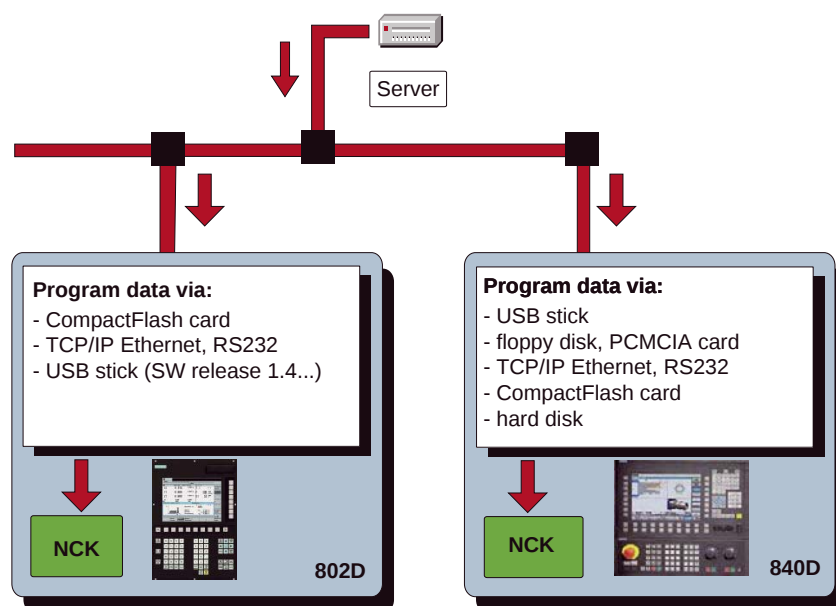


2.8 Program data transfer/managing programs

NC programs are stored in the control, if required, downloaded into the NCK working memory (RAM) and executed on the machine.

For mold making programs - that generally comprise technology and geometry programs - the geometry program with up to 100 MB is often so large that it can no longer be saved in the NCK working memory (RAM) and/or cannot be processed. This is the reason that mold making programs must be saved on an external memory so that they can be successively processed.



External program memory (hardware configuration)

Depending on the system, the existing operator interface (HMI) and the options that have been purchased, you can use external program memories with the following characteristics:

- TCP/IP Ethernet (network drivers), serial interface RS232/V.24 (low data rate)
- Hard disk (PCU 50)
- Compact Flash card (TCU, 802D)
- USB interface (USB stick)
- PCMCIA card (PCU 20)
- Floppy disk

Calling the program data in the external memory using EXTCALL

An EXTCALL command is programmed in the main program that in turn calls the geometry program that has been externally stored corresponding to the network path on the server, the USB interface, hard disk etc.

Using EXTCALL, you can subsequently download a program from the HMI in the execute from external mode.

In this case, all programs that can be reached through the directory structure of the HMI, can be subsequently downloaded and executed.

Procedure when calling the geometry program using EXTCALL

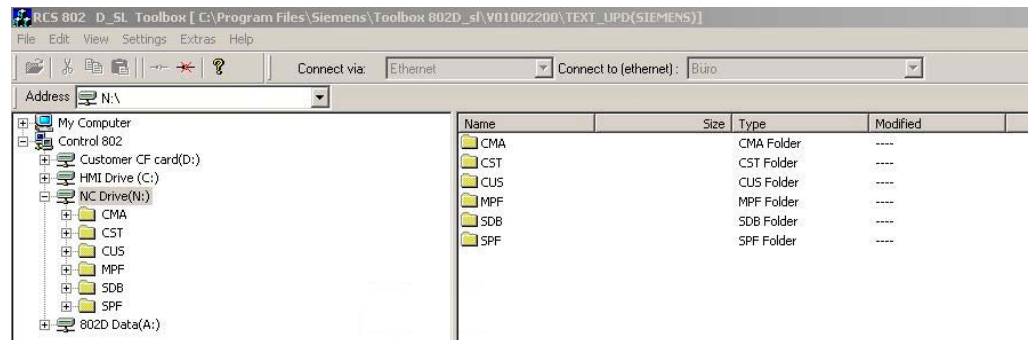
- ▶ In machine data SD 42700: EXT_PROG_PATH, define the source directory to the geometry program, e.g. to a server "\\R4711\workpieces\subprograms". The default setting is optional. The directory can first be specified when making the call with EXTCALL.
- ▶ Program the geometry program call, e.g. SAMPLE in the main program. The call differs depending on the control and where the data is saved.
 - PCU 50, subroutine is on the hard disk
EXTCALL "sample"
 - PCU 20, 802D, subroutine is directly on the CompactFlash card
EXTCALL "C:\sample.spf"
 - PCU 20, 802D, subroutine is in the directory on the CompactFlash card
EXTCALL "C:\programms\sample.spf"
 - Network connected to Ethernet and path in the machine data SD 42700
EXTCALL "sample.spf"
 - Network connected to Ethernet and no path in the machine data SD 42700
EXTCALL "\\myserver\programms\workpieces\sample.spf"

Managing large programs using the 802D sl – RCS tool

The RCS tool (Remote Control System) is an Explorer-type tool for your PC/PG to assist you in your daily work with SINUMERIK 802D sl.

The connection between the control system and the PC/PG can be established either via an RS232 cable, peer-to-peer cable, or a local network cable (optional).

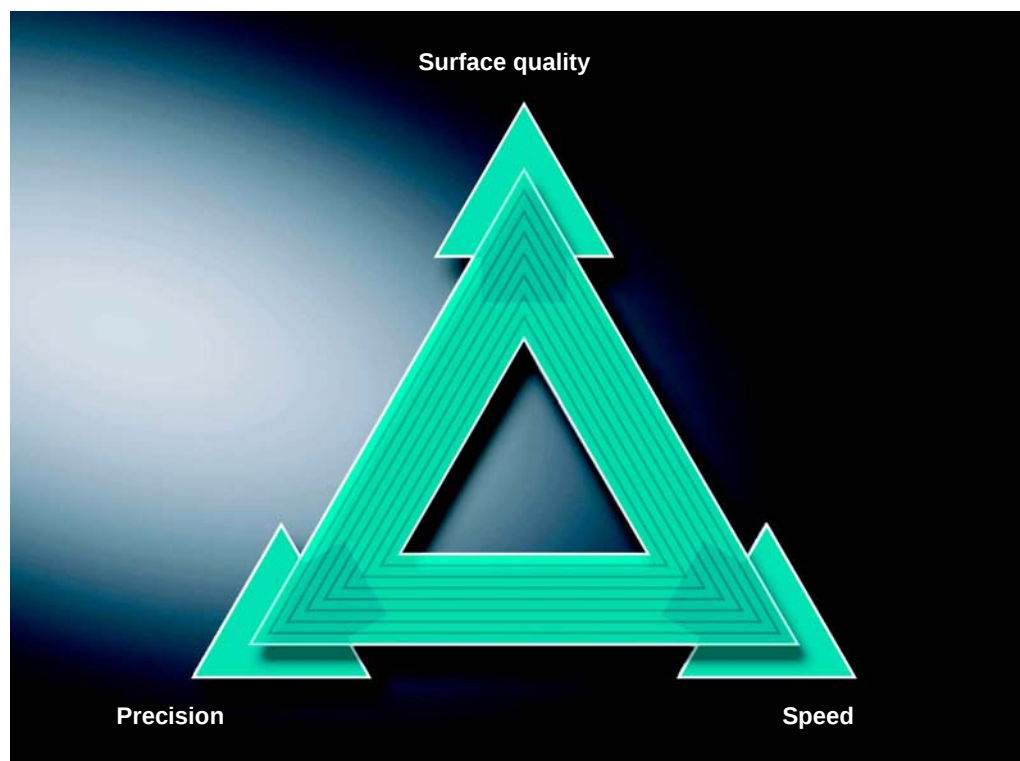
When executing from external with 802D sl please note that programs on the CF card cannot be edited. If these programs are larger than the NCK memory then you must externally edit the programs (e.g. PC). The RCS tool is provided for precisely this task. In a transparent Explorer-type display you can copy, shift and delete programs or other data.



2.9 High Speed Settings – CYCLE832

Application

You can influence the sequence of CAM programs using CYCLE832 of the SINUMERIK 840D. It is used to provide technological support when machining freeform contours in the 3-axis high-speed machining sector (High Speed Cutting - HSC). CYCLE832 combines the essential programming commands and G codes that are required for HSC.



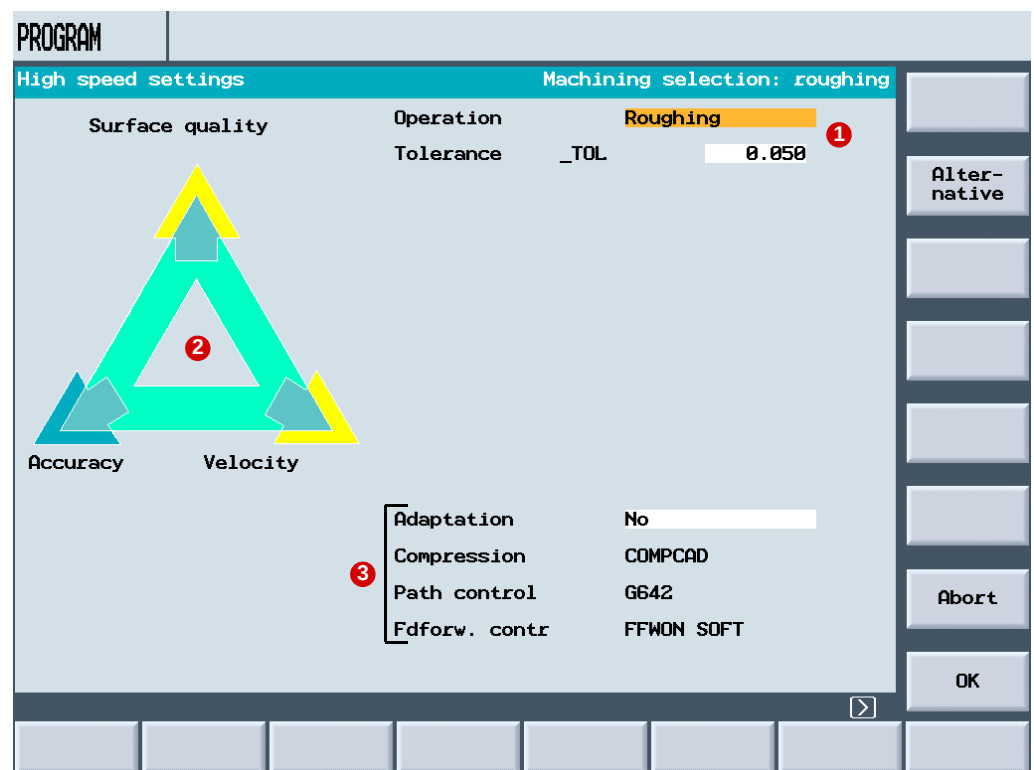
When executing CAM programs in the HSC area, the control has to process high feedrates with the shortest NC blocks. A good surface quality with high precision in the μm range at high machining feedrates $>10 \text{ m/min}$ is expected. By applying different machining strategies you can use CYCLE832 to fine tune the program.

- When **roughing**, the emphasis is on speed due to the blending of the contour.
- When **finishing**, the emphasis is on precision and surface quality.

In both cases, specifying a tolerance ensures that the machining contour is observed in order to achieve the desired surface quality and precision. Generally, when roughing, the tolerance is selected higher than when finishing.

Calling CYCLE 832 in HMI menu tree

Programs	Opens the "Programs" operating area.
Milling	Press the "Milling" softkey.
>>	Display additional softkeys.
High Speed Settings	Press "High Speed Settings". The cycle is called.

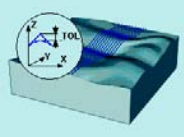
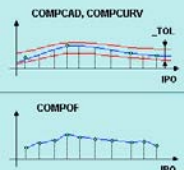
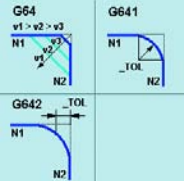
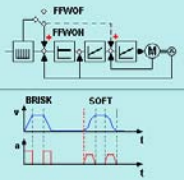


Corresponding to the parameter selection ① the yellow arrows ② either point towards "Speed", "Surface quality" or in the direction "Precision".

The other options ③ are released by the machinery OEM and are generally password protected.

Parameters for the High Speed Setting cycle

In the **Machining** field the user only has to select between finishing, pre-finishing and roughing and enter a value in the **Tolerance** field. The values in all of the other fields have already been entered by the machinery OEM. The machinery OEM can enable the other fields using the **Adaptation** field (password-protected).

Machining	☒ Finishing (default) ☐ Pre-finishing ☐ Roughing ☐ Deselection	By calling "De-selection" the modified machine/setting data are reset to the value generated by the machinery OEM
Tolerance_tol. 	☐ Chord tolerance (chord tolerance should be taken from the CAM system or weighted with a factor of 1.2 ... 1.5)	Tolerance of linear/rotary axes, default settings: → 0.01 mm/ 0.08° (finishing) → 0.05 mm/ 0.4° (pre-finishing) → 0.1 mm/ 0.8° (roughing) → 0.1 mm/ 0.1° (deselection)
Adaptation	☐ Yes ☐ No	→ The following fields can be modified → The following fields are invisible - and are released by the machinery OEM.
Compression 	☐ COMPOF (default) ☒ COMPCAD ☐ ☐ B SPLINE	→ Compressor off → Compressor on, continuous acceleration rate for mold making applications → Jerk-free for circumferential milling → Spline interpolation
Continuous path control 	☒ G642 (default) ☐ G641 ☐ G64	→ Blending with single-axis tolerances → Programmable blending clearance → Continuous path mode With an NC block compressor with COMPCAD, COMPCURV, G642 is always permanently selected.
Feedforward control 	☐ FFWOF SOFT ☐ FFWON-SOFT ☐ FFWOF-BRISK	→ Without feedforward control, with jerk limiting → With feedforward control, with jerk limiting → Without feedforward control, without jerk limiting The selection of feedforward control (FFWON) and jerk limitation (SOFT) requires that the machine manufacturer has optimized the control and the machining axes.

- Notes**
- CYCLE832 is based on the use of G1 blocks. The tolerance is not important when using G2/G3 and CIP programs.
 - When making changes you should align to the tolerance value to that specified in the CAM program. Tolerances that are lower than specified there are not practical.
 - Please note that there are dependencies between the fields: For instance, if **compression** is switched-out then various grinding types can be selected under continuous **path control**.

NOTE

Please refer to Chapter 3.5 for additional information; individual parameters are described in detail here.

Programming

Ideally, you program CYCLE832 in the higher-level NC master program that then calls the geometry program. This means that you can apply the cycle to the complete geometry or - depending on the transparency of the CAM program - to individual program sections or freeform surfaces.

