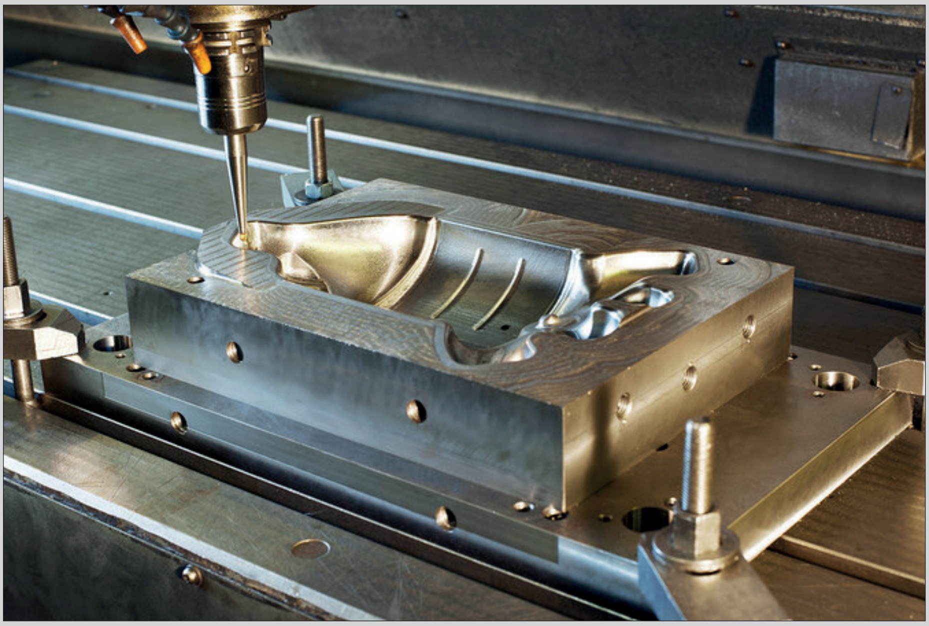
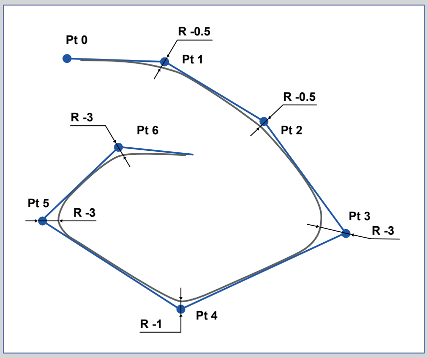
Flexium+ Five Axes and High Speed Machining (HSM)

Programming and Commissioning manual

M00052EN-00

Edition 03.2021





flexium+

www.num.com

num

CNC HighEnd Applications

Summary + Appendix

Page deliberately empty

1 2 3 4 A

Summary

Page deliberately empty

1 Foreword

1.1 Use of the manual

This manual is for use by qualified personnel only.

To reduce the risk due to an improper use of the product, do not perform any servicing other than that contained in this manual unless you are qualified to do so.

To ensure proper use of these products, please always read this User's manual carefully and retain for future reference. Should the unit require maintenance, please contact a NUM authorized service center only.

1.2 Use of symbols in the manual

Symbol	Meaning of the symbol
	Important notes on product use.
	You may find this symbol whenever a wrong operation may damage the product.

1.3 Document revisions

Date	Index	Reason for revision	See chapter
03 - 2021	00	Document creation	-

1.4 How to read the manual

This manual gives information about the Flexium+ Five-axes Machining Commissioning and Programming and its customization to a particular machine.

Chapter	Description
1	**Foreword** This chapter introduces: - The use of the manual.
2	**Five-axes Machining RTCP** This chapter introduces: - Description of the Five-axes Machining RTCP, - Interface for design of RTCP commissioning, - Programming RTCP, - Syntax of programming.
3	**iPLANE transformation** This chapter introduces: - iPLANE definition, - Prerequisites, - Interface for design of iPLANE commissioning, - Programming iPLANE.
4	**Kinematic measuring cycle248** This chapter introduces: - Cycle248 definition, - Prerequisites and application range, - Design of RTCP commissioning tool, - Procedure of kinematic measurement. - Pre-positioning of the probe center position, - Approximated location of calibration sphere, - Procedure of kinematic measurement, - Customization file Cycle248 Data File (10248.9), - Programming cycle248.
A	**Appendix** This chapter introduces: - Kinematic structures managed by the RTCP, iPLANE functions and Cycle248 , - Summary of the reference postures of Five-axes kinematics, - E80x parameters used by RTCP and by iPLANE functions, - Errors list handled by the RTCP & iPLANE functions and errors list for cycle248.

1.5 List of abbreviations

HSC	High Speed Cutting (or High Speed Machine (HSM))
MCS	Machine Coordinate System
ICS	Inclined Coordinate System
WCS	Workpiece Coordinate System
TCP	Tool Center Point
RTCP	Rotation Tool Center Point
iPLANE	Inclined PLANE function
CCS	Cartesian Coordinate System

1.6 NUM Agencies

The list of NUM agencies is given click here http://www.num.com

Page deliberately empty

1
2
3
4
A

Summary

Page deliberately empty

2 High Speed Machining

2.1 Introduction

Goal of High Speed Machining (HSM) is to keep very high contouring precision even at very high feedrate during work-piece machining.

There are many sources of contouring error on a CNC machine, for example machine poor stiffness, tool + material interactions, CAD/CAM generated profile poor smoothness, etc.

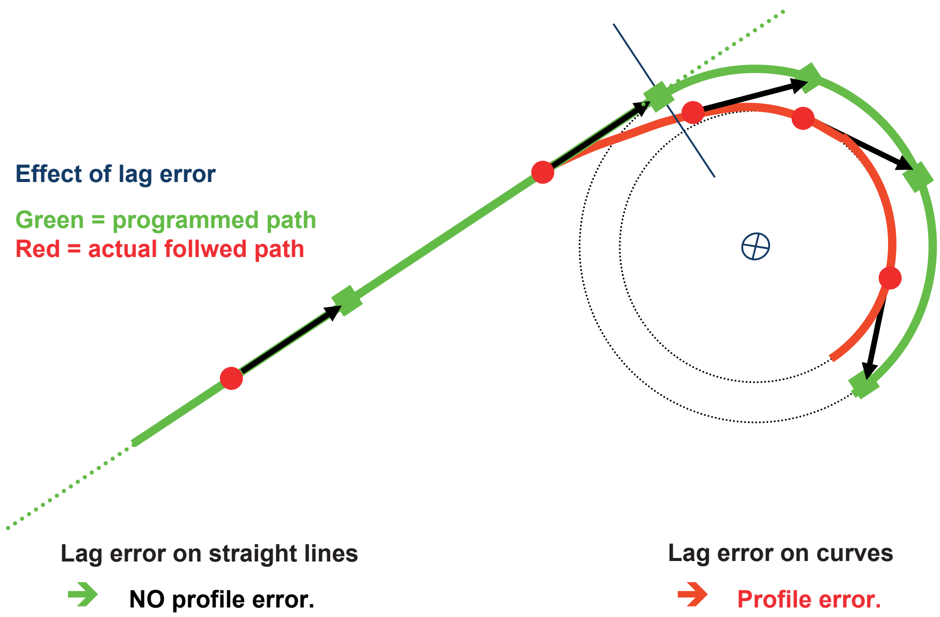
One source of contouring error (error on programmed profile), that is indeed completely controllable and removable by adoption of proper CNC functionalities, is the “Lag error” inherent to the implementation of position control in CNC servo-controlled systems:

Traditionally, CNC + servo-drive systems realize a position control of machine-tool axes by mean of a “linear proportional controller”: a difference between position reference, from CNC, and actual axis position is necessary to move the axis. Such needed difference is called “Lag error”.

Lag error is a sort of “drag effect” due to the fact that It is like the CNC “drags” the servo-controlled axis along the programmed trajectory.

Such Lag error depends on machining speed (feedrate): the higher the feedrate, the bigger the Lag error.

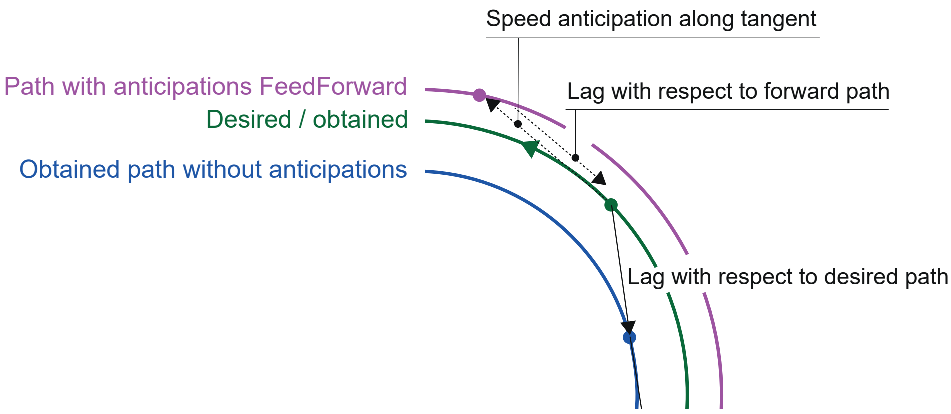
If the position loop gain of all interpolated axes are aligned, then, during machining of a straight line, the “Lag error” does not cause any contouring error, but, during machining of curvilinear paths, the “Lag error” causes a contouring error, that is bigger at higher feedrates.



The most commonly used CNC strategy to get high contour precision at high feedrates, despite the inherent Lag error, is to create a “path with anticipations” (feedforward path) able to compensate for the Lag error “delay” of the servo-controlled position regulator, making the residual Lag error almost null.

Such feedforwarded path consists on speed and acceleration anticipations added to the original programmed trajectory, in order to reduce Lag error as much as possible:

- Speed anticipation along the tangential direction of programmed path,
- Acceleration anticipation along the direction perpendicular to programmed path.



In the context of this chapter, the term "High Speed Machining" refers to the CNC control techniques that use speed anticipation and acceleration anticipation to eliminate the deviation from programmed profile due to Lag error.

This chapter gives an HSM setting methodology.
However, the first steps in the methodology also apply to normal feedback, i.e. without anticipation.
This methodology exploits the availability of some tools, developed to facilitate the settings:

- Position Loop Adjustment;
- Ball.Bar;
- Contour Accuracy.

(Details about the tools, and how to use them, can be found on 00044EN- 02 Flexium+ Commissioning manual - Chapter 6 § 6.3 - 6.4 - 6.5).

2.2 Prerequisites

High machining feedrates and high contouring precision are conflictual requirements.

In order to find the right trade-off between them, CNC and drives have to be tuned properly.

So, prerequisite to HSM setting are:

- Proper tuning of interpolated axes control loops (position loop, speed loop, anti-resonance filters, etc…)
- Proper tuning of feedrate handling along programmed trajectory, by CNC (jerk, corner crossing speed, etc…)

For all details about parameters cited in the following paragraphs, please take a look to M00041EN-01 Flexium+ Parameter manual

2.2.1 Prerequisite: choice of interpolated axes' position loop time constants

In order to minimize contouring errors, it is necessary to tune all axes' servo-drive control loops to their best performances, having checked that no servo-drive control loop saturation occurs.
An insight on how to tune optimally the axis' speed loop can be found in M00044EN- 02 Flexium+ Commissioning manual Chapter 6 § 6.8 related to "Auto-tuning".
For an insight on how to tune optimally the axis' position loop see paragraph 6.5 of the above cited manual, related to "position loop adjustment".

Associated with the position loop gain, there are two important entities, useful in tuning both, traditional and HSM, machining:

- The Lag error,: given by the relationship

$$\text{Lag error [mm]} = \frac{\text{Feedrate [mm/min]}}{\text{Position loop gain [mm/min/mm]}}$$

- The position loop time constant, given by the relationship

$$\textbf{P56 -} \text{ Position loop time constant [ms]} = \frac{60'000}{\textbf{V301 -} \text{ Position loop gain [mm/min/mm]}}$$

Position loop time constant will be one of the most important parameter used to compute HSM speed and acceleration anticipation throughout this document.

