SIEMENS

Application description • 02/2016

Interpolation turning with Cycle959

SINUMERIK 828D/840D sl, SW 4.5 SP2

Unrestricted

https://support.industry.siemens.com/cs/ww/en/view/109480429

Warranty and liability

Note

The application examples are not binding and do not claim to be complete regarding the circuits shown, equipment and any other eventuality. The application examples do not represent customer-specific solutions. They are only intended to provide support for typical applications. You are responsible for ensuring that the described products are used correctly. These application examples do not relieve you of the responsibility to use safe practices in application, installation, operation, and maintenance. When using these application examples, you recognize that we cannot be made liable for any damage/claims beyond the liability clause described. We reserve the right to make changes to these application examples at any time without prior notice.
If there are any deviations between the recommendations provided in these application examples and other Siemens publications − e.g. Catalogs − the contents of the other documents have priority.

We do not accept any liability for the information contained in this document.

Any claims against us – based on whatever legal reason – resulting from the use of the examples, information, programs, engineering and performance data, etc. described in this application example, shall be excluded. Such an exclusion shall not apply in the case of mandatory liability, e.g. under the German Product Liability Act ("Produkthaftungsgesetz"), in case of intent, gross negligence or injury of life, body or health, guarantee for the quality of a product, fraudulent concealment of a deficiency or breach of a condition which goes to the root of the contract ("wesentliche Vertragspflichten"). The damages for a breach of a substantial contractual obligation are, however, limited to the foreseeable damage, typical for the type of contract, except in the event of intent or gross negligence or injury to life, body, or health. The above provisions do not imply a change in the burden of proof to your detriment.

Security information

Siemens provides products and solutions with industrial security functions that support the secure operation of plants, solutions, machines, equipment, and/or networks. They are important components in a holistic industrial security concept. With this in mind, Siemens' products and solutions undergo continuous development. Siemens recommends strongly that you regularly check for product updates.

For the secure operation of Siemens products and solutions, it is necessary to take suitable preventive action (e.g. cell protection concept) and integrate each component into a holistic, state-of-the-art industrial security concept. Third-party products that may be in use should also be considered. For more information about industrial security, visit http://www.siemens.com/industrialsecurity.

To stay informed about product updates as they occur, sign up for a product-specific newsletter. For more information, visit http://support.automation.siemens.com.