NOTE

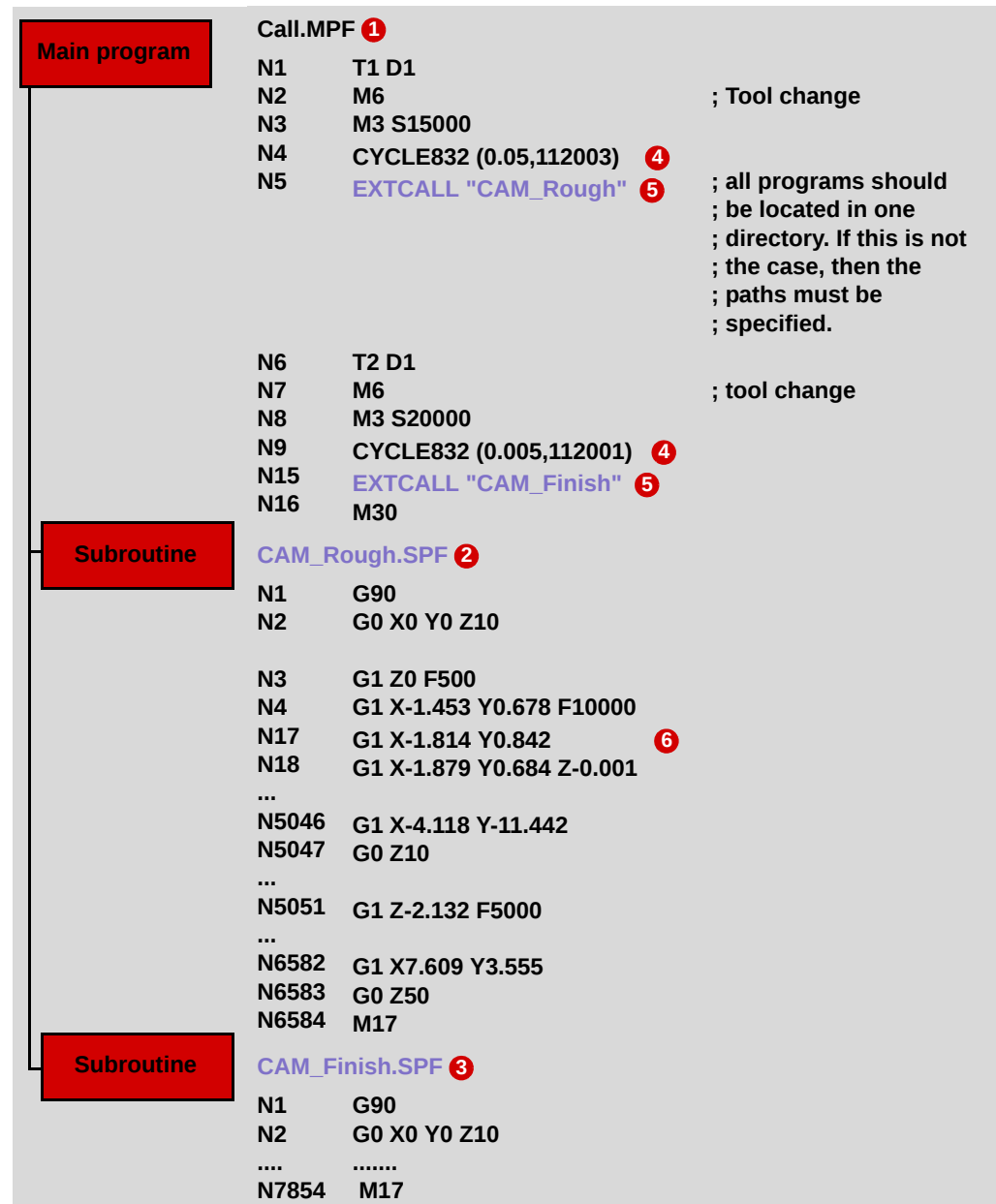
For information regarding an optimum program structure please refer to Chapter 1.4, specifically for CYCLE832, please observe the information in Chapter 2.10.

2.10 Program structure for mold making

Recommendation for a practical program structure with CYCLE 832

For machining a main program is generated ① that includes all technology data. The main program calls one or several subroutines ②, ③ that contain geometry data of the workpiece. The tool change defines the sub-division into subroutines.

Example



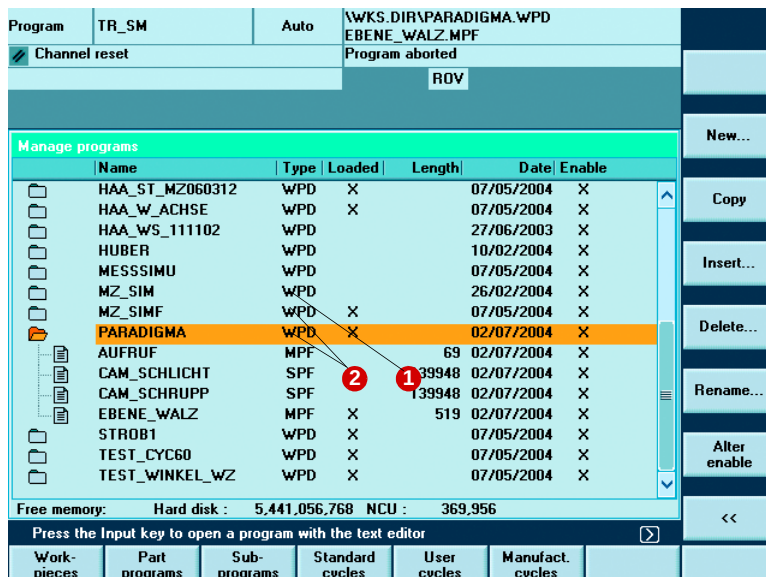
Main program: The main program includes two important functions for milling, CYCLE832 ④ and EXTCALL ⑤.

CYCLE832 ④ : CYCLE832 has been specifically developed for the program structure shown where technology and geometry data are separated. The machining technology for milling is defined in CYCLE832. For the roughing program "CAM_Rough" using T1, the parameters in CYCLE832 were set towards achieving a high speed. For the finishing program "CAM_Finish" the parameters were set towards achieving high precision and surface quality.

EXTCALL ⑤ : CAM programs are generally extremely large which is why they are stored in external memory. EXTCALL calls the subroutines from the external memory.

Subroutine: In the subroutine - G90 for absolute programming is immediately followed by the geometry sets. In our particular example these are the blocks for 3-axis milling ⑥.

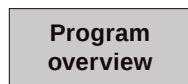
2.11 Selecting/starting/stopping/interrupting/continuing a program



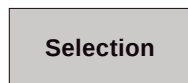
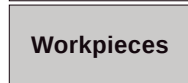
Select the "Machine" operating area.



Select the "Automatic" mode.



Press "High Speed Settings". The cycle is called.



In the workpiece directory highlight the part program **1** - in this case the program "Call.MPF" ("Aufruf.MPF") and press "Select" ("Anwahl").



Press "NC-Start" to start the part program. This calls the geometry programs "Roughing.SPF" ("Schrupp.SPF") **2** and "Finishing.SPF" ("Schlicht.SPF"), which are loaded block-by-block into the control system from the external memory during machining.



Press "NC-Stop" to stop the part program.



Press "Reset" to interrupt the part program.

Note A part program interrupted with "NC-Stop" can be continued by pressing "NC-Start". A part program interrupted with "Reset" is executed from the beginning if "NC-Start" is pressed - or, with a block search jumps to the point of interruption where it continues the program.

2.12 Interrupting a program

REPOS – repositioning after an interruption

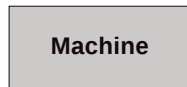
Function

When a program is interrupted - or after an NC-Stop - the tool can be moved away from the contour in the JOG mode, e.g. to check the cutting edge of a tool. The control saves the interruption point coordinates. The differential travel of the axes is displayed.

Operation



Initial situation: Program interrupted with "NC-Stop".



Select the "Machine" operating area.



Select the "JOG" mode.



Reposition after a program interrupt.



Select axes.



Move the axes to the point of interruption according to the differential travel displayed. It is not possible to pass the point of interruption.



Changeover from the "JOG" into the "Automatic" mode.



Continue machining

Accelerated external block search without calculation

Function

This SINUMERIK 840D function was specifically developed for programs that are called with EXTCALL. This means that it is admirably suited for large programs that are received from CAM systems.

After machining has been interrupted with "Reset" using the function "Accelerated external block search without calculation" any location in the part program can be selected at which machining is to be started or continued.

Operation



Initial situation: Program was interrupted with "Reset".

Example

```

Call.MPF ①
N1      G54
N2      T1 D1
N3      M3 S15000
N4      CYCLE832 (0.05,112003)
N5      EXTCALL "CAM_Rough"
N6      T2 D1
N7      M3 S20000
N8      CYCLE832 (0.005,112001)
N16     EXTCALL "CAM_Finish" ③
N10     M30

CAM_Roughing.SPF ①
N1      G90
N2      G0 X0 Y0 Z10

N3      G1 Z0 F500
N4      G1 X-1.453 Y0.678 F10000
N17     G1 X-1.814 Y0.842
N18     G1 X-1.879 Y0.684 Z-0.001

CAM_Finishing.SPF
N1      G90
  
```

Block
search

Press the "Block search" softkey.

Search pointer

Press the "Search pointer" softkey.

Break
point

Press the "Breakpoint" softkey.

Machine	TR_5M	Auto	\WKS.DIR\PARADIGMA.WPD	
Channel reset			AUFBRUF.MPF	
			Program aborted	
ROV				
Machine	Position	D. to go	Master spindle S1	
X1	0.0000 mm	0.0000	Act.	0.000 rpm
Y1	0.0000 mm	0.0000	Set	0.000 rpm
Z1	100.0000 mm	0.0000	Pos.	0 deg.
C1	294.4700 deg	0.0000		100.0 %
A1	65.5300 mm	0.0000	Power	0%
Search pointer				
WKS\PARADIGMA\AUFBRUF.MPF				
1: AUFBRUF.MPF	P	1	Type	N16
2: CAM_SCHRUPP.SPF	1	1	N3044	
3:				
4:				
5:				
Enter search target				
Search position	Search	Search	Search	Search

Pressing the "Breakpoint" softkey inserts the screen with the entire program sequence **1** up to the breakpoint:

In this example, the main program "Call.MPF" ("Aufruf.MPF") calls the subroutine "CAM_Roughing.SPF" ("CAM_Schrupp.SPF"). The EXTCALL for the subroutine is located in block N16 **3**. Block 3044 in "CAM_Roughing.SPF" ("CAM_Schrupp.SPF") is where the program was interrupted.

There are now two possibilities:

1. Jump directly to the breakpoint in the subroutine: Press the "External without calc." softkey. The program jumps immediately to block 3044.
2. To do this you must select a (search) type **2** - when searching in external programs always select type 3 for a string search. Then enter the type number and next to it the required search text - e.g. block or line number.

Press the "External without calc." softkey.

external
without calc.



Continue machining at the destination block.

Over-
store

Corrections

While making an entry for CYCLE Stop the "Overstore" function can be selected that so that you can correct the destination block before starting the program.

Machine	TR_5M	Auto	\WKS.DIR\PARADIGMA.WPD	
Channel reset			AUFBRUF.MPF	
			Program aborted	
ROV				
Machine	Position	D. to go	Master spindle S1	
X1	0.0000 mm	0.0000	Act.	0.000 rpm
Y1	0.0000 mm	0.0000	Set	0.000 rpm
Z1	100.0000 mm	0.0000	Pos.	0 deg.
C1	294.4700 deg	0.0000		100.0 %
A1	65.5300 mm	0.0000	Power	0%
Feedrate [mm/min]				
CYCLE832 (0.02) %				
1				
micro f				
Tool				
D14				
Preselected tool				
7				
G01				

A typical case is shown here, where the compressor tolerance needs to be subsequently changed. CYCLE832 was called to do this and the compressor tolerance was manually changed to 20 μ m **1**. This was possible by just entering one single parameter (tolerance = 0.02). CYCLE832 is now executed before the main program is started.



The tolerance is activated by pressing NC-Start.

2.13 Program overview/status of external programs

Function

When executing programs from external you can display the current status and the program runtime.

Displaying the status for 840D standard HMI

**Program
overview**

In the "Automatic" mode, select the softkey **Program Overview**. The program overview is displayed.

**External
programs**

Select the softkey **External Programs**.

FILE	JETFORM	WPD		10/06/2007	17:07:07	X	
FILE	MATHE_COS_EXP	WPD		02/02/2007	16:18:08	X	
FILE	MATHE_SPLINE_C	WPD		02/02/2007	16:18:08	X	
FILE	PLATE_BLOCK	WPD	X	12/09/2007	12:07:56	X	
FILE	DPWP	INI		9452 24/06/2004	11:17:52		
FILE	KONTUR	SPF		651 24/06/2004	11:17:52	X	
FILE	PLATE_BLOCK	MPF	EXT	3982 24/06/2004	11:17:52	X	
FILE	PLATE_FLANGE	WPD		02/02/2007	16:18:08	X	
FILE	PLATE_LEVER	WPD		02/02/2007	16:18:08	X	
FILE	WING	WPD	X	10/06/2007	13:13:33	X	
Free memory: Hard disk : 5.256.671.232 NCU : 1.707.272							
Work-pieces Part programs Sub-programs Standard cycles User cycles Manufact. cycles External programs							

The current status of the external program is displayed as a percentage in the overview that is displayed.

Machine	840D_Mill	Auto	\WKS.DIR\PLATE_BLOCK.WPD		
			PLATE_BLOCK.MPF		
Channel reset		Program aborted			
		ROV SBL1			
External programs					
Name	Type	Length	NC name	Channel	SP Status
\wks\plate_block\plate_block	MPF	23982	NCU840D	1	0 100%

2.14 Simulating the part program

Function

Using the simulation function you can obtain an overview of the individual machining steps and you can check the workpiece programming.

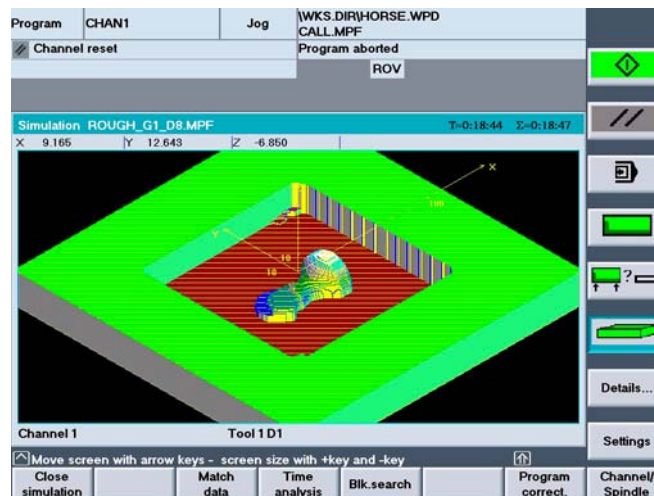
Selecting the simulation function:

- Select the Simulation softkey in the Program Editor.

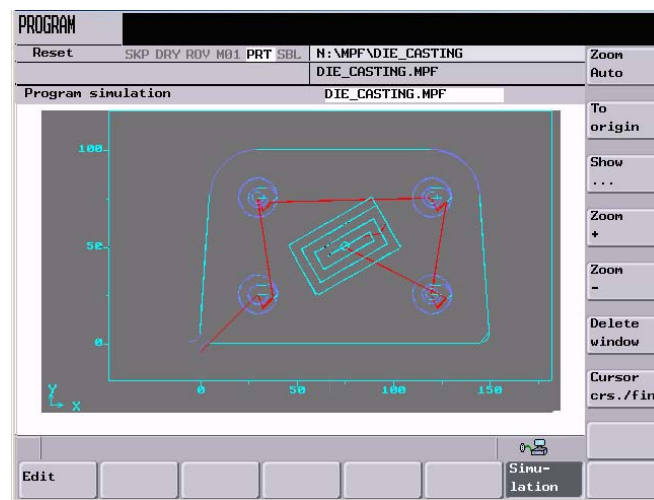
Simulation functions depending on the control:

- Simulation is displayed in the 3 planes (840D) or the machining plane (802D sl)
- Turning, scaling and zooming in/out
- 3-D volume model in any view and any sections (only 840D)

Example, simulation
840 D



Example, simulation
802D sl

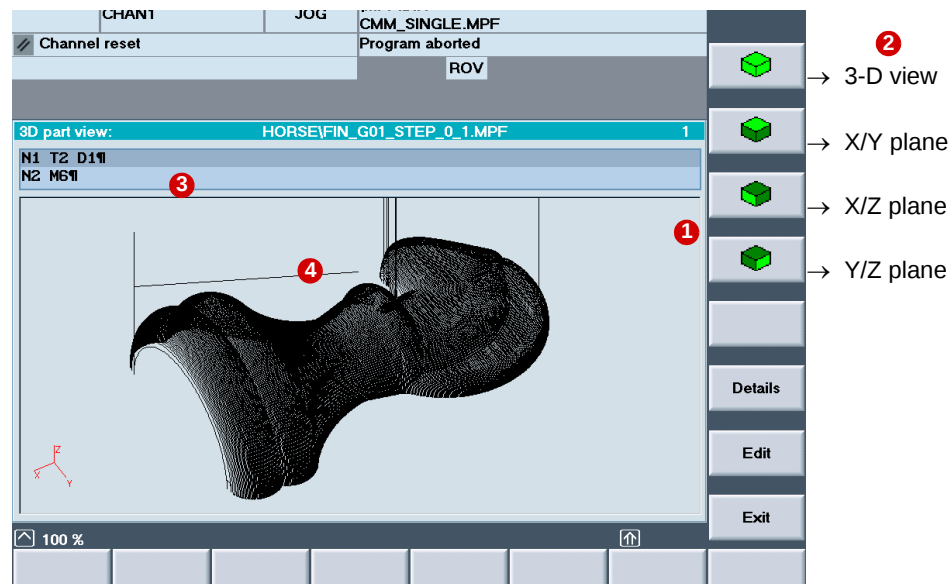


2.15 Quick View / fast display

Function

Quick View of SINUMERIK 840D with PCU50 allows mold making part programs that contain G01 blocks to be visualized. Program loops, polynomials, transformations and G02/03 blocks are not supported. Four views **1** are available: 3-D view **2**, X/Y plane, X/Z plane, Y/Z plane.