Once we have optimally tuned the speed loop of each axis, usually, in order to perform good machining in case NO HSM is involved, we have to "align" all position loop gains, of each interpolated axis, to the same value, that in order to keep the right geometry on path (straight line are followed without contouring error, and circles remain circles, even if a little bit smaller…)

Such strategy is called "position loop alignment"
The position loop gain common to all interpolated axes (channel position gain) must be chosen equal to the position loop gain of the less performant axis.

In case of HSM is involved, then we can choose between two different strategies:

1. **First strategy – Position loop alignment of interpolated axes, as for traditional machining:**
   - We set the position loop gain of each interpolated axis to the same value of the one with lowest gain;
   - We set the CNC parameter P56 (channel specific time constant) to the position loop time constant corresponding to the chosen common position loop gain;
   - We set to null all the words of CNC parameter P66 (time constant for axis).

   Advantages:
   - Easiness of tuning and handling;
   - Machining can be performed even deactivating HSM functionalities, without the need to change neither drive nor CNC parameters.

   Drawbacks:
   - Performances of axes are artificially reduced to the less performant one.

2. **Second strategy – Different position loop gain for each axis: each axis at its best performances:**

   - We set CNC parameter P66 (time constant for axis) to the position loop time constant of each interpolated axis;
   - We set CNC parameter P56 (channel specific time constant) to the position loop time constant corresponding to the lowest position loop gain;

   Advantages:
   - Each axis works at its best performances

   Drawbacks
   - Traditional machining cannot be performed without HSM functionalities active.

   HSM functionalities may work correctly with both described strategies.
   Obviously, HSM tuning will depend upon the adopted position loop strategy, as we will see later.

   In order to compute all needed HSM anticipations, the CNC will operate as follows:

   - In case P66 Nxx = 0
     (axis "XX" specific position loop time constant is null).
     Then the CNC will use common channel position loop time constant P56

   - In case P66 Nxx not null
     (axis "XX" specific position loop time constant different from 0). Then the CNC will use axis specific position loop time constant P66 Nxx

2.2.2 Prerequisite: optimal handling of feedrate on programmed trajectory.

As previously said, HSM techniques reduce the Lag error to an almost null value, thus obtaining an almost null deviation from programmed path, but, so doing, the healthy smoothing effect, introduced by Lag error as a side effect, is almost lost: it is, so, a CNC job to handle properly the feedrate along the programmed path, slowing down whenever abrupt change of direction or of curvatures occurs, in order to reduce shocks on machine and machine wearing as well.

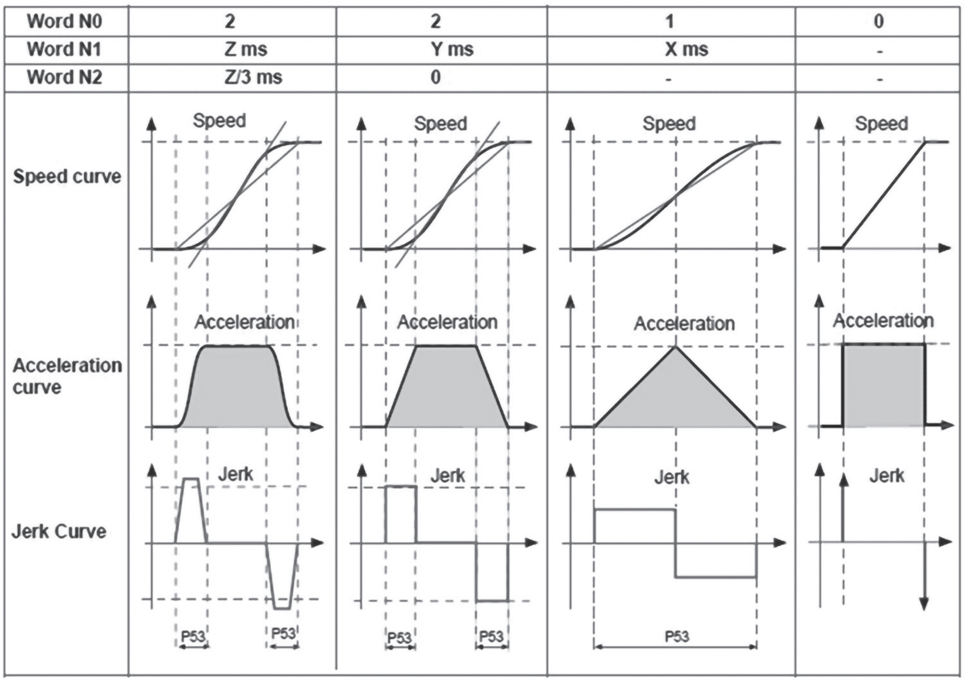
In the following paragraphs, we will describe main functionalities that allow proper feedrate handling by CNC.

Again, see M00041EN- 01 Flexium+ Parameter manual for all details about parameter cited in the following paragraphs, and take a look to M00040EN- 04 Flexium+ Programming manual for what concerns all cited external parameters Exxxxx.

2.2.2.1 Acceleration and jerk handling

In high-speed machining operation, it is of greatest importance to use gradual acceleration when varying the velocity along the path to limit the jerk on the machine. Jerk is the time rate of change of axis acceleration/deceleration and can be used to minimize mechanical wear and tear, reduce the contour error on curves, limit the synchronization errors on portal- and gantry-like machines, etc. The jerk is expressed in m/s^3; but to facilitate comprehension of this physical phenomenon it is preferable to express it as an acceleration build-up time.

The jerk limitation is entered with CNC parameter P53."Acceleration profile".
Its meaning depends upon the type of gradual acceleration you have selected with such parameter: Several acceleration and jerk profiles are available, by mean of P53 N0, N1 and N2 setting, as depicted in figure of M00041EN- 01 Flexium+ Parameter manual



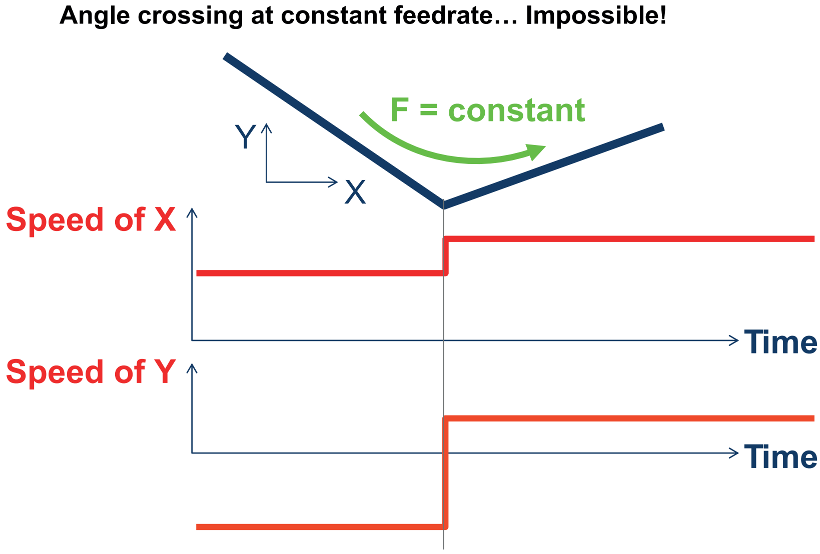
In case of HSM, the recommended profile is the one corresponding.
P53 N0 = 2
P53 N1 = 2 or 3 times the common channel position loop time constant (P56)
P53 N2 = 1/3rd of P53 N1.

The trapezoidal jerk profile allows a higher maximum acceleration (P32) to be set since the machine is less sensitive to acceleration effects because acceleration is gradual.

Obviously, by smoothing the acceleration profile (increasing P53 N1 and consequently N2) increases the machining time. To reduce the cutting time it is often preferable to increase the maximum acceleration (P32) within the limit of the machine performance rather than decreasing the jerk and acceleration build-up time.

2.2.2.2 Crossing-angle speed (approach speed)

Whenever a corner (angle on trajectory) is encountered along the programmed path, then it is impossible to execute it at constant feedrate without getting a contouring error and mechanical shock.

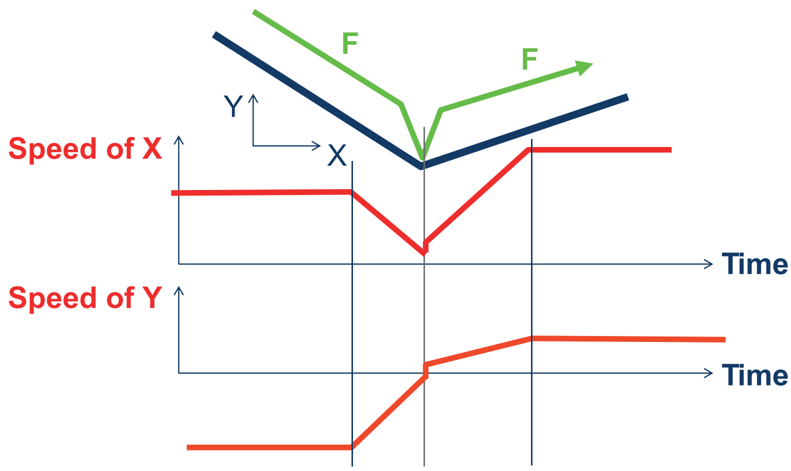


In order not to produce contouring error at angle crossing and to minimize mechanical shock, the feedrate should be reduced up to zero speed, at the corner; but so doing, the machining time would get longer.

CNC has, so, to find a proper trade-off between the two conflicting requirements: speed of machining and precision of contouring.

CNC parameter P33 (approach speed) allows the user to specify the maximum speed step at corner crossing for each interpolated axis.
Whenever CNC crosses an angle along the programmed path, then the feedrate is reduced in such a way that the axis subject to the greatest speed instant variation will experience a speed step not greater than corresponding word Nxx of parameter P33.

Feedrate reduction at angle crossing



The angle crossed at the reduced feedrate will produce an acceleration peak on all the involved axes, that, in case of NO HSM functionality active, will not exceed the value obtained from the formula below:

$$\text{Acceleration peak [mm/s}^2\text{]} = \frac{\text{16,67 * P33 approach speed [mm/min]}}{\text{Axis position loop time constant [ms]}}$$

That is due to the “Smoothing” side effect of the servo-drive position loop control based on Lag error.

As said above, in case of any HSM functionality active, such “smoothing” effect is lost, so the acceleration peak could be ten (10) times higher as well.
That’s why, in case of HSM, approach speed (P33) should be reduced with respect to the traditional machining, by an order of magnitude (10). Also it is necessary to check that the torque current of each axis never saturates due to such peaks of acceleration.

A useful tool to set up properly the approach speed P33, in both cases, with and without HSM functions active, is the “position loop adjustment” described in 00044EN- 02 Flexium⁺ Commissioning manual - Chapter 6 § 6.5

2.2.2.3 Definition of a curve by small elements

Very often ISO part-programs contain a succession of linear segments describing a polygonal path, with tangential discontinuities, in which case the feedrate has to be reduced in order to keep precision, as well as a succession of short linear segments representing the approximation of a curve approximated by a sequence of small elements.

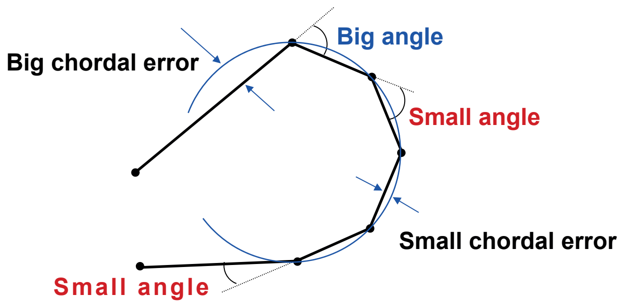
In this latter case the feedrate has to be kept constant despite the numerous angular points crossed along the approximated curvilinear path.

