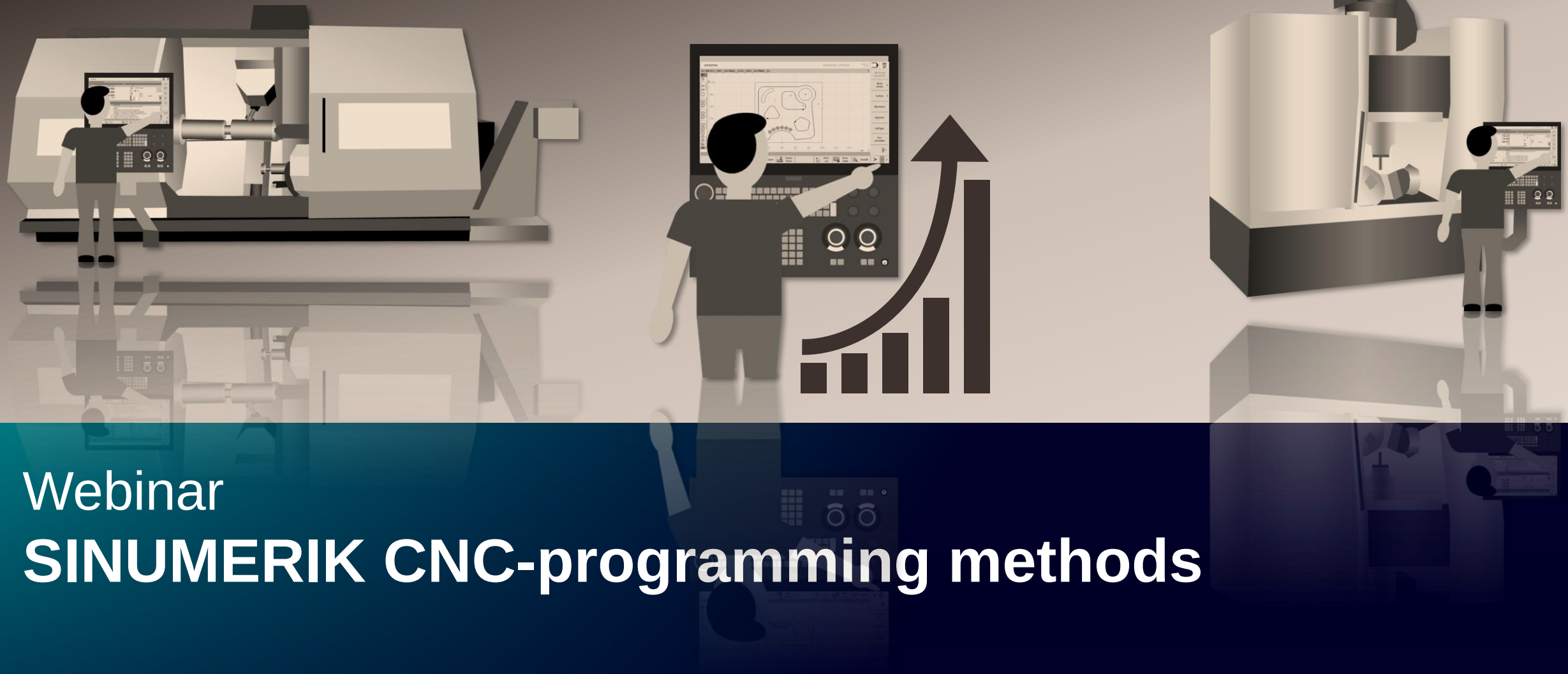


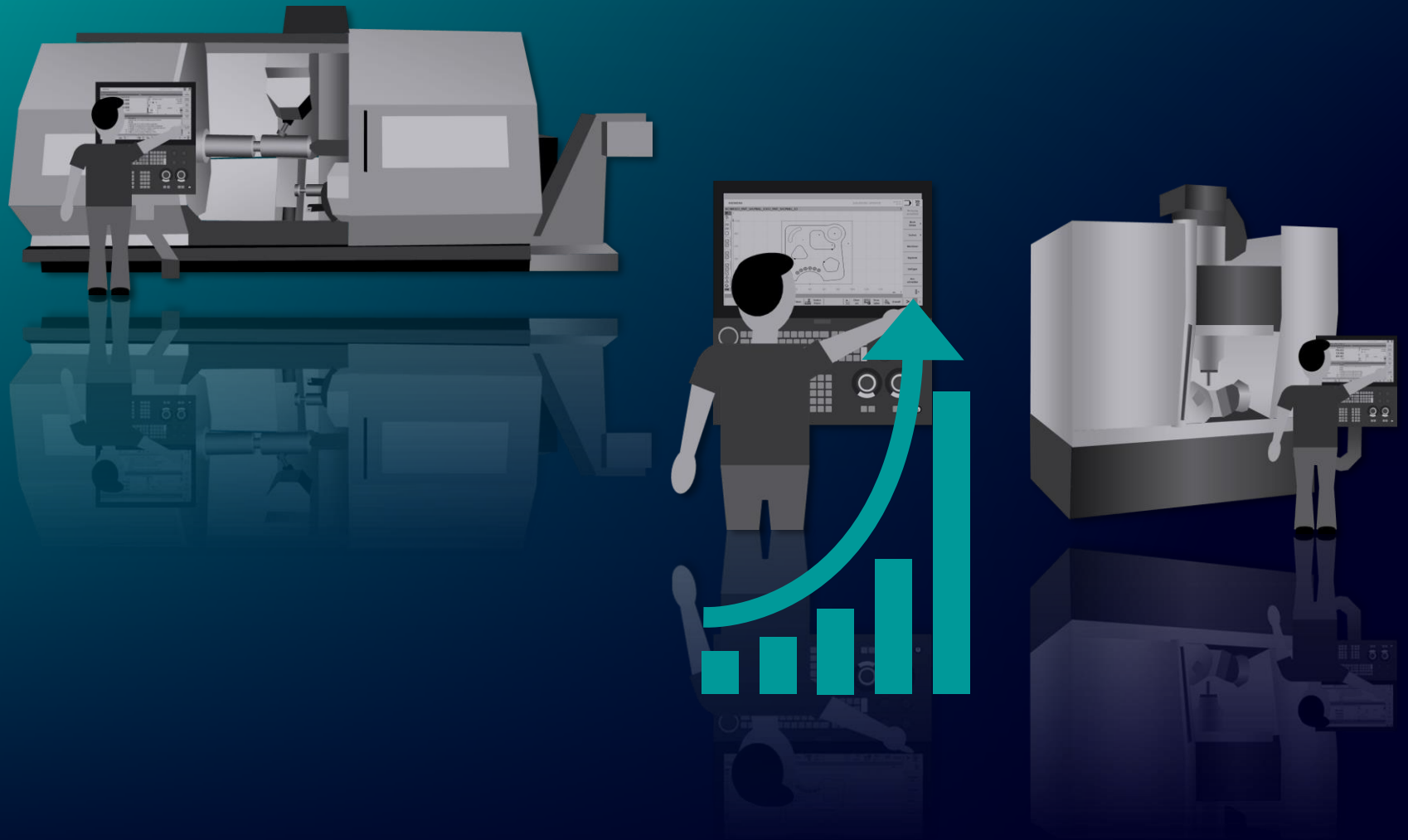
Webinar

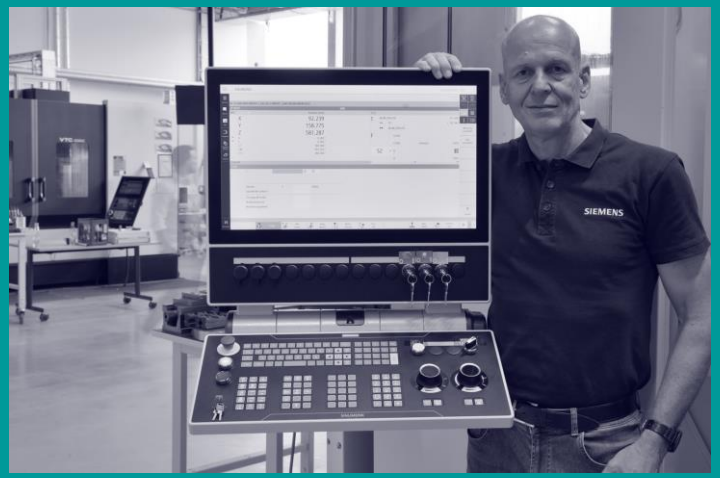
SINUMERIK CNC-programming methods

From work-steps to CNC-high level language – universal CNC-programming for every order!

SIEMENS

Webinar
- SINUMERIK CNC-programming methods





Andreas Grözinger

SIEMENS Digital Industries

Machine Tool Systems

Promotor for CNC-equipment
and digitalization of machine tools

SIEMENS

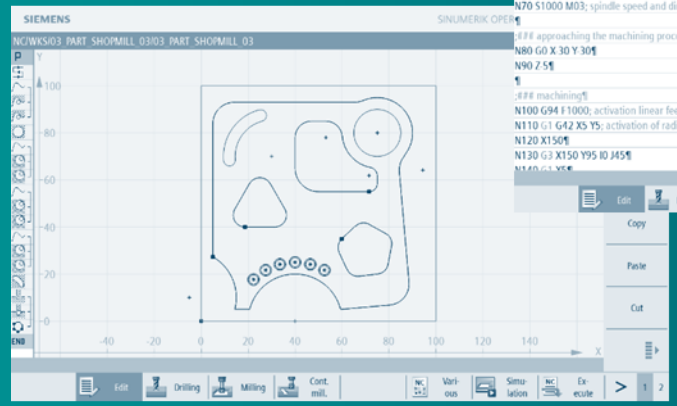
Agenda

1. **Overview**
2. Work-step programming ShopMill / ShopTurn
3. G-Code
4. programGUIDE
5. Machining cycles (ShopMill / ShopTurn, programGUIDE)
6. Offline-programming in the CNC-job preparation
7. CNC4you

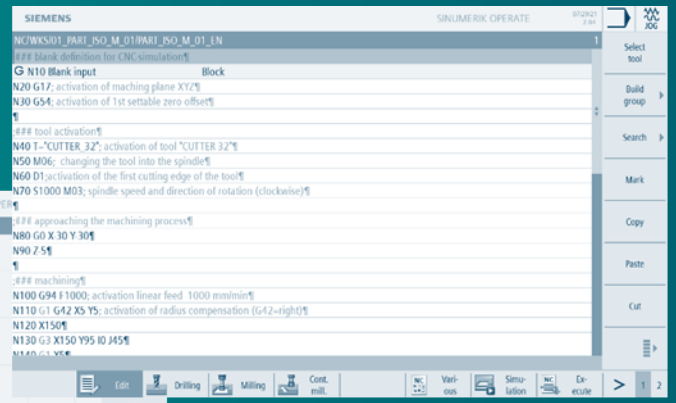
SIEMENS

SINUMERIK
- universal CNC programming for any job





ShopMill / ShopTurn machining step programming



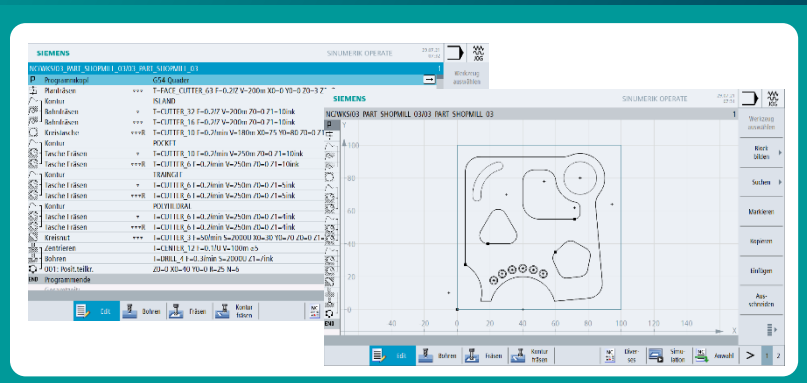
G-Code programming



A considerable benefit of SINUMERIK is having **two CNC programming methods which are well established in the market!**
An **WP-oriented**, highly productive **G-code programming** as well as a **workshop-oriented machining step programming**.

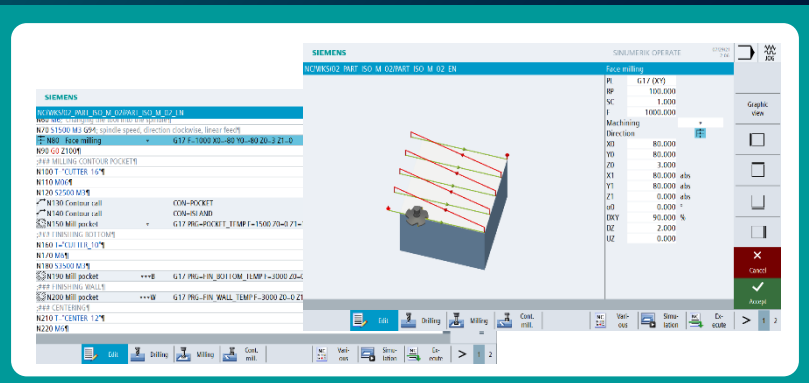
SIEMENS

SINUMERIK Operate
- perfectly prepared for all production tasks



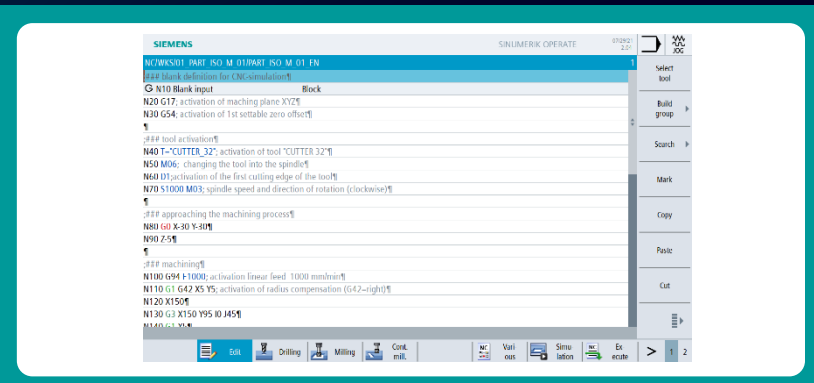
ShopMill / ShopTurn work-steps

- individual parts
- small series



programGUIDE

- mid-sized series



G-code programming

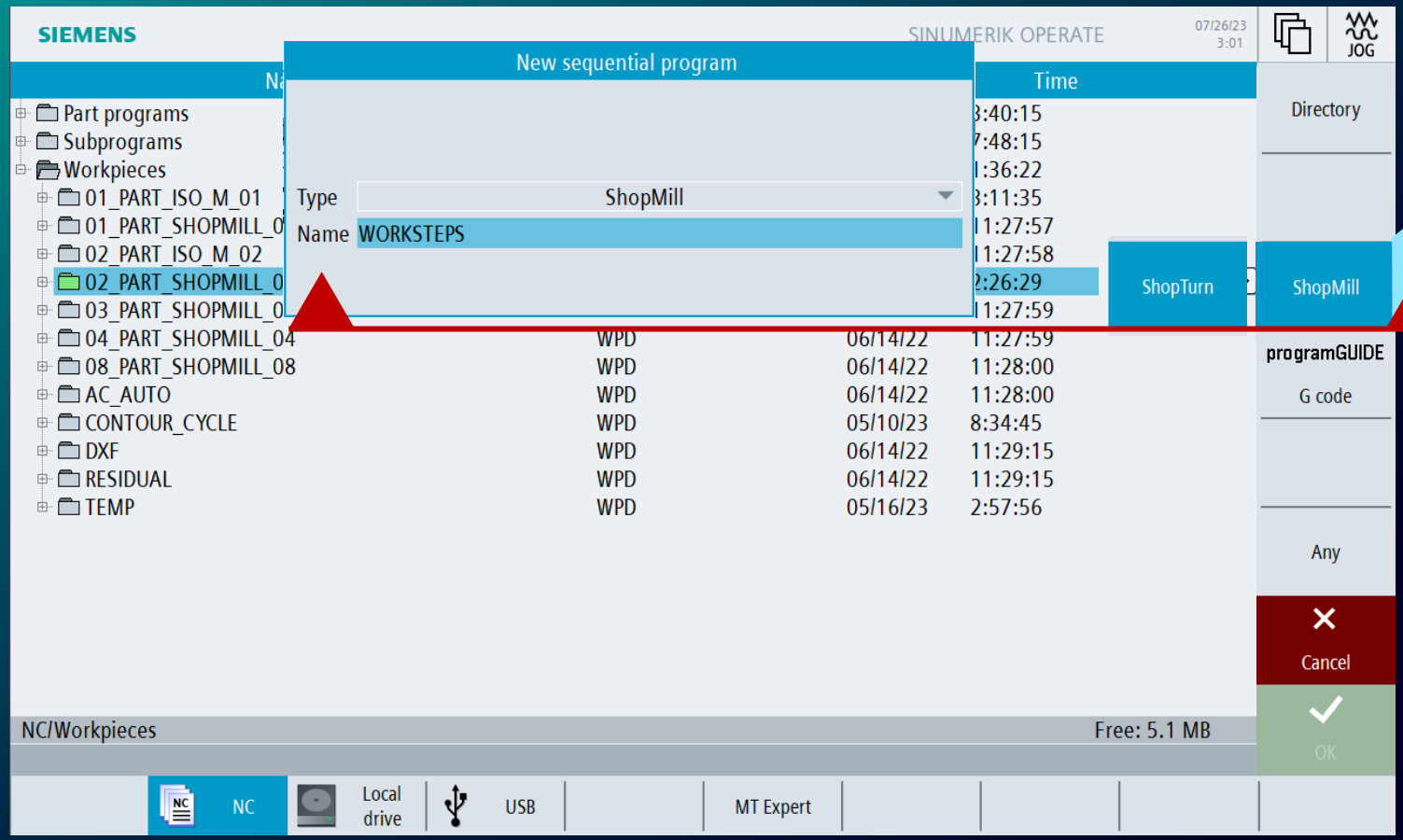
- large series
- special applications

SINUMERIK Operate

SINUMERIK Operate offers a **broad and consistent range of** CNC programming that is **unique** in the market.
Fully integrated and **without technological limits!**

SIEMENS

CNC-Programming methods
- the basic direction

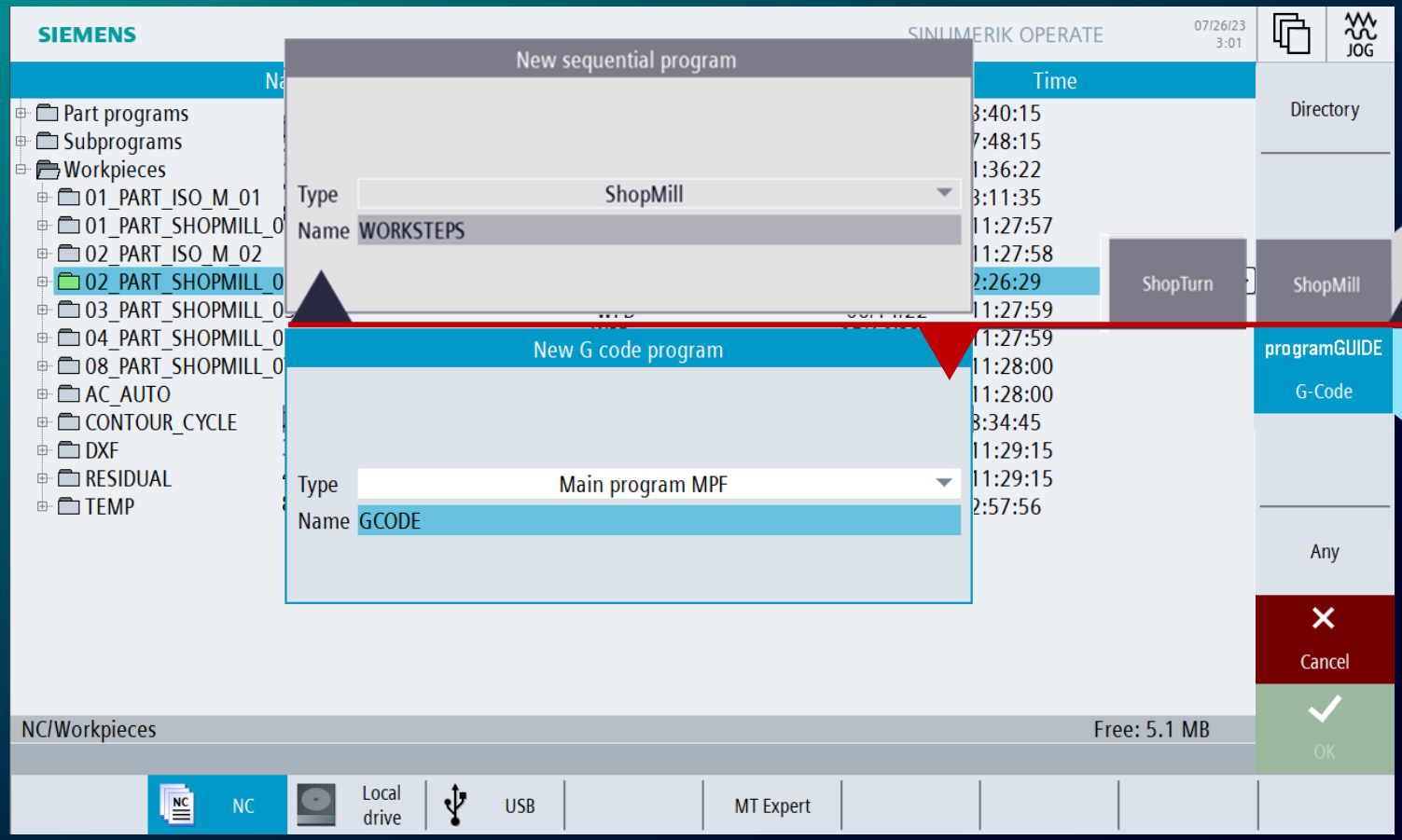


Alternative 1
ShopMill / ShopTurn
Work-steps

When the CNC program is created, the programming **method** used to create the CNC program is **determined**.