The two editor lines **3** display the block currently highlighted in the graphic. Scrolling through the editor window automatically highlights the position **4** in the graphic.

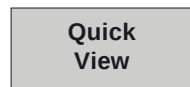


The following functions are also available

- Search for a specific block
- "Zoom in/out"
- Shift, rotate
- Measure the distance between two points
- Edit the NC part program displayed

Notes You can use Quick View (fast display) for the 840D with standard interface and for ShopMill. For 840D Standard Quick View is in the **Program Manager**, for ShopMill, you can open Quick View in the **Program Editor**.

Operation



Call the "Quick View" function.

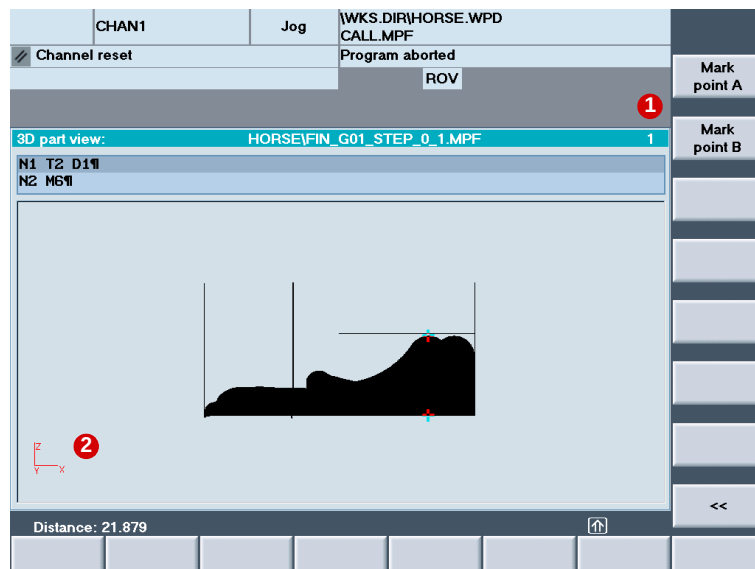


Choose the view you require - in this case the X/Z plane.



Use the cursor to select a point in the graphic. The associated block is displayed in the editor line.

Call the block, e.g. to change it in the program.



Measuring distances in Quick View

In Quick View you can also measure distances between two points. This function is extremely helpful, e.g. if you want to know the dimensions of a workpiece - because it is only very difficult to assess the size of a workpiece from the G1 blocks of a program.

By highlighting two points **1** you are shown the distance **2** between the points in the footer area.

2.16 ShopMill

For SINUMERIK 840D the user-friendly ShopMill interface provides a real alternative to the universal SINUMERIK 840D standard DIN/ISO user interface.

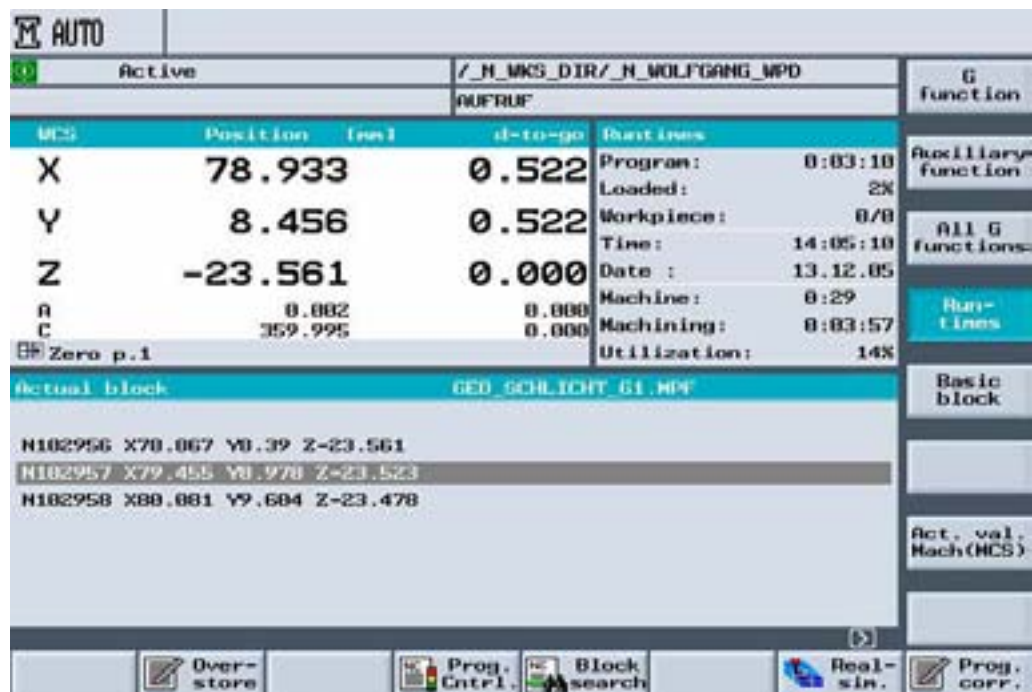
ShopMill has been supplemented by many mold making functions, greatly simplifying its use for mold makers.

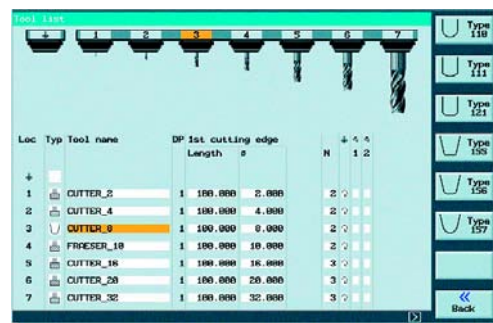
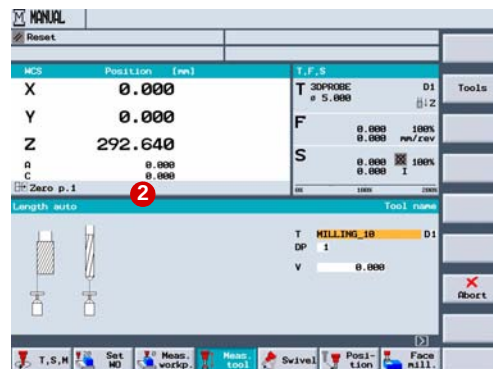
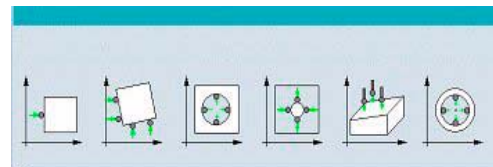
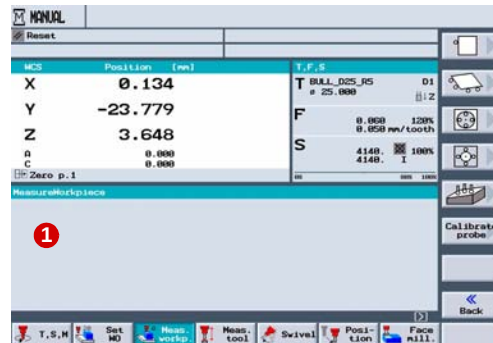
As a consequence, ShopMill is no longer restricted to sequencer programming using partial machining steps - and in fact it even supports demanding 3+2-axis and 5-axis applications.

The complete description of ShopMill functions can be found in "SINUMERIK 810D/840D Using and Programming ShopMill".

ShopMill user interface

Simple operation and programming in the workshop





ShopMill functions

Setting-up

Powerful setting-up functions in ShopMill ensure rapid and accurate detection of the workpiece position. Special measuring cycles simplify measuring tools and the workpiece. Any offsets are automatically compensated by the control system.

- 1 Measuring a workpiece
Edge, corner, pocket/ hole, spigot/ rectangle, plane))

The measuring functions are available when measuring in JOG. Measuring cycles support you when measuring in the AUTOMATIC mode.

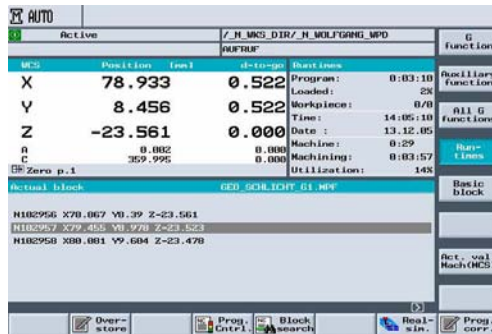
- 2 Measuring a tool

The measuring functions are available when measuring a tool in JOG and when measuring in the AUTOMATIC mode.

Tool management

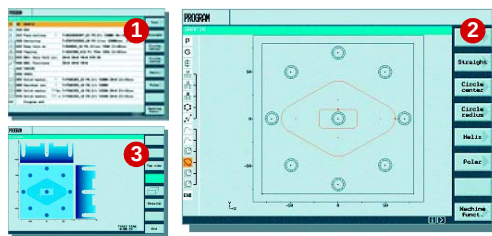
The ShopMill tool management is clearly structured and supports various tool types, tool names in plain text, daughter tools and the tool geometry with lengths, radii and number of cutting edges.

By entering the unit quantity, lifetime or wear parameters you can automatically monitor the time that tools are used - therefore ensuring a uniform machining quality.



AUTOMATIC

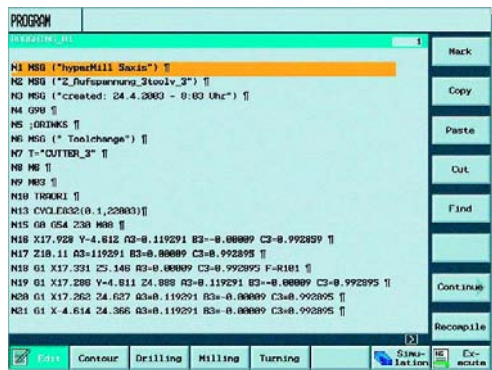
Displaying runtimes in the basic automatic screen.



Sequencers programming

The ShopMill sequencer programming permits simple programming of basic 2 1/2D machining tasks directly at the machine. This is an ideal add-on for mould makers.

- 1 Program
- 2 2D display
- 3 3D display



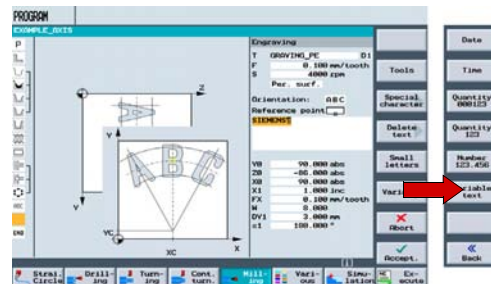
G code editor

ShopMill comes with an integrated, powerful G code editor, which supports mold making programs in a user-friendly fashion.



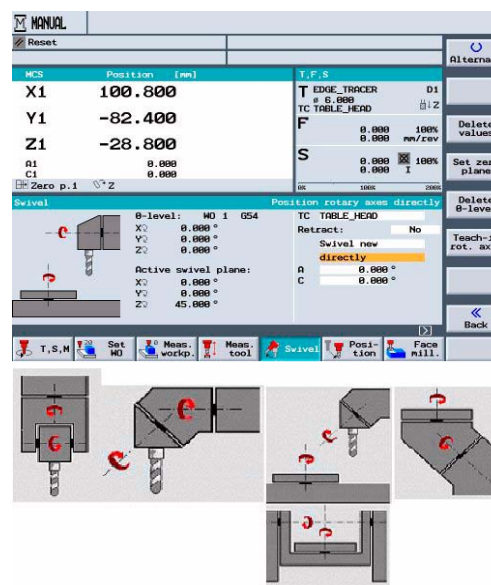
Block search

Fast block search in an external program (with and without calculation).



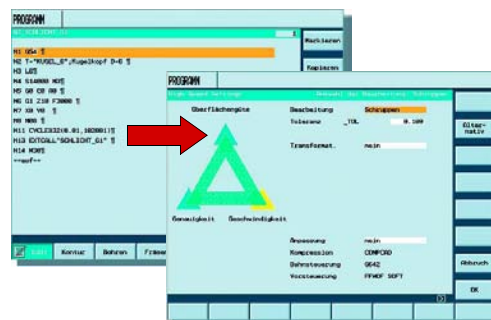
ShopMill cycle for engraving

- Text with special characters
- Date, time, workpiece counter, variable



Swiveling in JOG

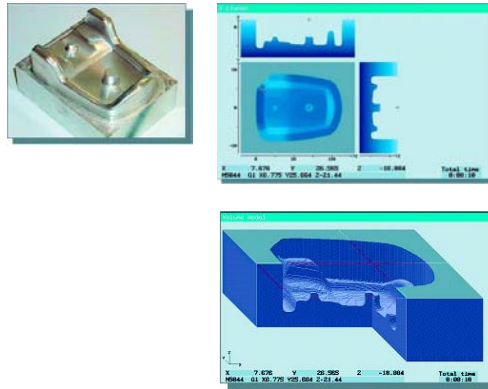
- Swivel cycle for all swivel tasks in the setting-up mode using softkeys
- Input is either directly or axis by axis
- All machine kinematics are supported



"High Speed Setting" cycle

The "High Speed Setting" cycle is now an integral component of the ShopMill user interface in the G code editor.

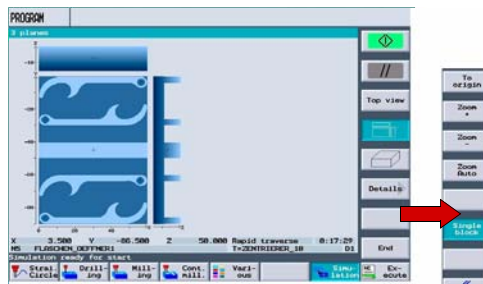
- Program editor
- CYCLE832, High Speed Settings



Simulation

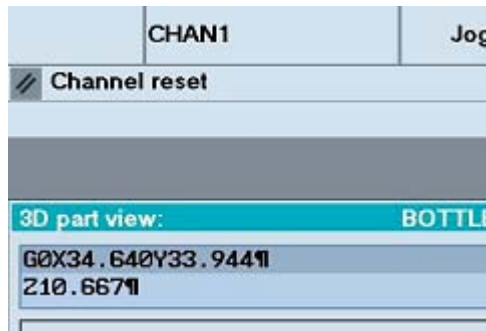
ShopMill provides various extensive and detailed simulation functions for displaying machining paths. To simulate the machining process, the control system completely calculates the currently selected program and displays the result in graphical form. You can select one of the following representation types for the simulation:

- Top view
- 3-plane view
- Volume model



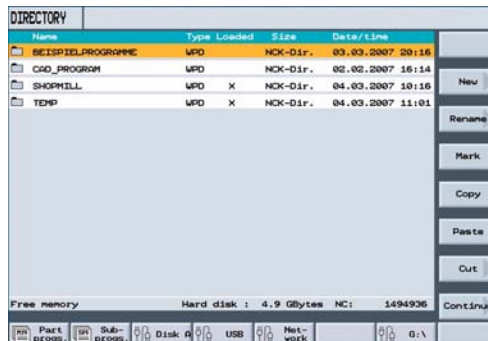
Program - simulation

- Start / stop / single block / reset of the simulation using softkeys
- Velocity can be controlled using the override function



Fast display / Quick Viewer for mold making

Quick display of traverse paths is possible for large part programs. With this dotted-line quick view display all programmed positions from G1 are displayed as resulting axis paths.



