adjacent paths can lead to

- 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

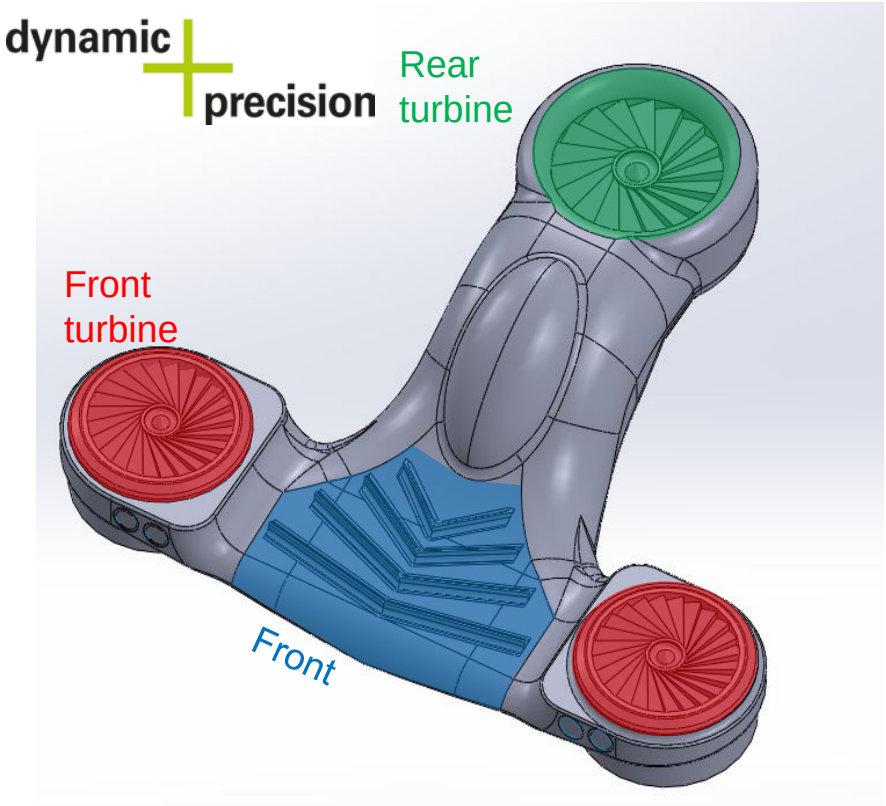
Motion Control – Application Example

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 Managment)

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

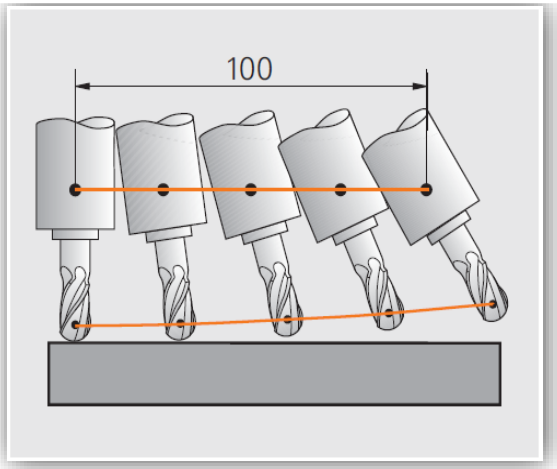
Inclination without M128:

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

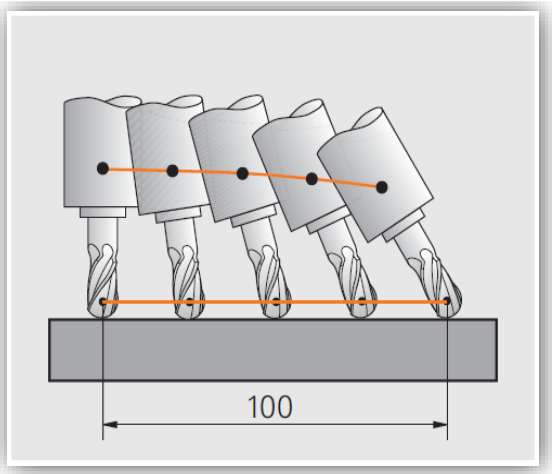
Inclination with M128:

- 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
```