The CNC possess proper parameters to discriminate between polygonal path and approximated curvilinear path.

Basically a sequence of linear segment is considered, by the CNC, as a “curve approximated by small segments” if and only if the following conditions are all satisfied at the same time:

- The chordal error on trajectory is smaller than CNC external parameter E32004
- The angle between segments is smaller than CNC external parameter E32003

If such conditions are satisfied, then the path is considered as a “curve approximated by small segments” and the CNC tries to execute it at programmed feedrate, keeping into account all considerations about curvature and speed limitations due to error on curvatures (see next paragraphs).



Estimates the chordal error along the curvilinear path approximated by small segments, by using the following relationship:

$$\text{Chordal error [mm]} = \frac{\text{Angle between small segments [rad] * small segment length [mm]}}{8}$$

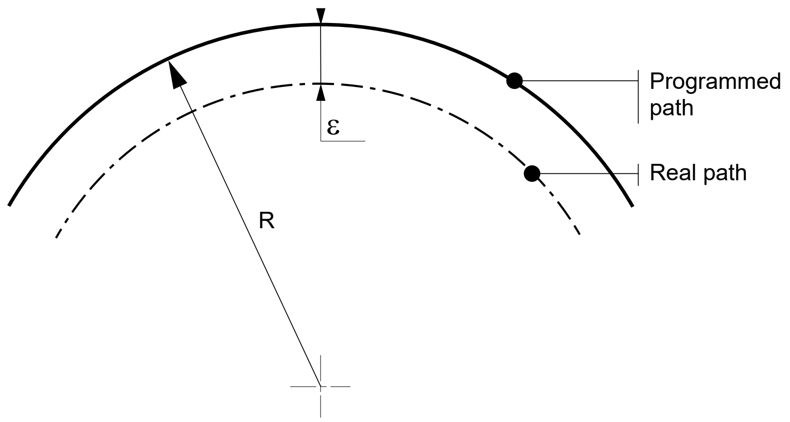
2.2.2.4 Radial error on curvilinear path

As described in the introduction, during machining of curvilinear paths, the “Lag error” of servo-controlled position loop causes a contouring error, that is bigger at higher feedrates.

In order to keep such error limited the feedrate has to be reduced.

CNC parameter P52 defines the maximal radial error tolerated on circles. Indeed it is applied to any curvilinear path (even approximated by small linear segments) based on its estimated curvature radius.

P52 Word N0 Radial error tolerated on circles.



$$\varepsilon = \frac{1}{2} \quad \frac{(F \cdot T / 60'000)^2}{R}$$

ε = Radial error [mm]

T = P56 common channel position loop time constant [ms]

F = Feedrate [mm/min]

R = Radius [mm]

In case of HSM active, then, such feedrate limitation could be relaxed due to the fact that the Lag error is almost null.

2.2.2.5 Radial acceleration management on curvilinear path

Due to possible elasticity of mechanical system of a CNC controlled machine, it is possible that, during machining at high feedrates, the centrifugal forces applied to mechanical structure, cause this latter to deform itself, producing an error of the real profile with respect to the desired profile.

Also it is possible that high centrifugal forces can cause mechanic to be damaged.

In order to limit such contouring error, and/or damages, the CNC is able to modulate the feedrate along curvilinear path in order to limit radial acceleration of axes to a percentage of the defined maximal axis acceleration (P32).

Such feedrate modulation is accomplished via CNC external parameter E32010.

The feedrate scheduling by CNC limits the feedrate on curves on the basis of the maximum acceleration on the machine's axes and the curve radius so that the radial acceleration does not exceed the following limit:

$$\text{Acceleration limit of axis "XX" on curvilinear path [mm/s}^2\text{]} = \frac{\text{P32 Nxx [mm/s}^2\text{] * E32010}}{256}$$

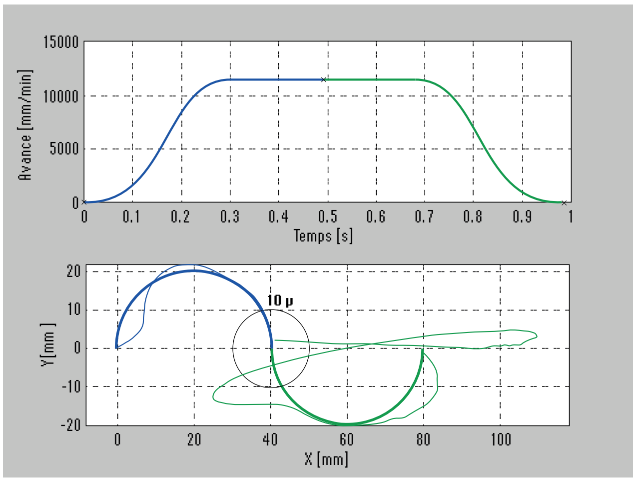
This limitation applies to arcs, polynomial curves and curves made of small line segments.

2.2.2.6 Speed management at inflexion point

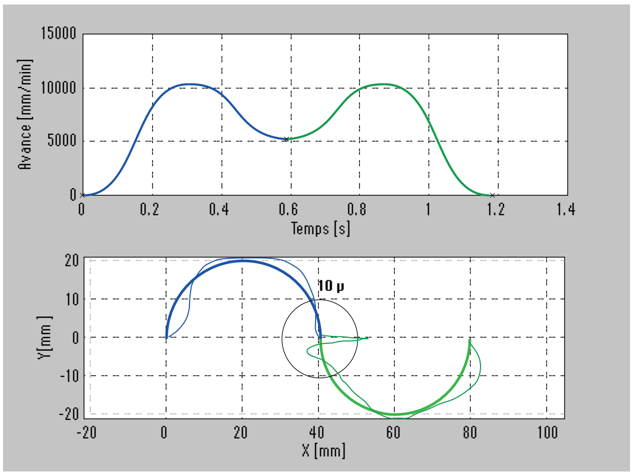
A sequence of two curves with different radii (even when the curves are tangent) results in a sudden change in radial acceleration. This may cause a path error due to the reaction of the servo system, especially in high-speed machining operation, because of cancellation of the following error by complete feed-forward control. To decrease the error, it is necessary to slow down at the curve inflexion point.

The CNC limits the feed rate according to external parameter E32009, which specifies the accepted contour error at inflexion point.

The following two figures illustrate the need to reduce the speed at a curvature inflexion point in order to limit the effect of the normal acceleration step on the contour error at the transition point.



Crossing an inflexion point (G2 → G3)
without *speed limit → Too large contour error.*



Crossing an inflexion point (G2 → G3)
with *speed limit → Acceptable contour error.*

When E32009 is equal to zero (default setting), curve radius variations are not analyzed.

E32009 has an effect ONLY in the following cases:

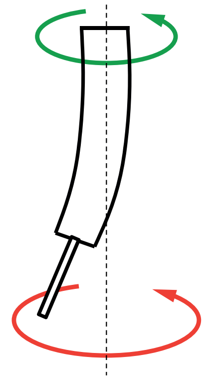
- circle-circle;
- circle-line;
- circle-polynomial;
- polynomial-polynomial;
- polynomial-line transition.

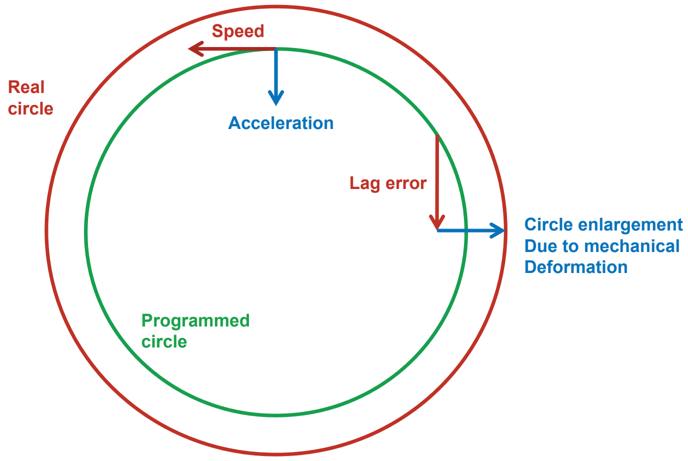
2.3 High Speed Machining according to NUM

HSM is mainly based on three basic actions:

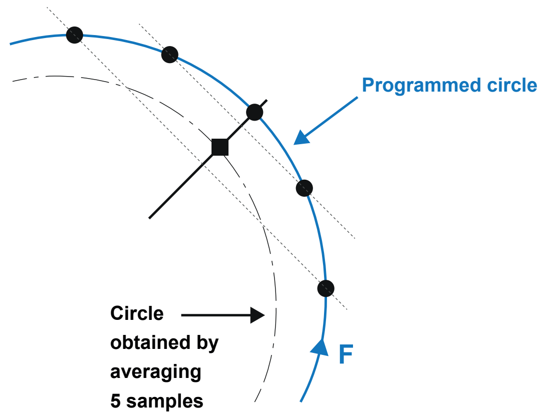
1. Speed anticipation;
   - Is needed to cancel the Lag error in order to achieve precision.

2. Acceleration anticipation;
   - Theoretically, such anticipation should be useful only to compensate for possible mechanical deformation under centrifugal forces during machining, due to unavoidable elasticity of the mechanics.
   - Practically, sometimes, acceleration anticipation is needed to compensate for some HSM implementation computational details.





3. Filering of the "Position reference signal".
    - It is necessary whenever machining curves by small segments in order to obtain a smooth surface instead of a lot of polygonal facets.
    - The use of reference filtering shifts the circular path to the inside of the circle in an effect similar to increasing the acceleration anticipation. Here follows an example of filtering by a simple average of 5 position reference samples:



Depending on the use and / or setting of these actions,NUM distinguishes between THREE main strategies of HSM:
- Simple speed feedforward (speed FFWD);
- Total anticipation High Speed Cutting (total HSC);
- Smart Servo Balance (SSB).

2.3.1 Description of the various kinds of HSM

As previously said, depending on the use and / or setting of speed and acceleration anticipations and position reference filtering, NUM distinguishes between THREE main strategies for High Speed Machining:

2.3.1.1 Simple speed feedforward (speed FFWD)

- Only speed anticipation is used
- Speed anticipation can be freely chosen between 0% and 100%
- It is possible to use such HSM scheme even with different position loop gain for each interpolated axis of the channel (only if speed anticipation = 100%).

Speed FFWD should be used in case of

- Small and compact machines, without mechanical deformation due to centrifugal forces, because of the missing acceleration anticipation;
- Machining that does not require any position reference filtering;
- Easiness of tuning.

2.3.1.2 Total anticipation High Speed Cutting (total HSC)

- It uses all type of anticipations:
  - Total Speed anticipation, fixed to 100%
  - Acceleration anticipation
- It allows also the insertion of a position reference filtering, by mean of a simple average, choosing the number of position reference samples involved in averaging:
  - 1 = no filtering
  - 2..13 = filtering active on 2..13 samples average.
- It is possible to use such HSM scheme even with different position loop gain for each interpolated axis of the channel;
- Such HSM scheme needs proper license option **(FXO000155)** to be used.

Total HSC can be used in any case, It is the most flexible NUM high speed machining scheme, but it has to be remembered that:

- It requires proper license option;
- Filtering of position reference is limited to the average of a maximum of 13 samples, it does not allow a very strong filtering.

2.3.1.3 Smart Servo Balance (SSB)

- It allows for both type of anticipations:
  - Speed anticipation.
  - Acceleration anticipation.
- It allows for the insertion of a special position reference filter, that behaves exactly like a virtual position loop with desired (parametric) position loop gain;
- It is possible to use such HSM scheme even with different position loop gain for each interpolated axis of the channel.

SSB (smart servo balance) should be used in case of:

- Machining does not require to work with null Lag error;
- Machining requires strong smoothing of position references (ex.: noisy part-programs generated via probing of a workpiece sample, or poor CAD/CAM systems).