SIEMENS

CNC-Programming methods
- the basic direction



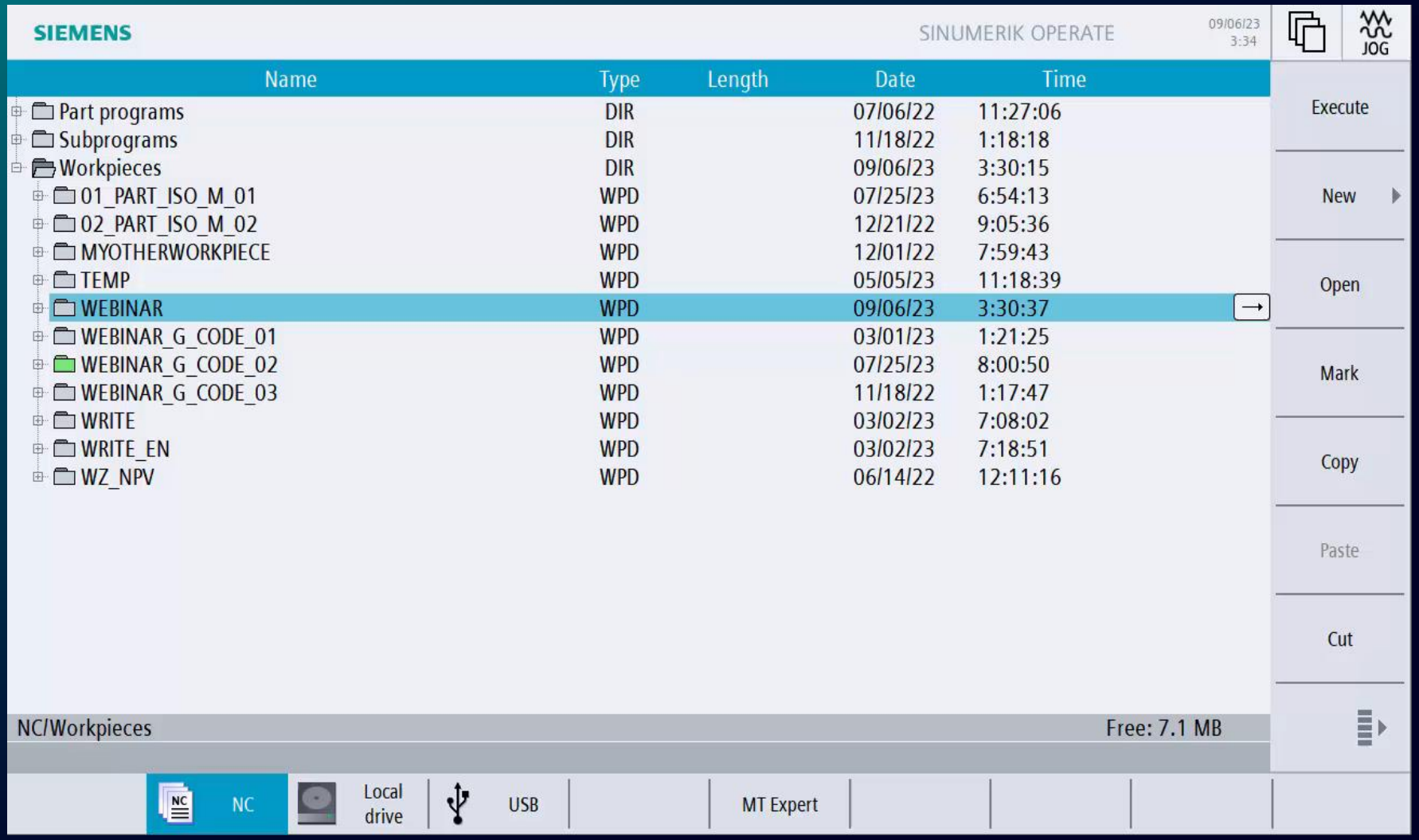
Alternative 1
ShopMill / ShopTurn worksteps

Alternative 2
- G-Code-programming
- With cycle support programGUIDE

When the CNC program is created, the programming **method** used to create the CNC program is **determined**.

SIEMENS

CNC-Programming methods
- the basic direction



SIEMENS

Agenda

1. Overview
2. **Work-step programming ShopMill / ShopTurn**
3. G-Code
4. programGUIDE
5. Machining cycles (ShopMill / ShopTurn, programGUIDE)
6. Offline-programming in the CNC-job preparation
7. CNC4you

SIEMENS

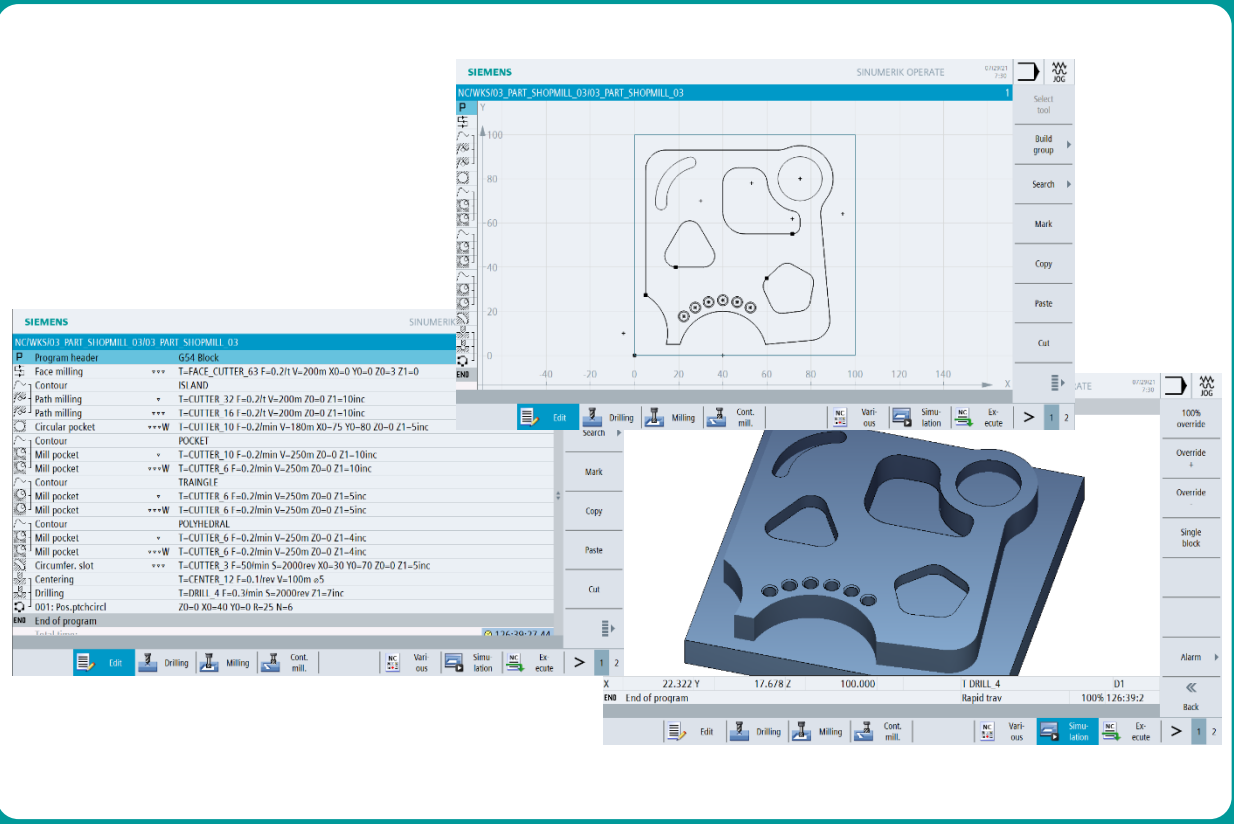
ShopMill / ShopTurn
- merged completely into SINUMERIK Operate



The originally completely **independent user interfaces ShopMill and ShopTurn** have meanwhile been completely **merged** into **SINUMERIK Operate** as technological forms.

SIEMENS

ShopMill work-step programming
- the specialist for individual parts and small series



program creation in work-steps designed for machining experts

switch to SINUMERIK without hurdles due to CNC programming without ISO knowledge

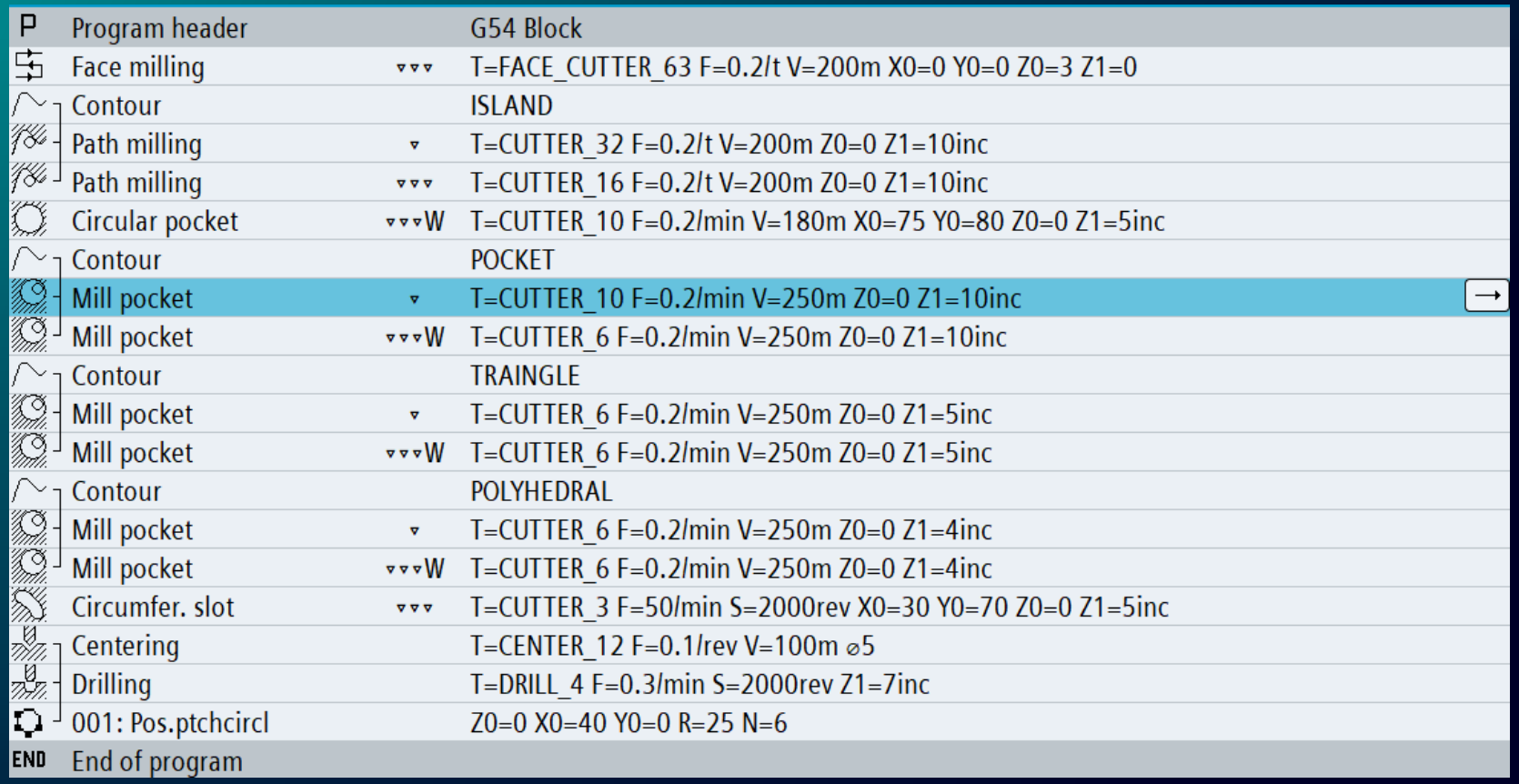
safe programming through automatic consistency check of the inputs

dynamic line graphics for immediate comparison of CNC program and workpiece drawing

Especially for **individual parts and small series**, **programming time** is the **productivity-determining factor**.
Therefore, SINUMERIK Operate offers graphically interactive **CNC programming in work-steps**
for the main technologies **milling and turning**.

SIEMENS

Expert-oriented program creation
- like programmers think!

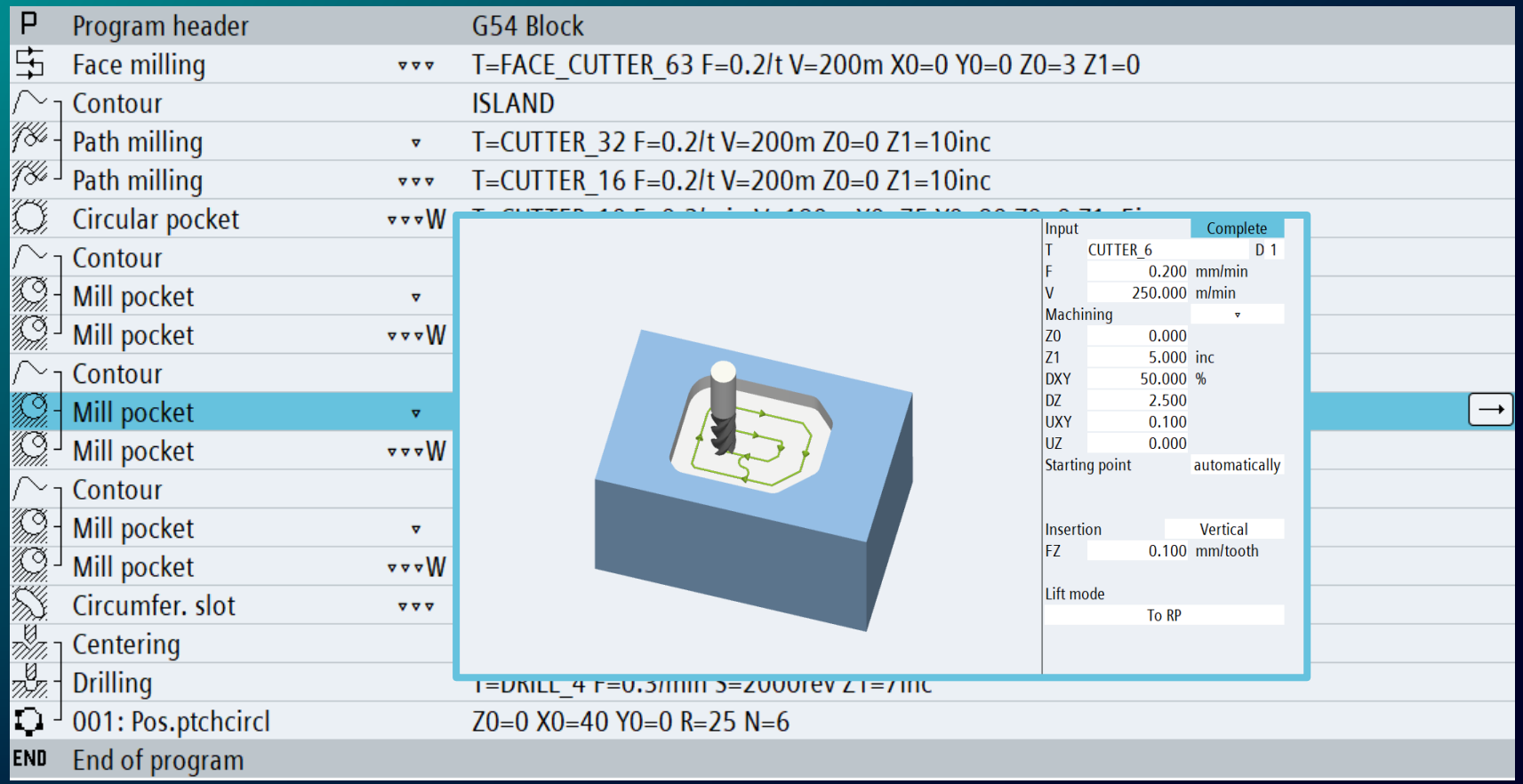


Clear representation in work-steps

Work-steps in ShopMill and ShopTurn represent the steps into which the **specialist** on the machine **mentally divides the machining**.

SIEMENS

Programming without ISO-knowledge
- changeover without hurdles



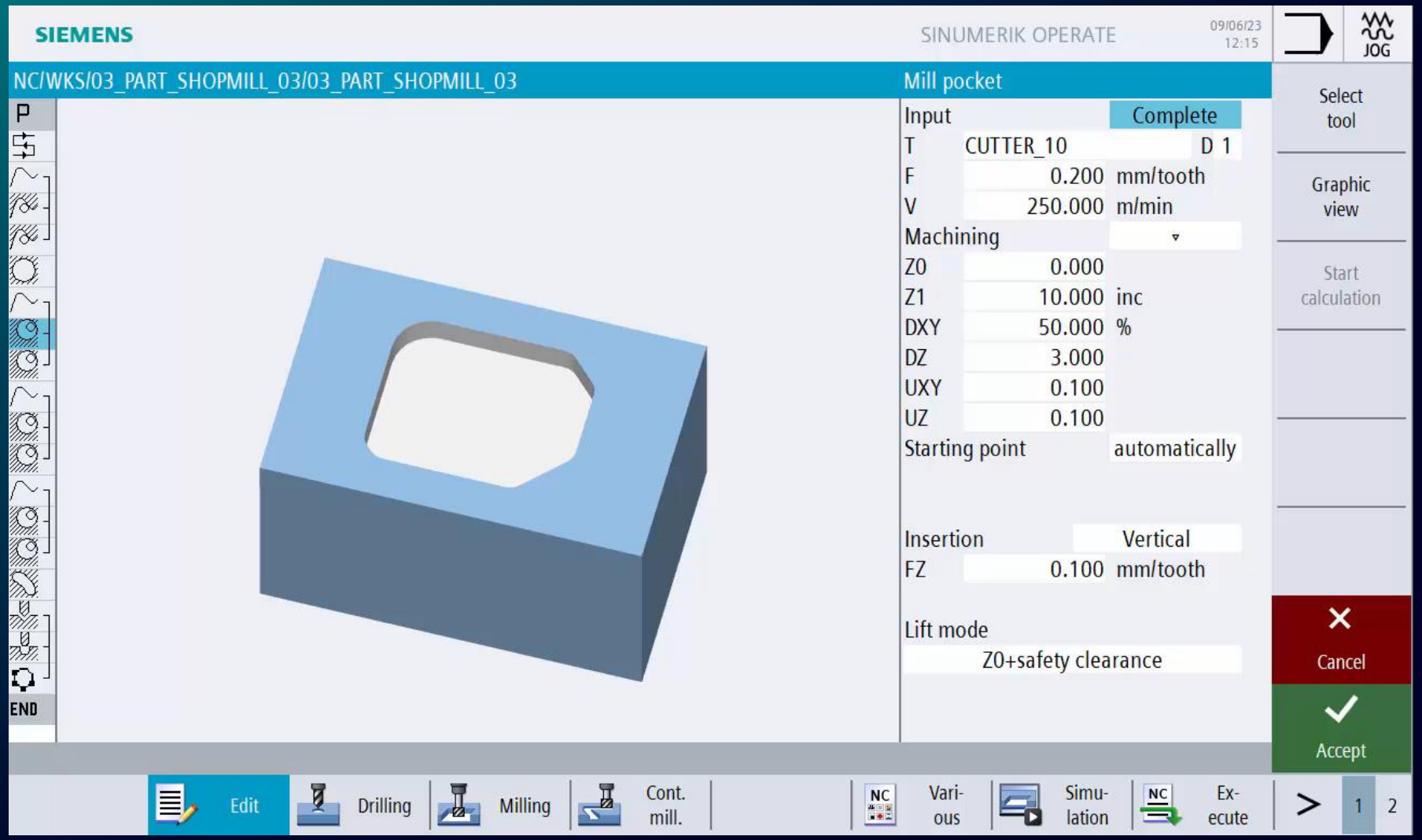
Programming without ISO-knowledge

+

Even CNC programs for the **most demanding machining tasks** can be created **without any knowledge of ISO** – switching to SINUMERIK without any hurdles!