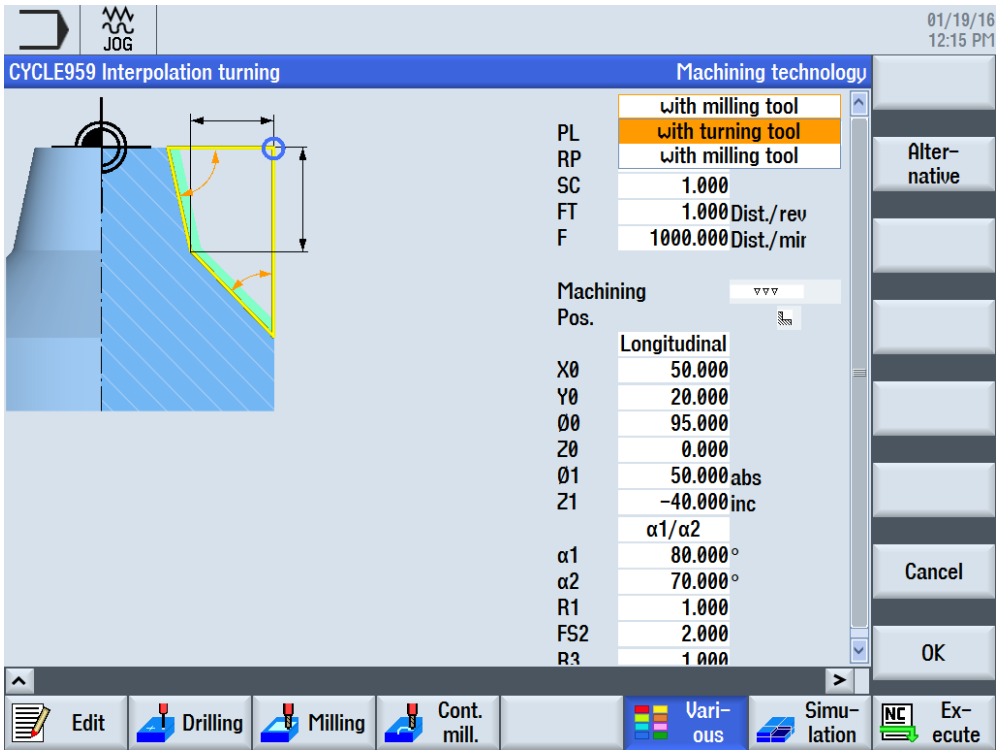
Table of contents

1 Interpolation turning

1.1 Turning on milling machines

The requirements placed on machine tools on a daily basis are continually rising - and as a consequence they must become increasingly more powerful and versatile. For instance, today, state-of-the-art machine tools are in a position to turn parts on milling machines without rotary axes. The machining technique employed is interpolation turning. The precondition to achieve this is that the main spindle of the machine can be switched over into position-controlled operation. This means that the main spindle machine can be used as turning axis with full functional scope.
The tool clamped in the main spindle is used for machining, which carries out circular motion on the X/Y plane. The workpiece (semi-finished product) to be machined is fixed to the machine table. Each individual circular tool motion in the spindle is realized around its own axis. As a consequence, the tool in the spindle is continuously aligned to the component in order that the cutting edge is brought into the optimum cutting position. The quality of the machined workpiece is comparable with the machining results of workpieces machined on conventional lathes. As a consequence, workpieces that are very complex to produce can be quickly and completely machined on just one machine.
The technical effort required and the time involved remain justifiable as additional equipping times are reduced or even eliminated. Technology cycle Cycle959 is used to generate the part program required. This is based on Run MyScreen - and supports the operator in the form of animated graphics in the part program generation for the interpolation turning.

Fig. 1-1 Selecting the technology parameters in Cycle959



In addition, the machine table can be swiveled at any location (Cycle800), and technology cycle Cycle959 used in the "new" plane. In the actual example, a cube is used as semi-finished product on a mill-turning machine with A/C kinematics. Using Cycle 959, the semi-finished product should be turned on a milling machine.

Note For roughing processes, it must be taken into account that not every milling machine is designed to handle the forces that occur when turning.

1.2 Supplementary conditions

The application software provided comprises a technology cycle and a user interface based on Run MyScreen. The software is suitable for Sinumerik 840D sl/ 828D from software release 4.5 SP2. The user is responsible for testing and correctly operating the software on the machine involved.

Restrictions

We would like to explicitly point out the following restrictions and constraints:

- Simulation/simultaneous recording under Sinumerik Operate is not possible.
- The application example is provided in German and English. Users must adapt the software to include additional languages using Run MyScreens.
- The help screens have resolutions of 640 and 1024 pixels. Users must adapt the system to achieve other resolutions.
- The alignment of the help screens is for coordinate systems 0/33/34. Users must adapt the system for other coordinate systems.
- The software supplied includes application examples for use under programmGUIDE and ShopMill. There is no application example for ShopTurn.

Run MyScreen

The following restrictions apply regarding Run MyScreen screen forms:

- A "Select tool" ("SELECT TOOL") is not available.
- The Sinumerik control does not respond when the "SELECT" button on the keyboard is actuated.
- The "Alternative" softkey of the cycle screen must be used to select either the turning or milling tool.
- The cycle cannot be recompiled if it was programmed externally.

1.3 Help screens

Presently, help screens exist for 3 positions of the coordinate system (MD52000-$MCS_DISP_COORDINATE_SYSTEM) 0-33-34 in 2 resolution versions (640 and 1024 pixels). The help screens that match the machine and hardware must be loaded.

If the cycle is to be installed on a machine, where there are still no screens for its coordinate system position, then the screens from directory ico640_COORD_SYS_0 must be adapted as follows corresponding to the position of the coordinate system:

1. Turning or mirroring.
2. The last two characters of the screen names, for example "0" of the corresponding numbers of the coordinate system (e.g. "15") must be renamed. (Exceptions, IPT01_0.png, IPT02_0.png, IPT05_0.png, IPT06_0.png).
3. Adapt the required size of 800/1024/1280/1600 pixels by increasing.