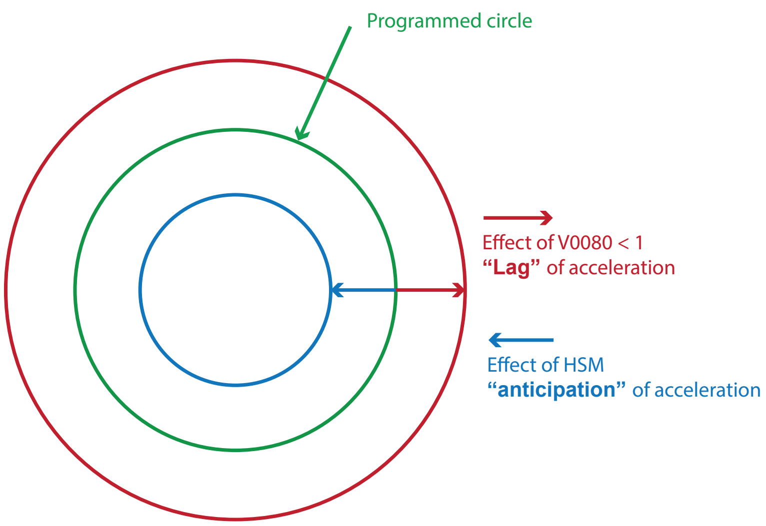
Here follows a table summarizing the characteristics of each NUM HSM scheme:

	Simple speed ffwd	Total HSC	SSB
License required	NO	**YES (FXO000155)**	NO
Speed ffwd	From 0% to 100%	Only 100%	Free, parametric
Acceleration ffwd	**NO**	Free, parametric	Free, parametric
Pos. Ref. Filtering	**NO**	Yes Small filtering	Yes Even strong filtering possible
Null Lag error	Yes If speed ffwd = 100%	Yes	Only possible if NO filter is used
Null error on profile (contour accuracy)	Only possible if NO mechanical deformation.	Yes Almost	**NO** Only almost possible if NO filter is used

Note that, due to flexibility of Flexium system (NCK + drive), it is possible to have a “fourth” HSM scheme:

Partial anticipation High Speed Cutting (partial HSC):

- It uses a **partial speed anticipation (less 100%) transforming the Lag error (dragging error) into a pure-delay, that does not produce any contouring error**;
- But, due to pure-delay between axis position reference and axis actual position, the “Lag error” displayed on HMI is no more null;
- It is possible to work with **null acceleration anticipation**, by exploiting **drive parameter V0080** (speed loop set-point-weighting); in fact, what drive parameter V0080 implements, is a sort of “acceleration Lag”, that operates in an opposite way with respect to acceleration anticipation, as shown in figure below.



- V0080 = 1.0 NO "Lag" of acceleration
- V0080 < 1.0 "Lag" of acceleration [ms] = [(1 – V0080) * V0052 [ms]]

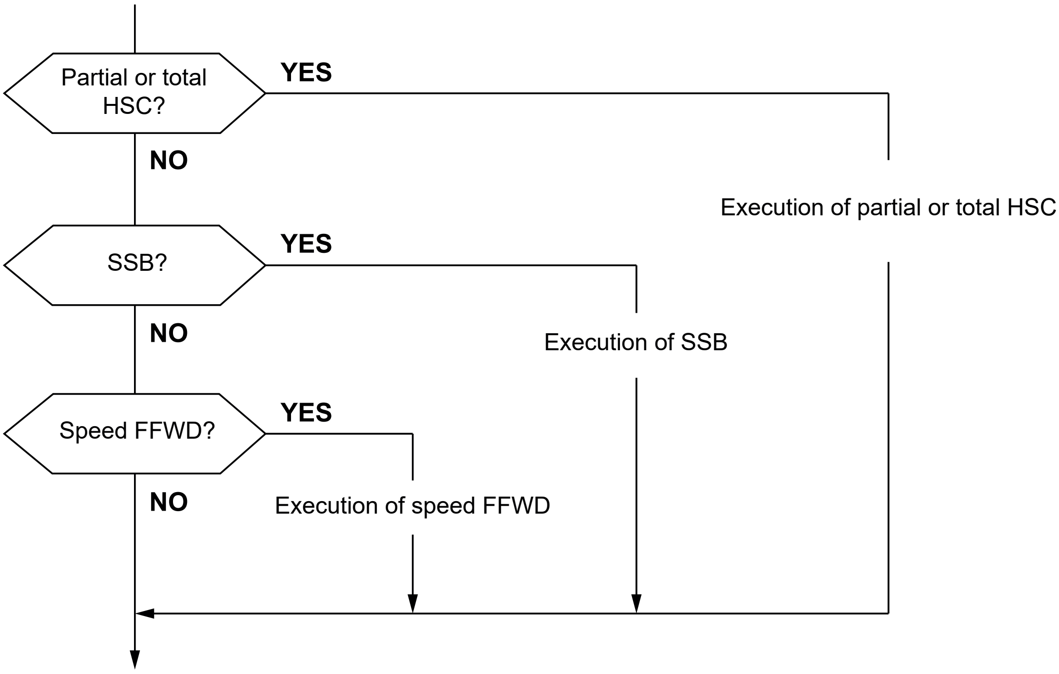
Advantages of such partial HSC versus the total HSC scheme are:

- Reduced anticipations (partial speed and null acceleration) produces less mechanical stress;
- Even if HMI displayed Lag error is not null, it is anyway possible to obtain an almost null error on profile, due to the fact that the effective Lag error has been transformed in a pure-delay between reference and actual axis position;
- It is possible to obtain contour accuracy performances even better than total HSC.

2.3.2 Rules of precedence between HSM types

Be careful: only one HSM type can be used at a time.

Here follows the “precedence logic” used by NUM

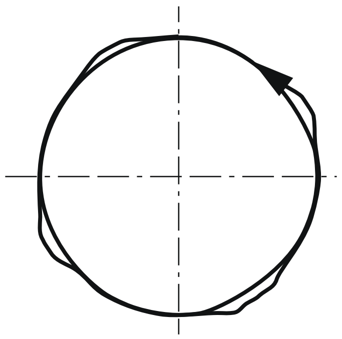


2.4 Dry friction correction (Anti-Pitch)

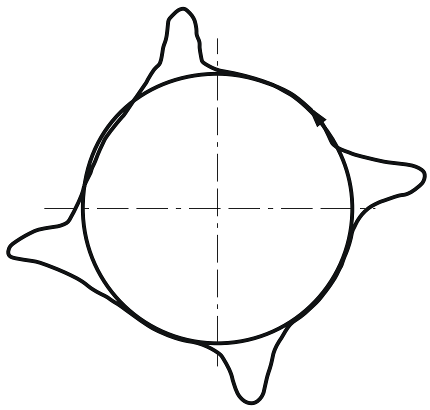
Aside of HSM configuration, it is possible to adjust the parameters related to the “dry friction correction” function, also known as “anti-pitch function:

Description

"Pitch" is an error occurring on "Ball-Bar" type traces that results in spikes pointing outward from the circle just after changing quadrants.



Real path with enlargement of the straight section



Display of the straight section on the polar trace of the radial error (BALL-BAR)

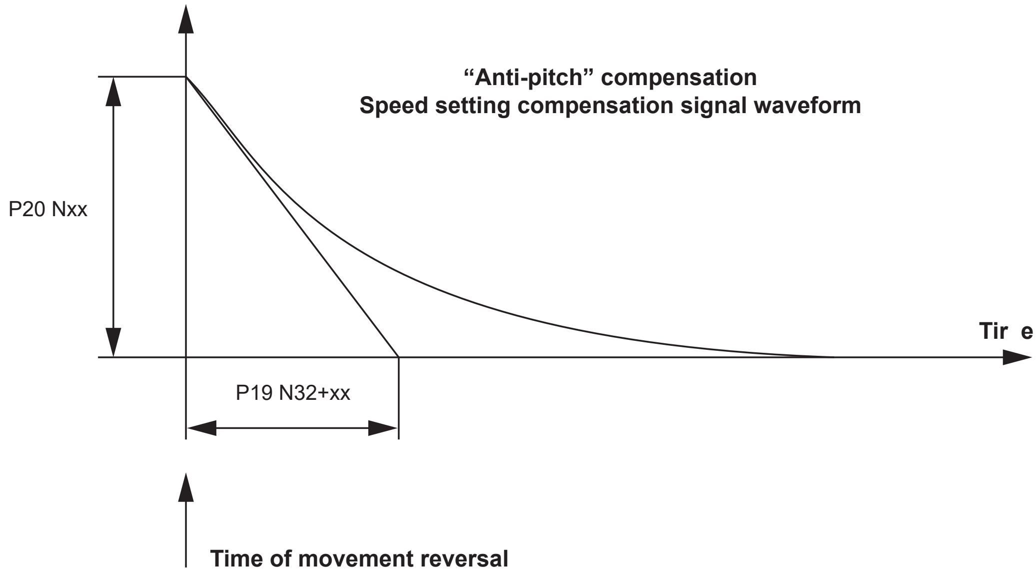
These trace errors actually identify the presence of straight sections in the real path which occur after changing quadrants. These straight sections amplified by the "Ball-Bar" trace appear as spikes on the screen.

These deformations are generally explained by the fact that when the direction of movement on an axis is reversed, a discontinuity occurs in the inertia component due to dry friction.

This component changes at that time from a finite value in one direction to a finite value in the other. The spikes are a consequence of this discontinuity. It is as though the servo-drive did not correct the effects of the discontinuity fast enough.

The principle used to compensate for this error was to accelerate movement on the axis whenever a direction reversal occurs, by "injecting a disturbance" in the feed rate setting. This disturbance is called "dry-friction compensation" or "anti-pitch" signal, sketched below.

The anti-pitch signal level decreases exponentially with time. Its waveform is defined by two CNC parameters: P20 dry friction compensation amplitude and P19 dry friction correction time constant (see M0041EN-01 Flexium+ parameters manual for details).



2.5 List of useful parameters & strictly related HSM parameters

Table of parameters related to optimal setting of feedrate management along the path (see M0041 Flexium+ parameter manual and M0040 Flexium+ programming manual)

Parameter	Description
P56 or E41006	Channel specific time constant. It is related to channel common position loop gain via the following relationship: time constant [ms] = 60’000 / position loop gain [mm/min/mm] It is also related to Lag error via the relationship: Lag error [mm] = feedrate [mm/min] / position loop gain [mm/min/mm] = = feedrate [mm/min] * time constant [ms] / 60’000
P66	Time constant for axis. Same thing as P56, but specific for each single axis. If different from 0 (zero) then it supersedes P56 of the channel in all feedforward HSM computations
P53 or E11013 E32006 E32008	Acceleration profile. It allows smooth acceleration and jerk variations whenever needed (ex. angle crossing, small radius curvatures, etc.), along the path.
E11015 E32003 E32004	Parameter used to distinguish a “curvilinear path made by small G1 segments” from a “real polygonal path”. Speed limitations are applied whenever the angle between G1 segments exceeds E32003 (if E11015 is enabled) or the “chordal error” of the supposed “curve path by small G1 segments” exceeds E32004. “chordal” error along a path is approximated by the following formula: chordal error [mm] = G1 segment length [mm] * angle [radians] / 8
P33 or E97300..31	Approach speed. It is the maximum permissible speed variation which is executed without feedrate reduction along the path.
P52 or E32002	Radial error tolerated on circles. Maximum permissible radial error on curvilinear paths. Feedrate on curvilinear paths is reduced in order not to exceed maximum tolerated error. It is applied also on “curvilinear path by small G1 segments”. It is always computed based on common channel time constant P56.
E32010	Radial acceleration (percentage of modulation). It is used to reduce feedrate along curvilinear paths in order to limit radial acceleration to a fraction of P33 (maximum axis acceleration).