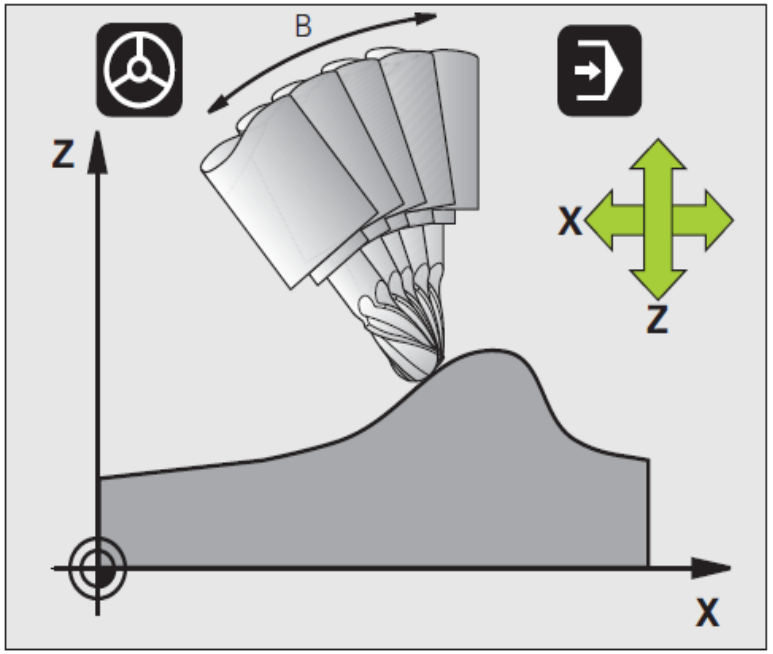


Functions for 5-Axis Machining

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.

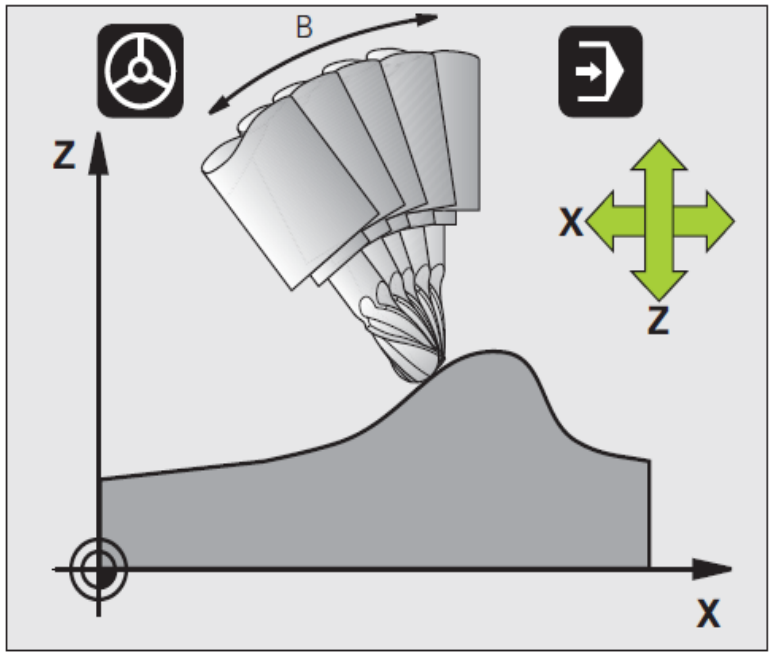
- Feed rate after M128 limits the feed rate for the compensation movement
- M129 resets M128
- The feed rate calculation can be selected via machine parameters



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)

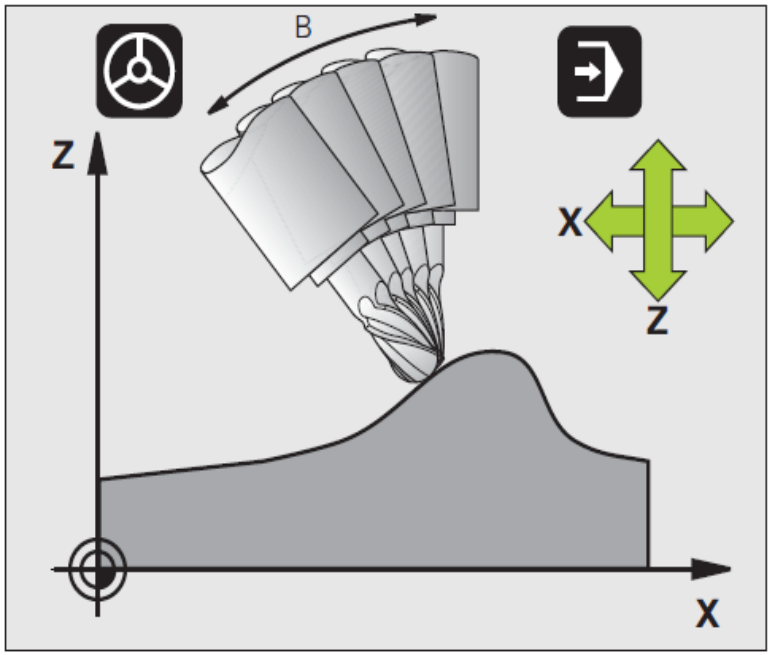


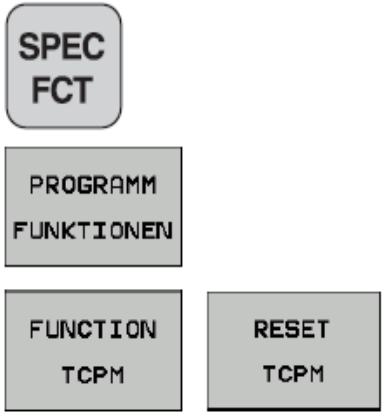
Functions for 5-Axis Machining

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