SIEMENS

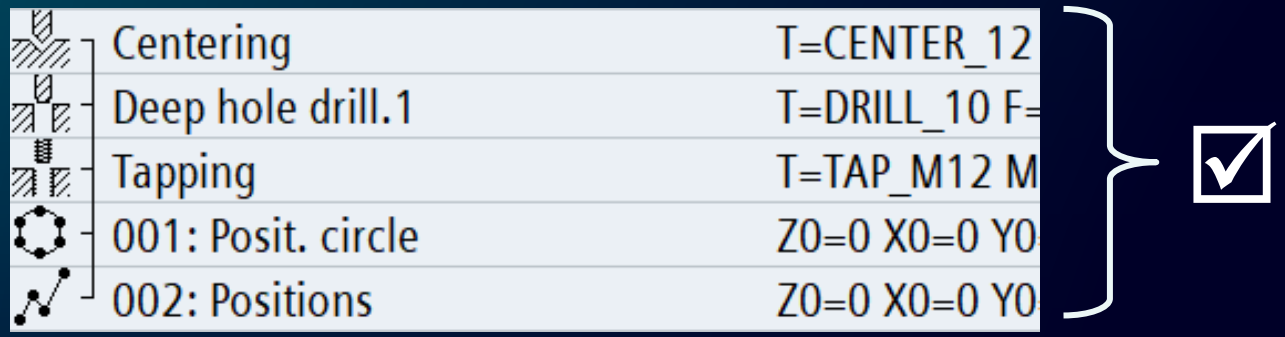
Programming without ISO-knowledge
- changeover without hurdles



SIEMENS

Consistency check
- more security when programming





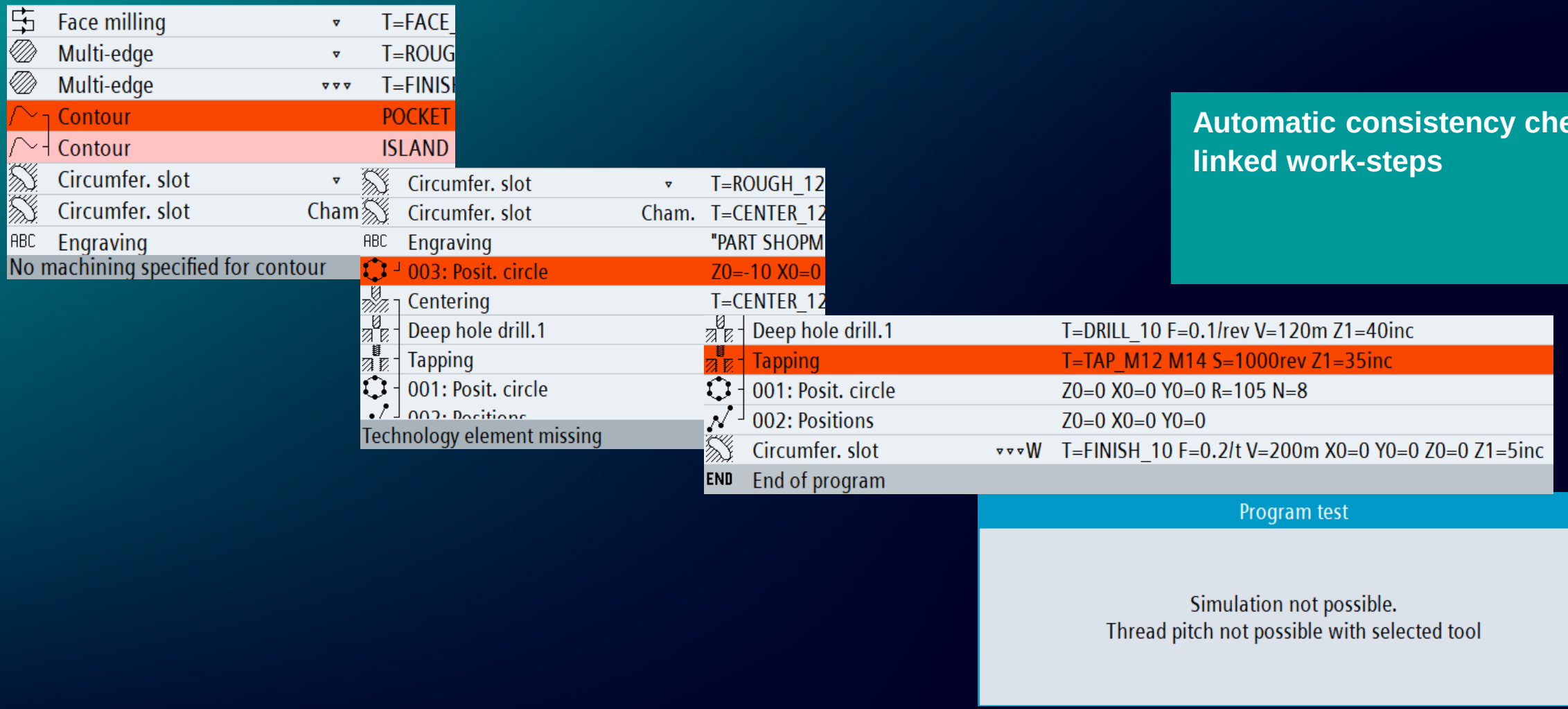
Automatic consistency check of linked work-steps

Linked work-steps are automatically **checked for consistency**. The graphic work-step programming thus provides **additional security when creating the CNC program**.

.

SIEMENS

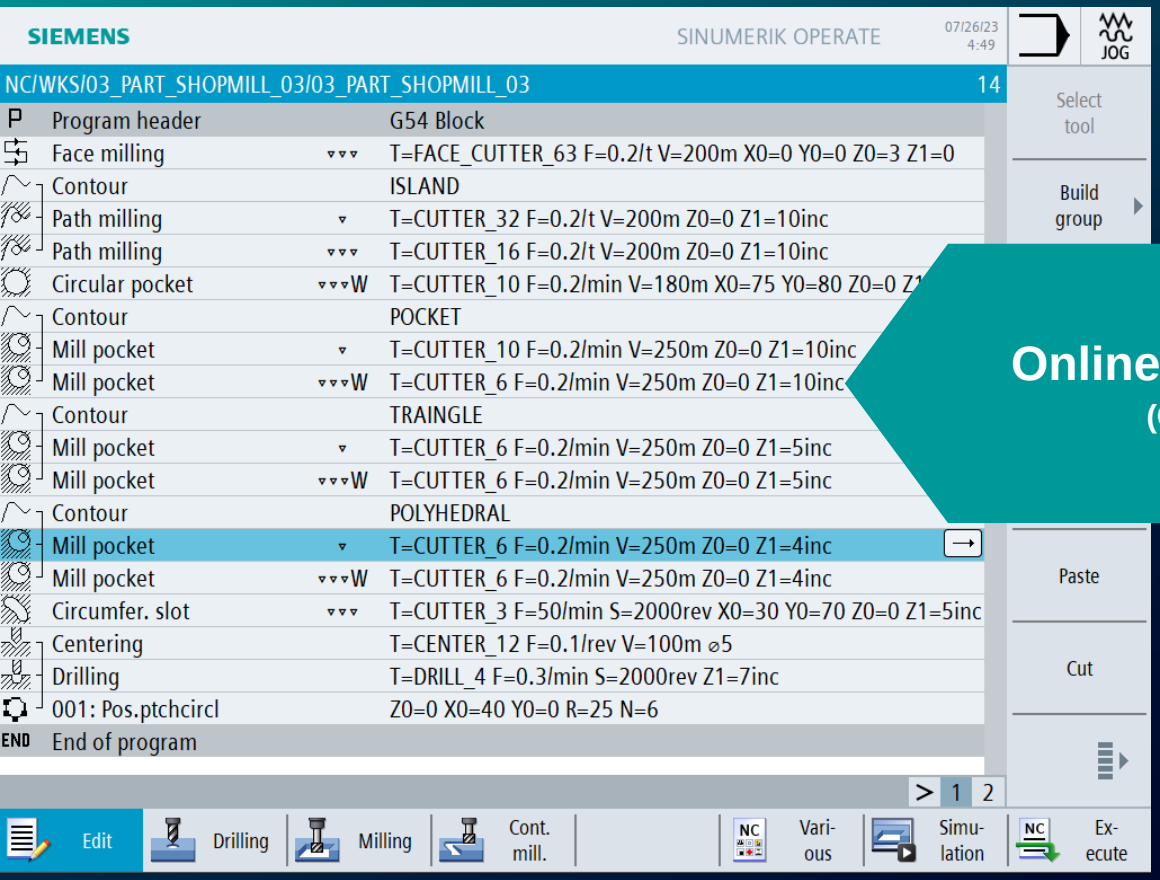
Consistency check
- more security when programming



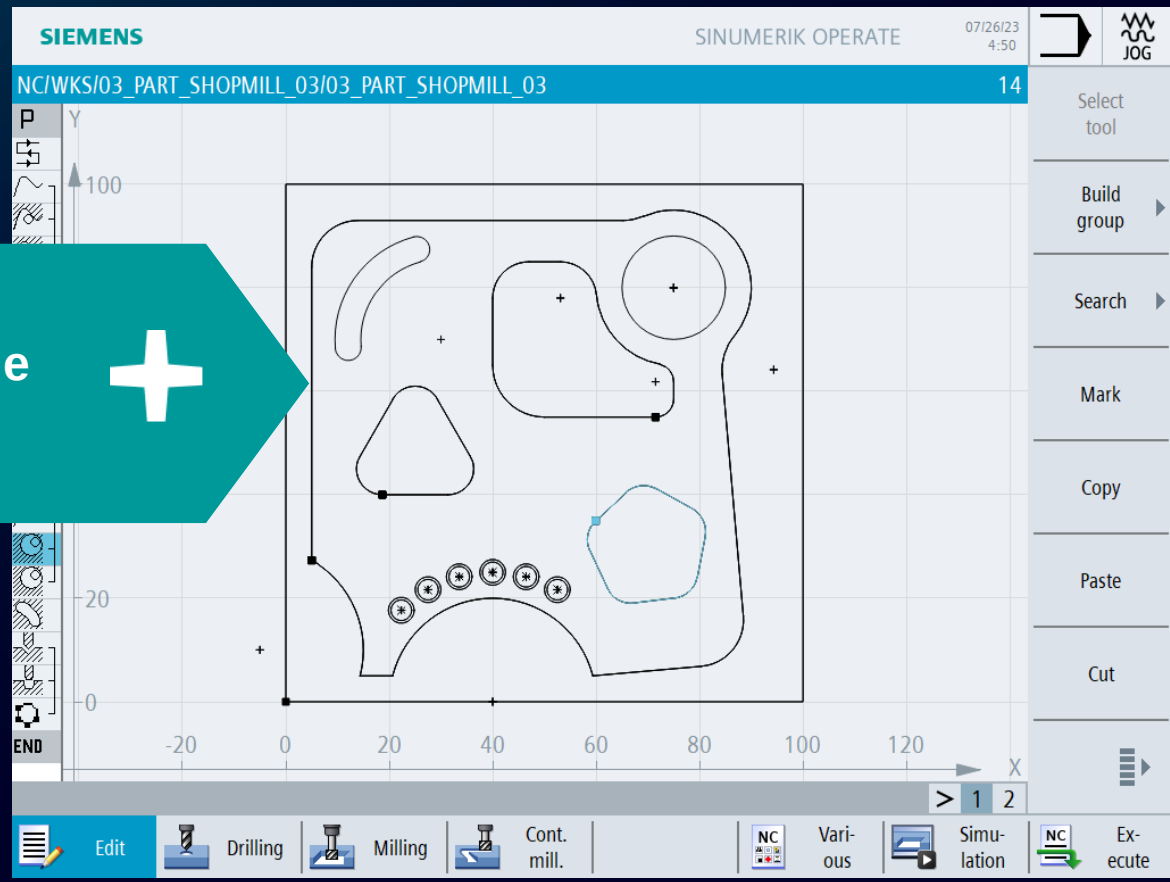
Automatic consistency check of linked work-steps

SIEMENS

Dynamic line graphics (1)
- the immediate comparison with the workpiece drawing



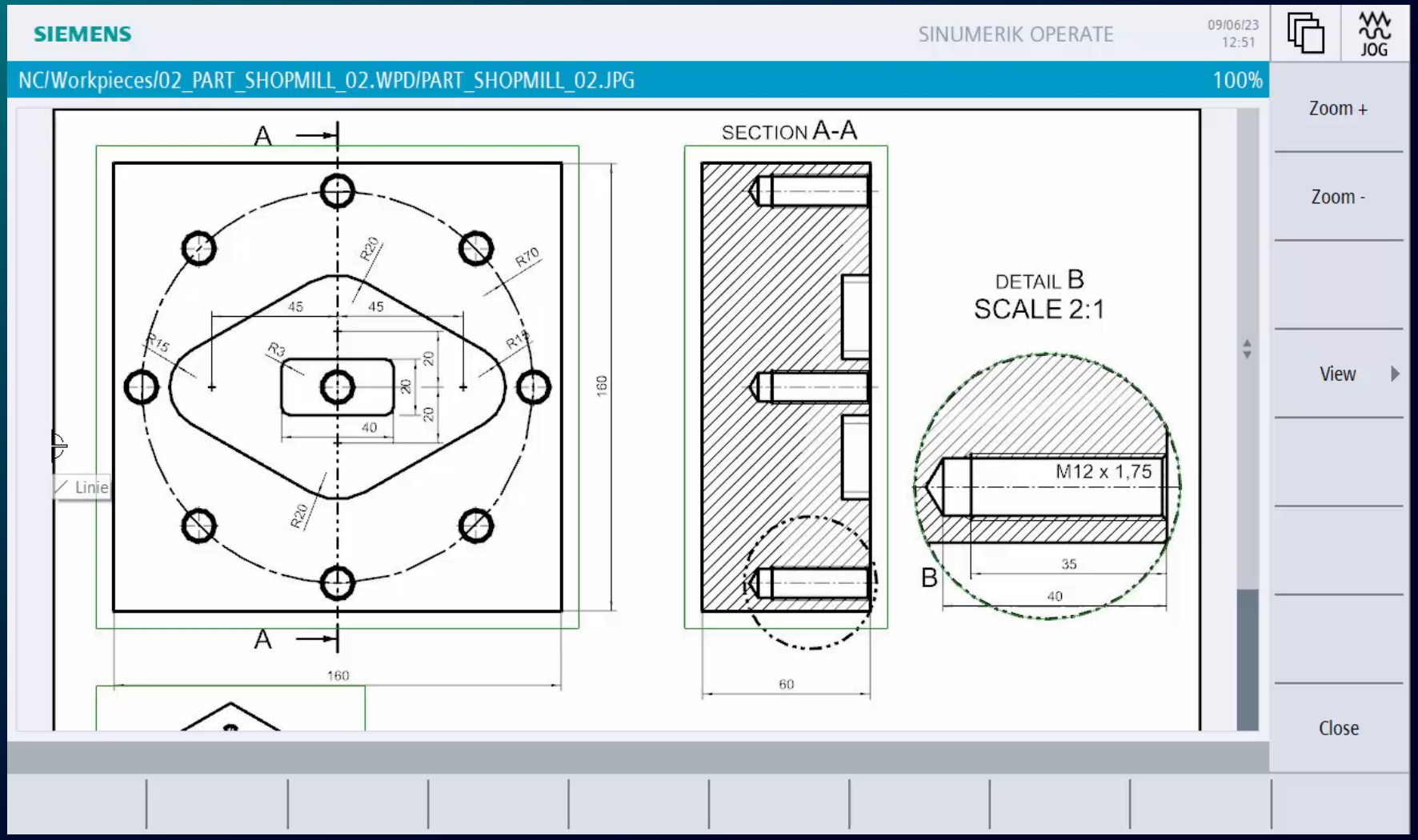
Online switchable
(Ctrl + G)



The machining operations **are shown to scale** as soon **as the program is entered**, even before the later simulation.
With this dynamic broken line graphic, **which is unique on the market**, the **process reliability** of the subsequent processing is noticeably increased.

