DOCUMENTATION ISG-kernel

Functional description
Coordinate systems and offsets

Short Description:
FCT-C2

ISG Industrielle Steuerungstechnik GmbH
STEP, Gropiusplatz 10
D-70563 Stuttgart

www.isg-stuttgart.de
support@isg-stuttgart.de

Documentation version: 1.04
07/11/2024

Preface

Legal information

This documentation was produced with utmost care. The products and scope of functions described are under continuous development. We reserve the right to revise and amend the documentation at any time and without prior notice.

No claims may be made for products which have already been delivered if such claims are based on the specifications, figures and descriptions contained in this documentation.

Personnel qualifications

This description is solely intended for skilled technicians who were trained in control, automation and drive systems and who are familiar with the applicable standards, the relevant documentation and the machining application.

It is absolutely vital to refer to this documentation, the instructions below and the explanations to carry out installation and commissioning work. Skilled technicians are under the obligation to use the documentation duly published for every installation and commissioning operation.

Skilled technicians must ensure that the application or use of the products described fulfil all safety requirements including all applicable laws, regulations, provisions and standards.

Further information

Links below (DE)

https://www.isg-stuttgart.de/produkte/softwareprodukte/isg-kernel/dokumente-und-downloads

or (EN)

https://www.isg-stuttgart.de/en/products/softwareproducts/isg-kernel/documents-and-downloads

contains further information on messages generated in the NC kernel, online help, PLC libraries, tools, etc. in addition to the current documentation.

Disclaimer

It is forbidden to make any changes to the software configuration which are not contained in the options described in this documentation.

Trade marks and patents

The name ISG®, ISG kernel®, ISG virtuos®, ISG dirigent® and the associated logos are registered and licensed trade marks of ISG Industrielle Steuerungstechnik GmbH.

The use of other trade marks or logos contained in this documentation by third parties may result in a violation of the rights of the respective trade mark owners.

Copyright

General and safety instructions

Icons used and their meanings

This documentation uses the following icons next to the safety instruction and the associated text. Please read the (safety) instructions carefully and comply with them at all times.

Icons in explanatory text

➢ Indicates an action.

⇨ Indicates an action statement.

⚠ DANGER

Acute danger to life!

If you fail to comply with the safety instruction next to this icon, there is immediate danger to human life and health.

⚠ CAUTION

Personal injury and damage to machines!

If you fail to comply with the safety instruction next to this icon, it may result in personal injury or damage to machines.

Attention

Restriction or error

This icon describes restrictions or warns of errors.

Notice

Tips and other notes

This icon indicates information to assist in general understanding or to provide additional information.

Example

General example

Example that clarifies the text.

Programing Example

NC programming example

Programming example (complete NC program or program sequence) of the described function or NC command.

Release Note

Specific version information

Optional or restricted function. The availability of this function depends on the configuration and the scope of the version.

Table of contents

List of figures

1 Overview

Task

By selecting a coordinate system (referred to as CS), the programming coordinate system (PCS) can be offset and rotated relative to the machine coordinate system (MCS). This permits simple NC programming even in slanted or twisted positions.

Offsets, e.g. zero offsets, can also be regarded as selecting a coordinate system.

Characteristics

Coordinate systems (CS) and offsets are always active from selection through to deselection or when offset parameters are changed. The offsets additionally programmed while a coordinate system is active are only valid until the coordinate system is deselected.

Programming and parameterisation

Some of the offsets can be defined in the NC program. They include the following:

- Offsets due to a coordinate system selection (#CS)
- Offsets due to a measurement run (G100)
- Offsets from position presets (#PSET)
- Reference point offsets (G92)
- Offsets due to manual mode in conjunction with parallel interpolation (/G202)

The following offsets are preset in configuration lists. They can be used directly in the NC program (G54..., D..). It is also possible to use the edit option during program runtime.

- Tool offsets: Parameterising using the tool parameter list [TOOL] or variable access (V.G.)
- Zero offsets: Parameterising using the zero point data list [ZERO] or variable access (V.G.)
- Clamping position offsets: Parameterisation by means of the clamp position offset data list [CLMP]
-

Mandatory note on references to other documents

For the sake of clarity, links to other documents and parameters are abbreviated, e.g. [PROG] for the Programming Manual or P-AXIS-00001 for an axis parameter.

For technical reasons, these links only function in the Online Help (HTML5, CHM) but not in pdf files since pdfs do not support cross-linking.

2 Description

Task of the coordinate system

The programming coordinate system (PCS) can be offset and rotated with respect to the machine coordinate system (MCS) by selecting a coordinate system (CS) (see figure below). This enables easy NC programming even in slanted or twisted positions.

Notice

NC program coordinates refer to the PCS.