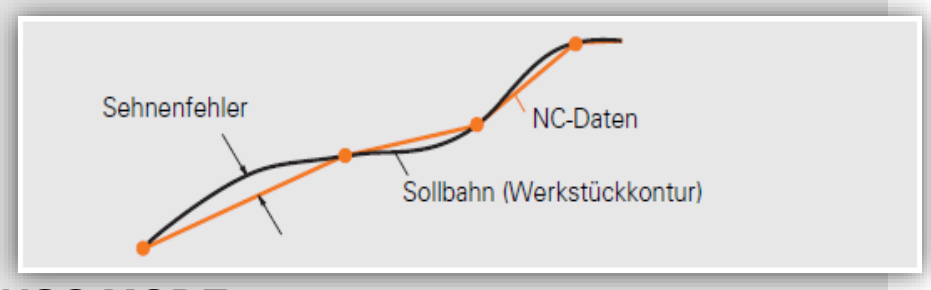
Functions for 5-Axis Machining

Cycle 32

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

Tolerance value T

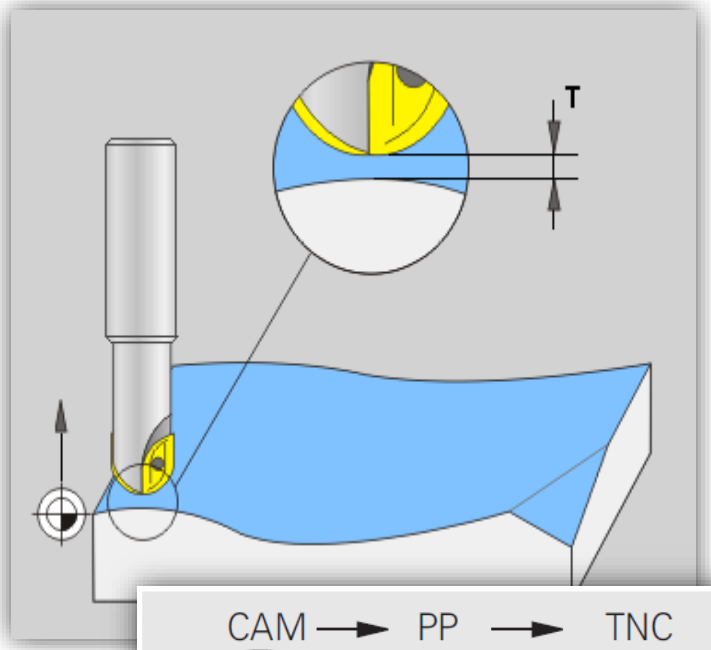
- Permissible contour deviation (1.1 to 2x chord tolerance from the CAM)

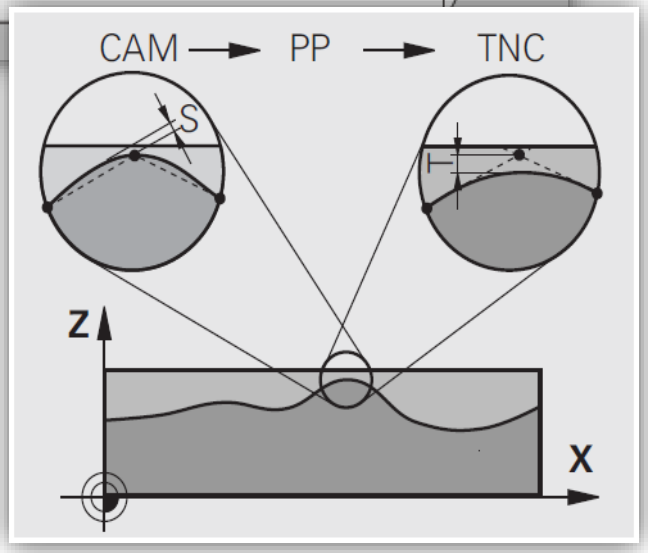


HSC MODE

- HSC MODE:0
- HSC MODE:1

  → Only effective with active HSC filter or Advanced HSC filter





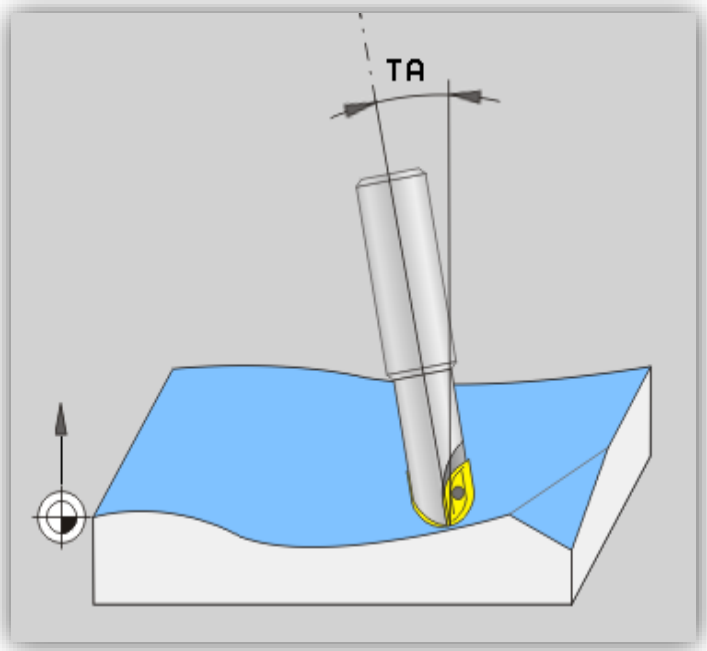
Functions for 5-Axis Machining

Cycle 32

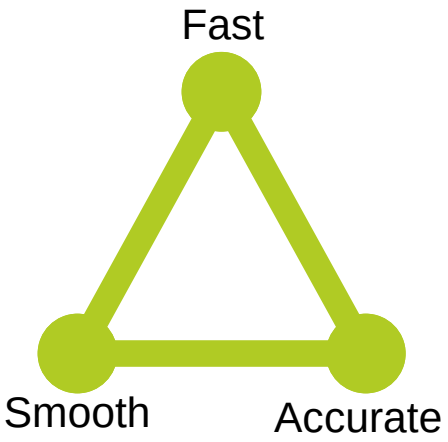