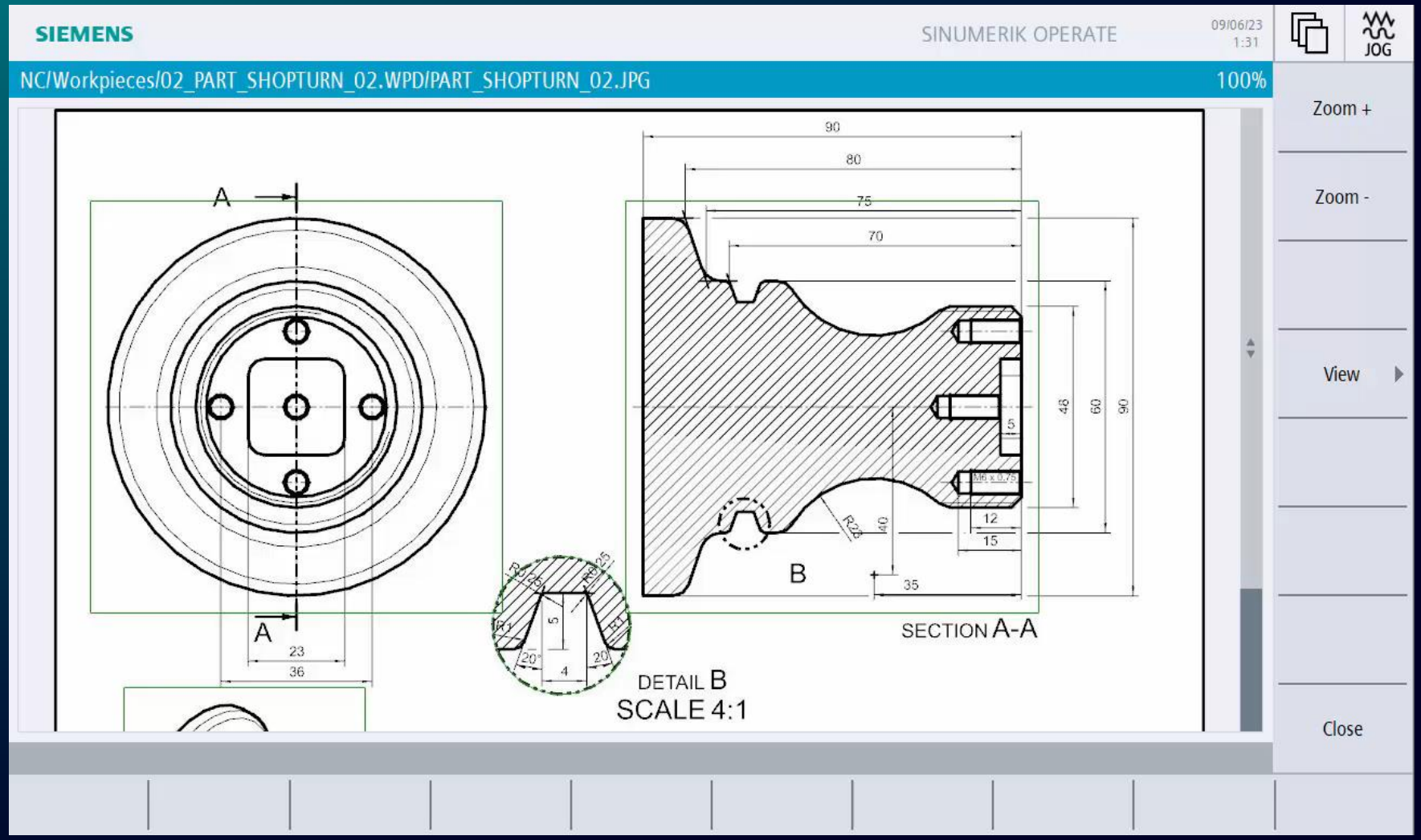
SIEMENS

Dynamic line graphics (1)
- the immediate comparison with the workpiece drawing



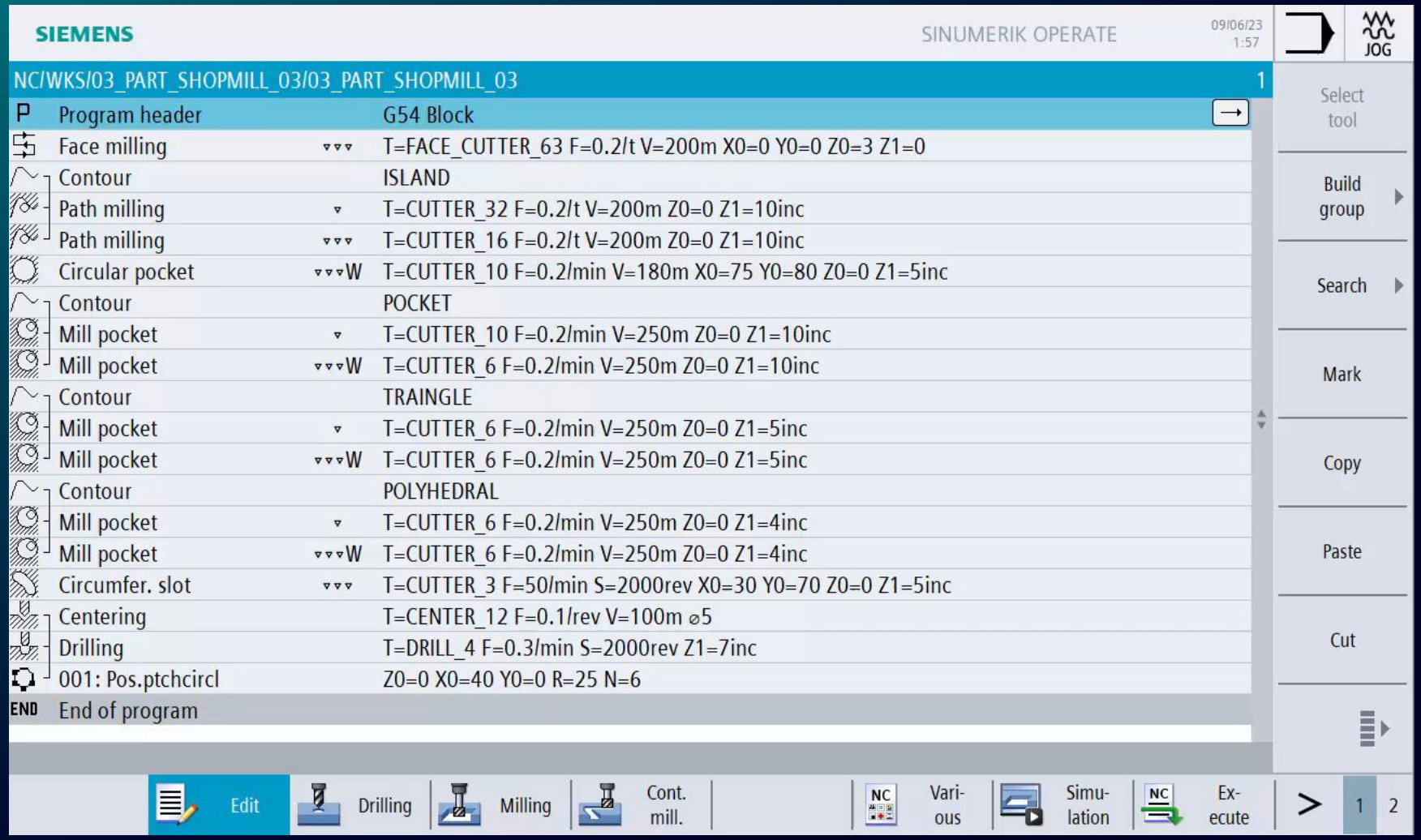
SIEMENS

Dynamic line graphics (1)
- the immediate comparison with the workpiece drawing



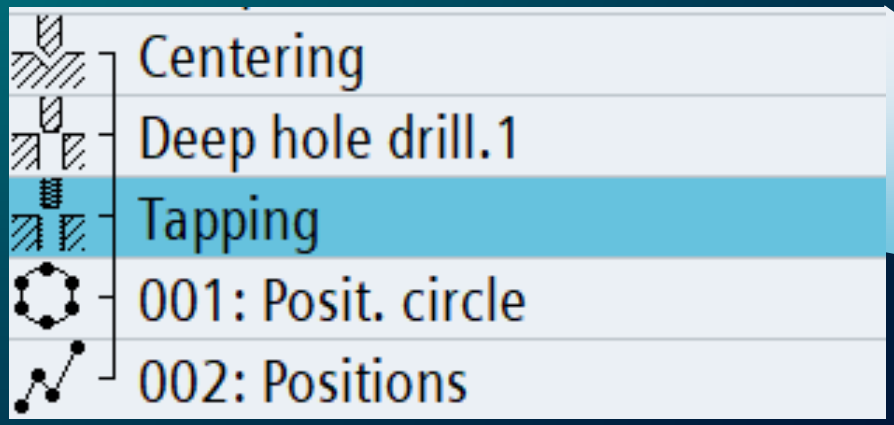
SIEMENS

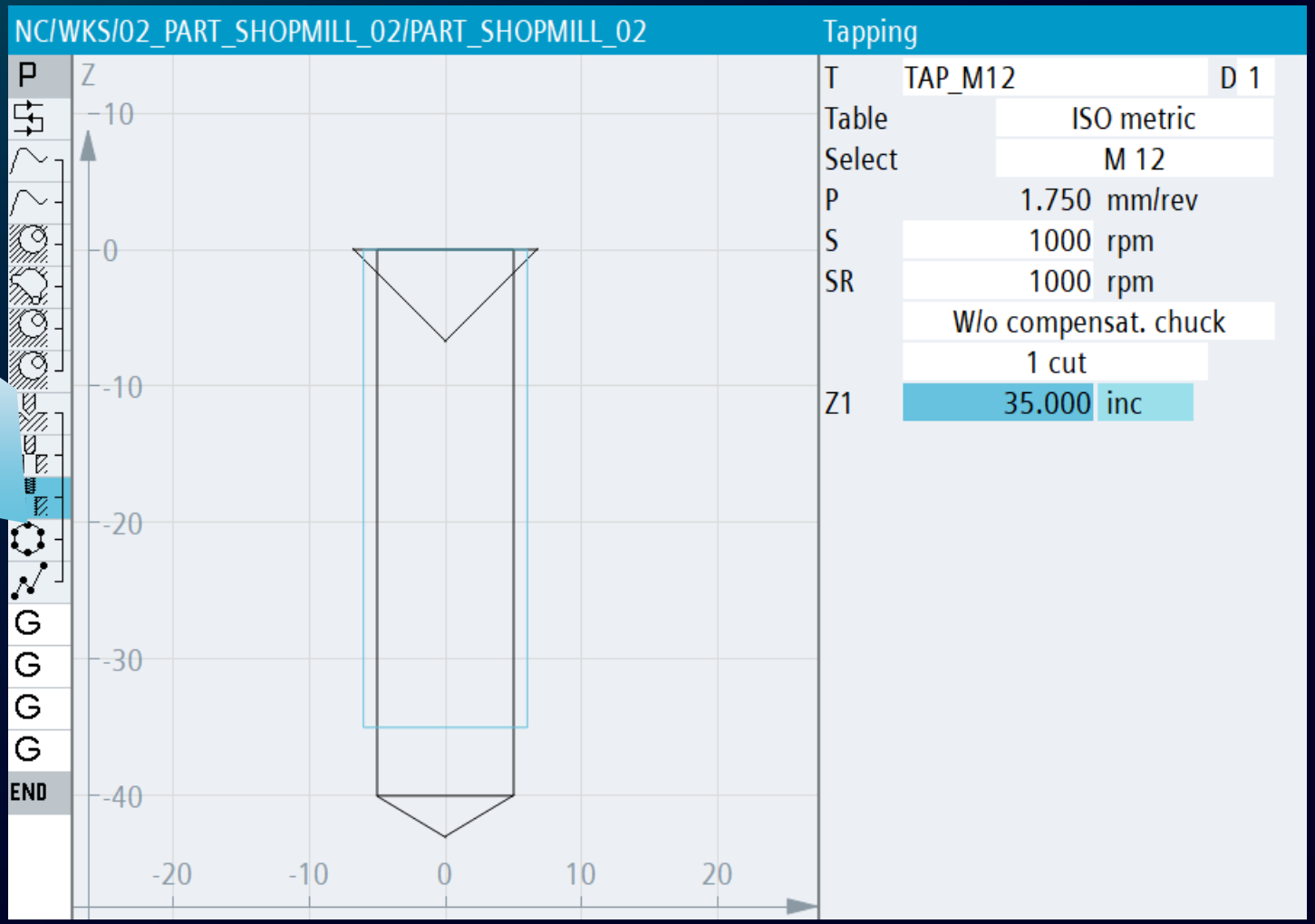
Dynamic line graphics (1)
- the immediate comparison with the workpiece drawing



SIEMENS

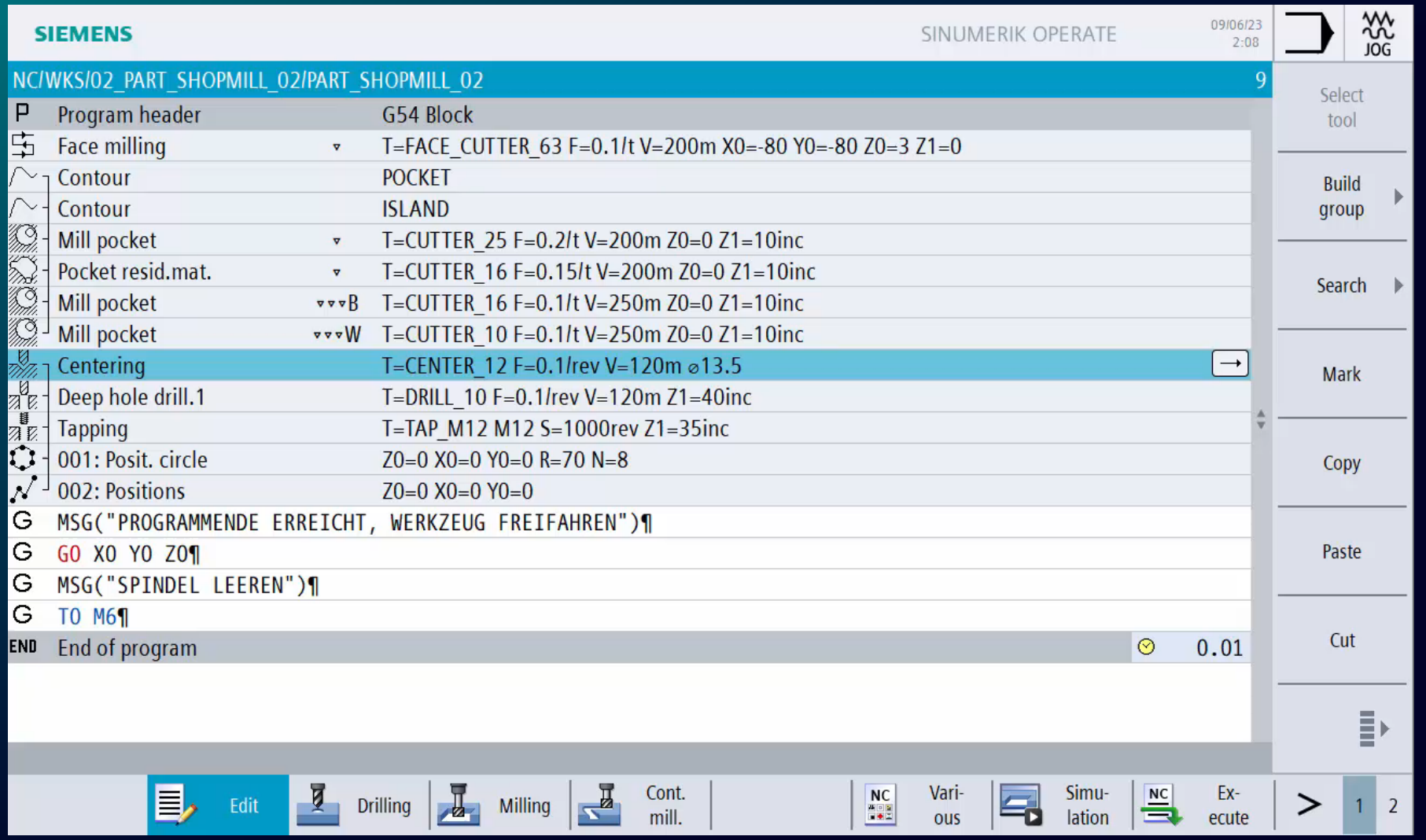
Dynamic line graphics (2)
- the immediate comparison with the workpiece drawing





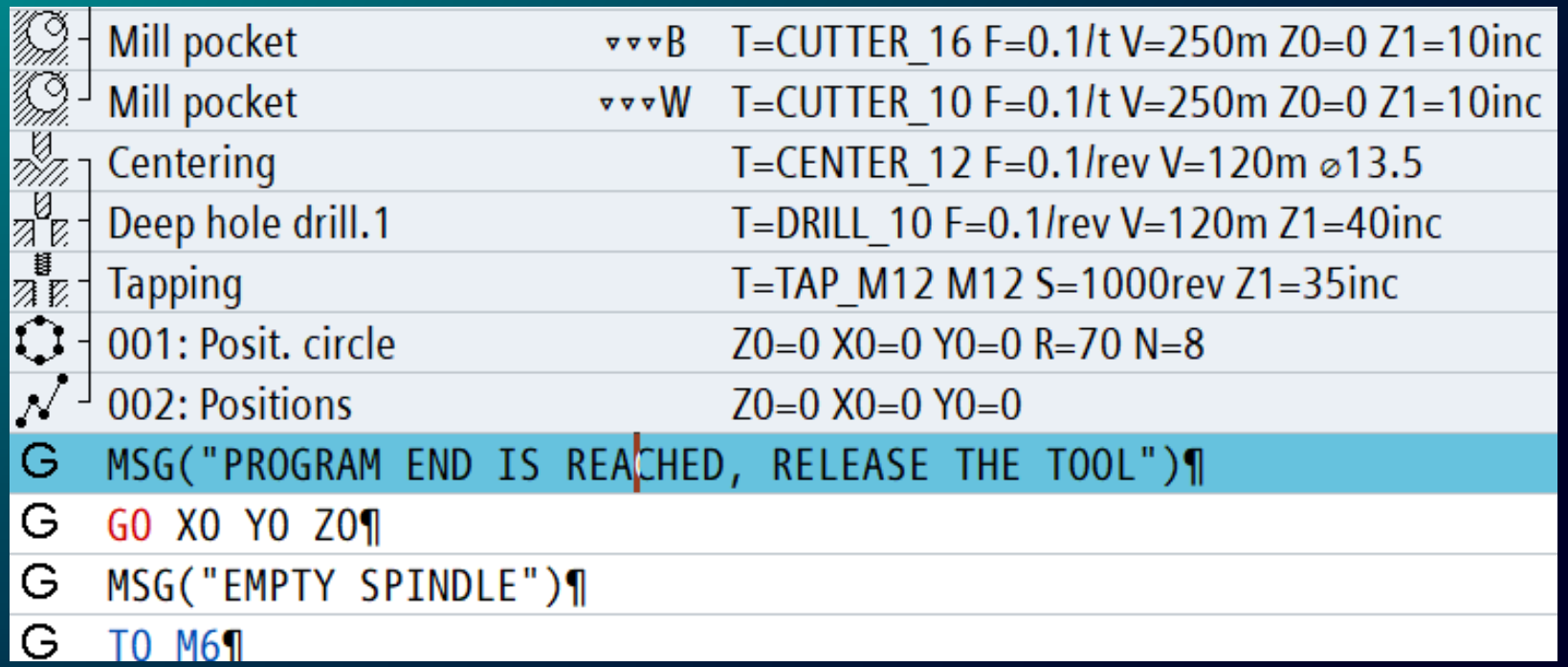
SIEMENS

Dynamic line graphics (2)
- the immediate comparison with the workpiece drawing



SIEMENS

Input of G-code lines
- the smooth transition to G-code programming



Additional flexibility



For special applications, **work-steps can be supplemented by G code**.
This gives the interactive work-step programming **additional flexibility**.

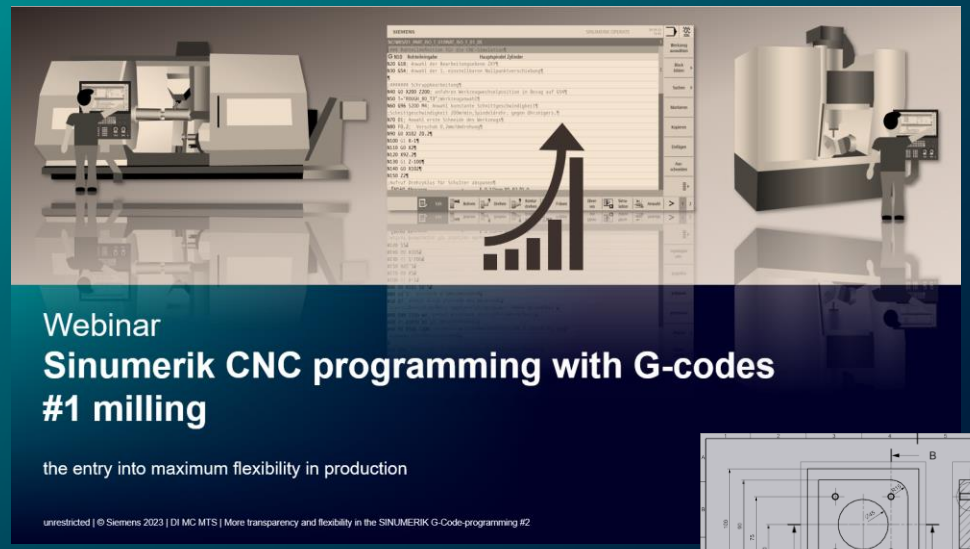
SIEMENS

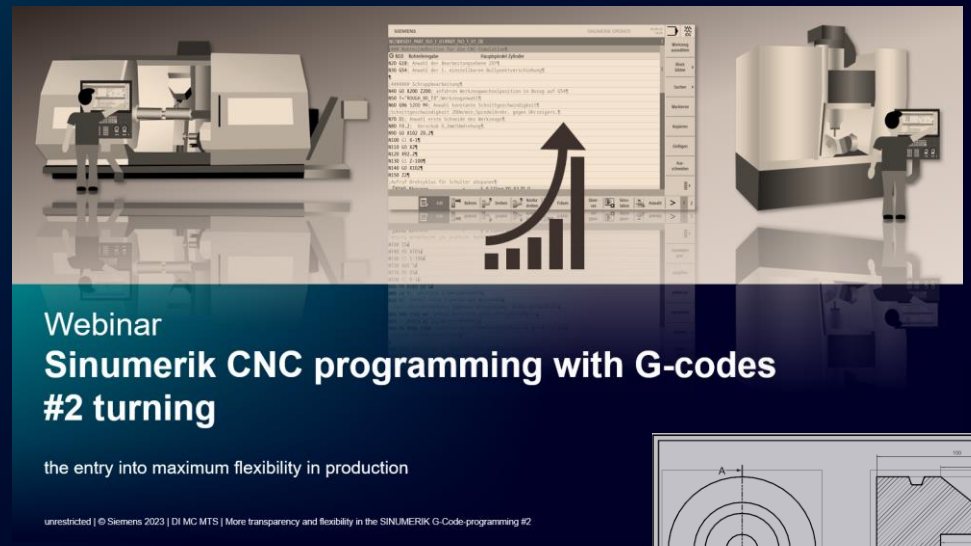
Agenda

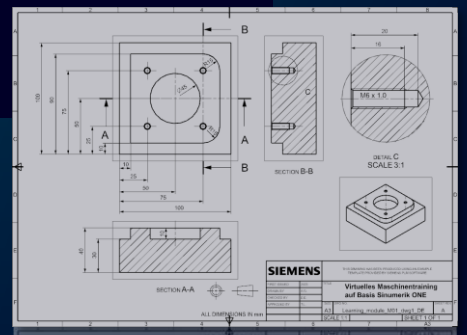
1. Overview
2. Work-step programming ShopMill / ShopTurn
3. **G-Code**
4. programGUIDE
5. Machining cycles (ShopMill / ShopTurn, programGUIDE)
6. Offline-programming in the CNC-job preparation
7. CNC4you

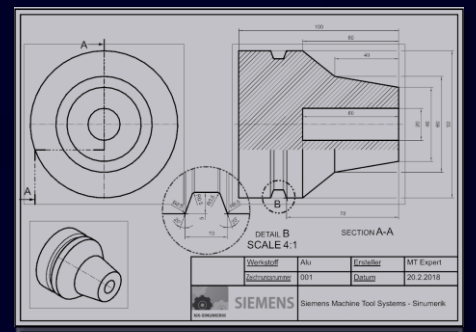
SIEMENS

Sinumerik CNC programming with G-codes
- own webinars





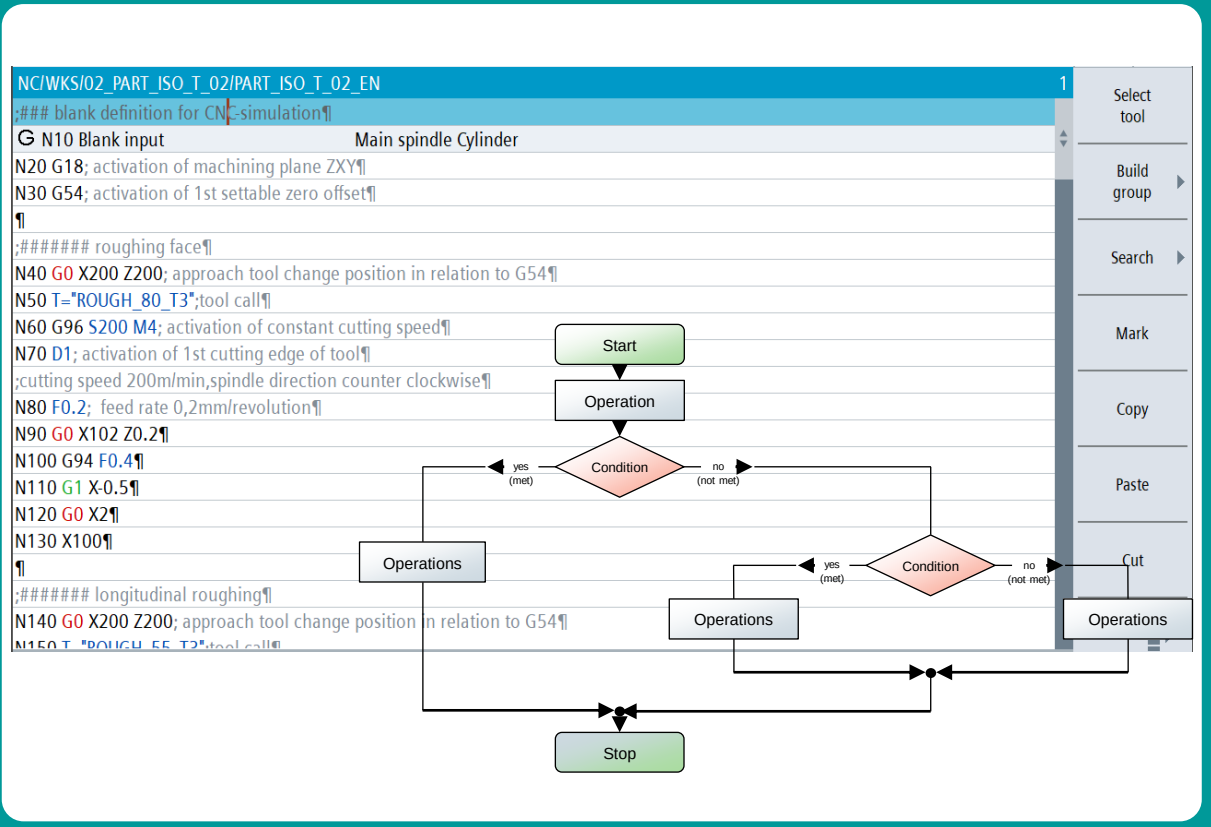




Do you want to **switch to** Sinumerik? Are you **new to** CNC programming?
We will show you the **basic creation of a Sinumerik G-code program** and how to use it to set the course for maximum flexibility in parts production with machine tools.