E32009	Maximum tolerated profile error at curvature variations feedrate along curvilinear paths is reduced whenever an abrupt curvature is encountered (eg. G2-G3 consecutive blocks) in order not to exceed maximum tolerated error. It is computed based on the common channel time constant P56.

Table of parameters related to anti-pitch (dry friction correction) function (see M0041 flexium⁺ parameter manual and M0040 Flexium⁺ programming manual)

Parameter	Description
P19 N32..N63 or E98300..31	Dry friction correction time constant. It is the time constant specific of the anti-pitch function.
P20 or E98200..31	Dry friction correction amplitude. It is the amplitude of the correction applied by the anti-pitch function.

Table of parameters strictly related to HSM functions
(see M0041 Flexium+ parameter manual and M0040 Flexium+ programming manual)

HSM type	Parameter	Description
Simple speed feedforward	P55 N0..N7 or E98500..31	Speed feedforward percentage. From 0% to 100% It is specific for each single channel. Related Exxxxx parameters are specific for each single axis. Speed anticipation calculation is based on: - P56 channel specific time constant, if and only if P66 of the axis is 0 (zero); - P66 of the related axis, if and only if it is different from 0 (zero):
Total HSC & Partial HSC	P55 N8..N15 or E11012 E32005	Number of position reference samples involved in smooth filtering. It applies a simple moving average filter on N samples (P55) of position reference before applying speed and acceleration anticipation. It is specific for each single channel. E11012 allows to quickly activate and deactivate total and partial HSC by part-program ISO command.
	P19 N0..N31 or E98100..31	Acceleration anticipation time constant. It is specific for each single axis. All anticipation calculation is based on: - P56 channel specific time constant, if and only if P66 of the axis is 0 (zero); - P66 of the related axis, if and only if it is different from 0 (zero).
	E98532..63	Partial Speed anticipation percentage. It allows to use a fraction of total speed anticipation: from about 0% to 100% It is specific for each single axis. Speed anticipation calculation is based on: - P56 channel specific time constant, if and only if P66 of the axis is 0 (zero); - P66 of the related axis, if and only if it is different from 0 (zero).
SSB	E92300..31	Activation of SSB on a specific axis.
	E98600..31	Lead time constant. It is the speed anticipation time constant.
	E98632..63	Lag time constant. It is the position reference filter time constant
	E98664..95	Acceleration anticipation time constant

Table of drive parameters related to HSM functions
(see M0031 NUMdriveX parameter manual and M0040 Flexium+ programming manual)

Drive Parameter	Description
V0080	Speed loop set-point weighting. Used to adjust acceleration “Lag” to fit ball-bar and contour accuracy needs.
V0052 or V1052 if “drive parameter set 1” is active…	Integral speed loop gain. It is the time constant of speed loop integral controller (PI). It is involved in the acceleration “Lag” together with V0080.

2.6 HSM tuning

2.6.1 Choice of HSM method to use: simple speed FFWD, total/partial HSC, SSB.

First of all, the HSM strategy must be defined, based on the following criteria:

- Do we need smoothing / filtering of position reference?
- Do we need to work with null Lag error?
- Do we need to compensate for mechanical deformation during HSM?
- Is NC cycle time (P50) comparable with position loop time constant of interpolated axes?

Here is a table that summarize which HSM kind is more suitable for user needs:

User need	Simple speed FFWD	HSC total / partial	SSB
Smoothing is necessary	**NO** No filtering possible.	YES Light filtering	YES Heavy filtering
Absolute need for null Lag error!	YES	YES	Yes only if NO filtering
Compensation of mechanical deformations due to centrifugal forces.	**NO** Due to missing acceleration ffwd	YES	YES
P50 comparable with position loop time constant	-	Do NOT use if 2*P50 is greater or equal to axis position loop time constant	-

2.6.2 Simple speed FFWD

Useful for its easiness of tuning and use.

It does not require any License option.

Its tuning consists of:

- Setting channel speed anticipation to 100% (see P55)
- Setting drive parameter V0080 of each interpolated axis to the following value:

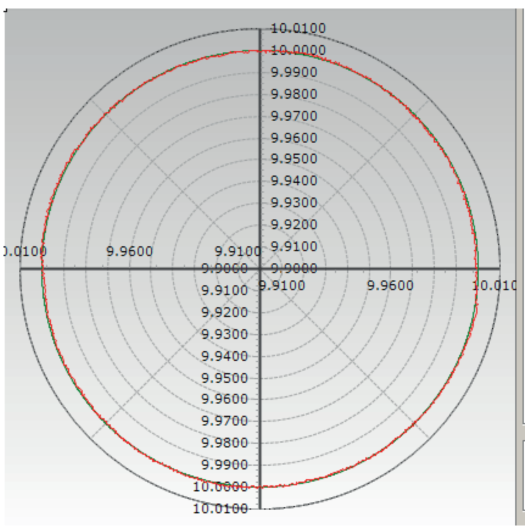
$$V0080 = 1.0 - \left(\frac{P50\ [ms]}{2 * V0052\ [ms]} \right)$$

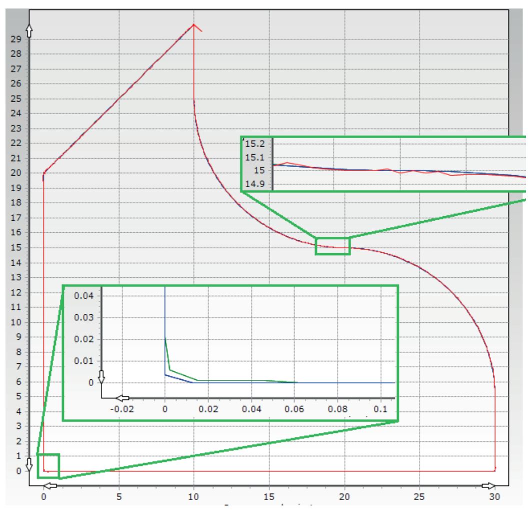
  Where:
  - V0052 is the “integral speed loop gain” drive parameter;
  - P50 is the NC parameter “sampling period”.
- Adjustment of drive parameter V0080 by mean of “ball-bar”
  - Increasing V0080 reduces real circle radius
  - Decreasing V0080 enlarges real circle radius

Example of speed FFWD tuning, with the aid of Ball-Bar and Contour accuracy tools, In case of:

Parameters	Descriptions
• P50 = 2 ms	Sampling period
• P56 = 20 ms	Channel specific time constant
• P66 Nxx = 0	Time constant per axis
• V52 of X and Y axis = 10 ms	“integral speed loop gain” drive parameter

Ball bar	R = 10 mm	F = 6’000 mm/min	Contour accuracy	
			Blue =	Programmed trajectory
Green =	Programmed circle		Green =	Actual trajectory
Red =	Actual circle		Red =	Profile error 40 times magnified
After optimization of V0080			After optimization of V0080	
V0080 = 0,87			V0080 = 0,87	





2.6.3 HSC total anticipation

It is the most flexible HSM mode, but it requires proper License option **(FXO000155)**.

Tuning procedure consists of:

1. Activate total anticipation HSC, by writing 1 (one) in the number of filter terms (see P55)
   From now on, the speed anticipation will be set to 100%.
2. Set acceleration anticipation (P19).

   $$P19\ Nxx = \frac{3}{2} * P50$$

   Where P50 is the NC parameter "sampling period"

3. Use "ball-bar" to tune properly the acceleration anticipation (see P19)
   - Increasing P19 reduces real circle radius
   - Decreasing P19 enlarges real circle radius.

4. Choice of position reference filtering:
   - **If NO filtering**;
     - The tuning procedure is terminated !

   - **If filtering has to be applied**;
     - Then set the number of filter terms to an odd number from 3 to 13 included.
       Best results are obtained if the following condition is respected:
       N*P50 [ms] ≤ position loop time constant [ms]

       Where:
       - P50 is the NC parameter "sampling period"
       - N is the number of filter terms (see P55)
       - Position loop time constant is given by P56 or P66 (if different from 0) (see paragraph …)

     - Obviously, after such setting the real circular path of "ball-bar" will be inside the theoretical (programmed) circle, but anyway we have NOT to modify the acceleration anticipation obtained with N = 1, in order not to make smoothing effect vanish!!

Here are example images of total HSC tuning, with the following parametrization:

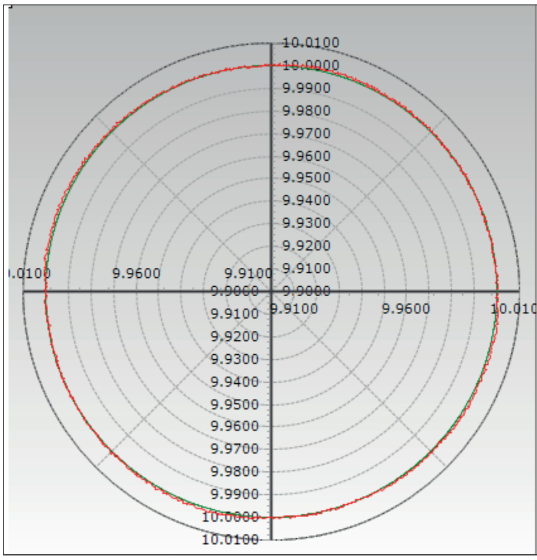
Parameters	Descriptions
• P50 = 2 ms	Sampling period
• P56 = 20 ms	Channel specific time constant
• P66 Nxx = 0	Time constant per axis
• V52 of X and Y axis = 10 ms	"integral speed loop gain" drive parameter

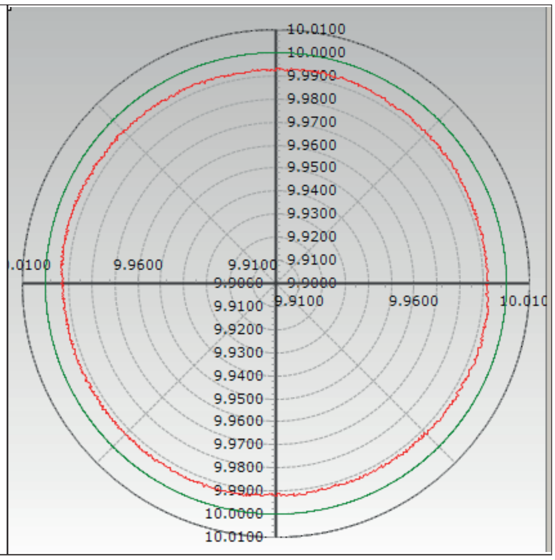
Ball bar	R = 10 mm	F = 6’000 mm/ min

Green = Programmed circle

Red = Actual circle

Final setting without filtering	Final setting with filtering
P55 N8 = 1 (1 sample average)	P55 N8 = 7 (7 samples average)
P19 N0 (acceleration anticipation) = 3050 us	P19 N0 (acceleration anticipation) = 3050 us





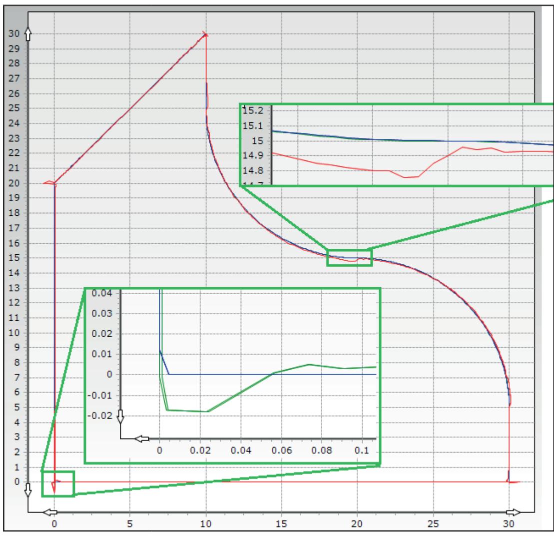
Contour accuracy

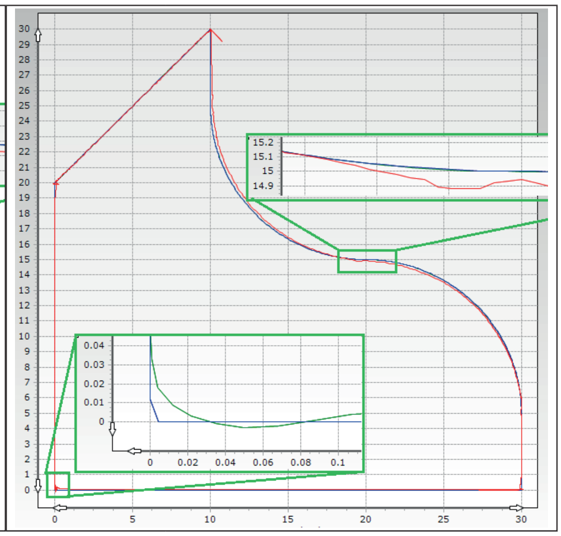
Blue = Programmed trajectory

Green = Actual trajectory

Red = Profile error 40 times magnified

Final setting without filtering	Final setting with filtering
P55 N8 = 1 (1 sample average)	P55 N8 = 7 (7 samples average)
P19 N0 (acceleration anticipation) = 3050 us	P19 N0 (acceleration anticipation) = 3050 us





2.6.4 Partial HSC

It is very similar to total HSC, except that it does not use acceleration anticipation and only a partial speed anticipation.