Modification of machining result with regard to precision, surface quality and speed

Tolerance for rotary axes

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



Functions for 5-Axis Machining



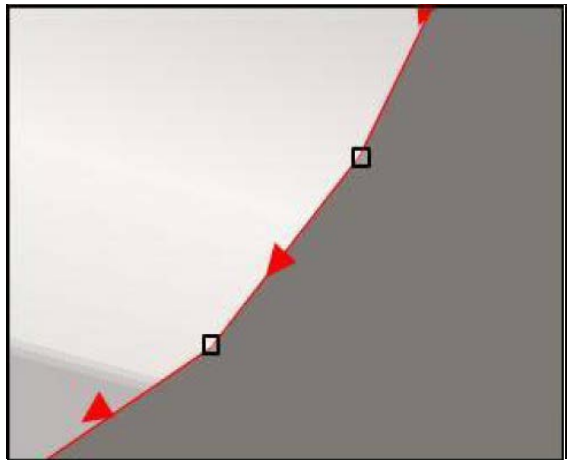
- Preference: **Rapid machining / roughing**
  - Large Cycle 32 tolerance
  - Select MP settings with higher jerk/accelerations
  - NC data with less data density is possible
  - → Poorer surface/precision ➔ **oversize**
- Preference: **Precise machining**
  - Small Cycle 32 tolerance