Function to access a drive

External drives

The ShopMill Program Manager allows direct access to external drives via Ethernet. Extensive mold making programs can be saved there.

- HMI hard disk (PCU 50)
- Flash card (PCU 20)
- Network drives
- USB stick



Information for programmers

Contents	Page
3.1 Introduction	62
3.2 What are frames?	63
3.3 Swiveling - CYCLE800	66
3.4 Programming example - swiveling	70
3.5 High Speed Settings – CYCLE832	72
3.6 Feedrate profile – FNORM, FLIN	80
3.7 Programming example with CYLCE832	81
3.8 Programming example without CYLCE832	84

3.1 Introduction

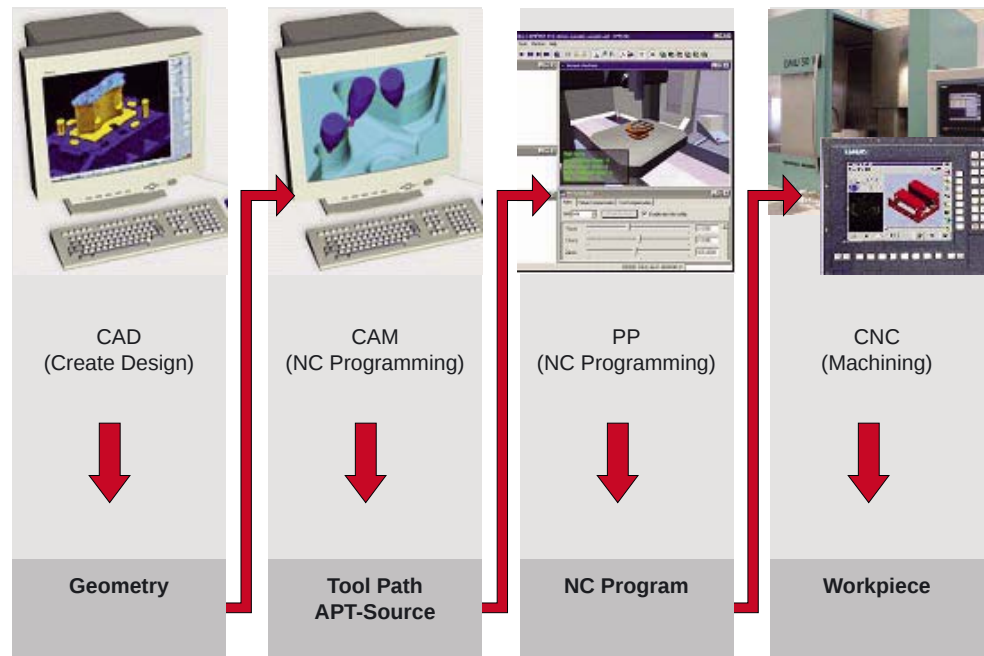
When programming freeform surfaces the entire CAD/CAM/CNC process chain is of vital importance.

The CAD system generates the geometry of the desired workpiece. Based on this geometry file the CAM system generates the corresponding machining strategy with the associated technology information.

The data format output from the CAM system is generally an APT or CL data file. This is converted in the post processor into an executable CNC code.

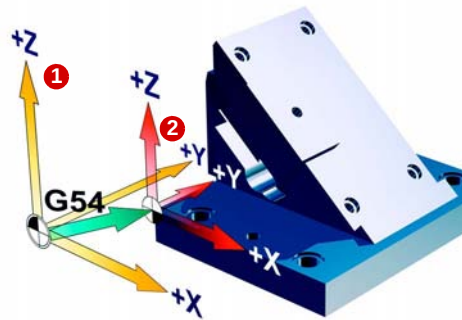
The upstream post processor is of the utmost importance in being able to fully utilize the capabilities and performance of SINUMERIK controls to the maximum.

The post processor must ensure that the higher-order functions of SINUMERIK controls - described in this Chapter - are activated in the best possible way. An overview of all of the higher-order SINUMERIK 840D functions can be found in Chapter .



3.2 What are frames?

Coordinate systems



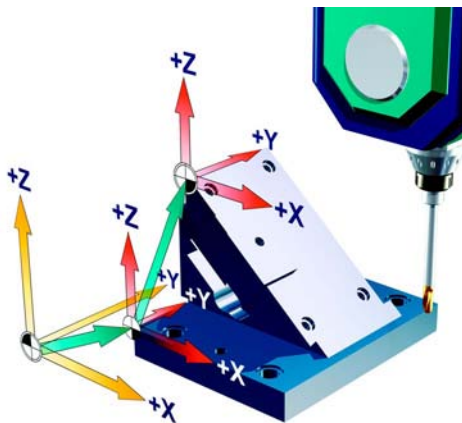
Machine coordinate system **1** with reference point and zero offset (G54, G55, ...) are known terms.

Using frames, coordinate systems can be shifted, rotated, mirrored and scaled so that they are aligned to the workpiece surface. This allows the programming time & costs to be reduced to a minimum.

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

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

Coordinate systems and traversing motion

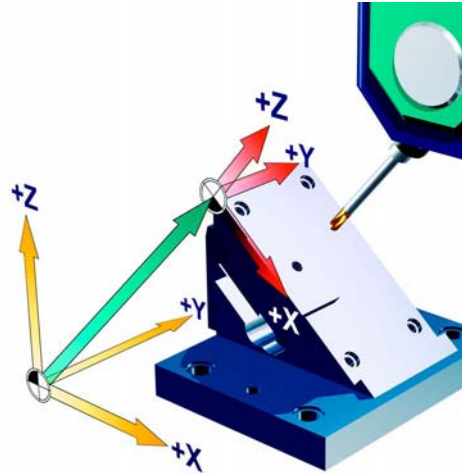


Using a 3+2-axis machine it is possible to machine surfaces that can be shifted and rotated in space as required.

The workpiece coordinate system only has to be shifted using frames and then rotated into an inclined plane.

This is precisely why we need **FRAMES**. All of the subsequent traversing commands now relate to the new workpiece coordinate system shifted using frames.

Using frames

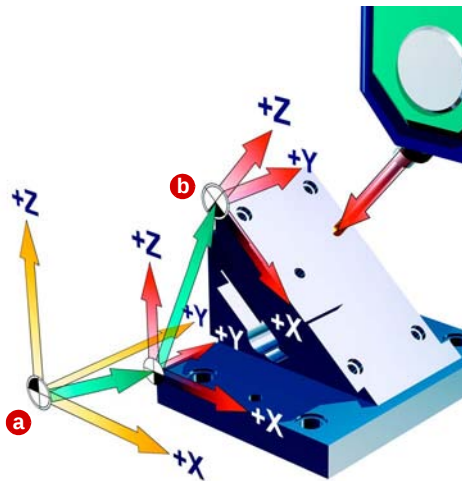


When the settable zero offset has been activated (G54, G55), the workpiece coordinate system is at the workpiece zero.

With the exception of special kinematics, the axes are now aligned parallel to the machine axes.

Using a **FRAME** this coordinate system can now be shifted and rotated anywhere in space.

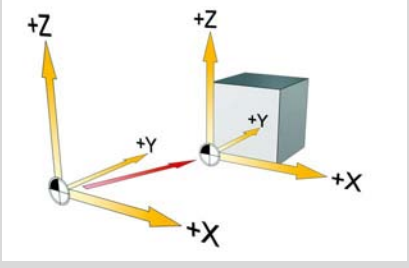
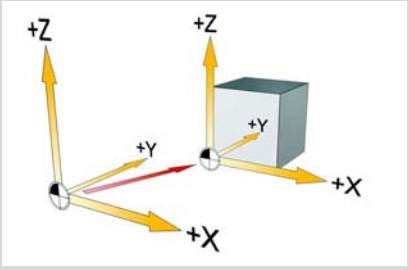
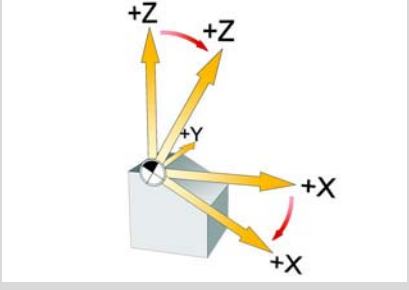
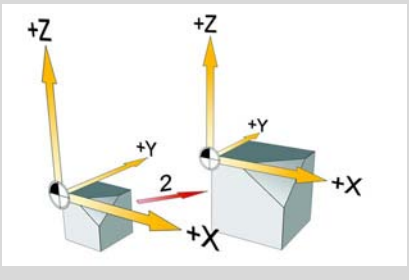
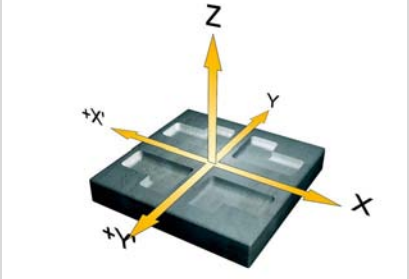
CYCLE800 must have been installed on the control (only 840D) in order to align the machine axes to the workpiece axes using graphic dialog.

Example Machining on an inclined plane

In the example, using frames the coordinate system is first shifted from **a** to **b** in two steps and is then rotated to the inclined surface.

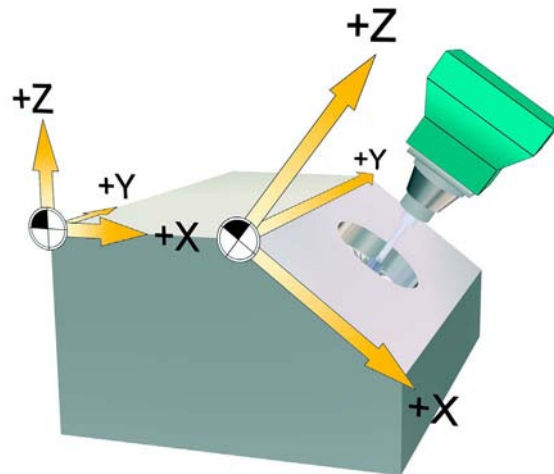
You now no longer have to take into account the inclination when programming. You program as usual, vertical to the workpiece surface and using the machining cycles - e.g. the drilling cycle.

Frames - programming components

Offset (coarse)		programmable with ■ TRANS ■ ATRANS (additive translation component) ■ CTRANS (zero offset for multiple axes) and ■ G58 (axial zero offset)
Offset (fine)		programmable with ■ C-FINE and ■ G59 (axial zero offset)
Rotation		programmable with ■ ROT / ROTS ■ AROT / AROTS and ■ CROTS
Scaling		programmable with ■ SCALE and ■ ASCALE
Mirroring		programmable with ■ MIRROR and ■ AMIRROR

3.3 Swiveling - CYCLE800

Function

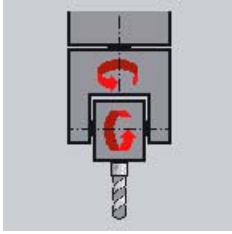
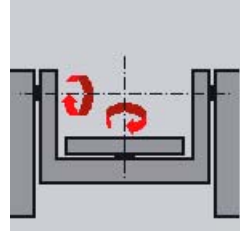
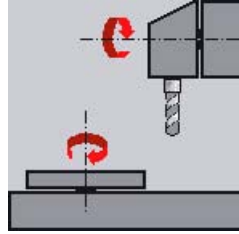


You can use swivel heads or swivel tables to set-up and machine inclined planes. Swiveling is possible in the JOG and AUTOMATIC modes. When parameterizing and/or programming swivel motion you are supported by transparent graphic displays.

You can program the swiveling axes of the machine (A,B,C) - or can simply specify the rotation around the geometry axes (X,Y,Z) of the workpiece coordinate system as described in the relevant workpiece drawing. The rotation of the workpiece coordinate system in the program is then automatically converted to a rotation of the relevant swiveling axes of the machine during machining.

The swivel axes are always rotated to place the machining plane perpendicular to the tool axis for machining. The machining plane remains fixed during machining. When the coordinate system is swiveled the previously set zero offset is automatically converted for the swiveled state.

Machine kinematics

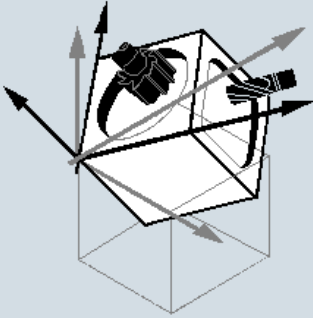
Swivel head (type T)	Swivel table (type P)	Swivel head + swivel table (type M)
Tool carrier that can be swiveled	Tool carrier that can be swiveled	Mixed kinematics
		

Procedure when programming swivel motion and subsequent machining:

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

Basic procedure when generating a swivel data set

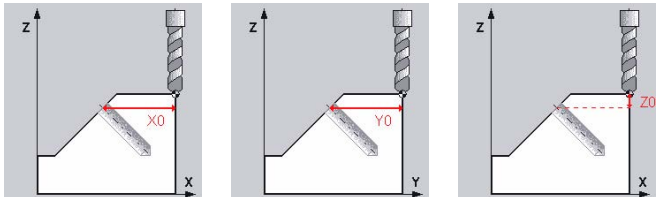
- ▶ Call the swivel function in the program.
- ▶ Select the name of the swivel data set **1**.
- ▶ Select yes for swivel if you wish to make a swivel movement. Select new as swivel movement if you wish to make a new swivel movement, or additive if you wish to base the movement on a previous swivel movement **2**.
- ▶ Specify the reference point before rotation (X0, Y0, Z0) **3**.
- ▶ Select the axis by axis swivel mode, directly, using the projection angle or the space angle **4**.
- ▶ Enter the angle through which the axis should swivel. In the axis-by-axis mode you can enter the angle for each axis **5**.
- ▶ Enter the zero point after the rotation **6**.

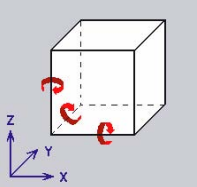
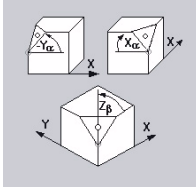
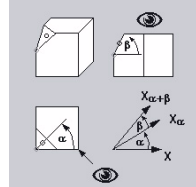
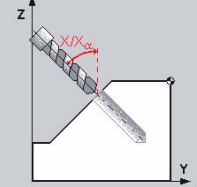
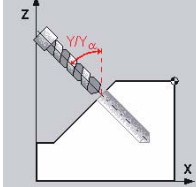
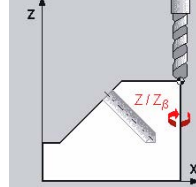
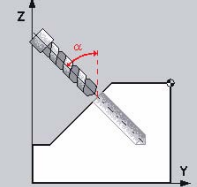
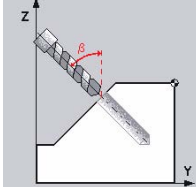
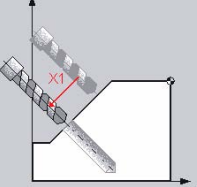
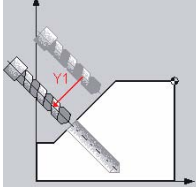
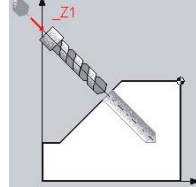
Swivel cycle/CYCLE800		Swivel rotary axes	
		Name: 1	TC_1
		Retract:	Z
		Swivel:	2 Yes
		Swivel plane:	Additive
		Ref. point:	X0 8.000
			3 Y0 0.000
			Z0 0.000
		Swivel mode: 4	axis by ax.
		Rot. around	X (A) 5 0.000
		Rot. around	Y (B) -8.000
Rot. around	Z (C) 0.000		
Zero point:	X1 0.000		
	6 Y1 0.000		
	Z1 0.000		
Direction:	Plus		
Tracking TL	No		

NOTE

You can program several swivel movements one after the other. A subsequent swivel motion can be based on a previous one (additive). This means that you can transparently represent swivel motion in the program code.