SIEMENS

G-Code programming
- the specialist for large series and special applications



Uniquely comprehensive CNC language scope
for manufacturing jobs of all types, up to special applications

CNC high-level language
for parameterizable CNC programs with variable workpiece part masters or own machining cycles

Convenient CNC editor
for the efficient creation of G-code programs

Machining time is the **productivity-determining factor**, especially for **large series**.
Special applications often require **maximum degrees of freedom** in CNC programs.
SINUMERIK Operate offers **unrivalled, globally established SINUMERIK G-code programming**.

SIEMENS

ISO 6983
- the basic framework

```
N20 G54; activation of 1st settable zero offset¶
¶
;####### turning¶
N30 G18; activation of turning plane ZXY¶
N40 G0 X200 Z200; approach tool change position in relation to G54¶
N50 T="ROUGH_80_T3";tool call¶
N60 G96 S200 M4; activation of constant cutting speed¶
N70 D1; activation of 1st cutting edge of tool¶
¶
N80 G0 X101 Z0¶
N90 G1 F0.1¶
N100 X-0.5¶
N110 G0 Z2¶
N120 X95¶
N130 G1 Z-100¶
N140 G0 X100¶
N150 M5; main spindle stopl¶
;##### milling on front face¶
N160 G0 X200 Z200; approach tool change position in relation to G54¶
N170 T="CUTTER_8_Face";tool call¶
```

Traditionally known CNC language commands

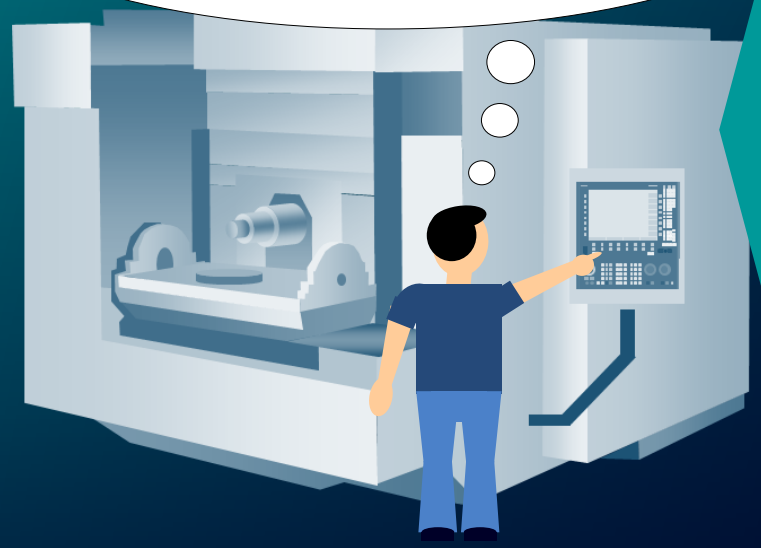


Using **language commands according to ISO 6983**,
CNC programmers can find a basic structure of traditionally known CNC instructions.

SIEMENS

Readable language commands
- important tools in "machine-related" CNC programming

- **More precision?**
- **More speed?**
- **Simply more possibilities !**



Readable language commands

- ⇨ Coordinate transformations (TRANS, ROT, SCALE, MIRROR)
- ⇨ Special motion commands
  - ⇨ Spline interpolation (ASPLINE, BSPLINE, CSPLINE)
  - ⇨ Polynomial interpolation (POLY)
- ⇨ Kinematic transformations (TRAORI, TRANSMIT, TRAANG, TRACYL)
- ⇨ Acceleration types (BRISK, SOFT)
- ⇨ Traversing with feedforward control (FFWON, FFWOF)
- ⇨ **Program control structures (IF, FOR, WHILE, REPEAT, CASE)**
- ⇨ …..

Going beyond programming according to ISO 6983, highly developed readable CNC language commands address a wide range of **powerful functions in "machine-related" CNC programming**.

SIEMENS

Definition of user variables
- individual variables for each use-case

DEF <type> <name>=<init_value>

NC/WKS/01_SINUMERIK_VARIABLES/SINUMERIK_VARIABLES_EN

```
;#################### local user data LUD¶
;## LUD only take effect during the runtime of the CNC prog
;## LUD must always be defined at the beginning of the progr
¶
DEF REAL AXISPOSITION;          example: floating point numb
DEF INT COUNTER;                  example: integer¶
DEF CHAR MYCHAR;                example: ASCII-character
DEF STRING[20] MESSAGE;       example: string of characters
DEF BOOL RESULT;                  example: logical valu
DEF REAL FIN_VAL=123.456;     example: direct value assignme
```

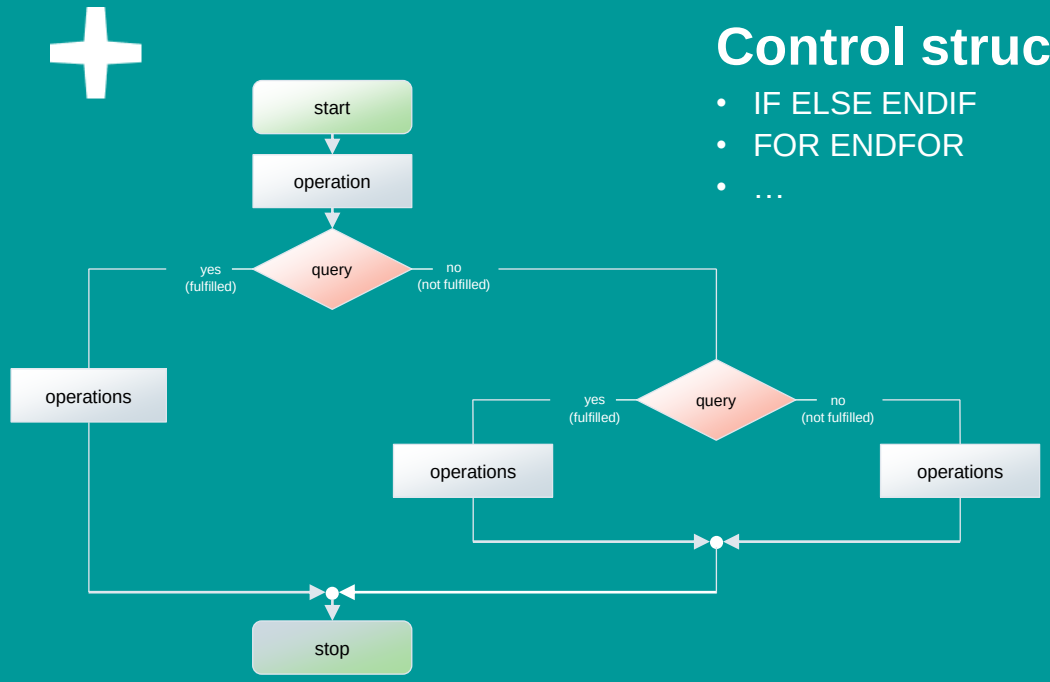
User variables



Any number of variables can be defined in CNC programs.
This opens up an undreamt-of flexibility, especially with **parameterizable CNC programs for workpiece families**.

SIEMENS

Control structures in the SINUMERIK CNC high-level language
- Flow charts become CNC programs



Control structures

- IF ELSE ENDIF
- FOR ENDFOR
- …

NC/WKS/03_FOR_ENDFOR/CONTROL_STRUCTURES_EN

```
;############### IF ELSE ENDIF
IF R0<=5
       MSG("R0 is smaller as 5")
ELSE
       IF R0<=10
              MSG("R0 is greater as 5 and smaller as 10")
       ELSE
              MSG("R0 is greater as 10")
       ENDIF
ENDIF

;############### REPEAT UNTIL
REPEAT
    POSITION=POSITION+10
    X=POSITION
UNTIL POSITION>=300
```

Complex production tasks, for example in user cycles, can be described using **clear flow charts** and **transferred directly to CNC programs** via **readable control structures** in the SINUMERIK CNC high-level language.

SIEMENS

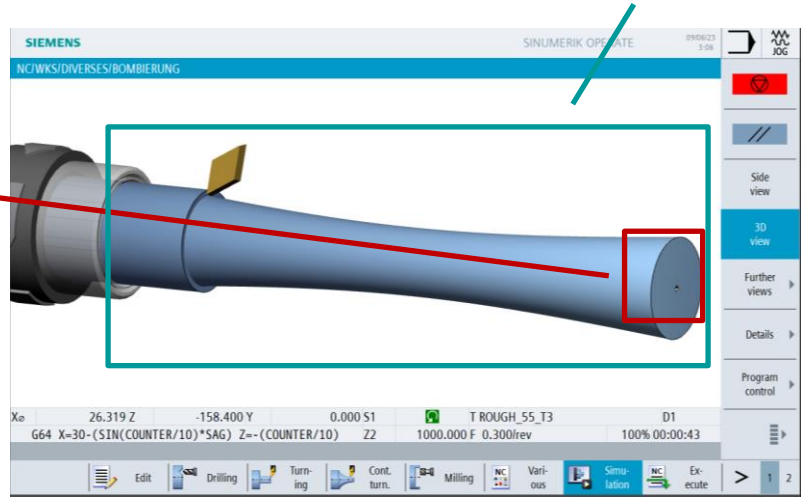
Control structures in the SINUMERIK CNC high-level language
- example counting loop

„machine-related “ ISO 6983

flexible programming with CNC high-level language

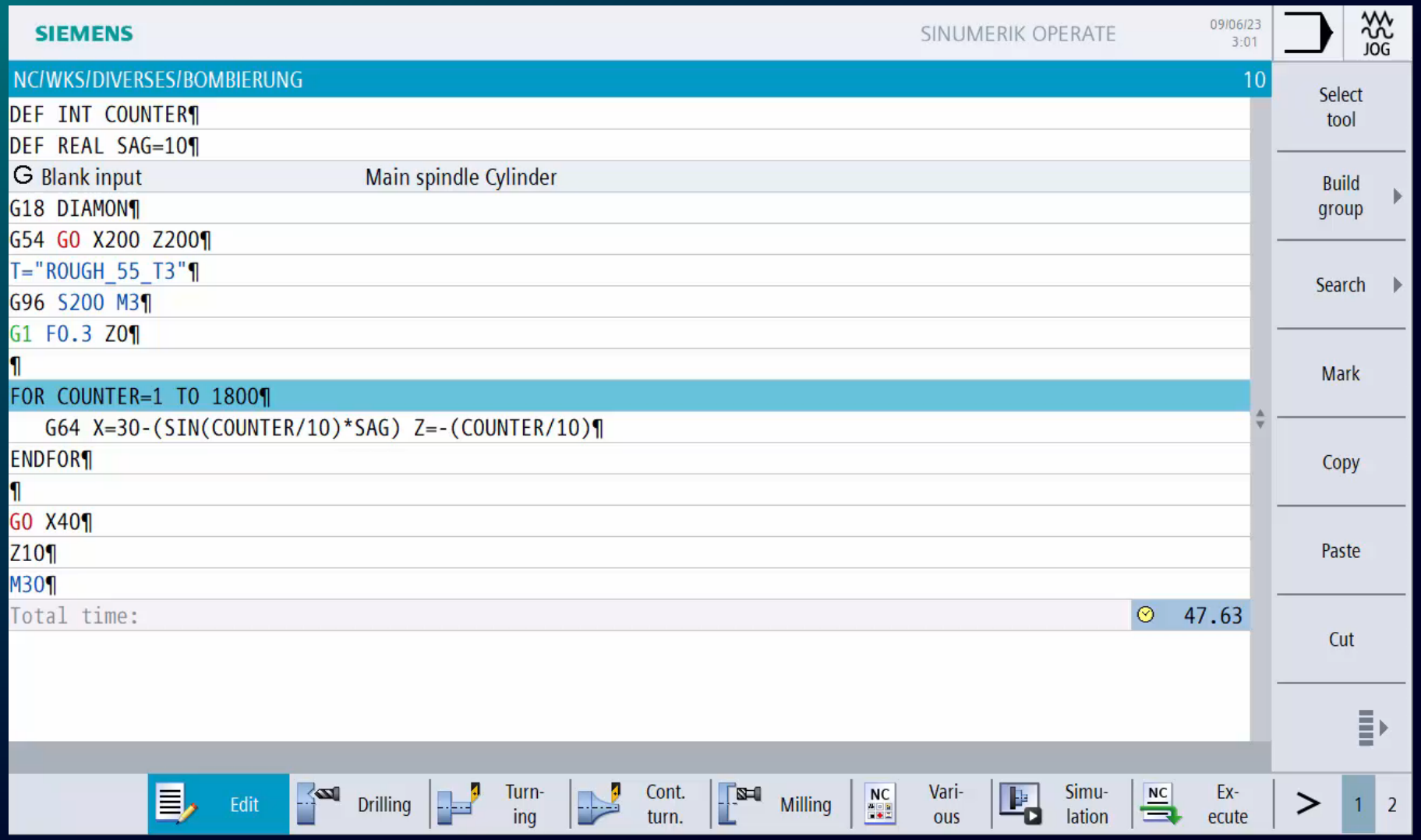
```
G64 G01 F500
Z-0 X0
Z-0.5 X1.396
Z-1 X2.792
Z-1.5 X4.187
Z-2 X5.581
Z-2.5 X6.972
Z-3 X8.362
Z-3.5 X9.750
Z-4 X11.134
Z-4.5 X12.515
Z-5 X13.892
Z-5.5 X15.265
Z-6 X16.633
Z-6.5 X17.996
Z-7 X19.354
Z-7.5 X20.706
Z-8 X22.051
….
```

```
G64 G01 F500
FOR COUNTER=1 TO 1800
  X=30-(SIN(COUNTER/10)*SAG) Z=-(COUNTER/10)
ENDFOR
```



SIEMENS

Control structures in the SINUMERIK CNC high-level language
- example counting loop



SIEMENS

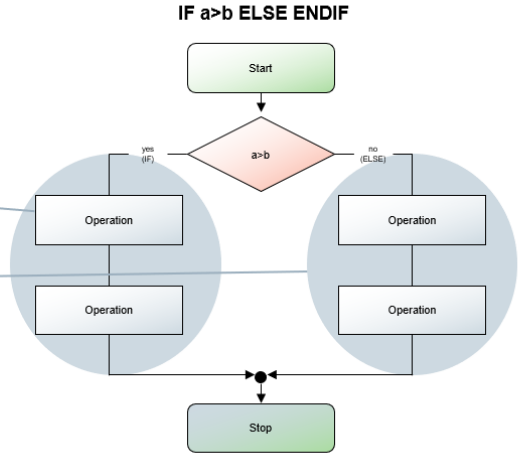
Flexible CNC-programs with SINUMERIK CNC high level language
- own webinar



IF … ELSE … ENDIF
- Conditional statement and branch

```
IF <condition>
    Program block_1
ELSE
    Program block_2
ENDIF
```

```
IF A>B          ;### condition¶
    G1 F2000    ;### program block 1¶
    X200¶
    X0¶
ELSE¶
    G1 F1000    ;### program block 2¶
    Y200¶
    Y0¶
    IF A==B     ;### nesting¶
        MSG("A is equal to B")¶
    ENDIF¶
ENDIF¶
```

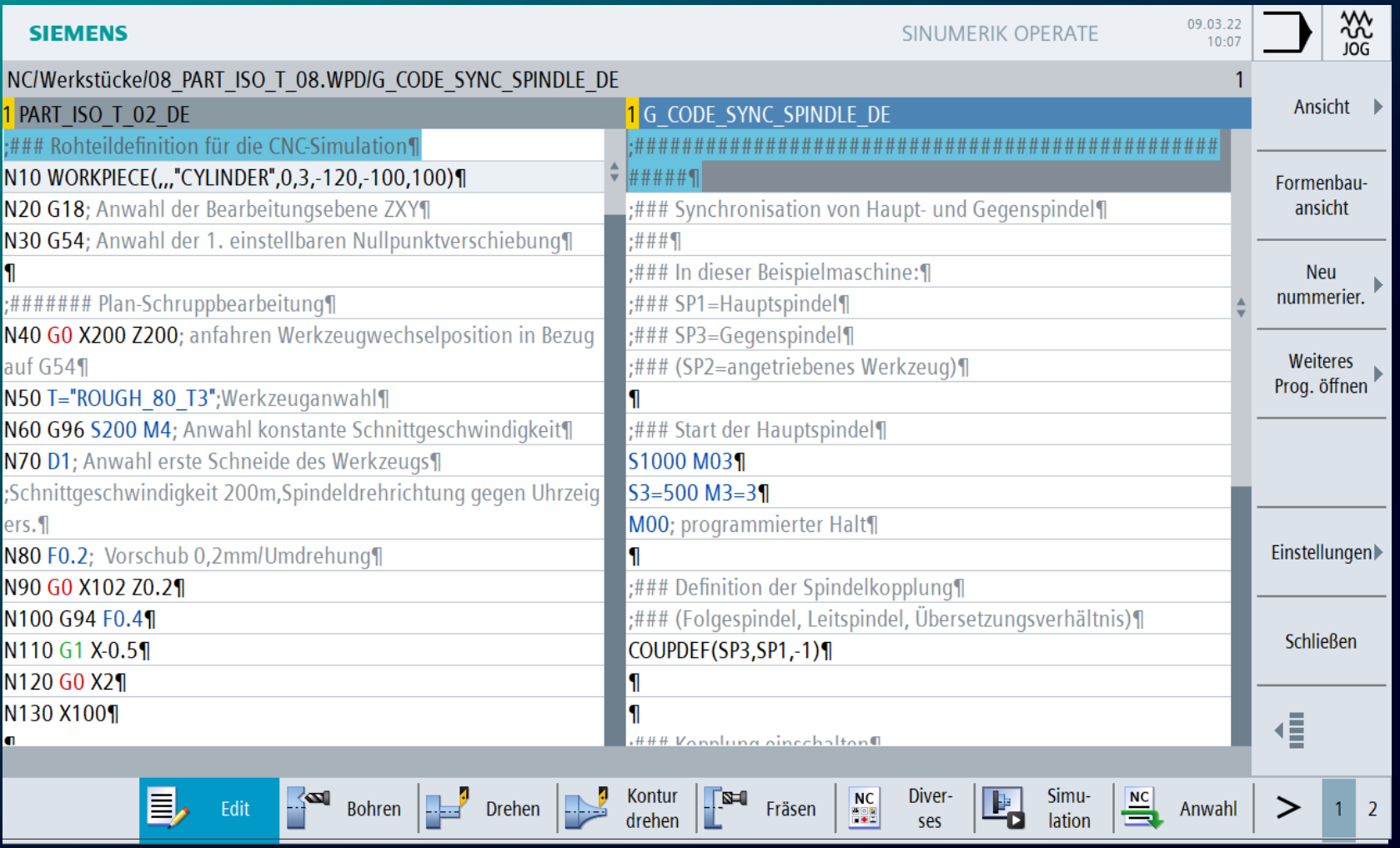


unrestricted | © Siemens 2023 | DI MC MTS | Flexible CNC-programs with SINUMERIK CNC high level language

SIEMENS

SIEMENS

The G-Code-Editor in SINUMERIK Operate
- a powerful and clear tool



Full overview through colored highlighting of certain commands

Powerful editor features (cut, copy, paste, search, replace ...)