It needs the same license option of total HSC **(FXO000155)**.

Such partial HSC has a more complex tuning procedure, but is able to get better performance, and, by reducing the total amount of anticipations, produces less mechanical stress and wearing with respect to total HSC modality.

Tuning steps are:

1. Activate total anticipation HSC, by writing 1 (one) in the number of filter terms (see P55).
2. Use the following formula to compute the partial speed anticipation for each interpolated axis:

$$E98532 + xx = 256 * \left(\frac{\text{Position loop time constant [ms] - (2 * P50 [ms])}}{\text{position loop time constant [ms]}} \right)$$

   - Position loop time constant is given by P56 or P66 (if different from 0) (see paragraph 2.2.1).
   - With such setting, the Lag error displayed by HMI, even at constant speed, will be no more null, but the obtained machining will be smoother as desired.

3. Set to NULL (zero = 0) the acceleration anticipation of each interpolated axis (see P19). In fact such kind of HSM does not use acceleration anticipation!
4. Set drive parameter V0080 of each interpolated axis to the following value:

$$\text{Set Point Weight speed loop gain} \quad V0080 = 1.0 - \left(\frac{\text{P50 [ms]}}{\text{2 * V0052 [ms]}} \right)$$

   Where:
   - V0052 is the “integral speed loop gain” drive parameter;
   - P50 is the NC parameter “sampling period”.
5. Fine adjustment of drive parameter V0080 by mean of “ball-bar”.
   - Increasing V0080 reduces real circle radius.
   - Decreasing V0080 enlarges real circle radius.
6. set position reference filtering, if needed.
   - **If NO filtering**;
     - The tuning procedure is terminated!
   - **If filtering has to be applied;**
     - Then set the number of filter terms to an odd number from 3 to 13 included:
       Best results are obtained if the following condition is respected:
       N*P50 [ms] ≤ common channel position loop time constant P56 [ms]
       Where:
       - P50 is the NC parameter “sampling period”
       - N is the number of filter terms (see P55)
     - Obviously, after such setting the real circular path of “ball-bar” will be inside the theoretical (programmed) circle, but anyway we have NOT to modify the V0080 settings obtained with N = 1, in order not to make smoothing effect vanish!!

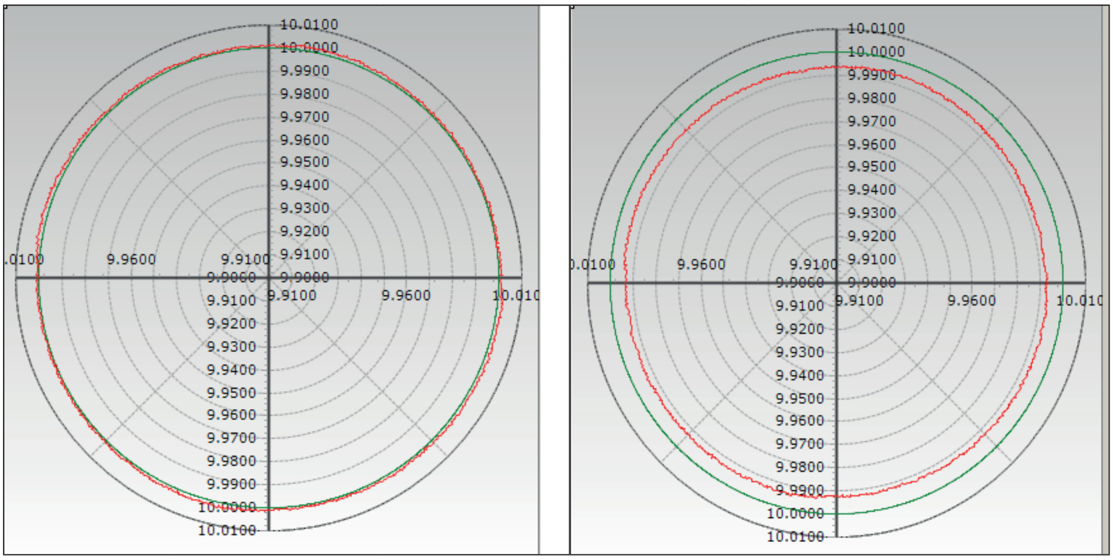
Here are example images of total HSC tuning, with the following parametrization:

Parameters	Descriptions
• P50 = 2 ms	Sampling period
• P56 = 20 ms	Channel specific time constant
• P66 Nxx = 0	Time constant per axis
• V52 of X and Y axis = 10 ms	“integral speed loop gain” drive parameter

Ball bar	R = 10 mm	F = 6'000 mm/ min
Green =	Programmed circle	
Red =	Actual circle	

Final setting without filtering
P55 N8 = 1 (1 sample average)
V0080 = 0,87

Final setting with filtering
P55 N8 = 7 (7 samples average)
V0080 = 0,87

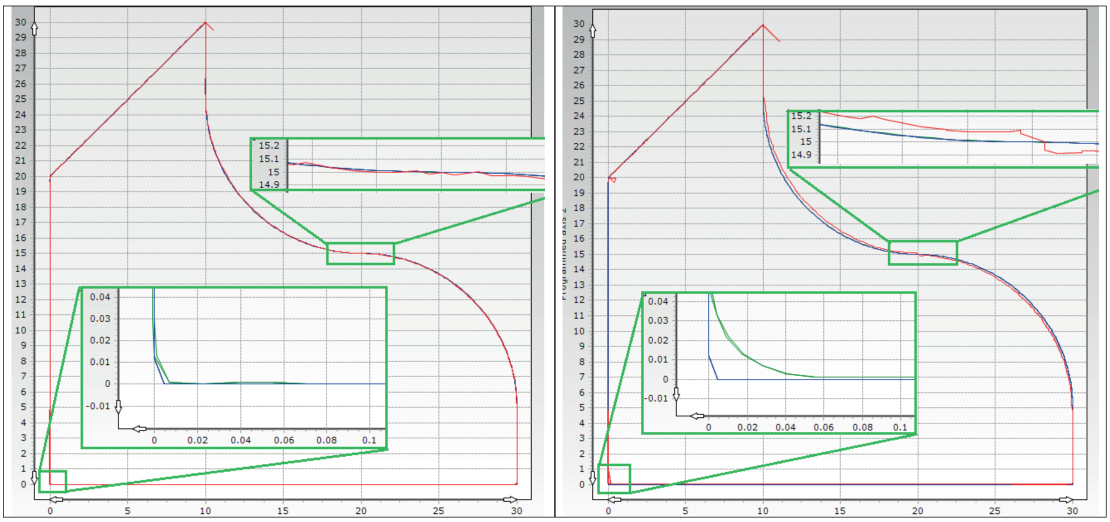


Contour accuracy

Blue =	Programmed trajectory	
Green =	Actual trajectory	
Red =	Profile error	40 times magnified

Final setting without filtering
P55 N8 = 1 (1 sample average)
V0080 = 0,87

Final setting with filtering
P55 N8 = 7 (7 samples average)
V0080 = 0,87



2.6.5 SSB smart servo balance

Such HSM mode does NOT require any License option.

It is able to adopt much heavier filtering than total / partial HSC.

The tuning consists in the following steps:

- Set “lead time constant” of each axis at the same value of its position loop time constant:
  - E98600 Anticipation time constant + xx = 100 * position loop time constant [ms]
  - Position loop time constant of each axis must be copied into P66, even if they are equal to P56.
- Set to NULL all acceleration anticipation:

  E98664 + xx = 0
- Preset NO filtering

  Set all “Lag time constant” to NC sampling period P50

  E98632 + xx = 100 * P50 [ms]
- Setting drive parameter V0080 of each interpolated axis to the following value:

$$V0080 = 1.0 - \left(\frac{P50\ [ms]}{2 * V0052\ [ms]} \right)$$

  Where:
  - V0052 is the “integral speed loop gain” drive parameter
  - P50 is the NC parameter “sampling period”

- Activate SSB for each interpolated axis:

  E92300 + xx = 1
- Adjustment of drive parameter V0080 by mean of “ball-bar”
  - Increasing V0080 reduces real circle radius
  - Decreasing V0080 enlarges real circle radius
- Set position reference filtering if needed:

  If NO filtering, then the procedure is terminated.

  If filtering has to be applied, then set “Lag time constant” to the chosen value:
  - E98632 + xx = 100 * chosen time constant [ms]
  - The "chosen time constant" has to be copied into P56!
  - Obviously, after such setting the real circular path of “ball-bar” will be inside the theoretical (programmed) circle, but anyway we have NOT to modify the V0080 settings obtained with NO filtering preset.
  - With such setting, the Lag error displayed by HMI, even at constant speed, will be no more null, but the obtained machining will be smoother as desired.

In order to have the same filtering in case of HSC and SSB, the following rule of thumb can be applied:
Number of HSC filter terms * P50 [ms] = 2 * SSB Lag parameter E98632 + xx [ms]

In such case, the two filtering method will be very similar.