Parameters of the input screen

Name of swivel data record _TC	The swivel data records that have been set-up can be selected (toggle). Every swivel data record is assigned a name. A name does not have to be declared if there is only one swivel data record. "0" → De-select swivel data record.
Retraction _FR (prior to swiveling the rotary axis)	■ Do not retract ■ Retract Z axis ■ Retract axis Z, XY ■ Maximum retraction in tool direction (with Cycles SW 6.5 and higher) ■ Incremental retraction in tool direction (with Cycles SW 6.5 and higher) The incremental value for the traversing path in the tool direction must be entered in the input field. The retraction positions can be entered in the CYCLE800 startup menu.
Swivel, direction _DIR	■ Swivel, yes → Rotary axes are positioned or are manually rotated into position ■ Swiveling, no (only calculated) → Rotary axes are not traversed, e.g. auxiliary swivel plane according to the workpiece drawing ■ Minus/plus direction When selecting the traversing direction the reference is rotary axis 1 or 2. The NCU can calculate two possible solutions as a result of the angular range of the rotary axes of the machine kinematics. Usually, one of these solutions is technologically suitable. The rotary axis (1st or 2nd rotary axis) that is to be used as reference for the two solutions is selected in the CYCLE800 startup menu. Selecting the "minus" or "plus" direction determines which of the two possible solutions is to be applied. Observe the machine manufacturer's instructions!
Swivel plane _ST	■ New Previous frames are deleted -> defined values -> swivel frame. ■ Additive The swivel frame is based additively on the swivel frame, programmed active frames (e.g. AROT ATRANS) and the currently effective rotation in the ZO are taken into consideration.
Reference points before rotation X0, Y0, Z0	Help displays refer to machining plane G17 (tool axis Z). 

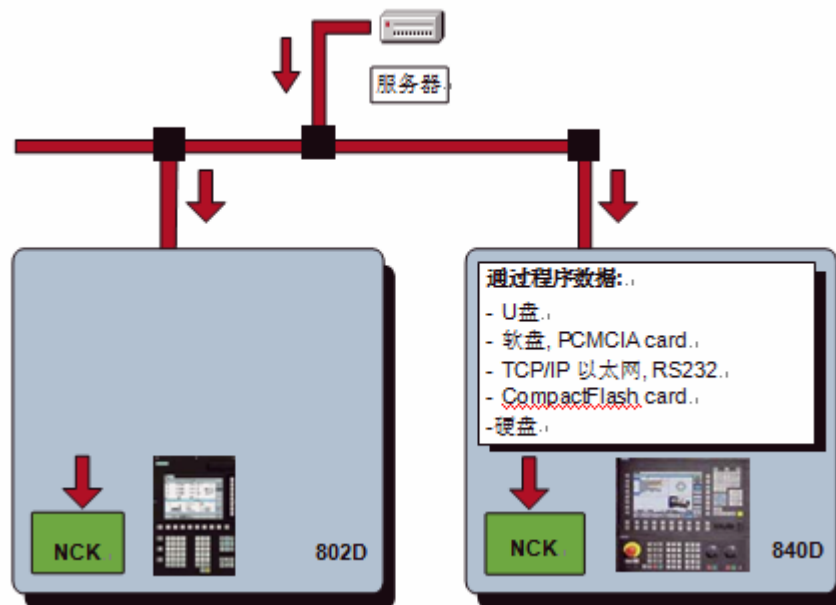
Swivel mode_MODE	   Axis by axis rotation around the individual axes of the coordinate system, Projection angle the angle of the swiveled surface is projected onto the first two axes of the coordinate system. Angle in space rotation first around the Z axis and then around the Y axis The axis sequence can be freely selected.
Rotation around A, B, C	   Rotation (axis by axis, projection angle)   Rotation (angle in space)
Zero point after rotation X1, Y1, Z 1	  
Correct tool	■ Yes When swiveling to a machining plane, the linear axes can be corrected to prevent the risk of collision. (Before-hand.:TRAORI and TOOLCARR.SPF have been adapted) ■ No Linear axes are not corrected when swiveling.

Observe the machine manufacturer's instructions. You can set the available parameters in the CYCLE800 startup menu.

2.8 程序数据的转移和管理计划

数控加工程序都被储存在控制器中，如果需要，程序将被下载到NCK工作存储器（RAM）并且在机器上执行。

模具制造程序，通常包括技术和几何程序，几何程序往往会达到100M之多，因此而不能被保存在NCK工作存储器（RAM）中，并且不能被处理，就是这个缘故，模具制造程序必须被保存在外部存储器中，这样他们就能够按顺序的被处理。



外部程序存储器（硬件配置）

根据购买的操作系统不同，操作界面和选项（人机界面），你可以使用的外部程序存储器有以下特点：

TCP/IP 以太网（网络驱动程序），串行接口RS232/V.24 (low data rate)

硬盘 (PCU 50)

Compact Flash 卡 (TCU, 802D)

USB接口(USB stick)

PCMCIA 卡 (PCU 20)

软盘

调用程序数据在外部存储器使用 EXTCALL

一个EXTCALL指令被编写在称为几何程序的主程序中，通过外部相应的网络途径存储在服务器，USB链接，硬盘等设备里。

使用EXTCALL指令时，你可以直接从外部设备中下载一个程序执行，这种情况下，所有能够出现在人机界面的程序，都可以用来下载执行。

通过EXTCALL调用几何程序

在 机器 数据 SD 42700: EXT_PROG_PATH 下, 定义 源 目录 为 几 何 程 序, 例 如 服 务 器
"\\R4711\workpieces\subprograms".默认设置是可选的.当EXTCALL信号出现该目录被先指定。

几何程序被调用时, 例如主程序中的样本. 根据控制和数据存放位置出现不同的呼叫.

PCU 50, 子程序在硬盘中

EXTCALL "sample"

PCU 20, 802D, 子程序直接存放在CompactFlash 卡里

EXTCALL "C:\sample.spf"

PCU 20, 802D, 子程序是在 CompactFlash 卡的目录里

EXTCALL "C:\programms\sample.spf"

网络被链接到以太网和机器的路径是 SD 42700

EXTCALL "sample.spf"

网络连接到以太网, 没有通过机器路径data SD 42700

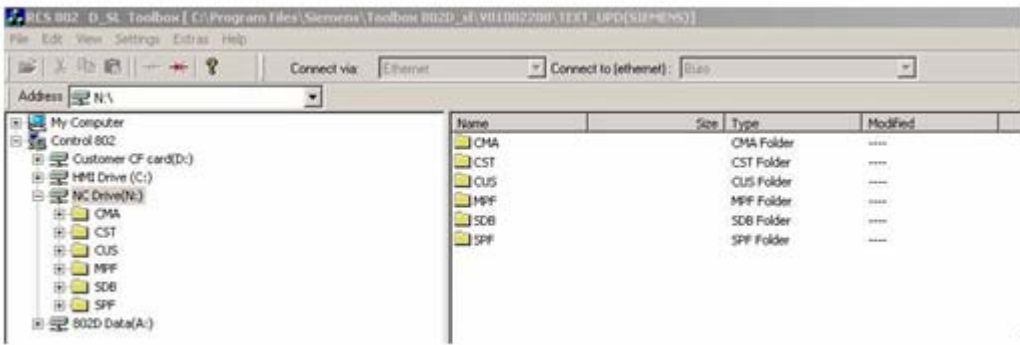
EXTCALL "\\myserver\programms\workpieces\sample.spf"

管理大型程序使用 802D sl – RCS 工具

RCS（远程控制系统）是一种帮助你在日常工作中通过PC/PG使用SINUMERIK 802D sl
的资源管理类型的工具。

控制系统和PC/PG之间的连接可以通过RS232电缆、点对点的电缆或者本地网络电缆（可选）。

当通过外部802D sl执行时请注意, 在cf卡中的程序是不能进行编辑的。当这些程序的大小超出NCK的存储空间时, 则必须从外部编辑程序（例如PC）。RCS正是为了这项任务提供帮助的工具, 在一个透明的资源管理器显示出可以复制、移动和删除程序或其他数据。

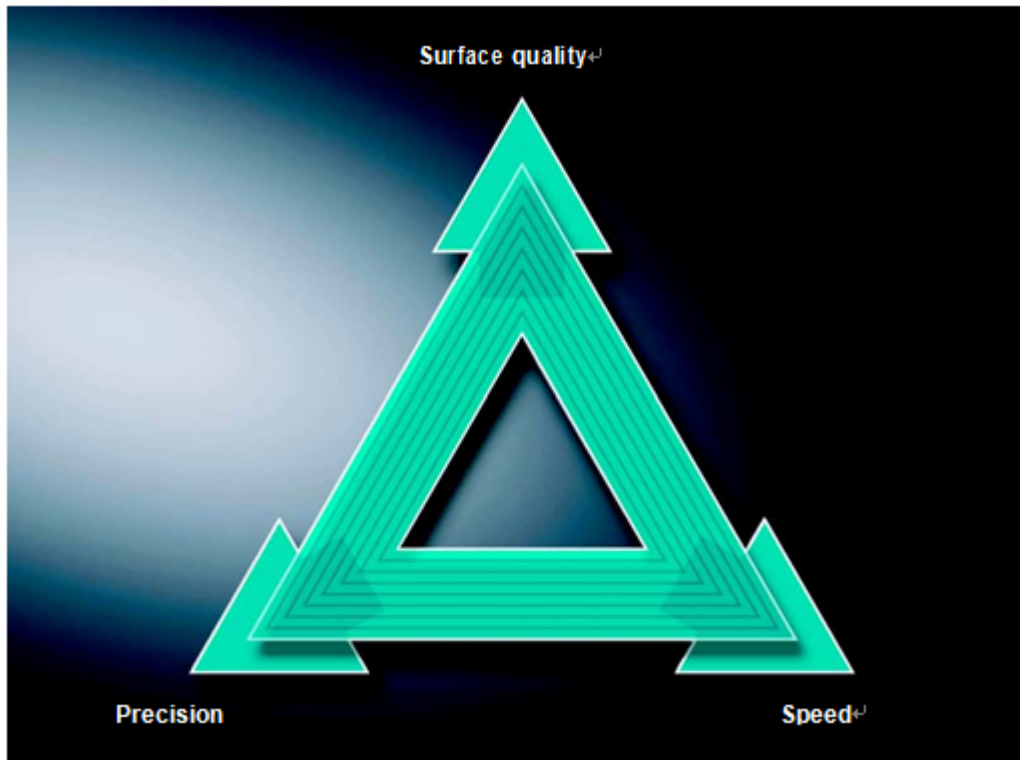


2.9告诉设置 – CYCLE832

应用

你可以使用840D中的CYCLE832指令控制CAM程序执行顺序.

它是为机器执行3轴高速加工自由曲面时提供技术支持（高速切削-HSC）CYCLE832指令结合基本编程指令和G代码为HSC提供需求。



在HSC领域中执行CAM程序，控制器必须用最短的NC模块处理高速进给.在微米级的高精度表面质量要求下，预计加工进给速度>10m/min.可以通过使用CYCLE832微调方案达到应用不同的操作方案。

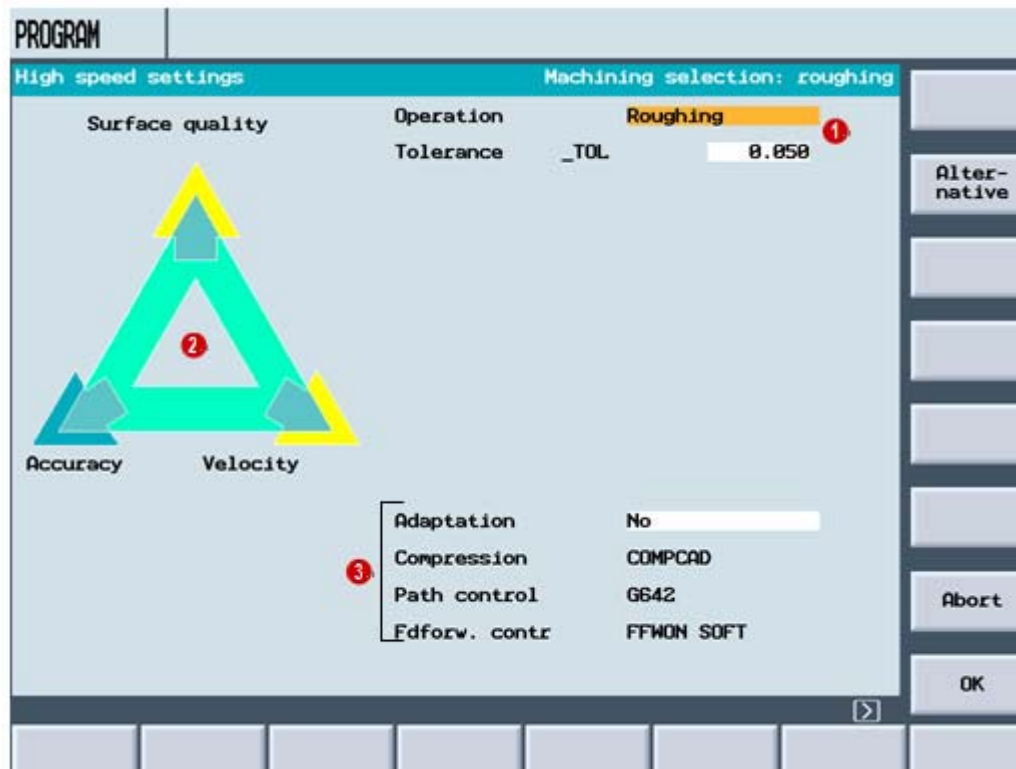
粗加工时，强调的速度是加工轮廓。

精加工时，强调的是加工精度和表面质量。

两种情况下，通过观察加工轮廓，确保达到公差要求，进而达到所需的加工精度和表面质量。一般而言粗加工的公差高于精加工。

在HMI目录中调用CYCLE832

- Programs** 打开程序操作区
- Milling** 按“铣削”键
- >> 显示其它选项
- High Speed Settings** 按“高速设定”，这个循环被调用



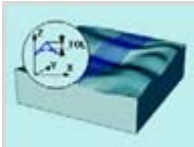
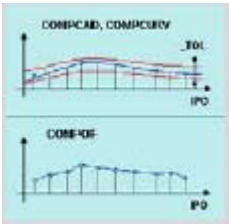
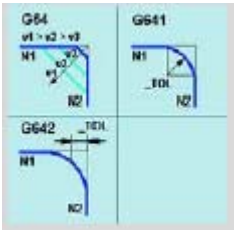
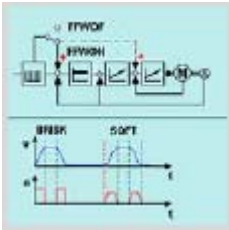
①相应的参数选择

②黄箭头可以指向“速度”、“表面质量”或者是“精度”