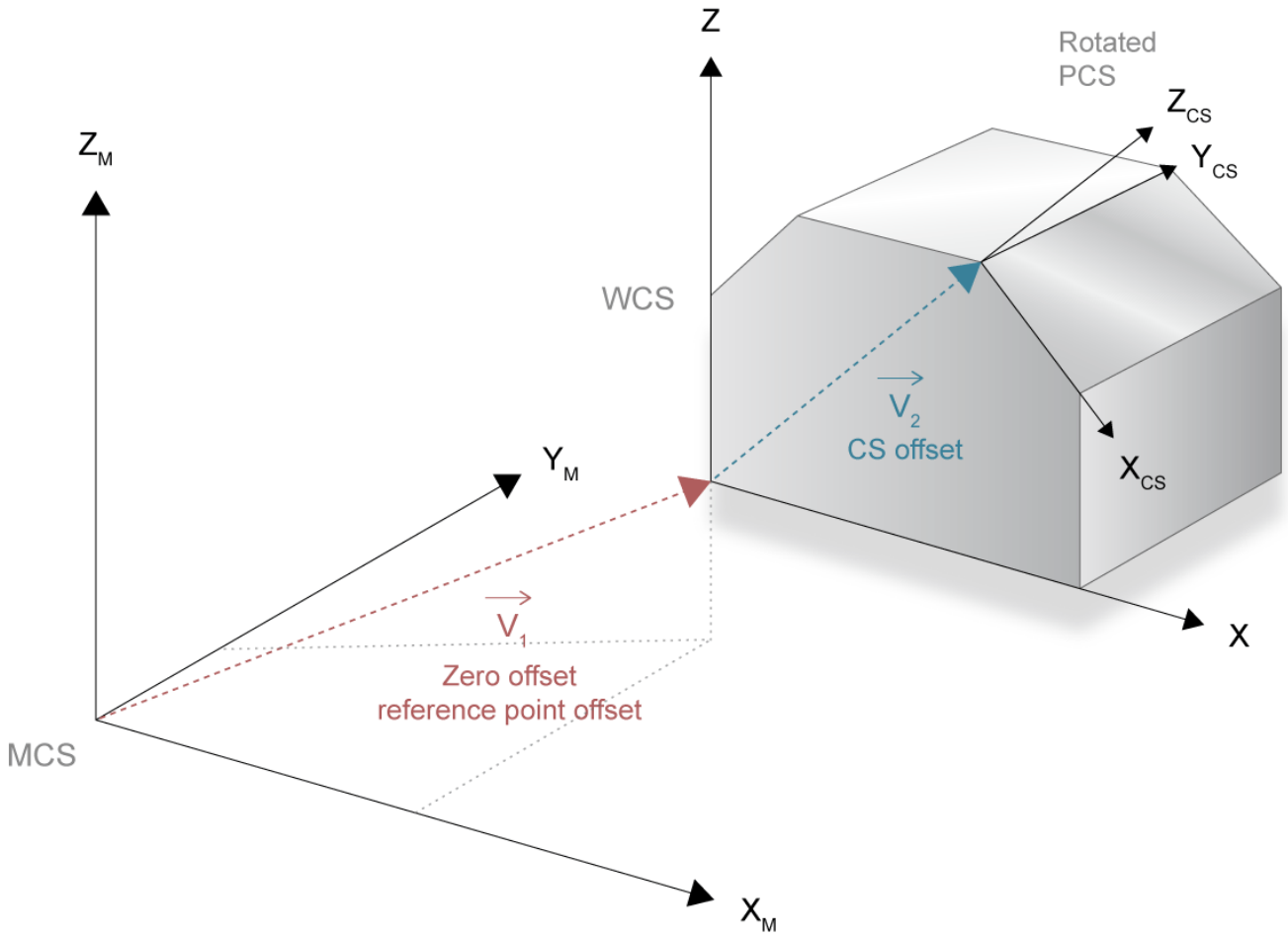

Fig. 1: PCS after selecting a coordinate system

Offsets

When a coordinate system is selected, an additional offset of the PCS is generated. The following offsets are available in the CNC system:

- Offsets due to selection of a coordinate system
- Offsets due to a measurement run
- Offsets due to manual operation with parallel interpolation
- Offsets due to setting an actual value
- Reference point offsets
- Tool offsets
- Zero offsets
- Clamping offsets

2.1 Offset types

Overview of clamp position offsets that are selectable in parallel:

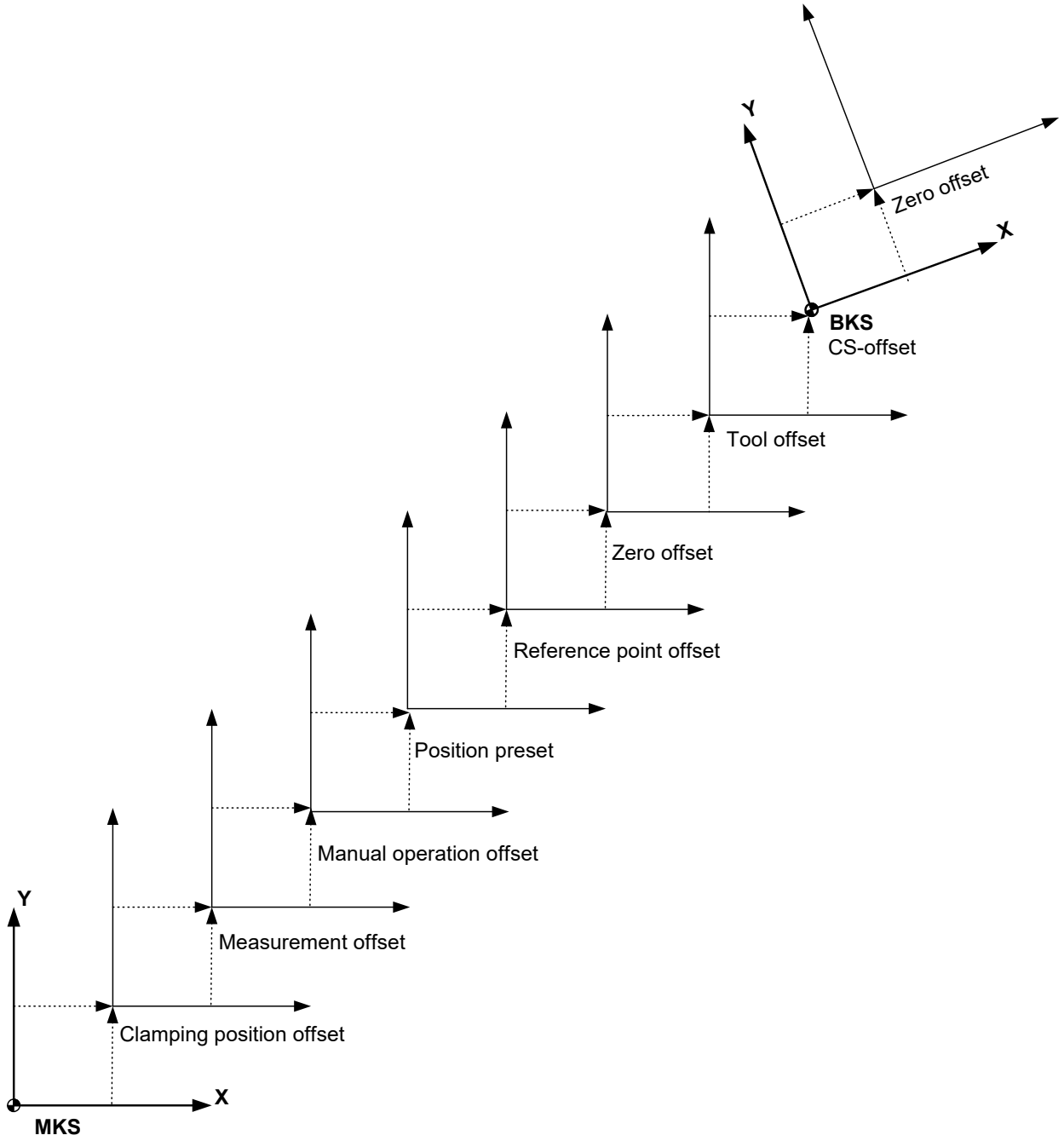

Fig. 2: Overview of all offset types

Notice

Special case in G91 mode:

Newly selected offsets are only effective after the next absolute programmed position (G90) since only the specified motion path may only be travelled in the case of relative programming (G91).

Tool offset exception: With appropriate parameterisation, tool offsets are travelled directly without programming a path movement when the D word (P-CHAN-00100) is selected.

Differences in definition and selection of individual offsets

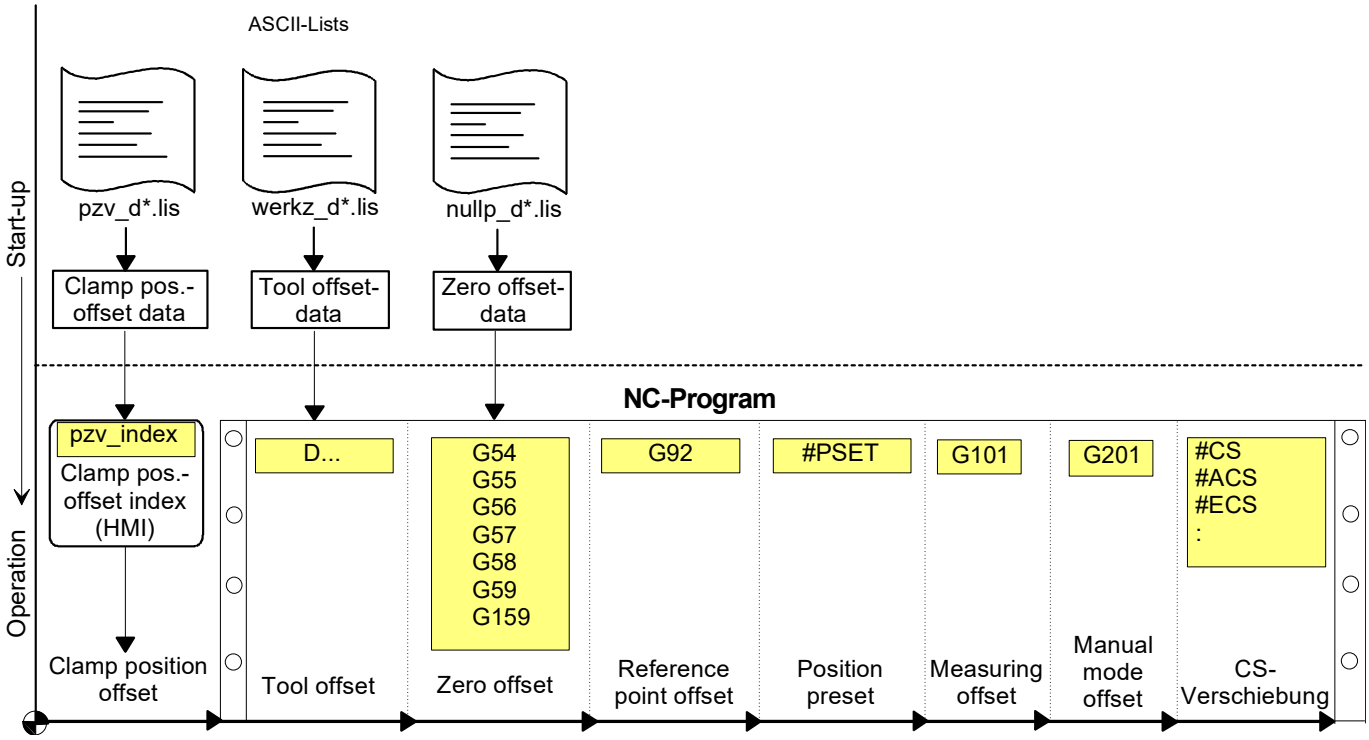

Fig. 3: Differences and selection of offsets

Offsets are selected or included in calculations by the following NC commands or parameters:

NC command	Description
#CS, **#ACS,** **#ECS,** **#ROTATION**	Select a coordinate system : : Rotate in the plane
G101	Include measuring offset in calculations
G201 **#GET MANUAL OFFSETS**	Include manual mode offsets in calculations
#PSET	Select position presets
G92	Reference point offset
G54, ... G59, G159	Select zero offsets
D	Tool geometry compensation

Variable	Description
pzv_index	Index for selecting position offset. This must be specified along with the NC program name when an NC program is commanded.

2.2 Coordinate systems

Coordinate systems are defined, selected and deselected by NC command. The section below deals briefly with programming. A detailed description of NC commands is contained in the Programming Manual (see PROG documentation).

2.2.1 Selecting a coordinate system (#CS)

Select: #CS ON

The PCS can be offset and rotated with respect to the MCS by selecting a CS. This permits simple NC programming even in slanted or twisted positions.

Use of this NC command depends on the axis configuration of the machine (see Channel parameters, Section: Configuration of the path axes).

Machining in a rotated PCS basically makes sense whenever existing rotary axes P-AXIS-00018 permit vertical orientation of the tool with respect to the machining surface (#TOOL ORI CS).

The axis designations P-CHAN-00006 remain unchanged in the CS.

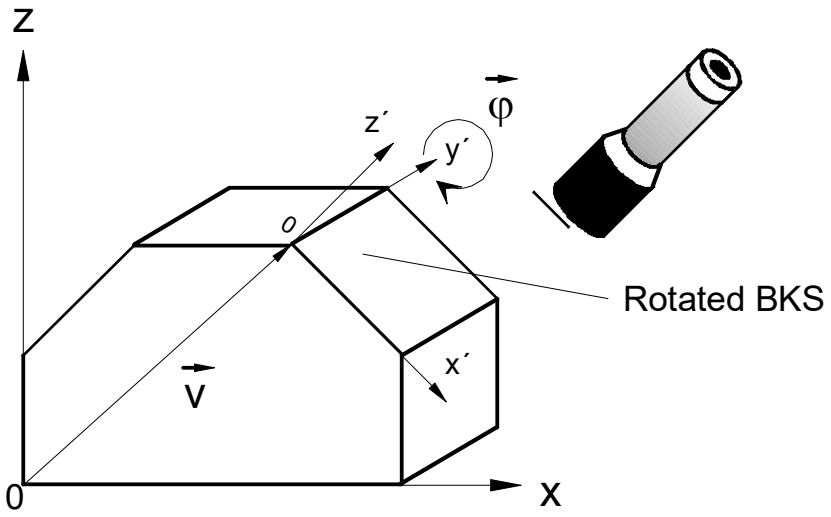

Fig. 4: Selecting a CS with one offset and one rotation

V Offset vector

φ Rotation vector

Effectiveness

- Active until cancelled or up to program end.
- Nesting/linking a CS is possible.

Zero offsets and reference point offsets may be programmed in the CS during machining. They are valid until the CS is deselected; they are not saved.

Deselect: #CS OFF

Notice

The NC command #ROTATION is provided for simple rotations in the plane.

2.2.2 Rotation in a plane (contour rotation)

Selection: #ROTATION ON

The coordinate system can be rotated within the main plane by using contour rotation. Rotation may also be applied within an already rotated coordinate system (CS, ACS).

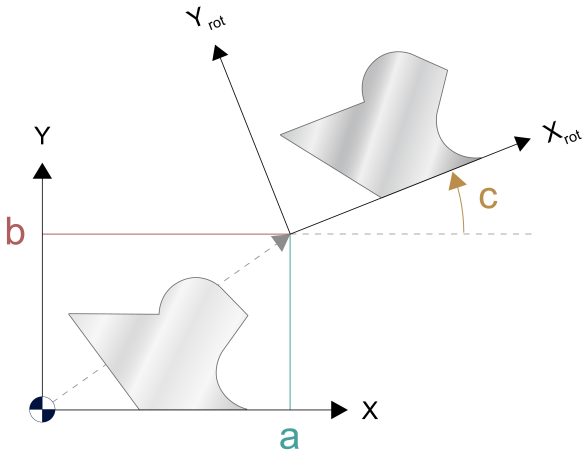

Fig. 5: Rotation of the main plane

Effectiveness

- Effective up to deselection or up to the end of the program.
- Contour rotation acts in the main plane only on the programmed axis coordinates **before** all other contour-influencing functionalities, i.e. all offsets and mirror-imaging operations can be used as before.

Attention

A change of plane with G17/18/19 automatically deselects an active contour rotation. The warning P-ERR-21143 is output.

Deselection: #ROTATION OFF

2.2.3 Coordinate system for fixture adaptation (#ACS)

Selection: #ACS ON

The fixture adaptation coordinate system ACS compensates a sloping position of the workpiece or workpiece palette. The ACS is independent of a CS.