Structuring of large CNC programs through block formation

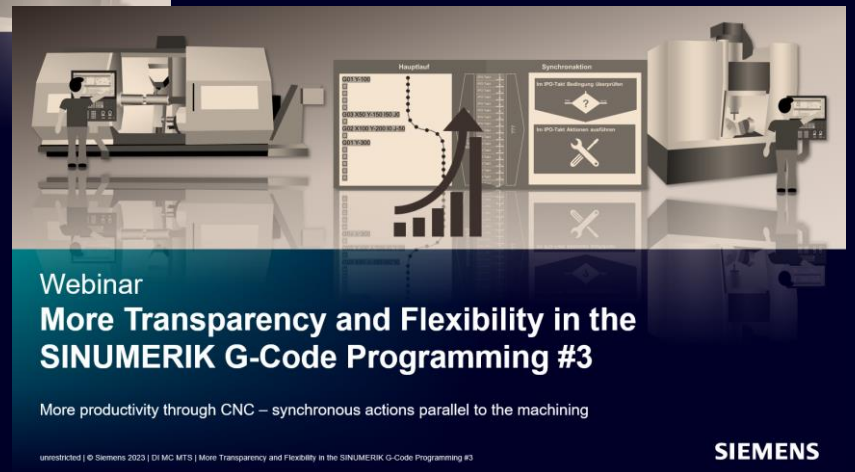
Simultaneous opening of several CNC programs

SIEMENS

More insight into SINUMERIK G-code programming
- own webinars







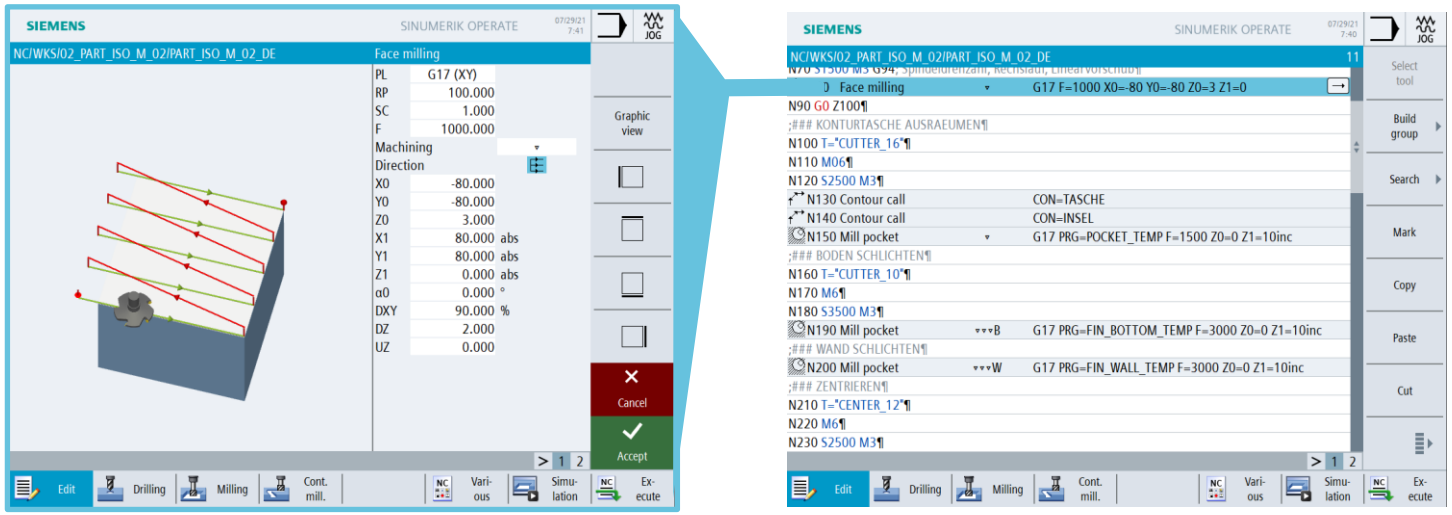
SIEMENS

Agenda

1. Overview
2. Work-step programming ShopMill / ShopTurn
3. G-Code
4. **programGUIDE**
5. Machining cycles (ShopMill / ShopTurn, programGUIDE)
6. Offline-programming in the CNC-job preparation
7. CNC4you

SIEMENS

programGUIDE
- the best from both worlds



Full flexibility of G-code programming

Ideal combination with a comprehensive spectrum of machining cycles

programGUIDE is the **integration of the principle** of easy-to-use, **graphical machining steps** into the environment of the highly flexible **SINUMERIK CNC programming language**. In this way, the benefits of both worlds can be **optimally combined**.

SIEMENS

programGUIDE
- Machining cycles shown as work steps!

```
;####### longitudinal finishing¶
N290 CYCLE952("OUTER_CONTOUR",,"",1101321,0.15,0.2,0,3,0.1,0.1,0.1,0.1,0.1,0,1,0.2,0.2,,,,,2,2,,,0,
¶
;###### threading¶
N300 G0 X200 Z200; approach tool change position in relation to G54¶
N310 T="THREAD_1.5_T8";tool call¶
N320 G96 S120 M4; activation of constant cutting speed¶
N330 D1; activation of 1st cutting edge of tool¶
N340 CYCLE99(0,48,-50,,1,2,1.22688,0,30,0,5,0,2,1110101,4,2,0.3,0.5,0,0,1,0,0.708339,1,,,,102,1)¶
¶
;###### radial groove¶
N350 G0 X200 Z200; approach tool change position in relation to G54¶
N360 T="GROOVE_3_T8";tool call¶
N370 G96 S120 M4; activation of constant cutting speed¶
N380 D1; activation of 1st cutting edge of tool¶
N390 CYCLE930(60,-70,4,7.639702,5,,0,20,20,1,0.25,0.25,1,0.2,3,1,10130,,1,30,0.1,1,0.2,0.2,2,110)¶
```

Settings	
Display cycles as machining step	Yes
Highlight selected G code commands	Yes
Font size	auto

```
;####### longitudinal finishing¶
N290 Stock removal ▿▿▿ PRG=OUTER_CONTOUR F=0.15 Longitu
¶
;###### threading¶
N300 G0 X200 Z200; approach tool change position in relation to
N310 T="THREAD_1.5_T8";tool call¶
N320 G96 S120 M4; activation of constant cutting speed¶
N330 D1; activation of 1st cutting edge of tool¶
N340 Thread long. ▿ P2mm/rev Outside X0=48 Z0=0
¶
;###### radial groove¶
N350 G0 X200 Z200; approach tool change position in relation to
N360 T="GROOVE_3_T8";tool call¶
N370 G96 S120 M4; activation of constant cutting speed¶
N380 D1; activation of 1st cutting edge of tool¶
N390 Groove ▿+▿▿ F=0.1 X0=60 Z0=-70 T1=5inc
```

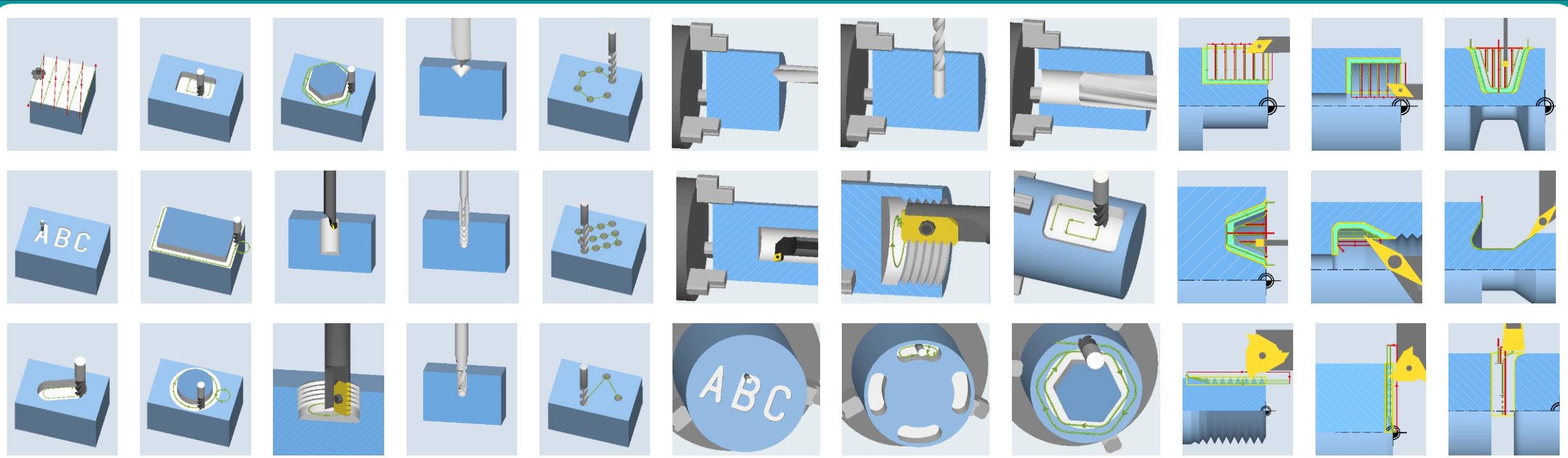

SIEMENS

Agenda

1. Overview
2. Work-step programming ShopMill / ShopTurn
3. G-Code
4. programGUIDE
5. **Machining cycles (ShopMill / ShopTurn, programGUIDE)**
6. Offline-programming in the CNC-job preparation
7. CNC4you

SINUMERIK machining cycles
- the universal assistants in daily CNC programming

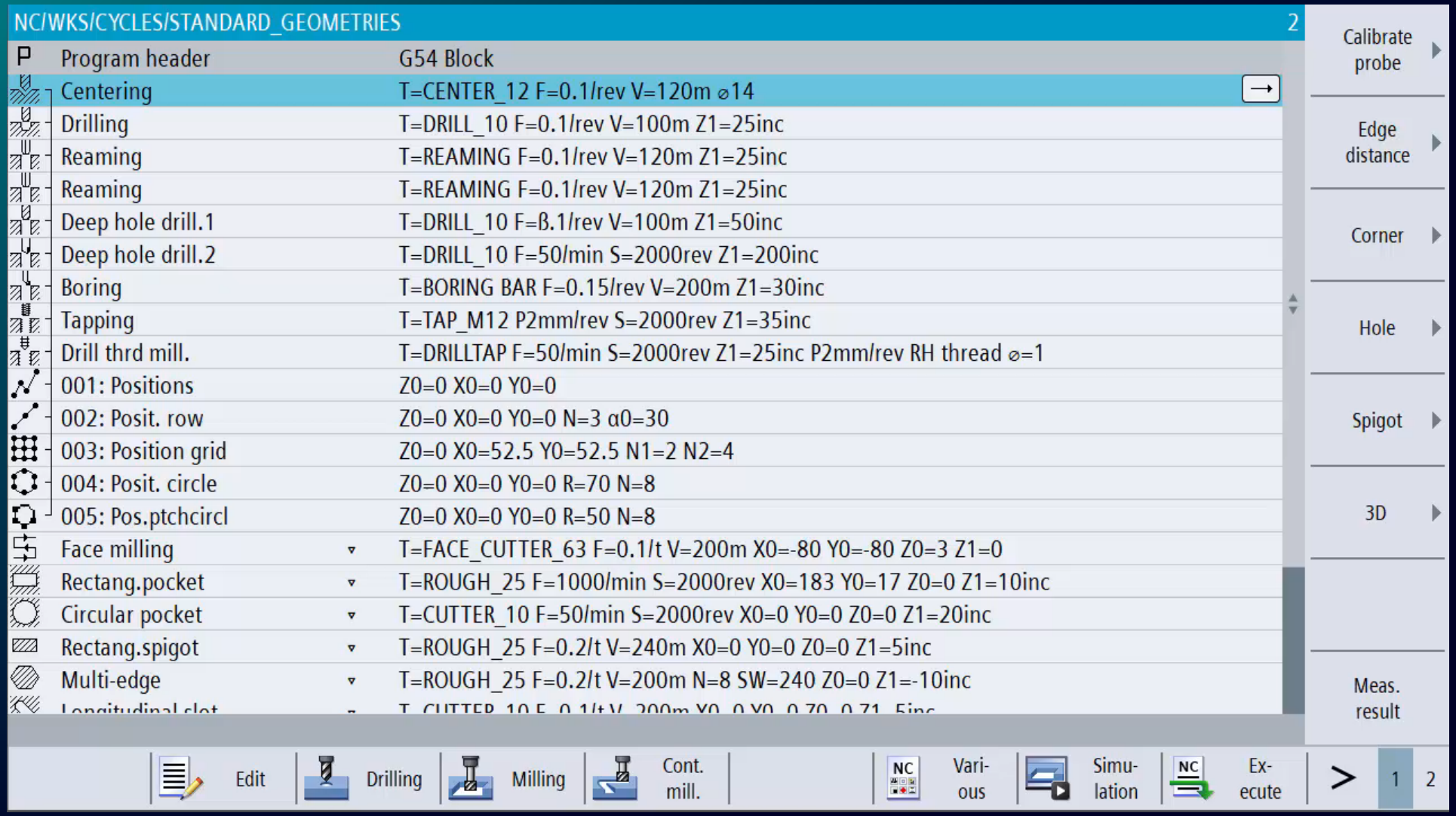
drilling, milling and turning cycles



SINUMERIK Operate offers a wide range of **machining cycles for drilling, milling and turning of standard geometries**.

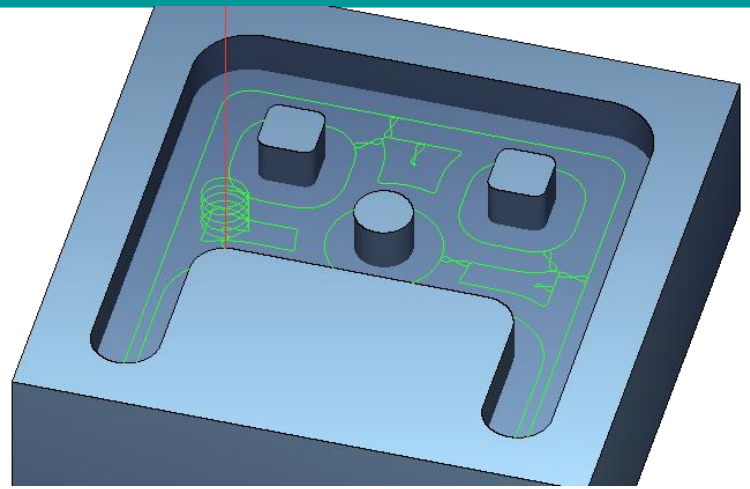
SIEMENS

SINUMERIK machining cycles
- the universal assistants in daily CNC programming



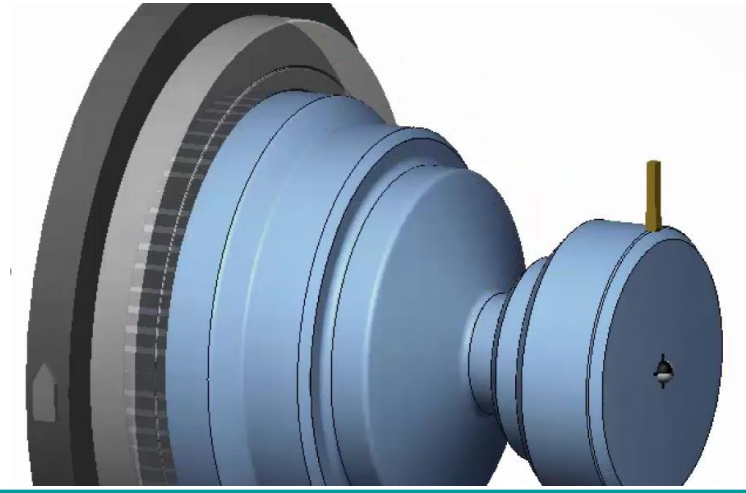
SIEMENS

Contour machining cycles with residual material detection
- CAM functionality on board



Contour milling cycle

- machining of contour pockets and spigots
- consideration of up to 20 island contours
- extensive approach/departure and insertion strategies



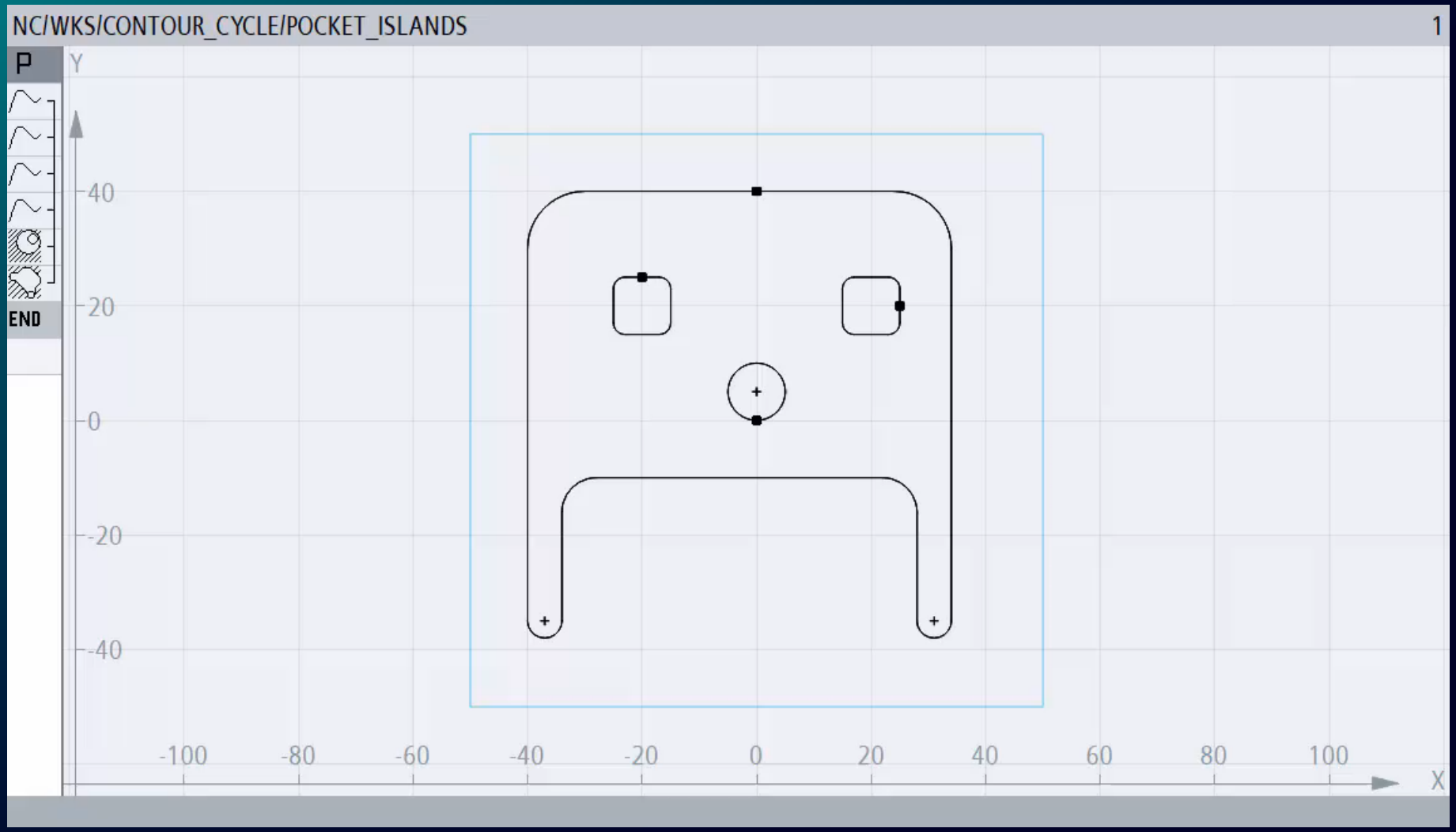
Contour turning cycle

- axis parallel stock removal, contour grooving, plunge turning
- arbityrary blank geometries
- arbityrary segmentation of machining sequences

SINUMERIK Operate enables the **programming of complex part geometries** without CAM system **directly at the machine**.
With the residual material detection, the optimal tools can always be used.