③其它选项，通过 OEM 和被保护的一般密码来释放

设定循环的高转速参数

在加工领域用户只需要输入一个公差范围去选择精加工、半精加工和粗加工。在其他领域中，数据已预先存入机器的OEM中。机器的OEM能够识别（密码保护）。

加工（默认）	精加工 初精加工 粗加工 取消选择	通过调用“取消选择”修改机器设置/设定数据的重置是由机器代工完成的
公差工具 	和弦公差(和弦公差应该从 CAM 系统/以 1.2。。1.5 系数加权)	公差线性旋转轴，默认设置 → 0.01 mm/ 0.08° (finishing) → 0.05 mm/ 0.4° (pre-finishing) → 0.1 mm/ 0.8° (roughing) → 0.1 mm/ 0.1° (deselection)
适应	Yes No	→ 以下字段可以被修改 → 以下字段是无形的，是由机器 OEM 发布
压缩 	COMPOF (default) COMPCAD	→ 压缩机关闭 → 压缩机开启，并对模具制造程序不断加速 → 挺举免圆周铣削 → 样条插值
继续轨迹控制 	G642 G641 G64	→ 融合了单轴公差 → 可编程混合过关 → 连续路径模式 COMPCAD, COMPCURV, G642 下的数控压缩机总是被使用
前馈控制 	FFWOF SOFT FFWON-SOFT FFWOF-BRISK	→ 不限制前馈控制和挺举 → 不限制前馈控制 → 没有前馈控制和挺举 前馈控制的选择 (FFWON) 和挺举限制 (SOFT) 要求制造商优化控制和加工轴

CYCLE832是基于G1模块的使用，G2/G3和CIP程序中公差不重要。

当你要更改OEM中指定的公差时，公差值低于指定值时是没有用的。

请注意不同领域之间的关系：例如，如果开关打开控制不同的研磨类型，可以选择连续轨迹控制。

请参阅3.5章的补充材料，在那里由各个参数的详细介绍。

编程

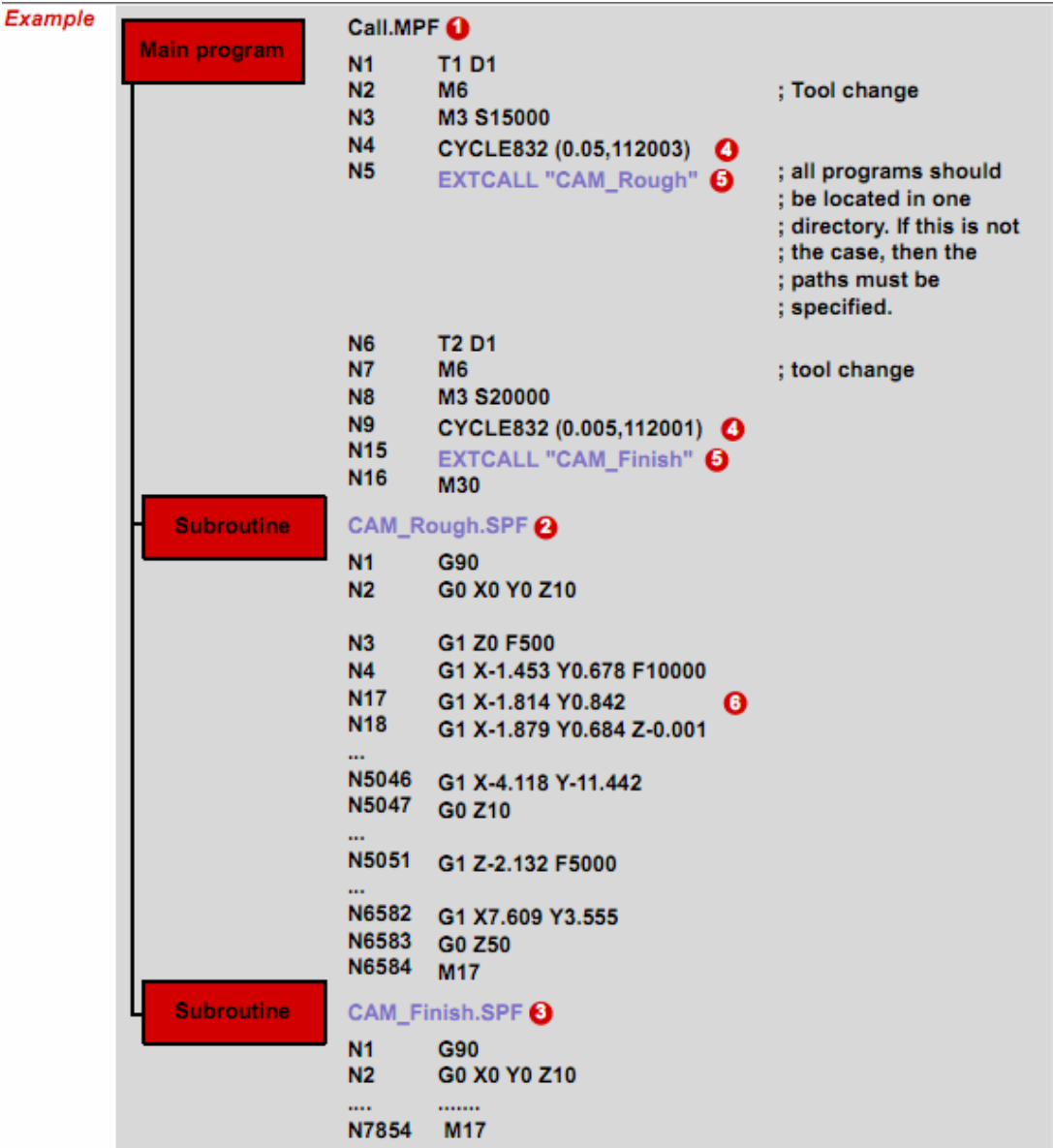
理想状态下，你的CYCLE832程序在上级NC掌控程序中，然后调用GEOM-etry。这意味着你可以申请到完整的循环，根据凸轮计划的透明度、个别课程部分或者自由曲面。

如果需要最佳方案结构，请参阅1.4章。CYCLE832请参阅2.10的资料。

2.10 模具制造的程序结构

对CYCLE832应用于实用程序结构的建议

- ①用于生成的主程序，包括所有的主程序资料
- ②主程序调用一个或多个子程序
- ③包含工件的几何数据，换刀程序定义在该组

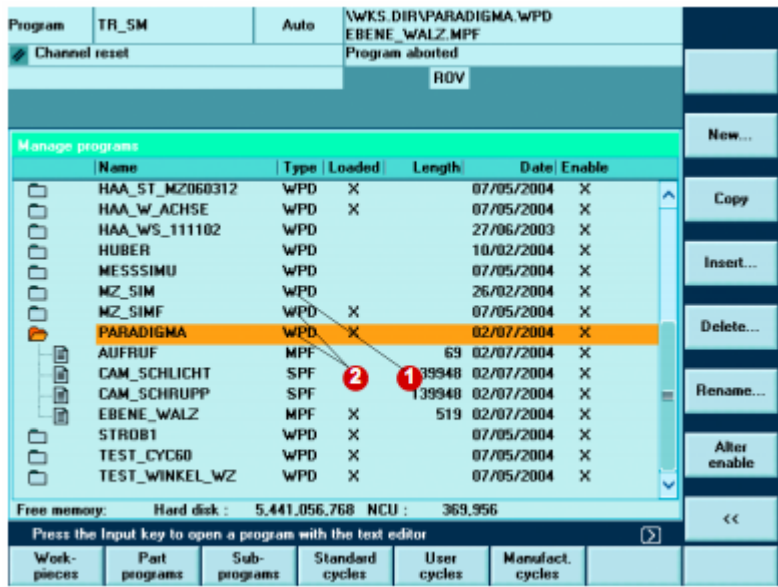


CYCLE832 : CYCLE832是专门为显示在科技开发和几何数据是分离的程序结构中，铣削加工技术被定义在CYCLE832中，对于粗加工程序“CAM—rough”使用T1，在CYCLE832参数设置中实现高速增长。为了实现高精度和良好的表面质量精加工程序“CAM-finish”的参数设置。

CAM程序一般都比较大，这就是为什么他们总是被存储在外部存储器中。EXTCALL从外部存储器中调用子程序.

子程序: 子程序G90相对于几何程序是绝对的，这些在我们的例子中为3轴铣削。

2.11选择/开始/停止/中断/继续程序



Machine 选择机器操作区

AUTO 选择自动模式

Program overview 按高速设定，周期被调用

Workpiece

Selection 在工件目录中的突出方案①在此情况下，程序"Call.MPF" ("Aufruf.MPF") 然后按选择 ("Anwahl").



按“数控启动”来启动零件程序. 这就要求几何程序 "Roughing.SPF" ("Schrupp.SPF") ② 和 "Finishing.SPF" ("Schlicht .SPF"),从机器中的外部存储器实现将块加载进控制系统



按 "NC-Stop" 以停止程序



按 "Reset"去中断程序

程序运行中“NC-stop”可以中断程序，按“NC-start”可以继续程序。程序执行中按下“Reset”程序将被中断，按“NC-start”程序将从头开始，或者是在配合搜索的条件下跳转到中断处继续程序。

2.12 中断程序

REPOS – 中断后重新定位

功能

当一个程序中断或执行“NC-stop”后，该工具可以避免进入JOG模式下。如：检查工具的最前沿。
保存中断控制点坐标。轴的差动行程显示。

操作



最初的情况：通过“NC-Stop”实现程序的中断

Machine 选择机器操作区



选择 "JOG" 模式



程序中断后冲洗定位



选择轴.



根据差动行程显示移动轴的中断点.这是不可能实现的断点

AUTO 从“点动”转换成“自动”模式



继续加工

没有计算的前提下加速外部搜索

功能

SINUMERIK 840D 系统的功能是专门开发被称作EXTCALL的程序，这意味着它可以从CAM中完美的调用大型程序.

使用“Reset”是加工中断后，使用“没有计算的前提下加速外部搜索”可以在任何程序的任何位置选择启动或继续加工.

操作



最初情况：程序因为“Reset”被中断

Example

```
Call.MPF ①
N1      G54
N2      T1 D1
N3      M3 S15000
N4      CYCLE832 (0.05,112003)
N5      EXTCALL "CAM_Rough"
N6      T2 D1
N7      M3 S20000
N8      CYCLE832 (0.005,112001)
N16     EXTCALL "CAM_Finish" ③
N10     M30

CAM_Roughing.SPF ①
N1      G90
N2      G0 X0 Y0 Z10

N3      G1 Z0 F500
N4      G1 X-1.453 Y0.678 F10000
N17     G1 X-1.814 Y0.842
N18     G1 X-1.879 Y0.684 Z-0.001

CAM_Finishing.SPF
N1      G90
```

Block search 按 "Block search"

Search pointer 按 "Search pointer"

Break point 按 "Breakpoint"

按断点软键将在屏幕上的整个程序的断点处插入序列。
在这个例子中，主程序 "Call.MPF" ("Aufruf.MPF") 调用子程序 "CAM_Roughing.SPF"

("CAM_Schrupp.SPF"). The EXTCALL 是位于"CAM_Schrupp.SPF"中的编号为N16的子程序。
"CAM_Schrupp.SPF"是在该装程序中被中断的



现在有两种可能性:

1. 直接跳转到子程序中的断点处: 按"External without calc."键, 该程序直接跳转到3004。
2. 做到这一点你必须选择一个搜索类型, 在搜索外部程序时总是选择搜索类型的字符串, 然后输入型号和邻近他的搜索文本, 例如: 块和搜索型号。

external without calc. 按 "External without calc." 键



继续在目标块加工

Overstore 更正当时用一个周期停止条目去实现“覆盖式”功能被选择时，这样就可以在程序开始前修正目标块



这里有一个典型的例子，压缩机的兼容性需要后面的改变. 调用CYCLE832 可以实现这些 并且压缩机的兼容性是通过手动改为 20μm①仅仅通过加入一个参数 (公差=0.02) 在主程序执行前执行 CYCLE832



公差通过数控启动被激活

2.13计划概述/外部程序状态

功能

当从外部执行程序是，你可以显示当前状态和程序运行时间.

人机界面显示为840D的标准状态

Program overview 在自动模式下，选择软件计划概述，方案概述显示

External 选择外部程序软键

[illegible]

The current status of the external program is displayed as a percentage in the overview that is displayed.

Machine	848D_Mill	Auto	WKS.DIR\PLATE_BLOCK.WPD			
			PLATE_BLOCK.MPF			
Channel reset			Program aborted			
			ROV SBL1			
External programs						
Name	Type	Length	NC name	Channel	SP	Status
\\wks\plate_block\plate_block	MPF	23982	NCU0480	1	0	

Programs

2.14模拟零件程序

功能

利用模拟功能，你可以获取个人加工步骤的概述，你可以检查工件的程序。

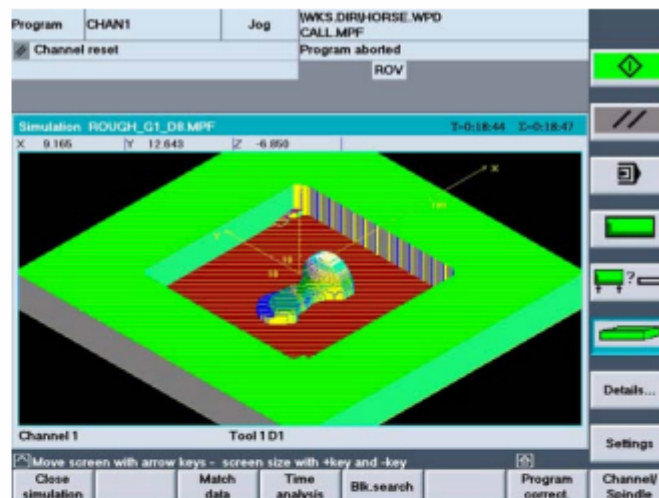
选择仿真功能:

选择在程序编辑器的仿真软键。

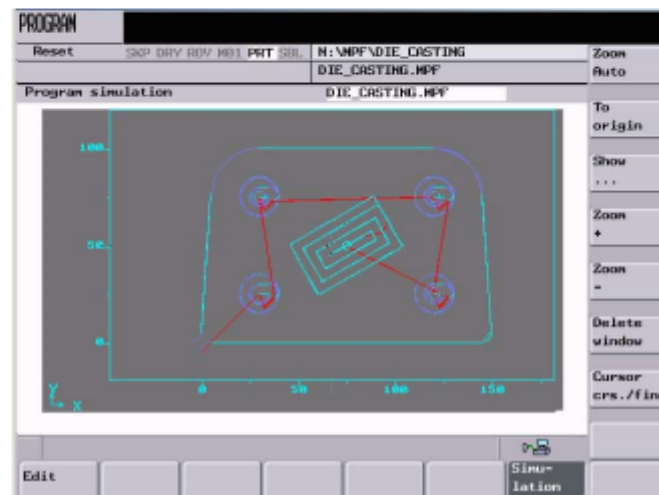
仿真功能取决于控制:

- „ 模拟显示在3坐标面内（840D系统）或加工平面（802D）
- „ 车削，缩放和放大/缩小
- „ 3-D模型在任何时候和任何部分

Example, simulation 840 D



Example, simulation 802D sl

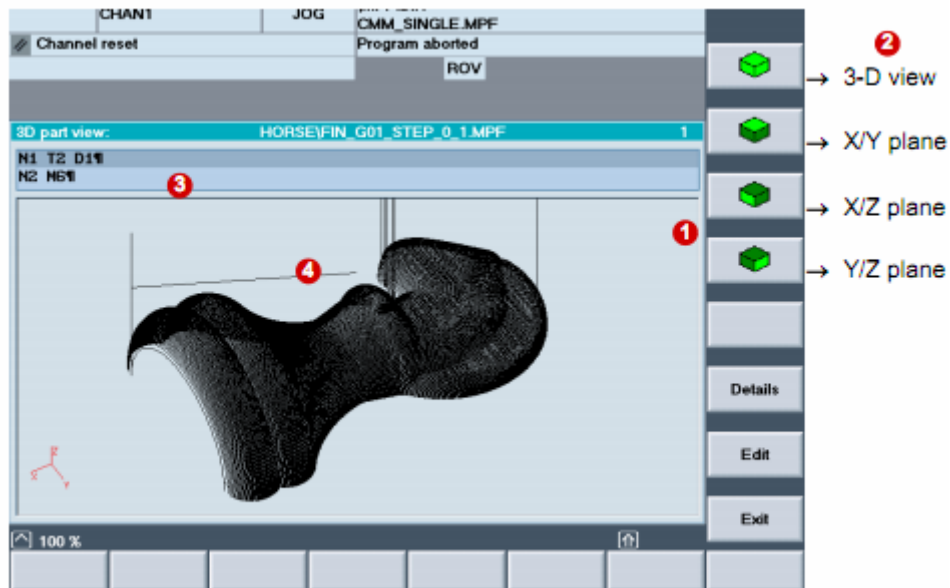


2.15快速浏览/快速显示

功能

SINUMERIK 840D 的快速查看是PCU50模具制造方案的一部分包含可视化G01.程序循环、多项式、转换和G02/G03不支持

4点①3-D 视图②X/Y 面, X/Z 面, Y/Z 面③编辑器中的两行显示④当前图形中突出部分。 通过编辑器窗口的滚动自动显示在图形中的位置



也可以实现以下功能

- „ 搜索特定快
- „ "放大缩小"
- „ 移动, 旋转
- „ 测量两点的距离
- „ 在显示中编辑NC

你可以使用标准接口和shopmill的D840快速浏览, 在程序管理器中对840D的标准快速查看, 为了shopmill, 你可以打开程序编辑器快速查看

操作

Quickview “快速查看”功能.