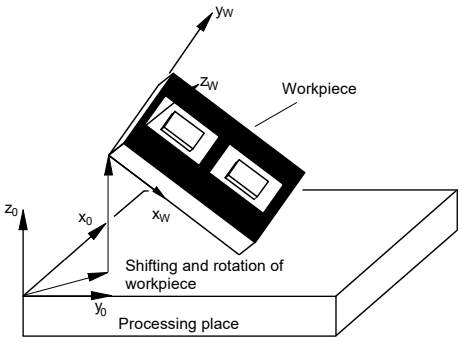

Fig. 6: Select a coordinate system for fixture adaptation

Effectiveness

- Effective up to deselection or up to the end of the program.
- Zero offsets and reference point offsets may be programmed in the ACS. However, they are only valid until the ACS is deselected and they are not stored.

Deselection: #ACS OFF

2.2.4 Linkage of coordinate systems

Machining on a sloping plane with a workpiece in an inclined plane is possible by combining an ACS with a CS.

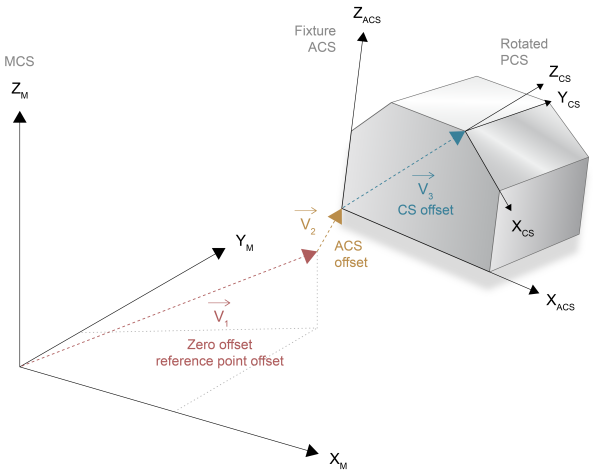

Fig. 7: Combination of ACS and CS

Effectiveness

- Effective up to deselection or up to the end of the program.

2.2.5 Effector coordinate system (#ECS)

Selection: #ECS ON

The ECS is used mainly to execute a withdrawal strategy after

- tool breakage
- NC reset or
- program abort

for machining with a tool in any orientation A machining plan that is perpendicular to the tool axis is determined automatically when determining the ECS.

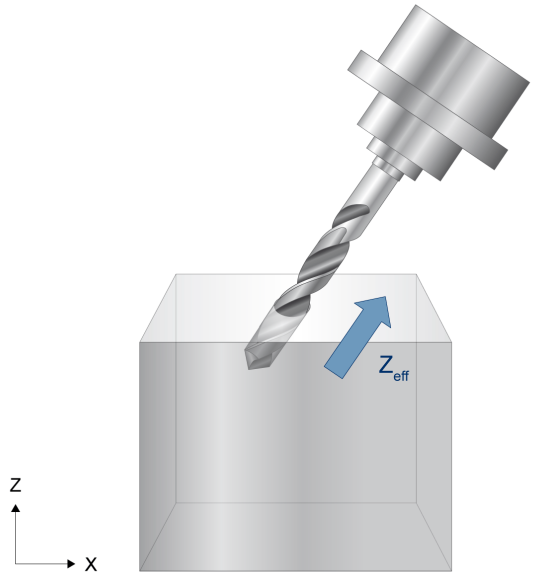

Fig. 8: Machining in a slanting hole

Effectiveness

- Effective up to deselection or up to the end of the program.

Notice

No other coordinate system may be active when ECS is activated, otherwise the error message P-ERR-20774 is output.

Deselection: #ECS OFF

2.2.6 Temporary transition to the machine axis coordinate system (#MCS)

Select: #MCS ON [**EX TOOL**]

With #MCS ON, active transformations and all offsets calculated in the axes can be temporarily deactivated so that machine axes can be directly positioned.

After leaving the MCS, the state before selection is restored.

Effectiveness

- Active until deselected or up to program end.
- With the ‘EX TOOL’ option, no tool offsets are included in the MCS calculation when the tool is changed so that machine axes can continue to be positioned directly. Only with #MCS OFF are tool offsets again included in the calculations.
- Default offsets (e.g. zero offsets) can be programmed in the MCS. However, they are only active until they are deselected and are not saved.
- It is not possible to select CS, ACS, ECS and TLC in the MCS.

Deselect: #MCS OFF

2.3 Measuring offset

Select: G101 <*Achsname*> <*Einrechnungsfaktor*>

Measurement offset is the offset between the recorded measured values and the target position.

With G101, the measurement offset is included in the axis coordinates as an additional offset in accordance with the inclusion factor specified.

Fig. 9: Measurement offset between measurement signal and target point

Effectiveness

- Active until deselected or up to program end.
- A new measurement offset has no effect until the next absolute programmed position (G90).

Deselect: G102 <*axis_name*><*Dummyzahl*>

2.4 Manual mode offset

Selection: #GET MANUAL OFFSETS

The additional command values generated during an active G201 result in the path axes being offset to the PCS plane. After manual mode is deselected with G202, these offset values can be requests from the interpolator using #GET MANUAL OFFSETS and included as additional offset in the calculations of the axis coordinates in the NC program.

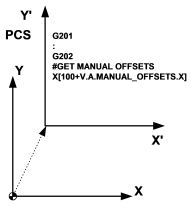

Fig. 10: Manual mode offset

Effectiveness

- Effective up to deselection or up to the end of the program.