1.4 Texts

The Run MyScreen texts used here have numbers in the range from 81800 up to 81899. **If this range is already assigned**, then the numbers must be appropriately modified. As otherwise, overlaps can occur. The text numbers in Run MyScreen screen forms cycle959.com and e_ipt.com must be appropriately modified.

Table 1-1

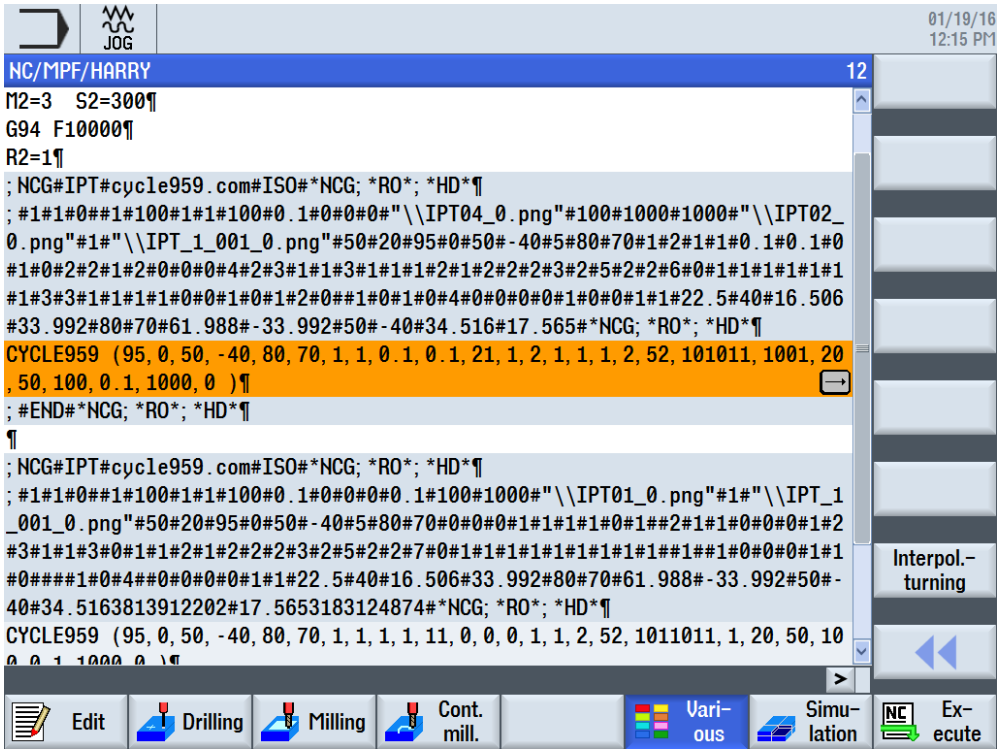
Cycle959.com and e_ipt.com al_ipt_deu.txt	81800 to 81899 (Run MyScreen texts)

2 Technology cycle "CYCLE959"

2.1 Overview

The objective of the technology cycle (Cycle959) is to execute turning processes on milling machines that do not have rotary axes. As a consequence, corners at outer or inner contours can be machined either longitudinally or transversely. This application represents a package comprising technology cycle and user interface based on Run MyScreen. By loading the data provided (Programme.zip) to the CF card of the machine involved, while generating the part program, the operator can utilize the technology cycle (Cycle959).

Fig. 2-1 Using Cycle959 when generating the part program



Advantages of the solution

The following advantages are obtained when turning a workpiece on a milling machine:

- Equipping times are reduced or eliminated
- Costs are reduced and time shortened
- Productivity and cost-effectiveness are increased
- The workpiece is completely machined on just one machine

2.2 Using the cycle

The following machining types are available when generating a part program.