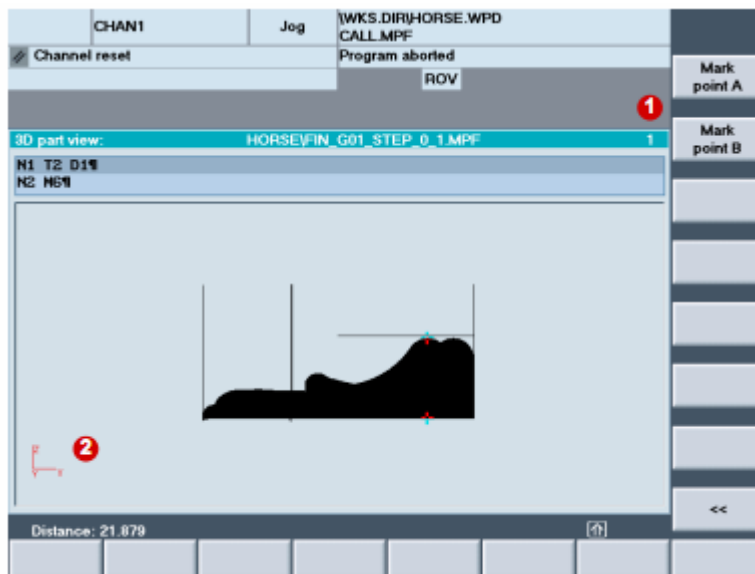


选择你需要的视图，在x/z平面内



使用光标在图形中选择一个点，相关的模块显示在编辑器的行列中

调用块，例如：在程序中改变它



快速查看测量距离

在快速查看中你还可以测量两个点之间的距离，这个功能是非常有用的，例如：你想知道一个工件的尺寸，应为从一个程序的G1模块下的工作面是很难评估的。

通过突出显示两个点：1你可以得到在页脚区的两个，2点的距离。

2.16ShopMill

由于SINUMERIK 840D ShopMill拥有良好的用户界面，而真正的替代了SINUMERIK 840D标准的DIO/ISO界面。

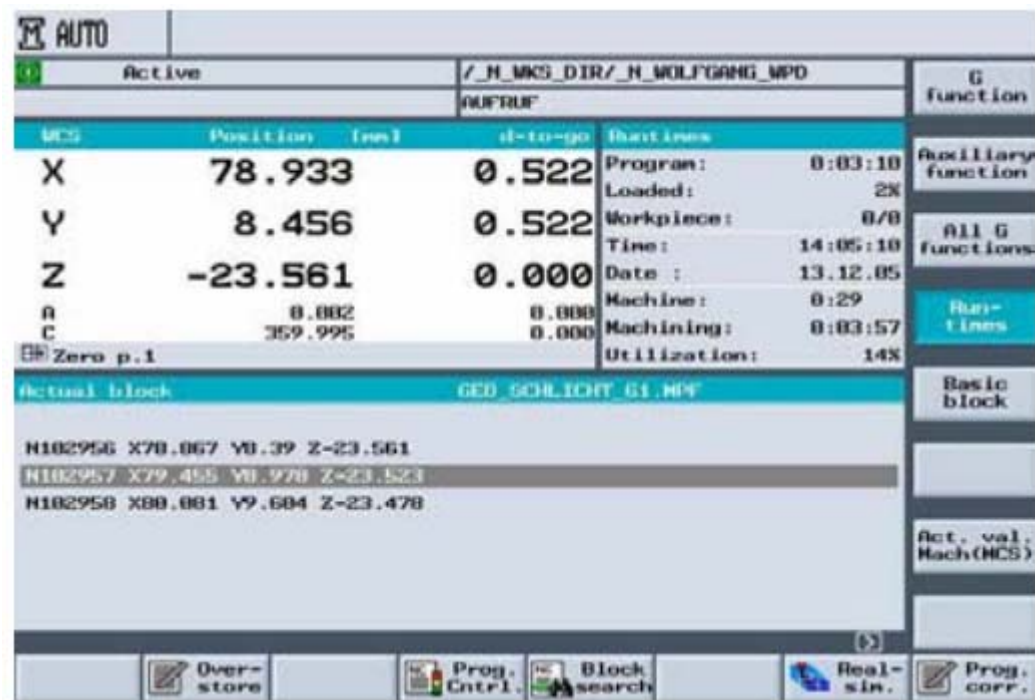
ShopMill已经被许多模具制造功能补充，大大的方便了模具制造商的使用。

因此, ShopMill不再被使用于偏程序加工步骤中，而实际上，它甚至可以在3+2轴或5轴机床上支持应用。

在ShopMill功能的完整描述中可以被证实“SINUMERIK 810D/840D使用ShopMill编程”。

ShopMill的用户界面

在车间中的简单操作和编程



ShopMill的功能

安装

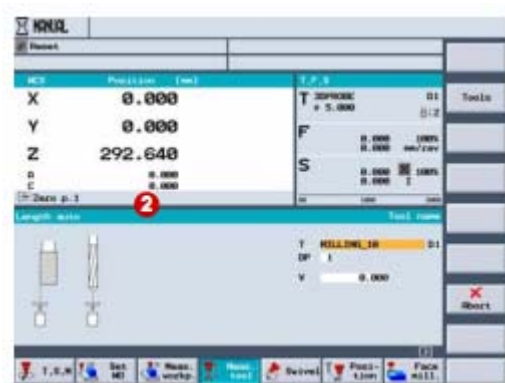
在ShopMill中强大的装置功能，保证了对工件快速准确的位置检测，特殊的测量循环简化了工件和测量工具，任何偏移量通过控制系统得到补偿。



①测量工件的边缘，角/孔，矩形，平面



测量功能在JOG测量时被使用，测量循环支持自动测量模式



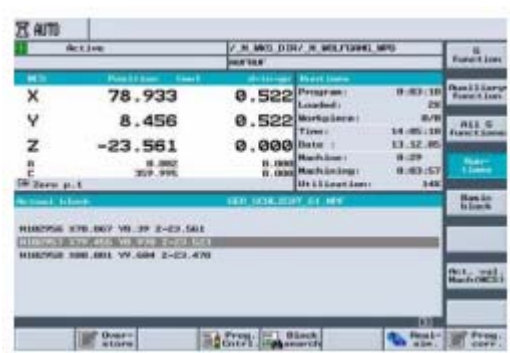
②测量工具，测量功能在JOG下测量工具和自动测量时使用



刀具管理

ShopMill 刀具管理结构是清晰的，并且支持多种类型的工具，在纯文本中的工具名称，女儿工具和刀具长度与几何形状，半径和切割边数。通过输入单位数量，寿命或磨损参数，你可以自动的检测工具的使用时间，确保统一的加工质量。

自动



在屏幕上基本自动显示运行时间

时序编程

ShopMill的时序编程允许直接在机器上的简单的程序像21/2D的基本任务，为了模具制造商，这是一个理想的补充

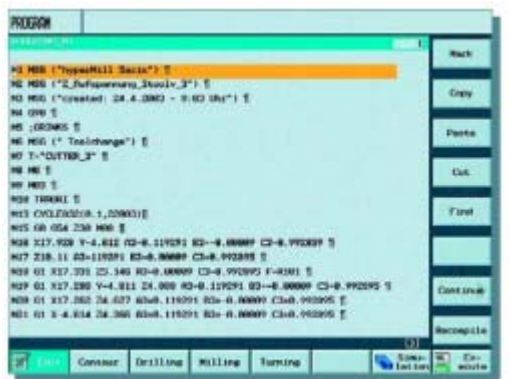


①Program

②2D display

③ 3D display

G代码编辑器



ShopMill 带有一个集成的并且功能强大的G代码编辑器，

在良好的状态下，它支持模具的制造程序。

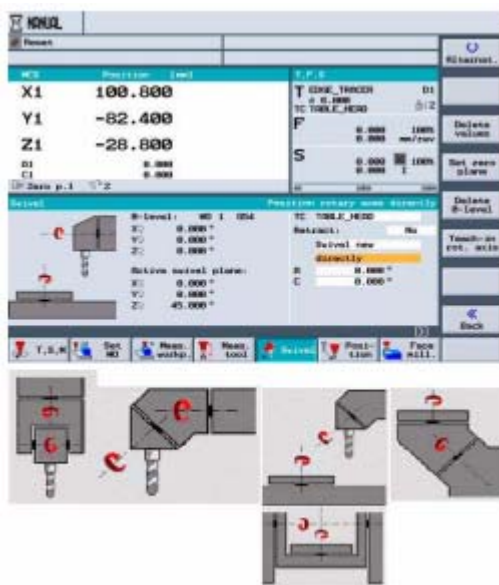
搜索模块

外部程序中的快速块搜索（计算机的有无）



ShopMill 周期雕刻

有特殊字符的文本；日期，时间，工件计数，变量
在JOG下的转体



旋转周期对所有旋转任务在安装模式下使用的软键；输入是直接的或通过轴与轴之间；支持所有的机器运动
周期的“高速设定”

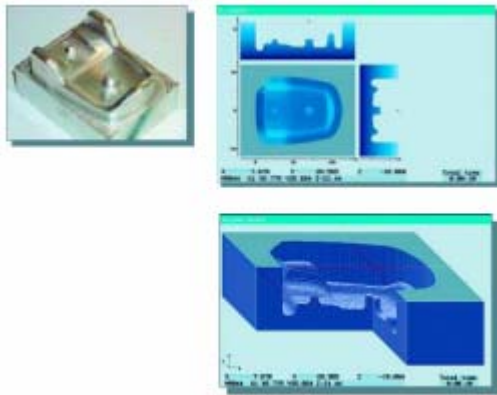


周期的高速设定已成为Shopmill用户界面在G代码编辑器的组成部分；
程序编辑器

CYCLE832, 高速设置

仿真

ShopMill的加工路径提供了各种丰富的展示和详细的仿真。为了模拟加工过程，控制系统完全计算出当前选择的程序，并以图形形式显示结果，你可以选择以下代表类型进行仿真：



1俯视图；23D视图；3立体视图

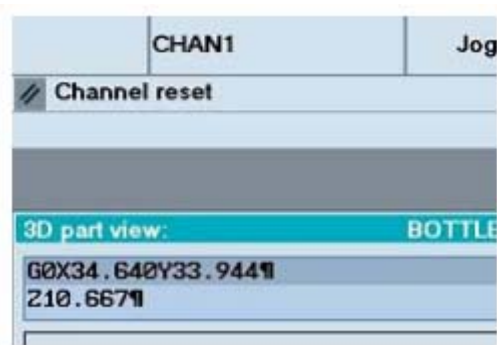
模拟计划



启动/停止/单块/重置模拟键；速度可以通过override功能控制

模具制造的快速查看、快速显示

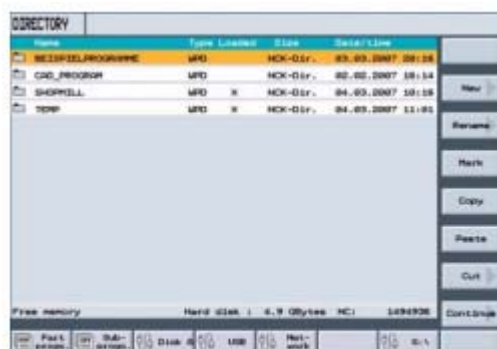
遍历路径的快速显示在大部分程序中是可行的。从G1中通过这个虚线的快速查看和显示，轴的路径被显示。



外部驱动器

ShopMill程序允许通过以太网直接访问外部驱动器，大量的模具制造程序可以被保存。

- 1、HMI硬盘 (PCU 50)；
- 2、Flash card (PCU 20)；
- 3、Network drives；
- 4、USB stick



第三章 信息程序员

目录	页数
3.1介绍	62
3.2什么是框架	63
3.3cycle800回转	66
3.4编程实例：回转	70
3.5高速设置cycle832	72
3.6进给速度剖面	80
3.7 cycle832下的编程实例	81
3.8无cycle832得编程实例	84

3.1介绍

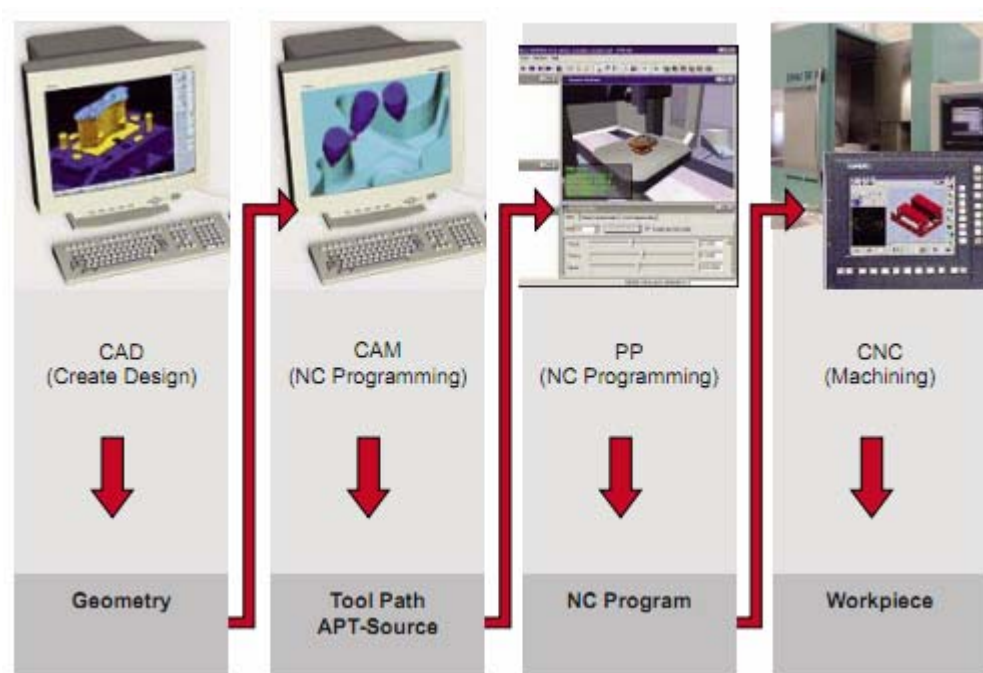
整个CAD/CAM/CNC加工链，在自由曲面的编程时是至关重要的。

CAD系统生成所需几何形状的工件,基于CAM系统这种对应相关技术信息的加工策略。

来自CAM系统的数据输出通常是一个数据文件中的APT或CL. 这是在后处理器中转换的可执行的数控代码。

上游后处理器是有充分利用能力的和把SINUMERIK性能控制到最大的最重要部分。

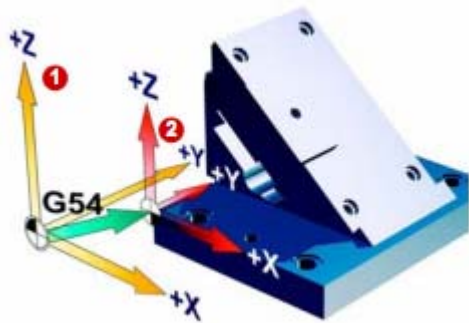
后处理器必须确保SINUMERIK的高阶函数控制在本章描述中,以最好的方式启动. 在本章可以找到一个较高阶SINUMERIK840D系统的所有功能。



3.2什么是框架？

坐标系

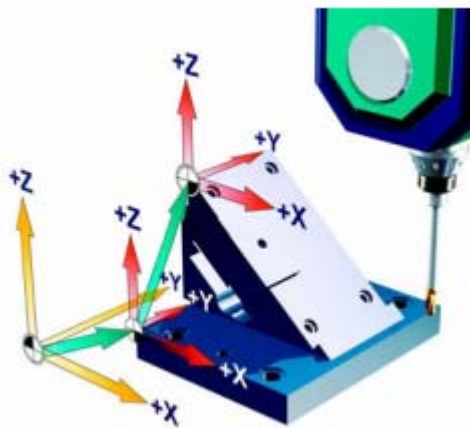
有参考点的机床坐标①系统和零偏移(G54, G55, ...) 是已知的。使用框架坐标系可以移动、旋转、镜像和缩放让他们的工件表面对其，这样可以使编程时间和成本减少到最低。通过使用框架，实际工件坐标系的目标是通过指定位置坐标定义坐标系和角度②



1、基本框架（基本偏移）(G500); 2、可设定框架(G54, G55...); 3、可编辑的框架 (TRANS, ROT...)