Deselection: -

2.5 Position preset

Select: #PSET *<axis_name> <Neue Istposition> ...*

This NC command can assign a new actual value to the current axis position. The resulting offset is called the position preset.

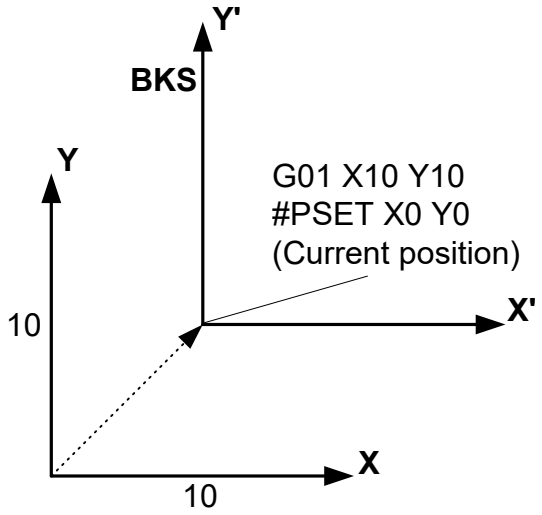

Fig. 11: Position preset

Effectiveness

- Active until deselected or up to program end.
- Position reset is cancelled by a homing run (G74).
- A new offset has no effect until the next absolute programmed position (G90).

Deselect: #PRESET {*<axis name><Dummywert>}*

The position presets are cancelled in all axes if #PRESET is programmed without specifying an axis.

2.6 Reference point offset

Selection by G92 <*Achsbezeichnung*> <*Verschiebung*> ...

G92 permits a reference point offset by a freely programmable value in the specified axes. Depending on the settings of G90/G91, the currently programmed reference point offset is set absolutely or relatively.

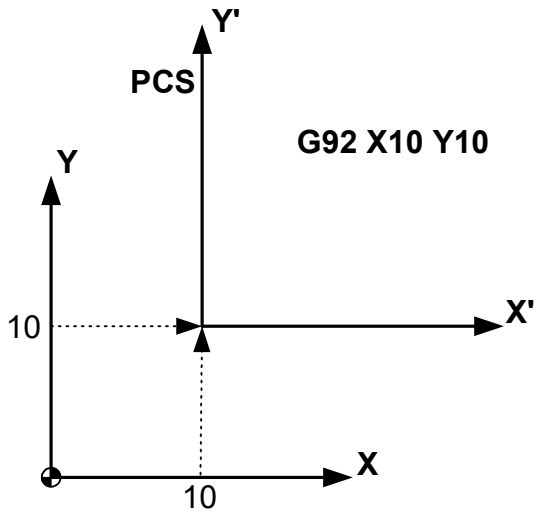

Fig. 12: Reference point offset

Effectiveness

- Active until the next reference point offset is programmed or up to program end.
- A new offset has no effect until the next absolute programmed position (G90).

Deselect: G92 <*Achsbezeichnung*> **0**

2.7 Zero offset

Select: G54, ... G59, G159

The G54 - G59 or G159 commands select the corresponding zero offsets from the zero offset table [ZERO].

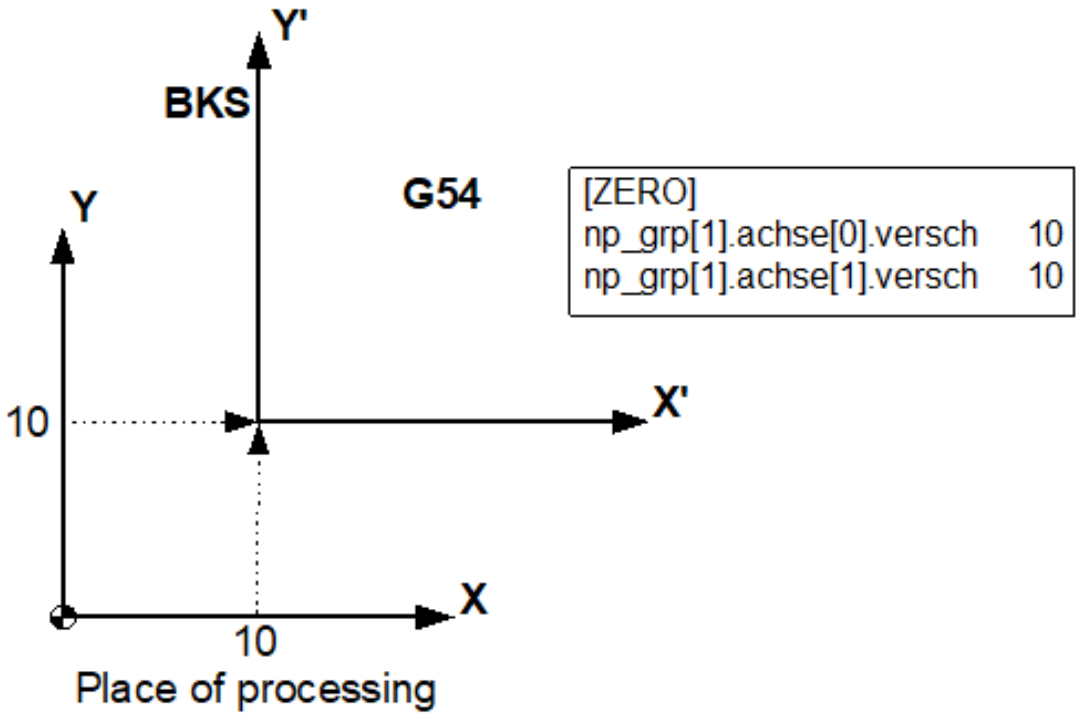

Fig. 13: Zero offset

Effectiveness

- Active until deselected or up to program end.
- A new offset has no effect until the next absolute programmed position (G90).