  - Select MP settings 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:

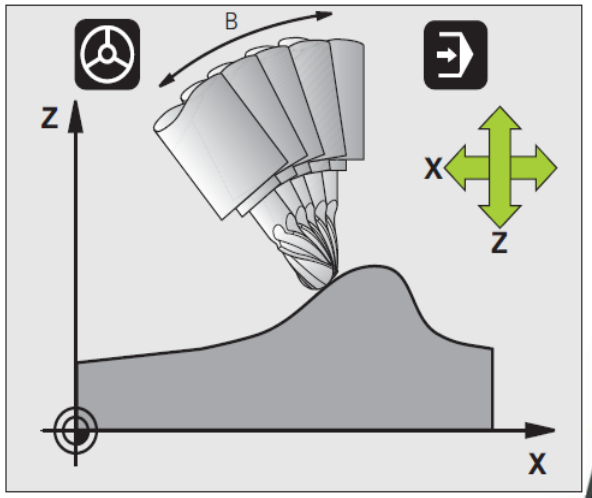
In Parameter T you define a minimal point distance, up to which the TNC does not consider points during processing

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 superimpositioning 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

$$\frac{point\ distance\ [mm] * 1000 * 60}{block\ processing\ [ms]} = Fmax[\frac{mm}{min}]$$

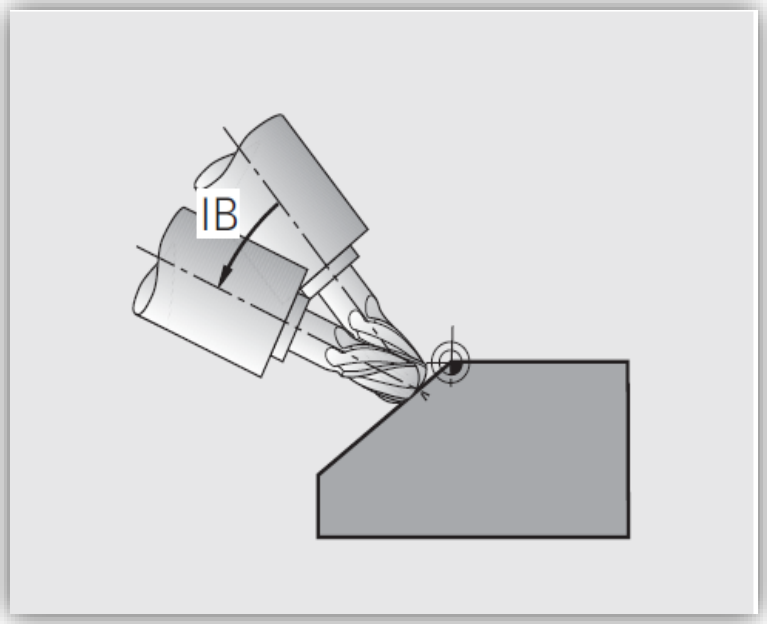
$$\frac{0.2mm * 1000 * 60}{0.5ms} = 24000\frac{mm}{\text{min}}$$