2.2.1 Machining methods

Roughing

For roughing applications, paraxial cuts are machined to the finishing allowance that has been programmed. If no finishing allowance was programmed, stock is removed when roughing up to the final contour.
While roughing, the cycle reduces the programmed infeed depth D if necessary so that it is possible for cuts of an equal size to be made.
For example, if the overall infeed depth is 10, and an infeed depth of 3 has been specified, then this would result in cuts of 3, 3, 3 and 1. The cycle now reduces the infeed depth to 2.5 so that 4 cuts of equal size are created. The angle between the contour and the tool cutting edge determines whether the tool rounds the contour at the end of each cut by the infeed depth D, in order to remove residual corners, or is raised immediately.
The angle beyond which rounding is performed is stored in a machine data element. If the tool does not round the corner at the end of the cut, it is raised by the safety clearance or a value specified in the machine data at rapid traverse. The cycle always observes the lower value; otherwise, stock removal at inner contours, for example, could cause the contour to be damaged.

Finishing

Finishing is performed in the same direction as roughing. During finishing, the cycle automatically calculates the tool radius compensation for turning tools. For milling tools, only for ball mills and mills with corner rounding.

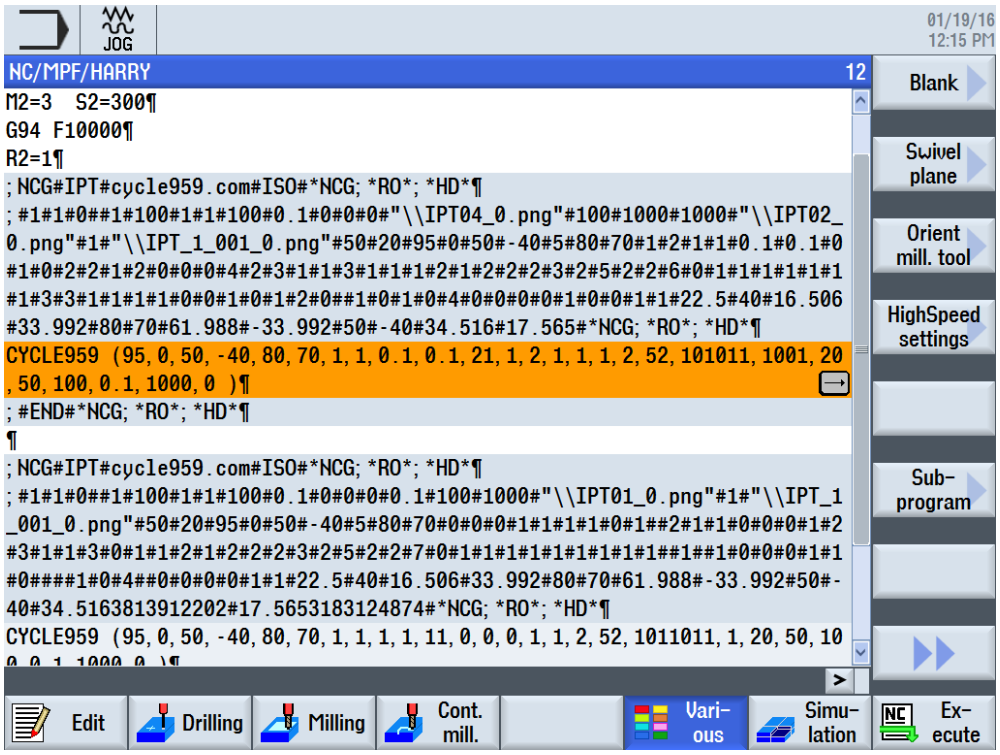
2.2.2 Approach/retraction

1. The tool being used moves to the programmed retraction plane in the Z direction.
2. The tool rapidly traverses in the X and Y direction to the starting point of the machining operation calculated internally in the cycle (reference point & safety clearance).
3. The tool rapidly traverses in the Z direction to the starting point of the machining operation calculated internally in the cycle (reference point & safety clearance).
4. The tool rapidly traverses to the first infeed level.
5. The first cut is made with the machining feedrate.
6. The tool rounds the contour at the machining feedrate or is raised with rapid traverse.
7. The tool is moved in rapid traverse to the starting point for the next infeed depth.
8. The next cut is made with the machining feedrate.
9. Steps four to six are repeated until the final depth is reached.
10. The tool retracts to the safety clearance with rapid traverse.
11. The tool retracts at rapid traverse in the X and Y direction to the starting point of the machining operation calculated internally in the cycle (reference point and safety clearance).
12. The tool traverses to the programmed retraction plane in the Z direction.