Here are example images of SSB tuning, with the following parametrization:

Parameters	Descriptions
• P50 = 2 ms	Sampling period
• P56 = 7 ms	Channel specific time constant
• P66 of X and Y axis = 20 ms	Time constant per axis
• V52 of X and Y axis = 10 ms	“integral speed loop gain” drive parameter

Ball bar   R = 10 mm   F = 6'000 mm/min

Green = Programmed circle

Red = Actual circle

Final setting without filtering

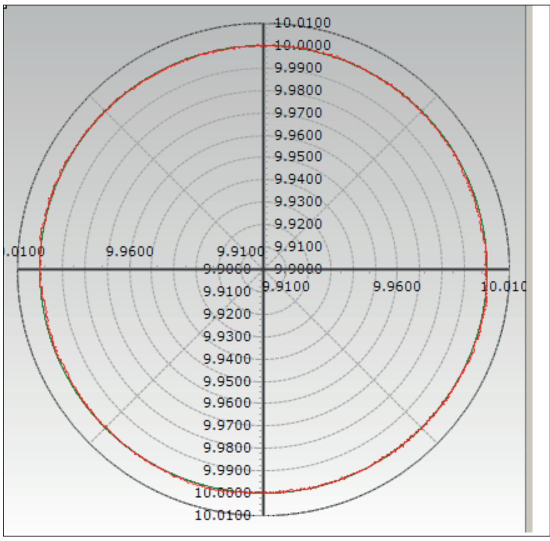
E98632 = 200 (= P50)

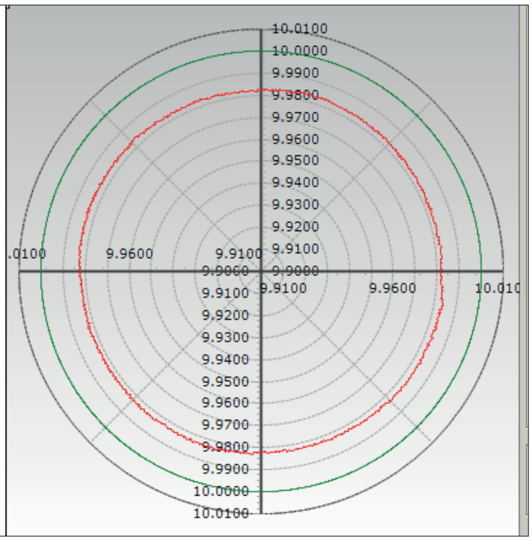
V0080 = 0,87

Final setting with filtering

E98632 = 700 (7 ms filtering)

V0080 = 0,87





Contour accuracy

Blue = Programmed trajectory

Green = Actual trajectory

Red = Profile error   40 times magnified

Final setting without filtering

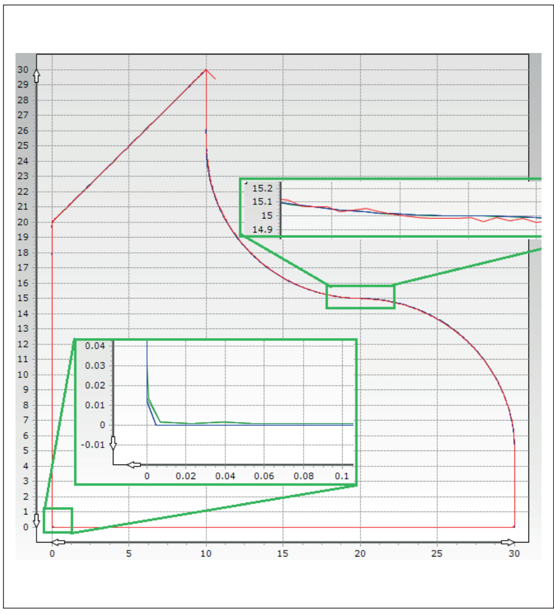
E98632 = 200 (= P50)

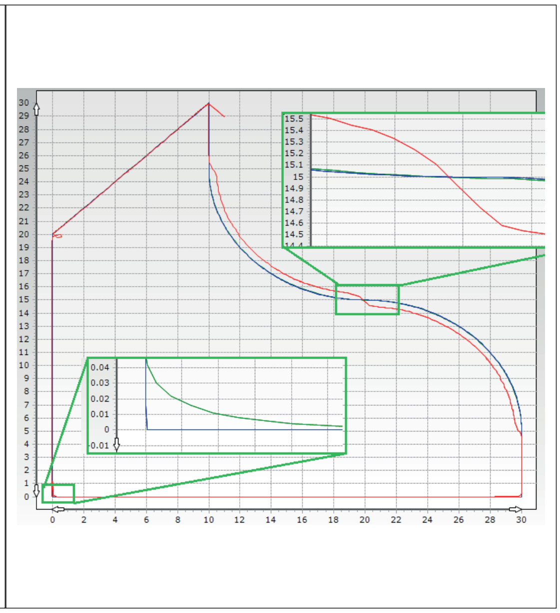
V0080 = 0,87

Final setting with filtering

E98632 = 700 (7 ms filtering)

V0080 = 0,87





2.6.6 Dry friction correction

The “dry friction correction” function, also known as “anti-pitch”, is much easily tuned as soon as HSM functionality has been tuned, by mean of “ball-bar” tool; that is because HSM removes error on circular path, emphasizing the possible presence of “pitch” phenomenon.

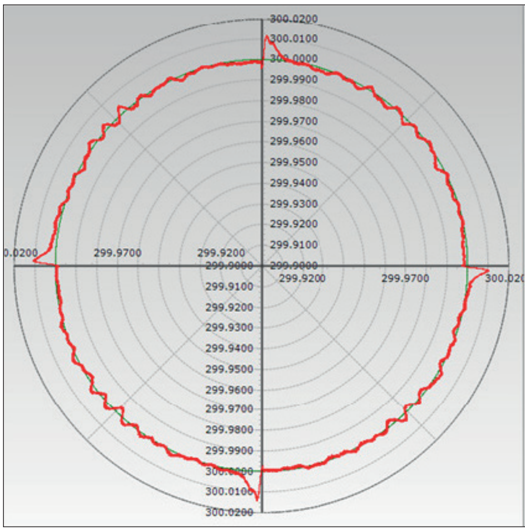
In order to tune properly “anti-pitch” function, operate as follows:

- for each type of HSM, just before the application of position reference filtering, it is possible to use “ball-bar” tool to adjust anti-pitch:

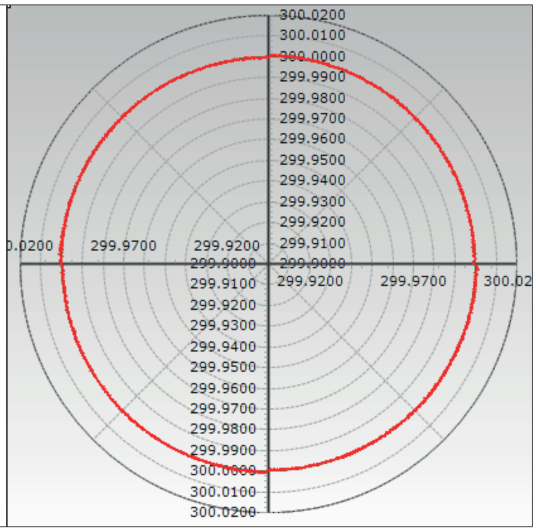
  1. By setting the Dry friction correction time constant (see P19), as good starting value, to the same value of drive parameter V0052 (speed loop integral gain time) of related axis.

  2. By Setting, via trial and error, the Dry friction correction amplitude parameter (see P20 parameter).

BEFORE dry friction correction



AFTER dry friction correction



Page deliberately empty

Page deliberately empty

Summary

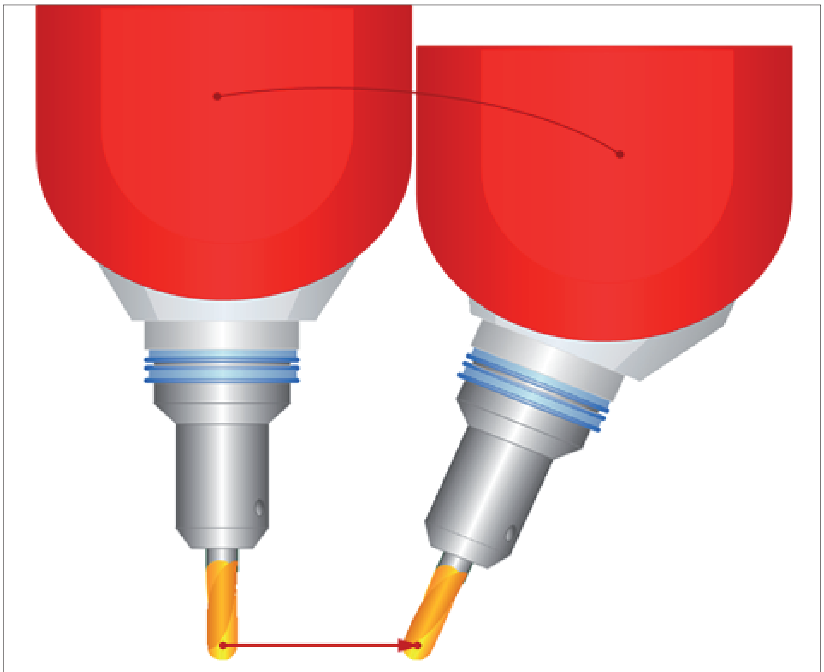
Page deliberately empty

3 Five-axes Machining RTCP

3.1 Description

In complex machining, the machines have additional axes for orienting the tool with respect to the workpiece. These types of rotation kinematics are described by two ideal pivot-axes (two revolute joints) and two offset vectors in addition the three linear axes X, Y and Z.

In Five-axes transformation, the RTCP function (Rotation around Tool Center Point) allows the tool center to retain its position along the specified path when the tool orientation changes.

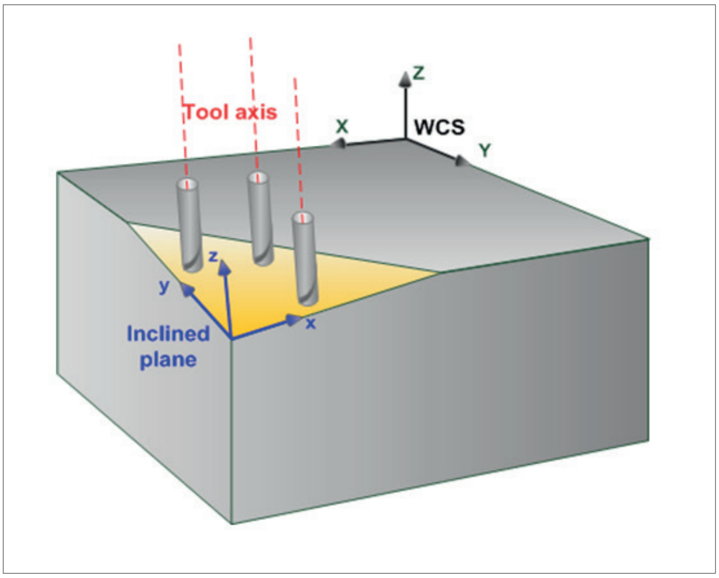


Straight line motion with change of tool orientation under active RTCP control

When enabling RTCP function the programmed trajectory is linked to the Tool Center Point (TCP) and so NC acts at pivot point (controlled point) with the mainly axes XYZ of the machine.

When used alone, the RTCP control requires only an enable command or an inhibit command.

However, a tool coordinate system can be defined to express the TCP position with respect to the axes of that inclined frame. It requires programming the features of this coordinate system with respect to the current part program origin when enabling RTCP.



An inhibiting statement or a re-enabling statement can be programmed to respectively to return to machine coordinate system for a tool change or to recall the last context of programming of inclined frame.

In addition, it is possible to program the TCP in teaching mode with definition of an inclined frame after positioning the tool in a known position from the origin of this inclined frame.

Starting with version 4.1.20 it is possible to define offsets on the rotary RTCP axes. Further explanation will be available in the Programming and Extended Programming manuals.

3.2 Prerequisites

RTCP function requires having:

- the option of validating RTCP transformation **(FXSO000154)**

The data of kinematic structure, linked to the RTCP function, have to describe accurately the actual Five-axes machine.

The values of kinematic offsets should be very close to the exact dimensions to guaranty the Five-axes machining accuracy.

A kinematic measuring **cycle248** with 3D touch probe is available in option for calibrating the Five-axes kinematics supported by RTCP function.

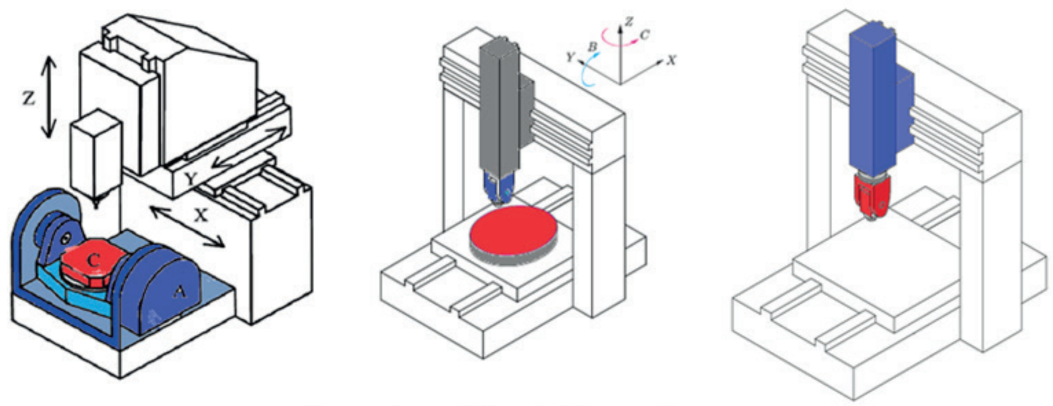
3.3 Targeted kinematic chains

This transformation takes in charge of the Five-axes machines with the following milling unit:

- A tilting turntable
- A swivel head and a rotary table combined
- A dual twist head but also a head with nutating[1] spindle.