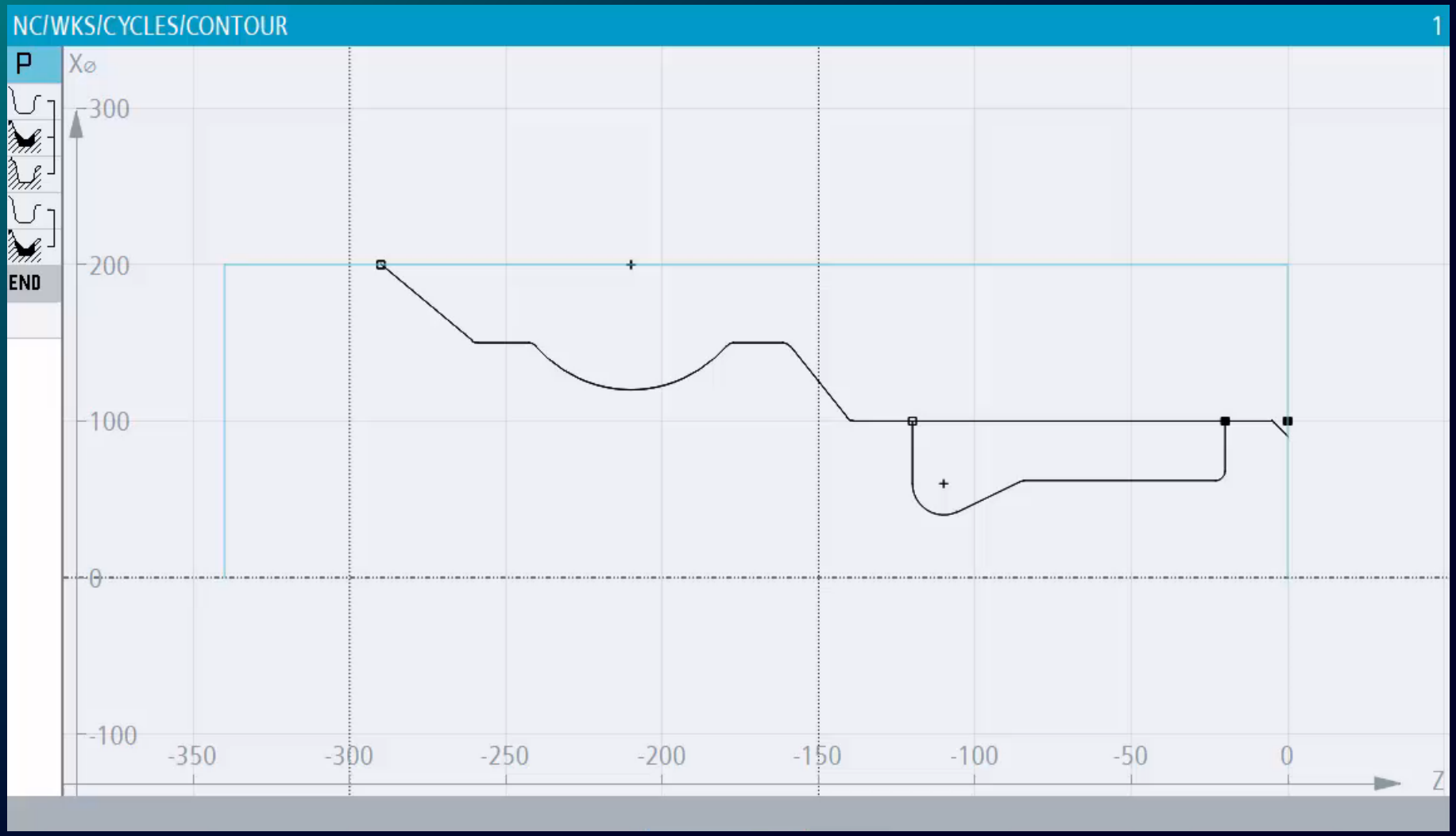
SIEMENS

Contour machining cycles with residual material detection
- CAM functionality on board



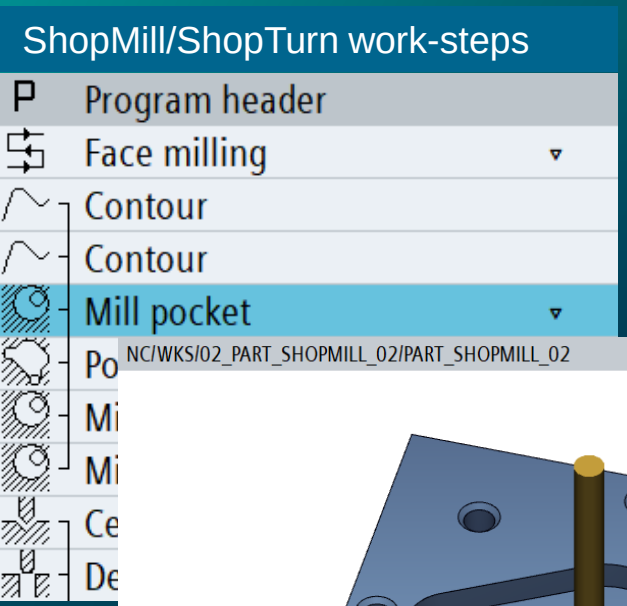
SIEMENS

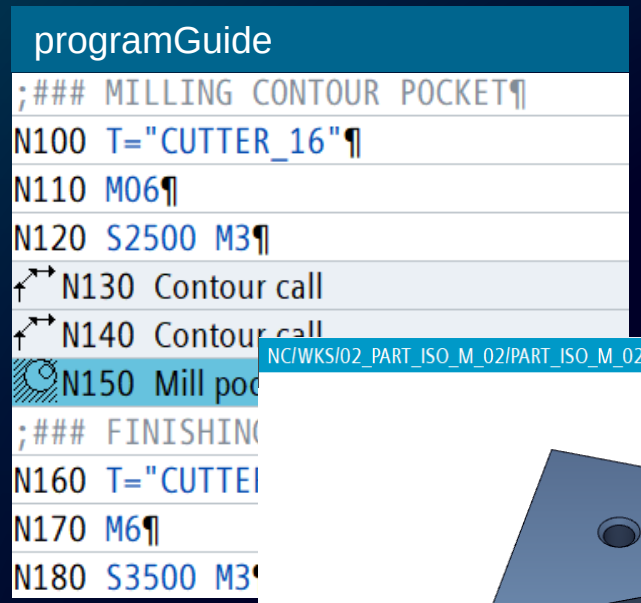
Contour machining cycles with residual material detection
- CAM functionality on board

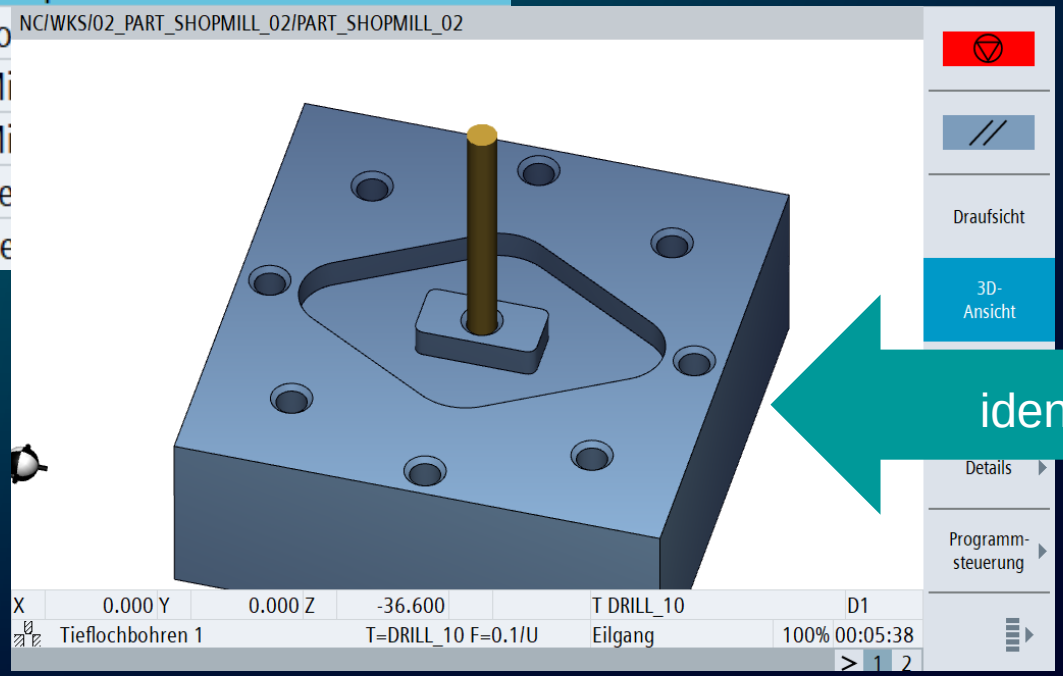


SIEMENS

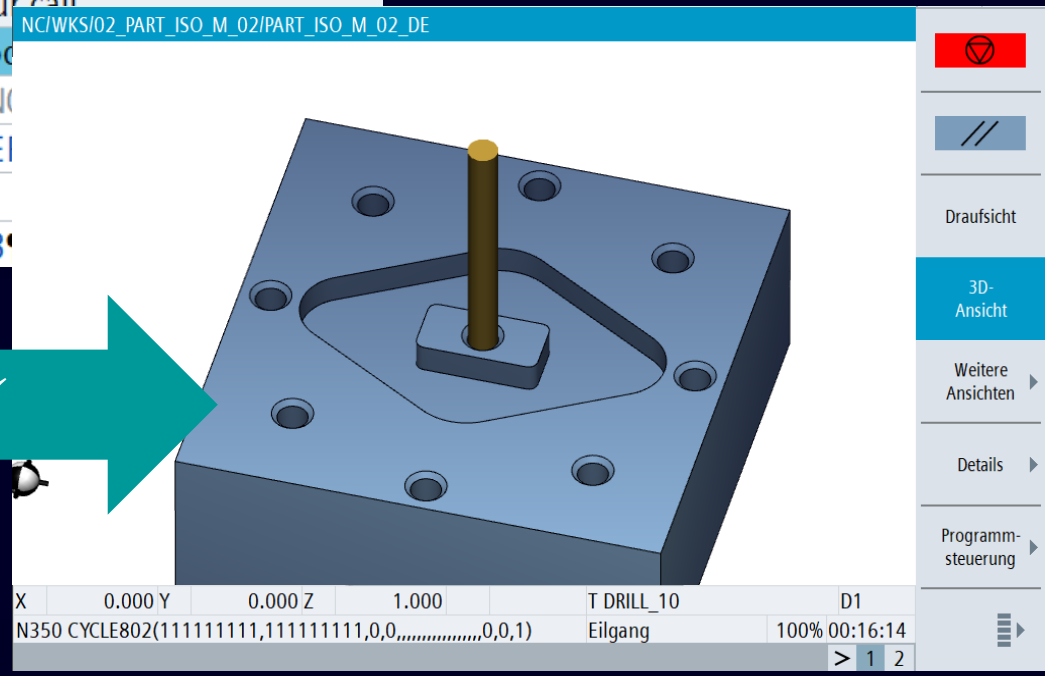
Machining cycles in work-step programming and programGUIDE
- identical behavior!







identical behavior ✓

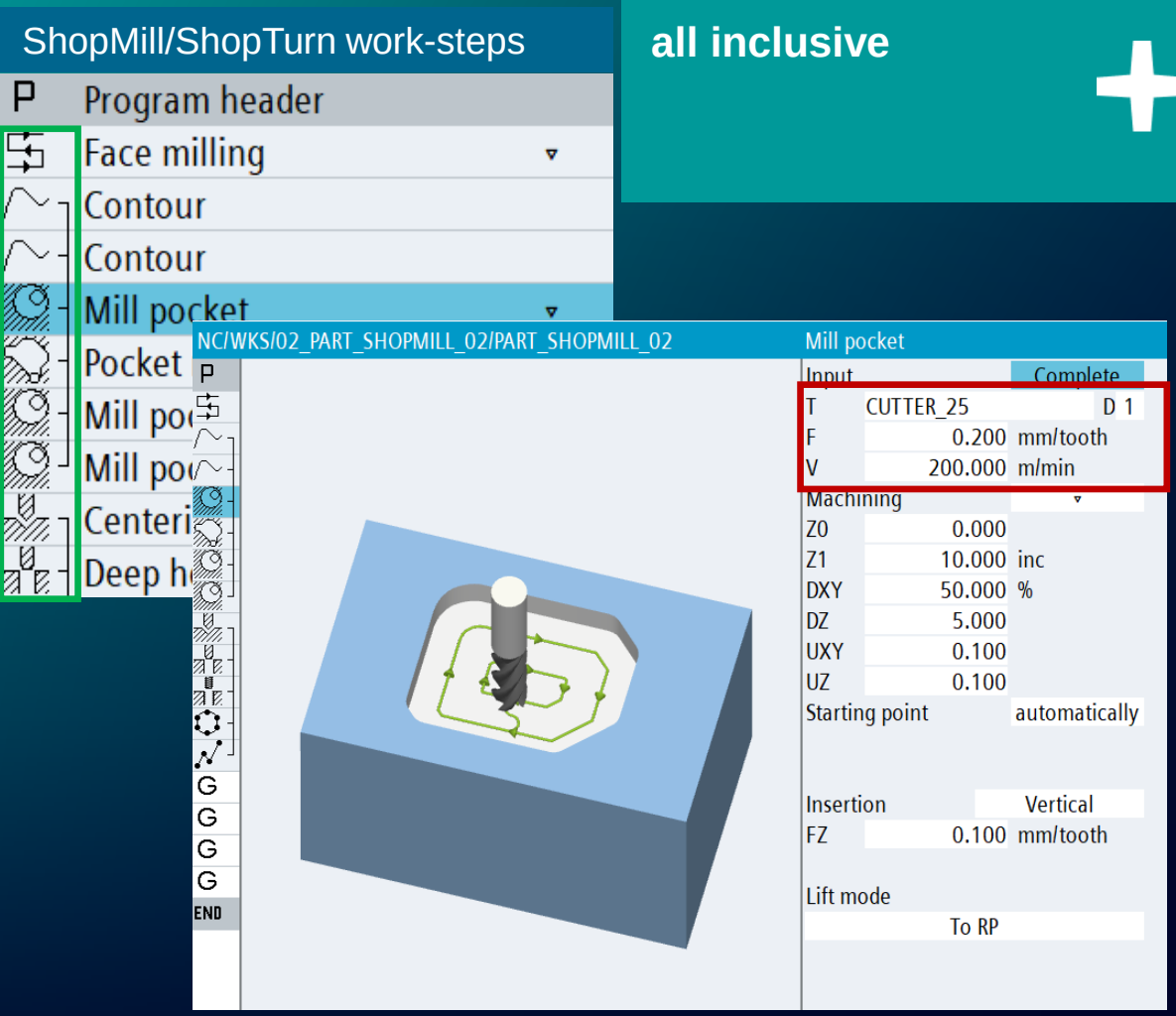


SIEMENS

Machining cycles in work-step programming and programGUIDE
- all inclusive versus further degrees of freedom

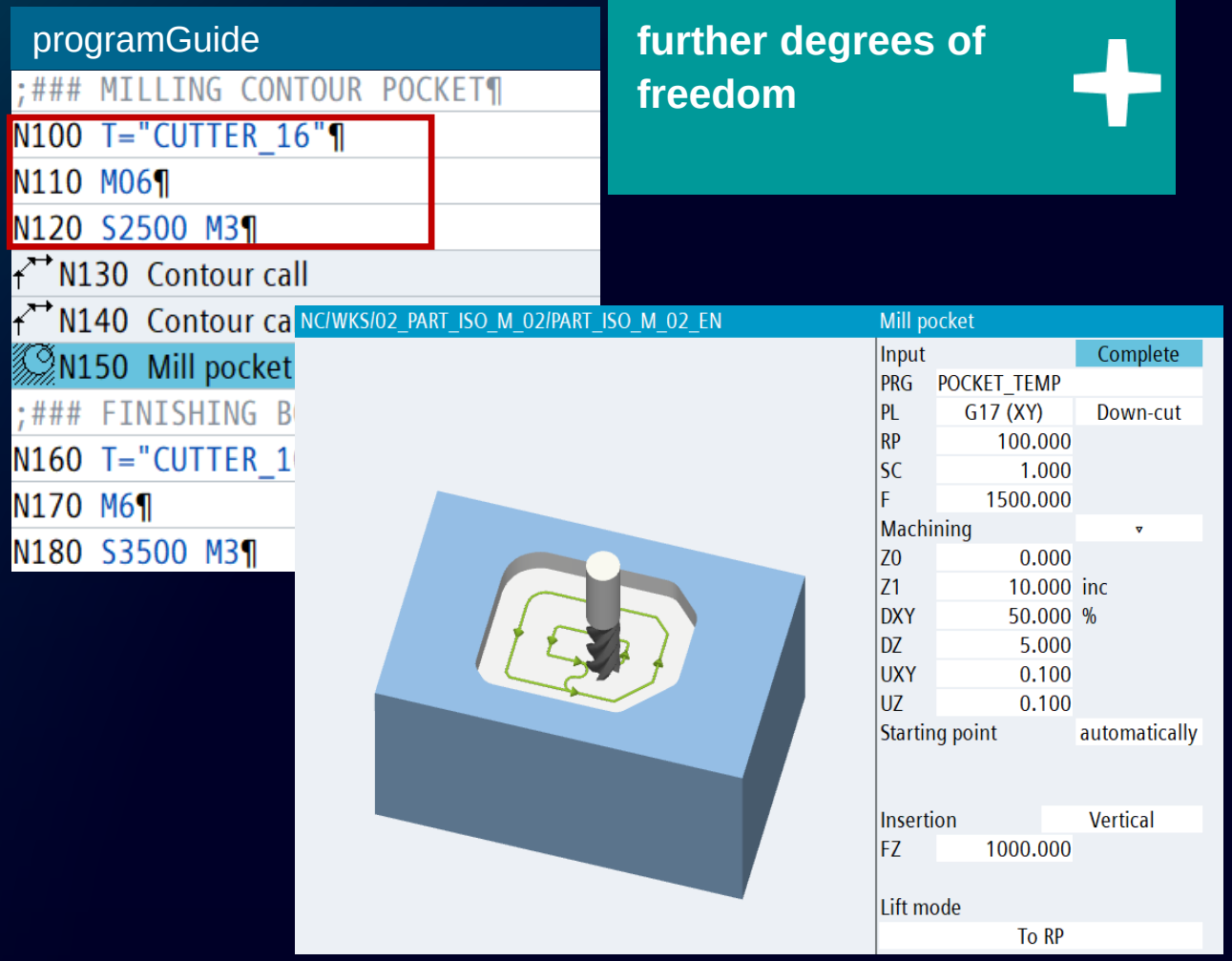
ShopMill/ShopTurn work-steps

all inclusive +



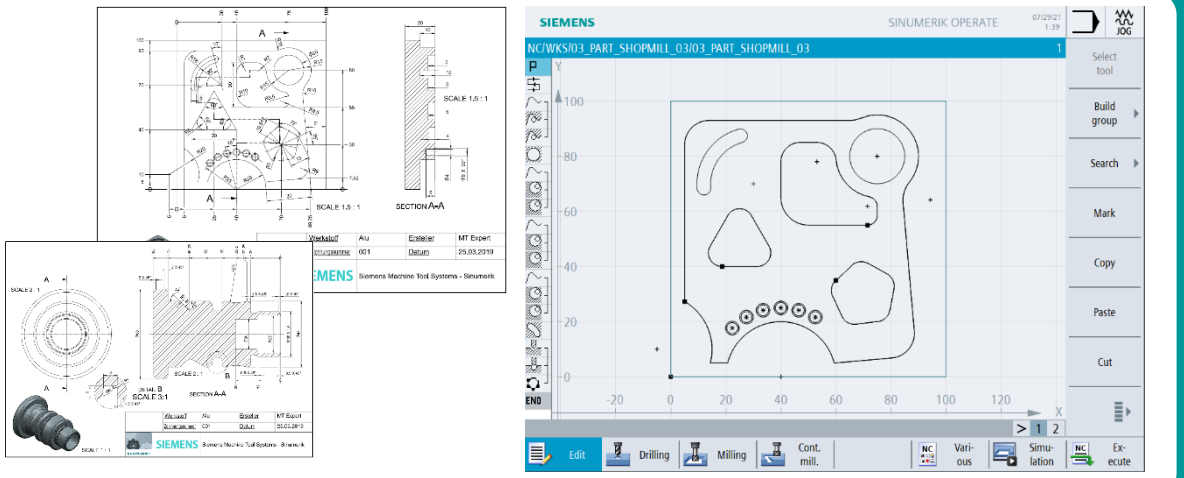
programGuide

further degrees of freedom +



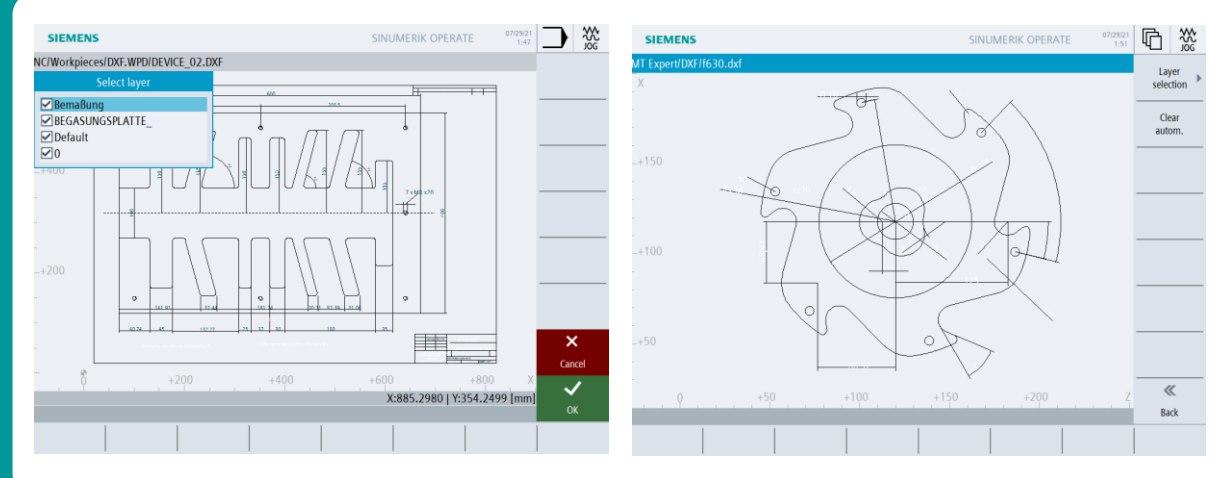
SIEMENS

Integrated contour calculator, DXF reader
- CAD functionality on board



Integrated contour calculator

- comfortable creation of milling and turning contours
- automatic calculation of partly defined elements