2.2.3 Procedure

The part program or ShopMill program to be edited has been created and you are in the editor.

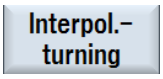
Fig. 2-2 From the navigator, navigate to interpolation turning



Interpolation turning is called using the subsequent key combination.

- Press the "Various" softkey.

- Press the "Expansion" softkey.

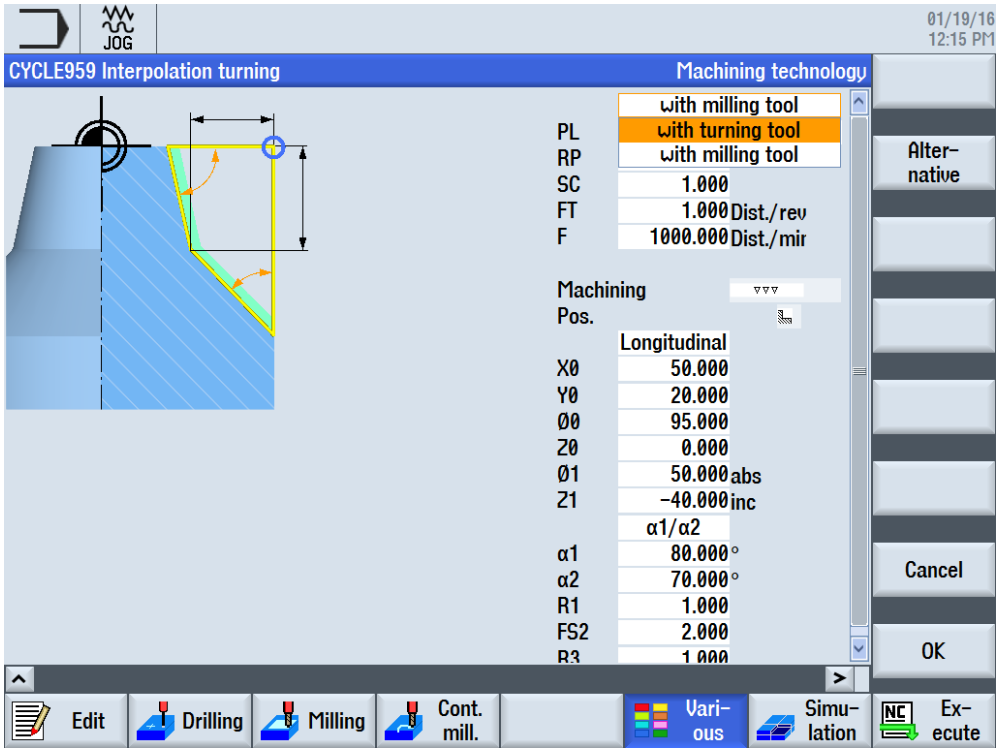


- Press the "Interpol. turning" softkey.

.

The following input screen is displayed. A detailed description of the parameters of the machining technology is provided in Chapter 2.2.4 Parameter selection.

Fig. 2-3 Screen form interpolation turning with Cycle 959



2.2.4 Parameter selection

programGUIDE parameters

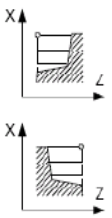
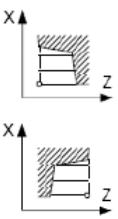
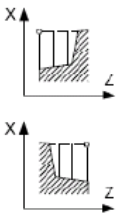
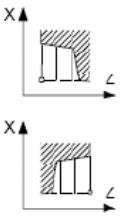
G-code program parameters					
Alter-nativ Machining technology					
With milling tool			With turning tool		
PL	Machining plane	Direction of rotation Alter-nativ Down-cut/up-cut	**PL**	Machining plane	
RP	Retraction plane		**RP**	Retraction plane	
SC	Safety clearance		**SC**	Safety clearance	
FT	Pitch of the machining helix	distance/rev	**FT**	Pitch of the machining helix	distance/ rev
F	Milling feedrate	mm/min / inch/min Alter-nativ mm/tooth / inch/tooth	**VT/** **ST**	Constant cutting rate Turning Alter-nativ Turning speed	m/min inch/min Alter-nativ rpm
			SPOS	Start position of the tool spindle	°