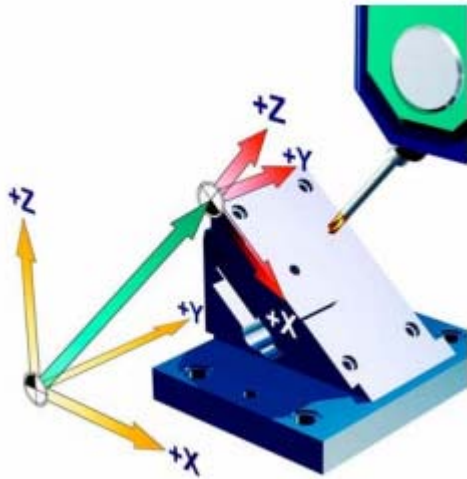
坐标系统和遍历议案

使用3+2轴机床，他可以实现空间中转移和旋转的加工曲面要求，使用框架可以使工件坐标系转移，然后旋转成一个斜面，这就是为什么我们需要框架，随后的所有便利命令现在涉及到新的工件坐标系转向使用框架。



框架的使用

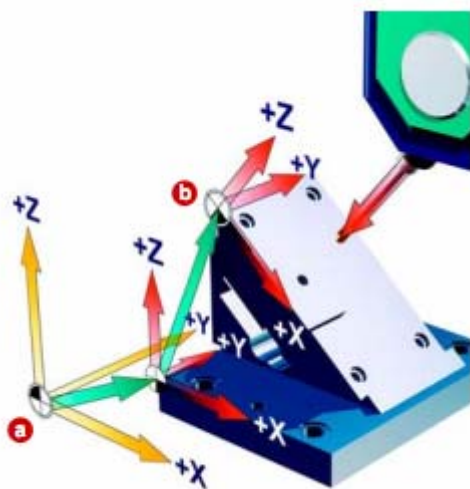
当可设置零点偏执已被激活 (G54, G55), 工件坐标系位于工件零点, 除去特殊运动, 和特别运动学, 现在的轴平行于轴机, 使用框架这个坐标系, 可以在空间实现旋转和移动, 为了使机轴工件轴使用图形对话, CYCLE800必须安装在控制器中 (仅限840D系统)。



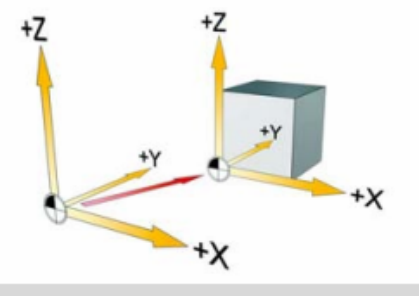
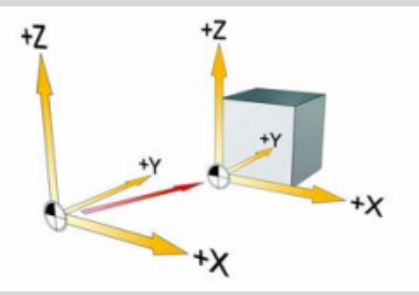
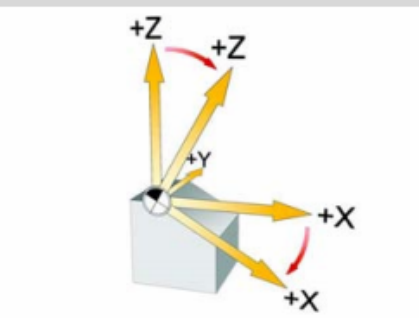
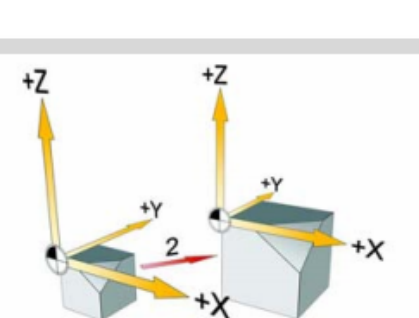
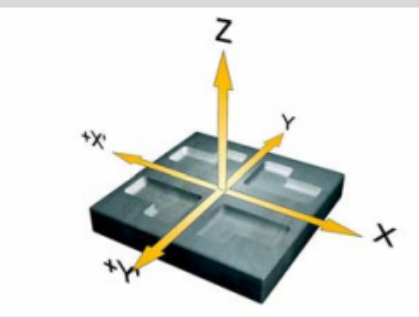
斜面上加工

在这个例子中, 使用框架系统坐标有2步, 第一从A移动到B, 然后使面旋转倾斜;

现在你再也不用考虑编程时间的因素, 你的程序像往常一样, 垂直于工件表面, 利用加工周期, 例如: 钻井周期。

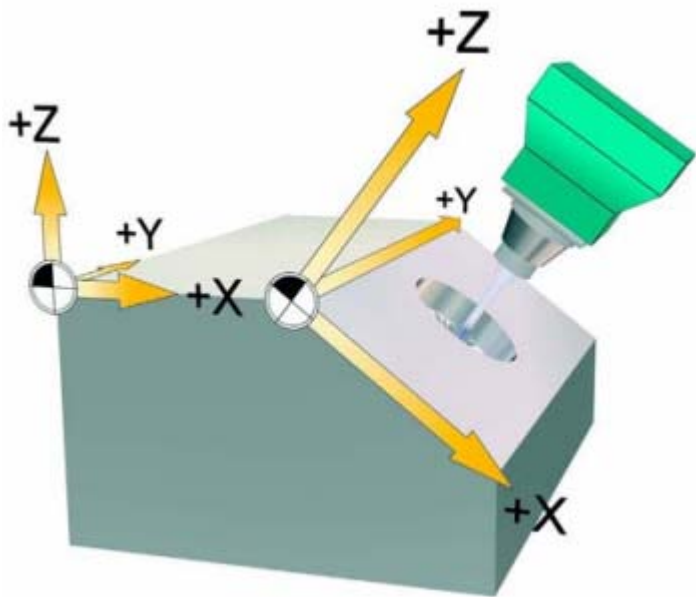


框架——编程组件

Offset (coarse)		programmable with ■ TRANS ■ ATRANS (additive translation component) ■ CTRANS (zero offset for multiple axes) and ■ G58 (axial zero offset)
Offset (fine)		programmable with ■ C-FINE and ■ G59 (axial zero offset)
Rotation		programmable with ■ ROT / ROTS ■ AROT / AROTS and ■ CROTS
Scaling		programmable with ■ SCALE and ■ ASCALE
Mirroring		programmable with ■ MIRROR and ■ AMIRROR

3.3 回转 - CYCLE800

功能



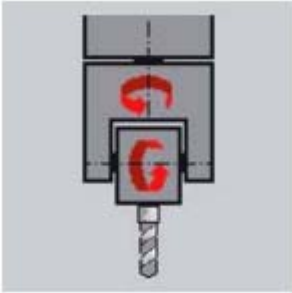
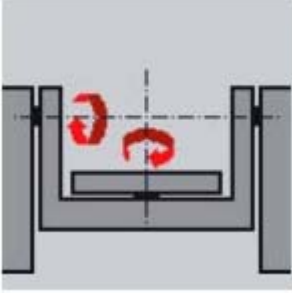
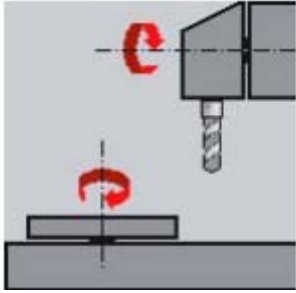
你可以使用旋转头或旋转面来设置机械斜面. JOG和AUTOMATIC模式下回转是可以的. 可通过透明图形显示支持旋转运动的参数和编程。

在有关图纸中，你可以对机器的旋转轴（ABC）进行编辑- 也可以直接定义工件周围的几何轴（XYZ）的旋转坐标系中. 在程序中的旋转工件坐标系统是自动转换到加工过程中的有关旋转轴转动的机器。

旋转轴的旋转加工平面总是垂直于刀具轴. 该加工面在加工中保持不变。

旋转状态自动转换为零偏执在旋转坐标设定前。

机器运动学

旋转头（T型）	旋转面（P型）	旋转头和旋转面（M型）
可旋转刀架	可旋转刀架	混合运动
		

编程后的程序旋转运动和后续加工:

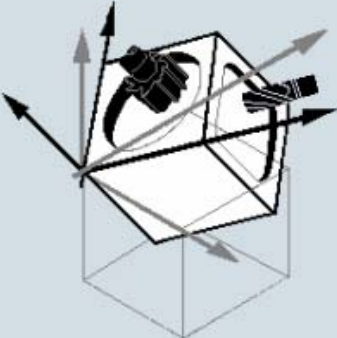
- X 被加工的旋转平面坐标系.
- X 加工程序像依旧在x/y面.
- X 旋转坐标系恢复到原来位置.

基本程序产生一个旋转数据集

- X 在程序中调用旋转功能.
- X 选择旋转数据集的名称。1
- X 如果你想做出一个旋转运动，选择旋转。如果你想做出新的旋转运动，将作为旋转运动的新选择，如果你想在之前旋转运动的基础上运动，或者添加。2
- X 转动前指定参考点（x0y0z0）。3
- X 通过旋转轴选择轴模式，可以直接利用旋转角度或投影角。4
- X 通过旋转轴线输入角度.在轴轴模式下，你可以输入每个轴的夹角。5
- X 转动后输入零点。6

Swivel cycle/CYCLE800

Swivel rotary axes



Name: 1

TC_1

Retract:

Z

Swivel:

2 Yes

Swivel plane:

Additive

Ref. point:

3 X0

8.000

Y0

0.000

Z0

0.000

Swivel mode:

4 axis by ax.

Rot. around

X (A)

5 0.000

Rot. around

Y (B)

-8.000

Rot. around

Z (C)

0.000

Zero point:

6 X1

0.000

Y1

0.000

Z1

0.000

Direction:

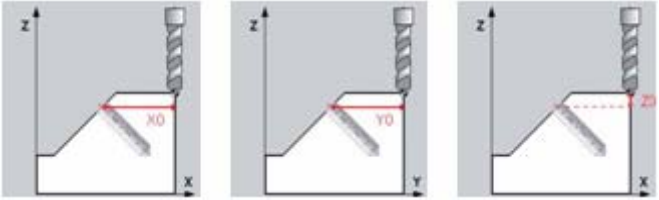
Plus

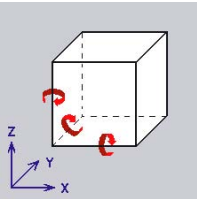
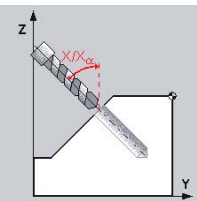
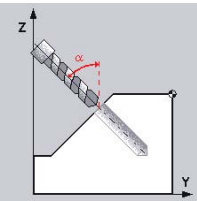
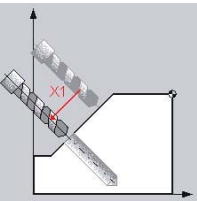
Tracking TL

No

你可以一个个编辑旋转运动 后一个旋转运动可以依据前一个. 这意味着你可以清晰的将旋转运动反映在程序代码中。

参数的输入界面

旋转式数据记录-TC	旋转式数据可以选则以设置的记录（切换）。 每一个旋转数据被记录一个名字.如果只有一个旋转数据，那么不被记录. "0" → 取消旋转记录.
回缩（前旋转的旋转轴）	■ 不要回收 ■ 回缩Z轴 ■ 回缩轴Z，XY面 ■ 在工具的最大收缩方向（Cycles SW 6.5或更高版本） ■ 在刀具方向的增量收缩 (Cycles SW 6.5或更高版本) 穿过刀具方向的增量值必须输入在输入栏. CYCLE800中的菜单可以进入回缩位置。
旋转，方向 _DIR	■ 选择旋转 → 旋转轴的位置或手动旋转到位 ■ 不旋转 (只计算) → 不经过旋转轴，例如根据辅助平面旋转工件图纸 ■ 正/负方向 选择运动方向是参考1或2个旋转轴。NCU可以计算作为机床运动的旋转轴角度范围导致的两种可能的解决方案. 通常，技术是解决方案之一. 在CYCLE800启动菜单中的旋转轴作为第二种方案选择。选择“加”或“减”的方向决定了哪种方案被应用. 注意观看机器制造商的说明！
旋转平面_ST	■ 新的 前一框架被删除>定义的值>旋转架. ■ 添加 该框架是基于旋转相加的旋转架，活动框架的编程（例如：AROT ATRANS） 和目前考虑有效转动.
设X0Y0Z0为旋转前的参考点	帮助显示是指加工平面G17号（工具轴Z）。 

旋转模式_MODE	 <table><tr><th>轴与轴</th><th>投影角</th><th>空间角度</th></tr><tr><td>围绕个别坐标轴 旋转</td><td>旋转表面的角度 投影到前两个 坐标轴的轴心</td><td>首先绕Z轴然后是 绕 Y轴</td></tr></table> 轴的循序可以自由选择。	轴与轴	投影角	空间角度	围绕个别坐标轴 旋转	旋转表面的角度 投影到前两个 坐标轴的轴心	首先绕Z轴然后是 绕 Y轴
轴与轴	投影角	空间角度					
围绕个别坐标轴 旋转	旋转表面的角度 投影到前两个 坐标轴的轴心	首先绕Z轴然后是 绕 Y轴					
围绕A, B, C旋转	 旋转 (轴与轴, 投影角)  旋转(在空间角)						
零点在 X1, Y1, Z 1旋转后							
正确的工具	■ 对的 当旋转到加工平面, 为防止发生碰撞, 线性轴被纠正. (事前.:TRAORI 和TOOLCARR.SPF 已被调整) ■ 错的 旋转时线性轴不被纠正。						

观察机床制造商的指示, 可以再CYCLE800启动菜单中设置可用参数。