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



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

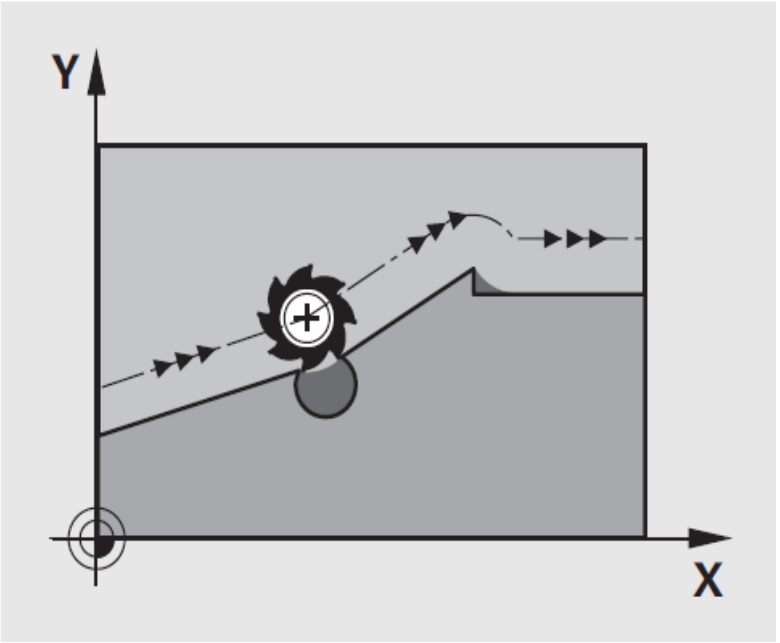
M120 must be in an NC block that

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



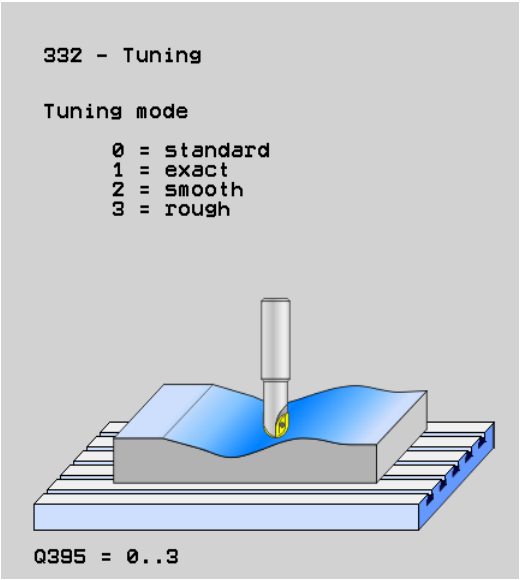
Functions for 5-Axis Machining

"Tuning" cycle

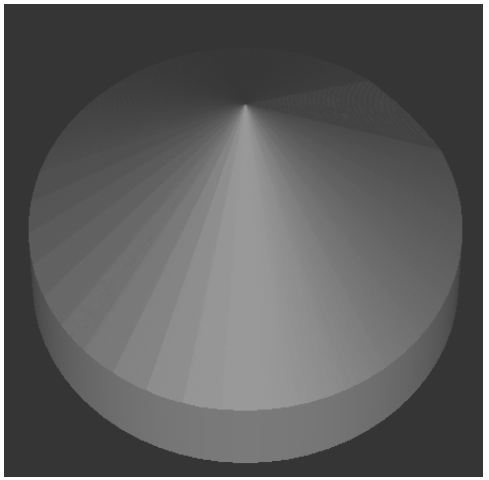
Activation of various MP subfiles via user cycles.

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



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