Deselect: G53

2.8 Tool offset

Select: D <*Nummer des Korrekturdatensatzes*>

The D word selects the corresponding tool geometry compensation data from the tool list [TOOL].

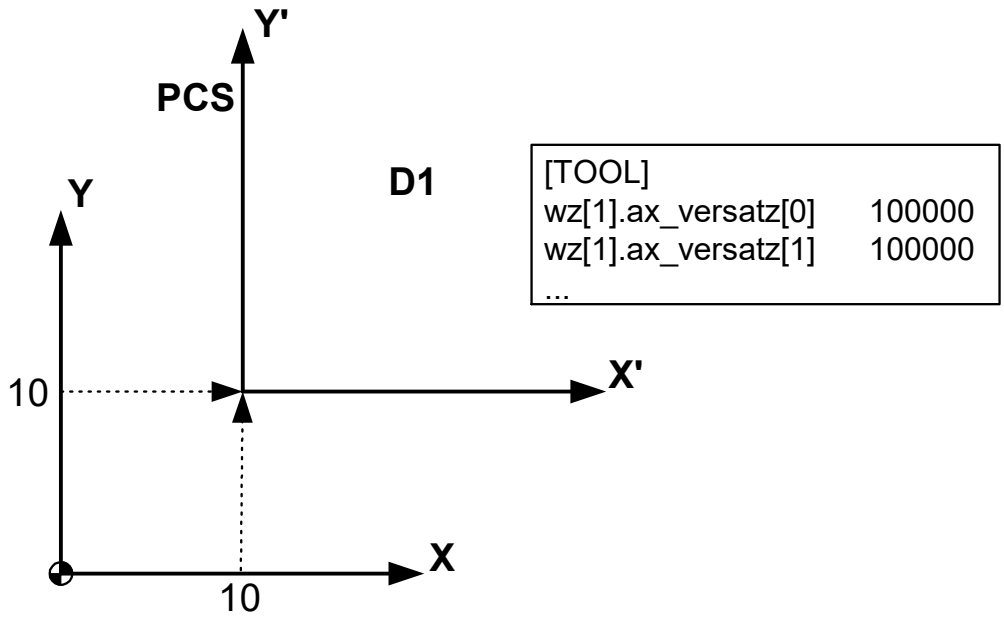

Fig. 14: Tool offset

Effectiveness

- Active until deselected or up to program end. In order to use kinematic transformations, special tool offset parameters may have to be assigned (wz[i].kinematic.param[j]).
- Depending on the parameterisation, a new offset becomes active either directly when the D word is selected or until the next absolute programmed position (G90) from (P-CHAN-00100).
- Depending on P-TOOL-00010 tool offsets act in the axis directions of the machining coordinate system or the machine coordinate system.

Deselect: D0

2.8.1 Tool length compensation (#TLC)

Select: #TLC ON [<tool length compensation> **]**

TLC permits the reuse of NC programs which were created by a programming system and which consider a specific tool length, even if the tool length was changed on the machine.

Effectiveness

- Active until deselected or up to program end.
- No new offsets or radii of the tool can be compensated, only the length.
- A new offset has no effect until the next absolute programmed position (G90).

Abwahl #TLC OFF

2.9 Clamp position offset

Select

When the NC program is commanded, the clamping position offset compensation data [CLMP] is selected together with the NC program name. In the NC program, the currently valid position offset index can be read via the V.G.AKT_PLATZ variable.

```
#PZV1
pzv_grp[1].achse[0].versatz     -10
pzv_grp[1].achse[1].versatz     88
pzv_grp[1].achse[2].versatz     0.32
#PZV2
pzv_grp[2].achse[0].versatz     13
pzv_grp[2].achse[1].versatz     153
pzv_grp[2].achse[2].versatz     -8.76
.
.
.
```