ShopTurn parameters

ShopTurn program parameters							
Alter-nativ Machining technology							
With milling tool				With turning tool			
T	Tool name	**D**	Cutting edge number	**T**	Tool name	**D**	Cutting edge number
FT	Pitch of the machining helix		mm/rev / inch/rev	**FT**	Pitch of the machining helix		mm/rev / inch/rev
F	Milling feedrate		mm/min / inch/min Alter-nativ mm/tooth / inch/tooth	**VT/** **ST**	Constant cutting rate Turning Alter-nativ Turning speed		m/min inch/min Alter-nativ rpm
S/ **V**	Milling tool speed Alter-nativ Constant cutting rate Milling		rpm Alter-nativ m/min	**SPOS**	Start position of the tool spindle		°

Common parameters for programmGuide and ShopTurn

Parameter	Description		Unit
Machining Alter-nativ	• ▽ (roughing) • ▽▽▽ (finishing)		
Position Alter-nativ	Machining position 		
Machining direction Alter-nativ	Stock removal direction (transverse or longitudinal) in the coordinate system		
	Parallel to the Z axis (longitudinal)	Parallel to the X axis (transverse)	

	außen	innen	außen	innen	
					
X0	Machining center point in X				mm
Y0	Machining center point in Y				mm
Ø0	Reference point in X ∅ (abs, always diameter)				mm
Z0	Reference point in Z (abs)				mm
Ø1	End point X (abs) or end point X in relation to Ø0 (inc)				
Z1	End point Z (abs) or end point Z in relation to Z0 (inc)				
D	Maximum depth infeed – (not for finishing)				mm
UX	Finishing allowance in X – (not for finishing)				mm
UZ	Finishing allowance in Z – (not for finishing)				mm
FS1...FS3 or R1...R3 Alter-nativ	Chamfer width (FS1...FS3) or rounding radius (R1...R3)				mm

Alter-nativ	Parameter selection of intermediate point The intermediate point can be defined using a position or angle. The following combinations are possible • ØM ZM • α1 α2 • α1 ZM • ØM A2	
ØM Alter-nativ	Intermediate point X Ø (abs) or intermediate point X in relation to X1 (inc)	mm
ZM Alter-nativ	Intermediate point Z (abs or inc)	mm
α1	Angle of the 1st edge	mm
α2	Angle of the 2nd edge	mm

3 Installing technology cycle Cycle959

All of the preconditions for using the "Run MyScreen" tool must be fulfilled before using the technology cycle. A detailed list of the necessary criteria can be found in the Commissioning Manual "SINUMERIK 840D sl Basesoftware and operating software".
In this chapter, only the detailed sequence in the "Run MyScreen" tool for activating Cycle959 is discussed. All of the necessary files, which, within the scope of using Run MyScreen are used to activate technology cycle Cycle959 (easyscreen.ini, .com files etc.) are available as download in the attachment to this application. Depending on the hardware components being used/available, the file specifications are subdivided for PCU 50 and NCU.