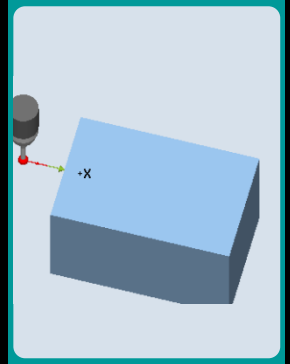
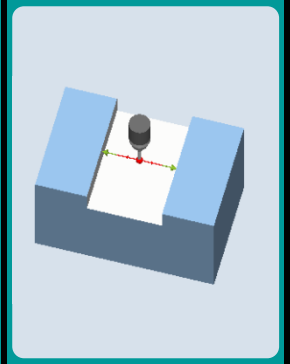
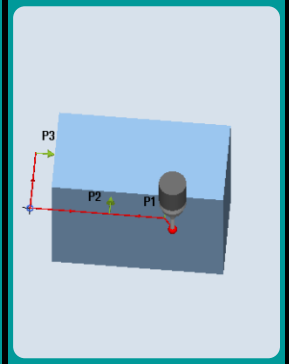
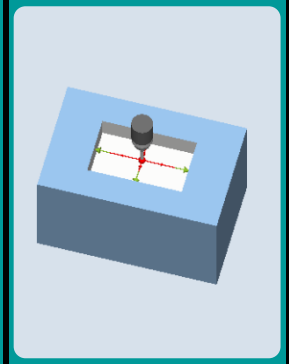
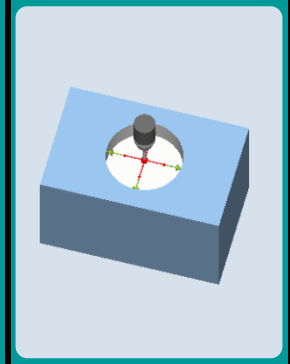
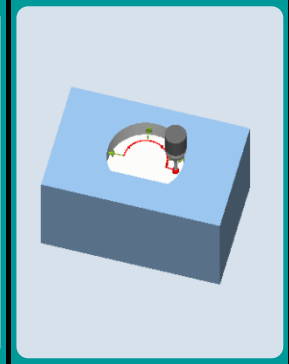
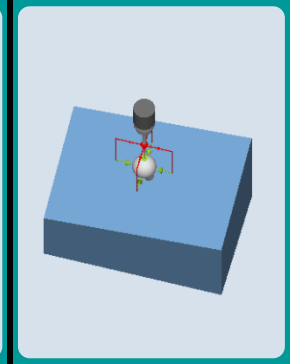
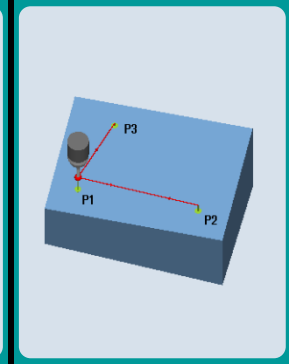
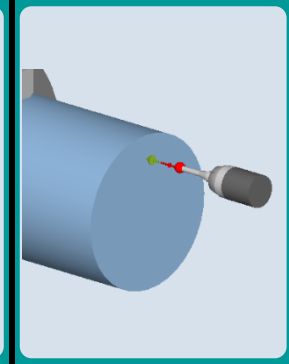
DXF viewer / reader

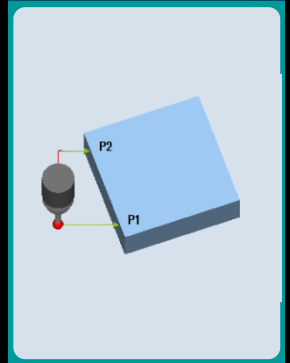
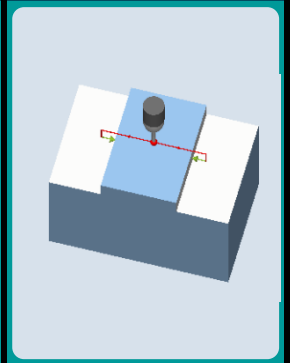
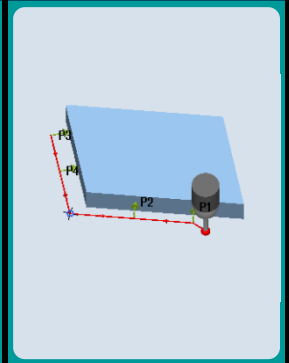
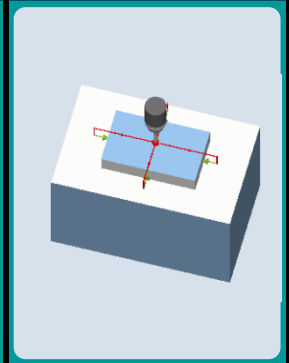
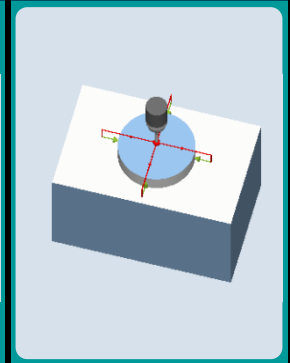
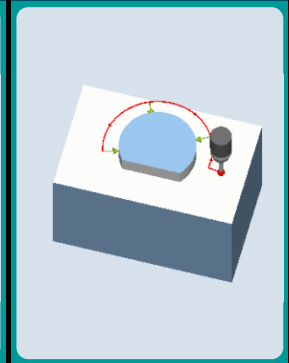
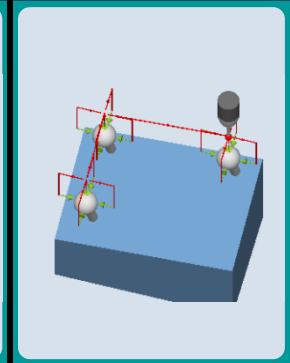
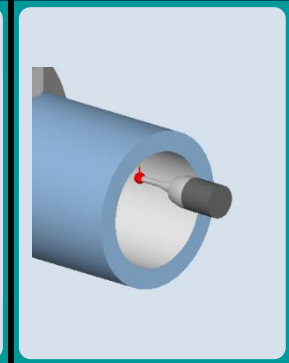
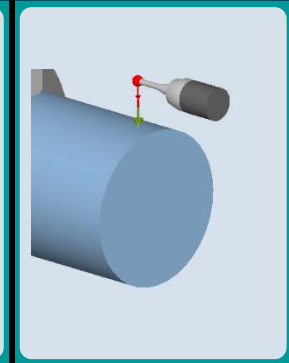
- conversion of any 2D-contours in SINUMERIK format
- conversion of any position patterns in SINUMERIK format

SINUMERIK Operate enables the **programming of any geometries** without a pocket calculator or CAD system **directly on the machine**.

SIEMENS

SINUMERIK measuring cycles
- in-process measurements à la carte

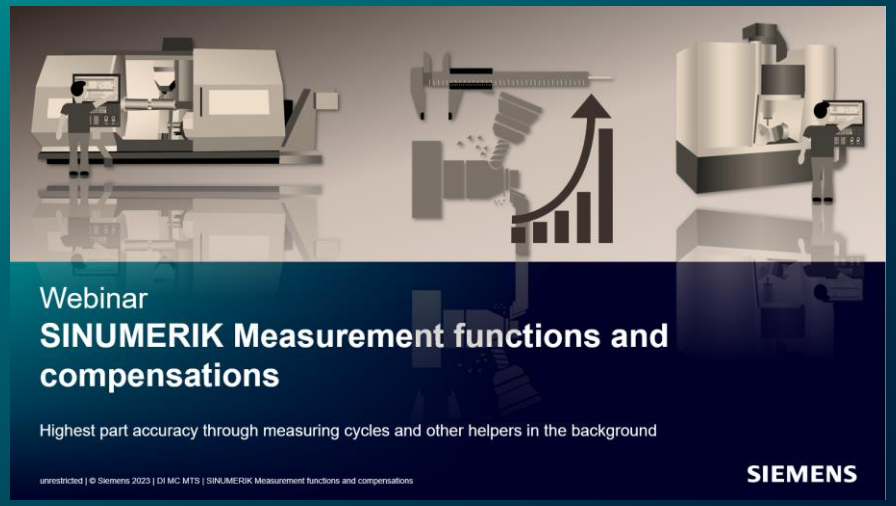
Edge	Slot	Rectangular corner	Rectangular pocket	Hole	Inside circle segment	Ball	Align plane	Front edge
								

Align edge	Web	Any corner	Rectangular spigot	Circular spigot	Outside circle segment	3 balls	Inside diameter	Outside diameter
								

With a **comprehensive package of measuring cycles**, SINUMERIK CNCs ensure maximum **work-piece accuracy even for large batch sizes**.

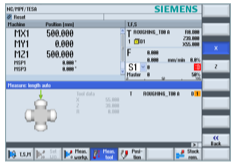
SIEMENS

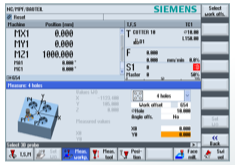
SINUMERIK Measurement functions and compensations
- own webinar



The basic measurement methods
- measurement in the setup mode and process measurement

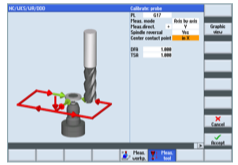
Measurement in JOG (setup mode)

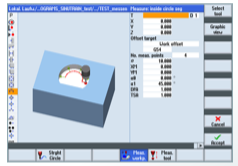




- Measurement in JOG serves to **prepare the machine** for the machining task.
- Measurement in JOG serves to measure **unknown** work-piece or tool geometries.
- The measurement is performed „semi-automatically“ as an **interactive operation** in the setup mode of the machine (JOG mode).

Measurement in Automatic (process measuring)





- Measurement in Automatic serves to **control the accuracy of the work-pieces**.
- Measurement in Automatic serves to control **“known”** work-piece or tool geometries.
- The measurement is performed „automatically“ by the **call of a measurement cycle** in the CNC-program (AUTOMATIC mode).

Basically there are **two different use cases** for measuring inside the machine – **measurement in the setup mode** (JOG) and **in-process measurement** (AUTOMATIC).

unrestricted | © Siemens 2023 | DI MC MTS | SINUMERIK Measurement functions and compensations

SIEMENS

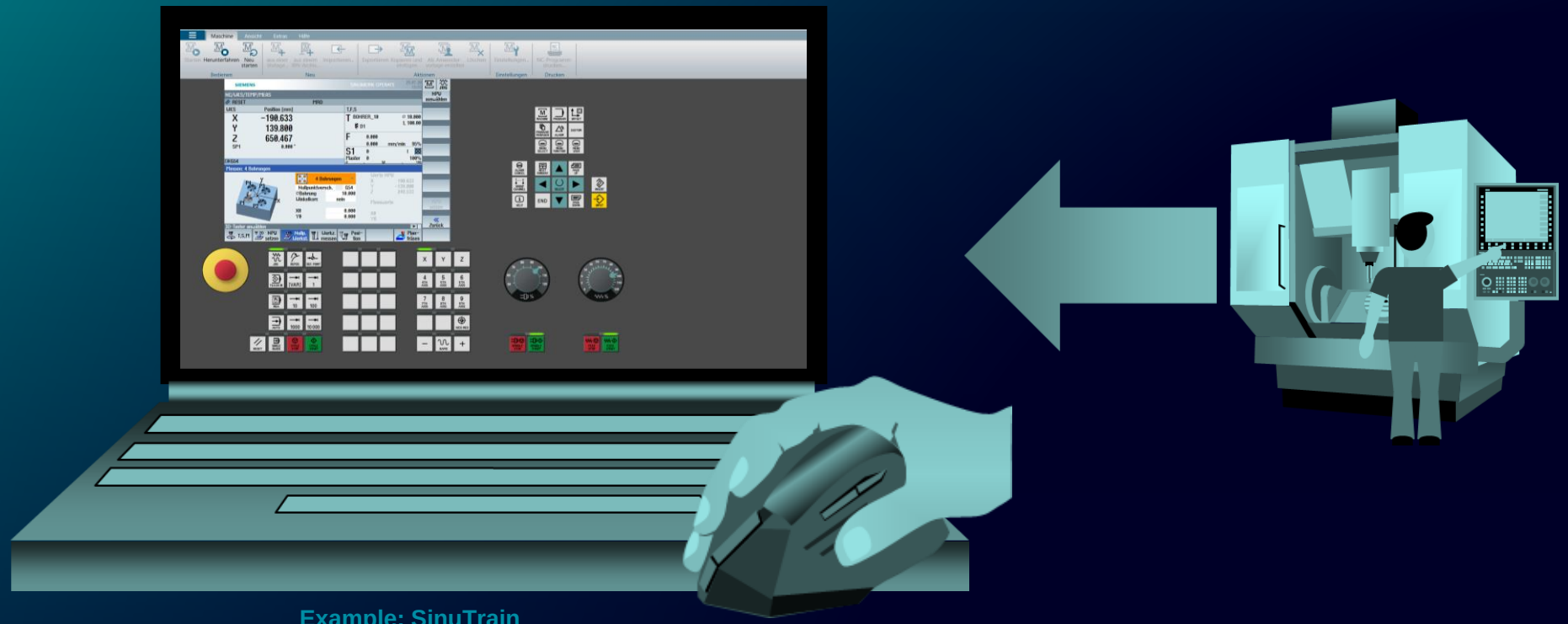
SIEMENS

Agenda

1. Overview
2. Work-step programming ShopMill / ShopTurn
3. G-Code
4. programGUIDE
5. Machining cycles (ShopMill / ShopTurn, programGUIDE)
6. **Offline-programming in the CNC-job preparation**
7. CNC4you

SIEMENS

CNC-job preparation with SINUMERIK Operate
- all CNC programming features also available offline !



Example: SinuTrain

+ The ideal introduction to CNC work preparation
+ The ideal addition to CAD/CAM systems

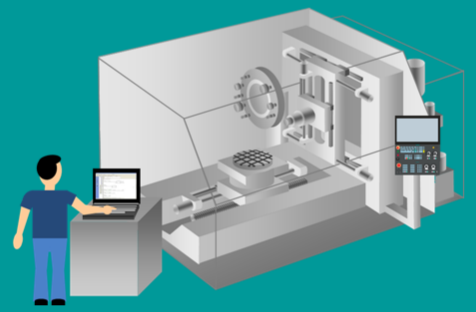
SIEMENS

Digital Twins in the CNC job preparation
- own webinar



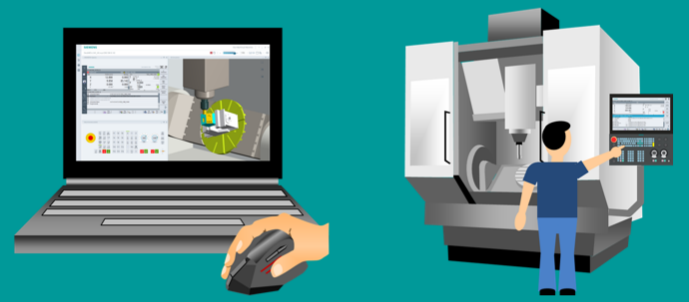
The two types of digital twin
- the principle difference

Automation of the machine



- ⇨ primary use for automation engineering
- ⇨ engineering without real prototype machines
- ⇨ **more efficient, time optimized engineering**

Application of the machine



- ⇨ primary use for the operation of the machine
- ⇨ testing of CNC programs in job preparation
- ⇨ **higher process reliability on the shop-floor**

Agenda

1. Overview
2. Work-step programming ShopMill / ShopTurn
3. G-Code
4. programGUIDE
5. Machining cycles (ShopMill / ShopTurn, programGUIDE)
6. Offline-programming in the CNC-job preparation
7. **CNC4you**

SIEMENS

CNC practical knowledge: production, job shop and production planning

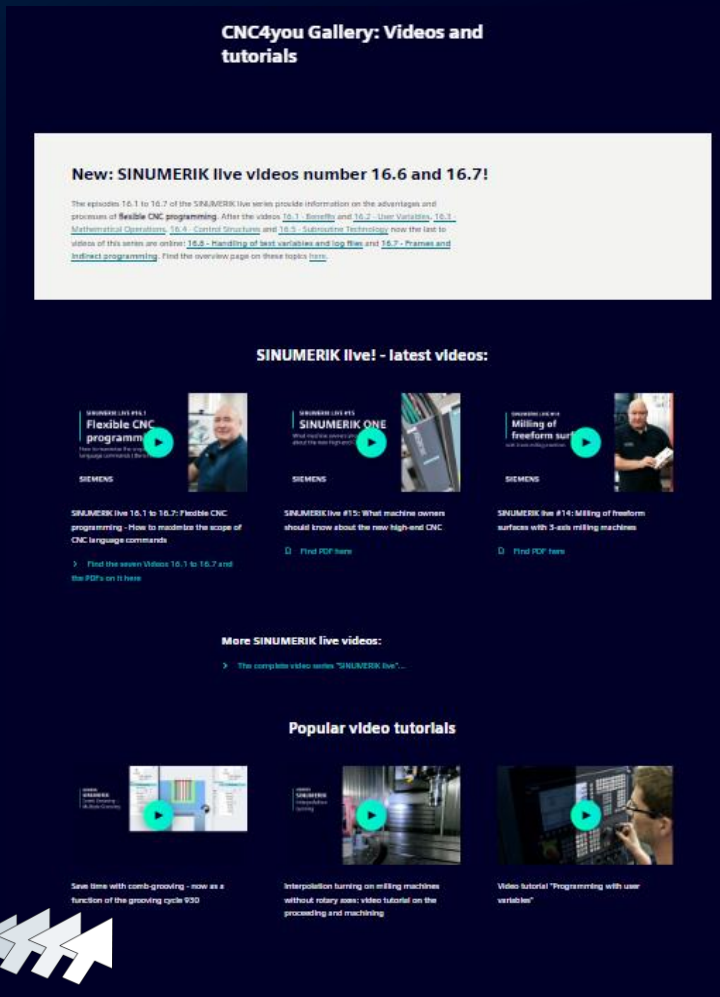
Do you work with SINUMERIK CNCs? Here you will find the matching PC programming station SinuTrain, tips and tricks for NC programming, information on CNC training, CAD/CAM integration and digital manufacturing support in workshops and industry.

The CNC4you magazine



SINUMERIK LIVE
- an alternative learning medium





SIEMENS

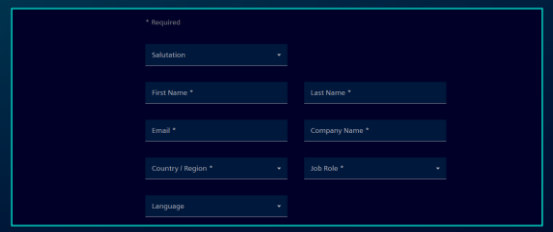
CNC4you newsletter
- always up to date about the SINUMERIK application

- **Dates for the next webinars**
- new SINUMERIK LIVE videos
- each new CNC4you-emagazin
- and more….

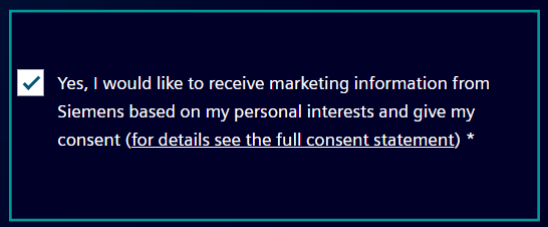
CNC4you newsletter

Here you can subscribe to the CNC4you newsletter. It is published about 6 times a year and informs you about news around CNC4you. You will also receive our CNC4you e-Magazine automatically and free of charge via the newsletter.

1



2



3



General Data Protection Regulation

SIEMENS

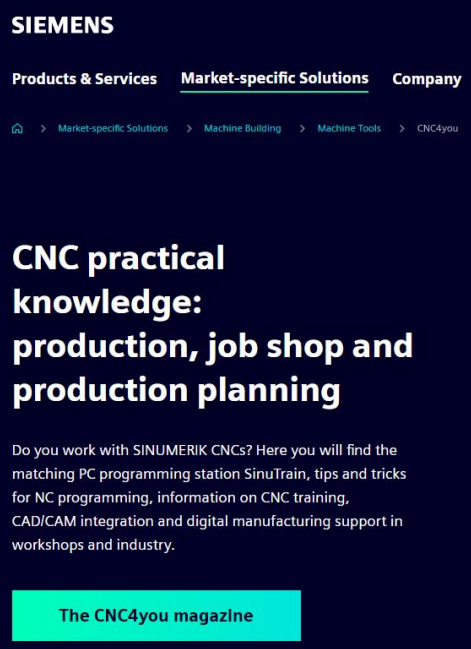
Disclaimer

© Siemens 2023

Subject to changes and errors. The information given in this document only contains general descriptions and/or performance features which may not always specifically reflect those described, or which may undergo modification in the course of further development of the products. The requested performance features are binding only when they are expressly agreed upon in the concluded contract.

All product designations may be trademarks or other rights of Siemens AG, its affiliated companies or other companies whose use by third parties for their own purposes could violate the rights of the respective owner.

Contact



CNC4you



Contact



SIEMENS