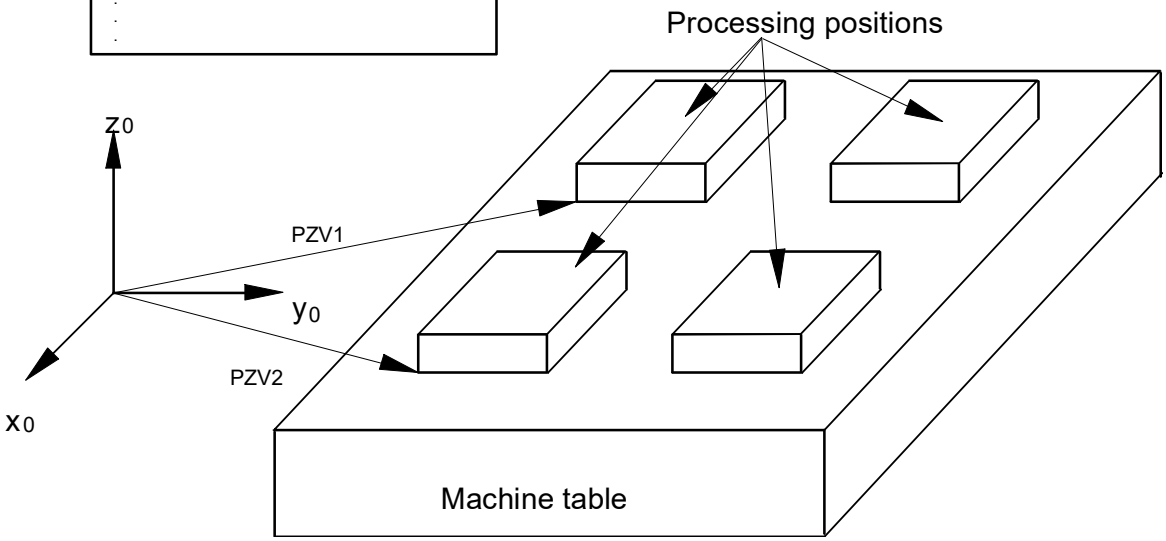

Fig. 15: Clamp position offset

Effectiveness

- Effective up to program end.

Deselect: -

2.10 Suppressing offsets

Select: #SUPPRESS OFFSETS [ZERO PSET ...] *<Achsname><position> ...*

In combination with a motion block, this command causes an execution of the programmed axis positions without consideration of the active offsets.

All offsets in the NC block are suppressed if a particular offset type is not specified.

Effectiveness

- Valid only in the programmed NC block.

Deselect: -

3 Parameter

3.1 Overview

ID	Parameter	Description
P-AXIS-00018	achs_typ	Axis type
P-CHAN-00006	bezeichnung	Axis identifier
P-CHAN-00100	move_tool_offsets_directly	Effectiveness of tool compensation data

3.2 Axis parameters

P-AXIS-00018	**Axis type (linear axis, rotary axes, spindle)**	
Description	This parameter specifies the axis type of an axis.	
Parameter	kenngr.achs_typ	
Data type	STRING	
Data range	Linear axis (ACHSTYP_TRANSLATOR) : 0x0001 Rotary axis (ACHSTYP_ROTATOR) : 0x0002 Spindle (ACHSTYP_SPINDEL): : 0x0004	
Axis types	T, R, S	
Dimension	T: ----	R,S: ----
Default value	ACHSTYP_TRANSLATOR	
Drive types	----	
Remarks	Depending on the axis type that is set, special functionalities are addressed in the NC kernel. Examples: - modulo calculation for rotary axes, - speed monitoring for spindles	

3.3 Channel parameters

P-CHAN-00006	Name of an axis in the NC channel
Description	This variable specifies the axis designations of all axes in the interpolator. The related corresponding logical axis number is defaulted by the parameter P-CHAN-00035.
Parameter	gruppe[i].achse[j].bezeichnung
Data type	STRING
Data range	Maximum 16 characters (length of axis designation, application-specific)
Dimension	----
Default value	*
Remarks	The axis designations must begin with the letters A, B, C, U, V, W, X, Y, Z or Q. After that, all letters and digits are possible. The axis designations must be unique. Caution: The axis identifiers ‘A1‘ to ‘A32‘ may not be used if the parameter P-CHAN-00253 is active. In this case, these identifiers are explicitly permitted in the axis lists (P-AXIS-00297) only. Parameterisation example: An axis group with 3 axes is to be configured. The first axis is addressed with ‘X_ACHSE1’ in the NC program. The second axis is designated ‘A’. The third axis is addressed with ‘W1’. *gruppe[0].achse[0].bezeichnung X_ACHSE1* *gruppe[0].achse[1].bezeichnung A* *gruppe[0].achse[2].bezeichnung W1+* * Note: The default value of variables is a blank string.

P-CHAN-00100	Time of effectiveness of tool compensation data
Description	This parameter defines the time of the effectiveness of the tool compensation data (tool length compensation, tool position compensation) programmed by the D word
Parameter	move_tool_offsets_directly
Data type	BOOLEAN
Data range	0: The compensation movement to consider the new tool compensation data only takes place after the D word for an axis when the next traverse information **absolute** (G90) is programmed (default). 1: The compensation movement in the corresponding axes is executed immediately with the D word without programming an absolute path condition (in compliance with DIN 66025).
Dimension	----
Default value	0
Remarks	The parameter has no effect if the kinematic transformation is active (#TRAFO ON). New tool compensation data are only considered with the next absolute travel information.

4 Appendix

4.1 Suggestions, corrections and the latest documentation

Did you find any errors? Do you have any suggestions or constructive criticism? Then please contact us at documentation@isg-stuttgart.de. The latest documentation is posted in our Online Help (DE/EN):

QR code link: https://www.isg-stuttgart.de/documentation-kernel/

The link above forwards you to:

https://www.isg-stuttgart.de/fileadmin/kernel/kernel-html/index.html

Notice

Change options for favourite links in your browser;

Technical changes to the website layout concerning folder paths or a change in the HTML framework and therefore the link structure cannot be excluded.

We recommend you to save the above "QR code link" as your primary favourite link.

PDFs for download:

DE:
https://www.isg-stuttgart.de/produkte/softwareprodukte/isg-kernel/dokumente-und-downloads

EN:
https://www.isg-stuttgart.de/en/products/softwareproducts/isg-kernel/documents-and-downloads

E-Mail: documentation@isg-stuttgart.de

Keyword index

P

ISG Industrielle Steuerungstechnik GmbH
STEP, Gropiusplatz 10
D-70563 Stuttgart

www.isg-stuttgart.de
support@isg-stuttgart.de