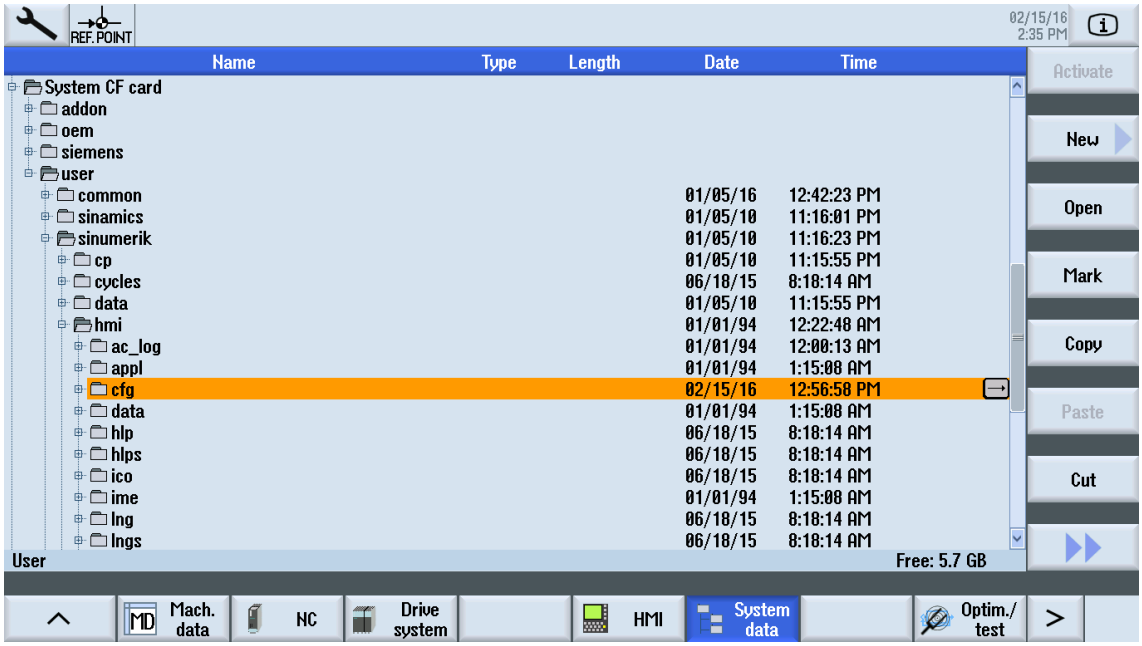
3.1.1 Run MyScreen - screen forms

Note All of the .com file names used must have lowercase letters.

File easyscreen.ini, from directory cfg of file Programme.zip must be copied into the subsequent Sinumerik directory.

- PCU 50: F:HMI/hmisl/user/sinumerik/hmi/cfg/
- NCU: SystemCF card/user/sinumerik/hmi/cfg/

Fig. 3-1 Creating the Run MyScreen user interface



If, in the Sinumerik directory, there is already an easyscreen.ini file, then this must be supplemented.

The two Run MyScreen screen forms cycle959.com and e_ipt.com from directory cfg/SPF_File must be copied to the following directory.

- PCU 50: F:HMI/hmisl/user/sinumerik/hmi/proj/
- NCU: SystemCF-card/user/sinumerik/hmi/proj/

3.1.2 Screen form text

The Run MyScreen texts al_ipt_xxx.txt must be copied from directory lng into the following directory.

- PCU 50 (Win XP/ 7): F:HMI/hmisl/user/sinumerik/hmi/lng
- NCU: SystemCF-card/user/sinumerik/hmi/lng

Note The text files used must be processed and saved as UTF-8 format.

3.1.3 Help screens

The help screens from directories ico/ico640 bis ico1024 correspond to the position of the coordinate system (MD52000 $MCS_DISP_COORDINATE_SYSTEM) and must be copied into the directory in the relevant display size folder.

Note The information regarding the display position is included in the last two digits of the directory name, e.g. Ico640_COORD_SYS_33.

The correct target directory for the help displays is:

- PCU 50 (Win XP/ 7): F:HMI/hmisl/user/sinumerik/hmi/ico
- NCU: SystemCF-card/user/sinumerik/hmi/ico

3.1.4 Turning tools

If, at the machine, turning is not set (MD52000=2 and MD52200=2), then for creating the tools, turning tools are not available. In order to still be able to select turning tools, the **sltmlistconfig.xml** file provided must be copied from directory **cfg** into the following directory.

- PCU 50 (Win XP/ 7): F:HMI/hmisl/user/sinumerik/hmi/cfg
- NCU: SystemCF-card/user/sinumerik/hmi/cfg

3.1.5 NC cycle

Copy cycle **cycle959.spf** and **e_ipt.spf** - without comments - from directory **cycle/SPF_File** to directory **NC_Daten/Zyklen/Standard-Zyklen**.

Note NC and HMI must then be restarted.

4 Contact

Siemens AG

Industry Sector
DF MC MTS PPM
Frauenauracher Straße 80
91056 Erlangen
Germany
mailto: MC-MTS-APC-Tech-Team.i-dt@siemens.com

5 History

Table 5-1

Version	Date	Revision
V1.0	02/2016	First edition