It is well adapted to conventional 4-axes machine, equipped with either a single turntable or with a single swiveling axis head.

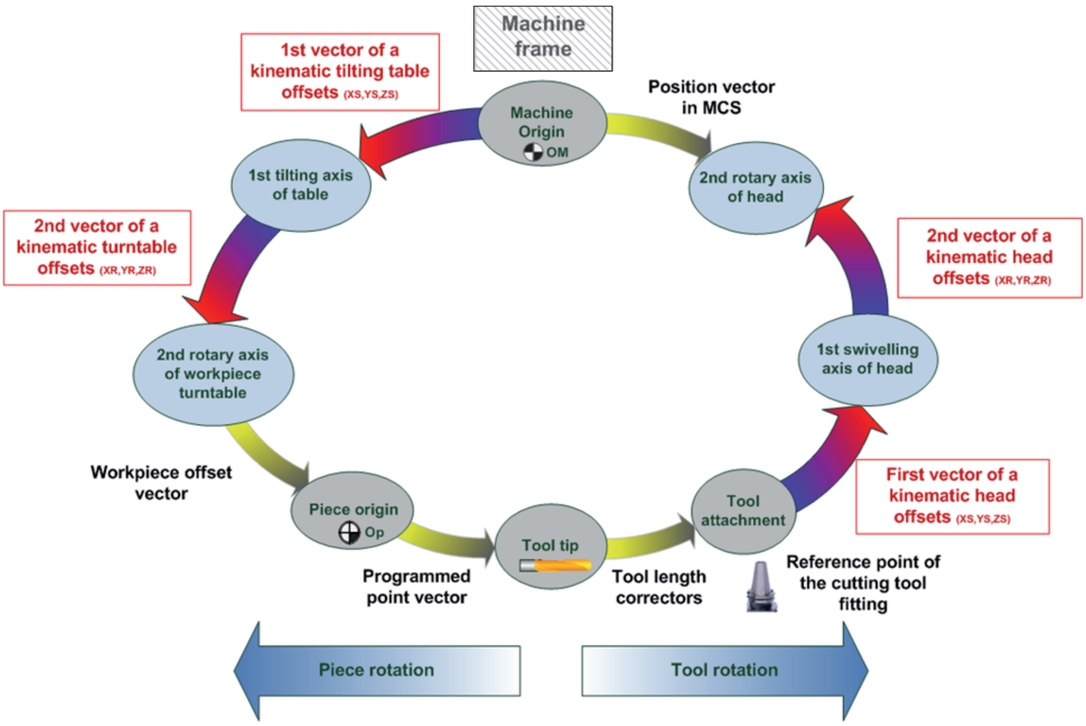
1) The word of “nutating” meaning an oscillatory motion or a slow spinning about an axis of a rotating body, a nutating head has a rotating pivot axis in an inclined plane.



Examples of 5 axis kinematic

The description rule for a Five-axes kinematic changed from version 4.1.0.0 of the Flexium Tool, it starts from the spindle tool holder to the machine base and from the machine base to the workpiece.

3.3.1 Five-axes kinematic chain



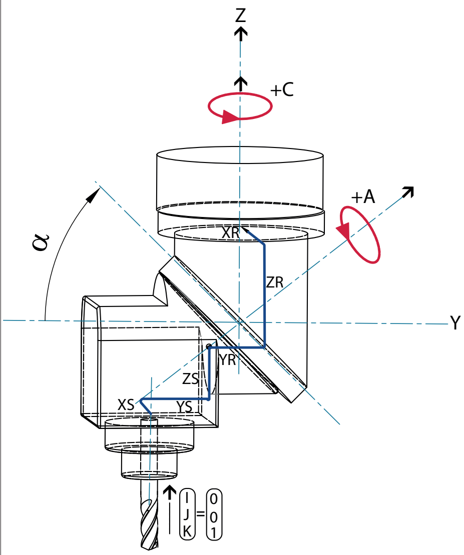
A Five-axes kinematics is mainly characterized by a swivel axis of the tool holder mechanical part or by a tilting axis of a turntable. "S" axis indicates the swivel/tilt axis.

The other, named “R” axis denotes a rotary axis of the tool kinematic part or of a kinematic part carrying the workpiece.

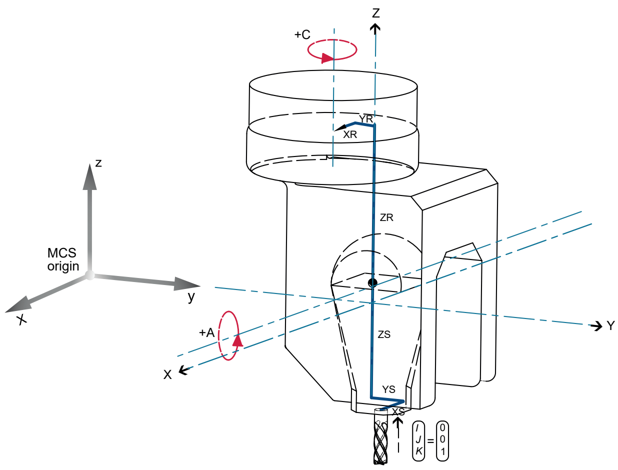
The first vector of kinematic offsets features either a swiveling axis, from the tool holder to the pivot axis or either a tilting table from the machine frame to the pivot axis. (XS, YS and ZS are its spatial components along xyz direction).

The second vector describes the kinematic offsets between the two pivot axes (XR, YR and ZR are its spatial components along xyz direction).

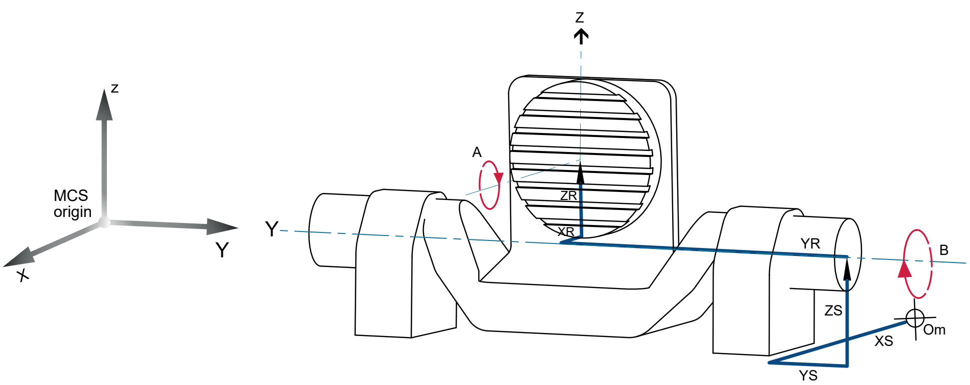
These two vectors are expressed with respect to machine frame O-xyz.



Shift vectors of a nutating head



Shift vectors of a dual twist head



Shift vectors of a tilting turntable

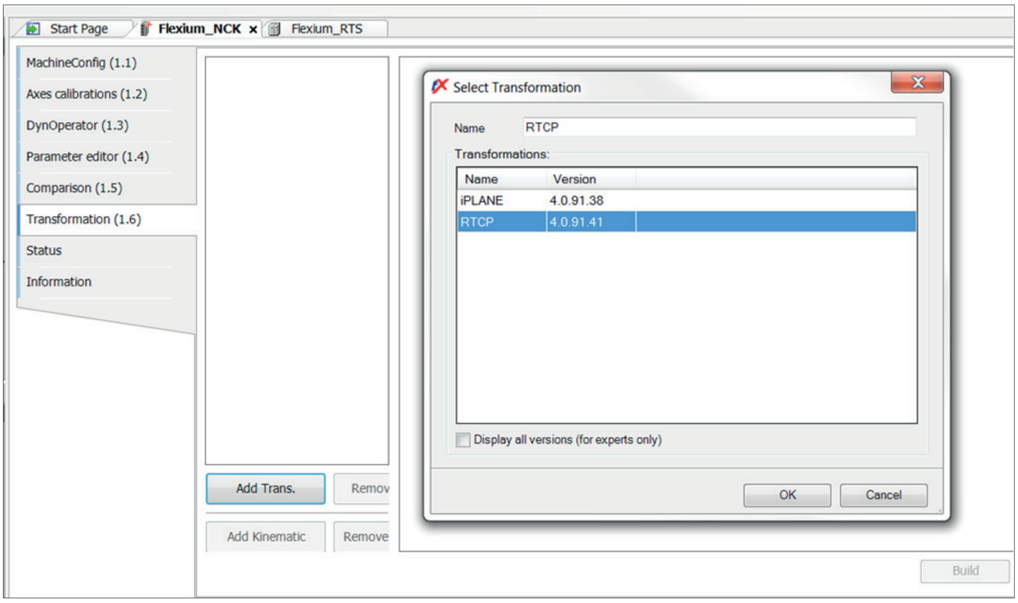
A list of different configurations managed by the RTCP function is described in annex A.

3.4 Interface for design of RTCP commissioning

Flexium Tools offers a new design for customizing data managed by the RTCP function or by *iPLANE* function.

The *"Transformation"* tab provides access to the tool of both RTCP customization and iPLANE customization projects.

The *"Add Trans"* button opens a library of configurations linked to a RTCP customization project or a library of configurations linked to an inclined plane project (iPLANE customization project).



Project selection dialog

By default two most current projects are displayed in this dialog:

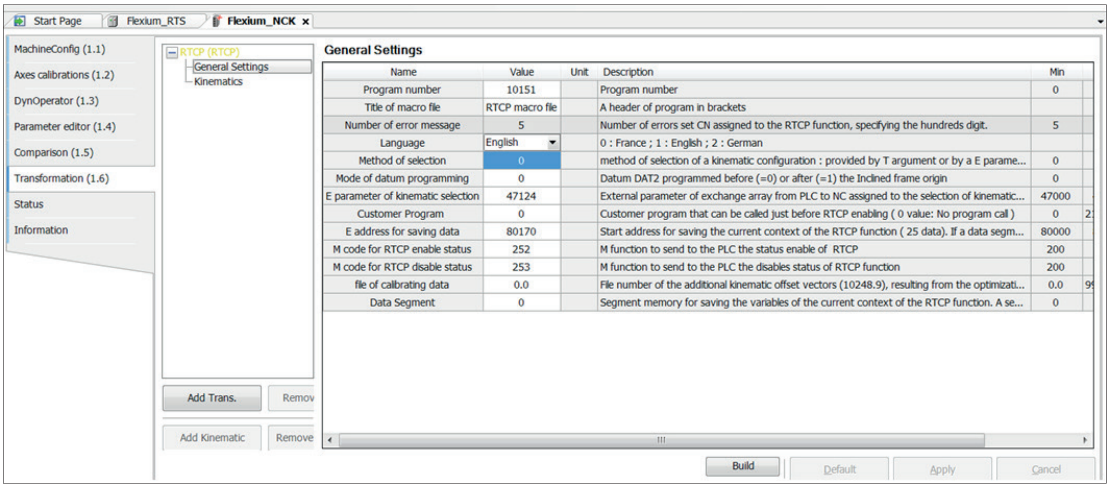
- iPLANE project dedicated for customization of an inclined plane function,
- RTCP project dedicated for customization of a RTCP function.

After selecting a RTCP project, two folders are displayed under the root of RTCP tree:

1. *"General Settings"* folder for editing the common parameters of the project
2. *"Kinematics"* folder from which you can add some kinematic configurations.

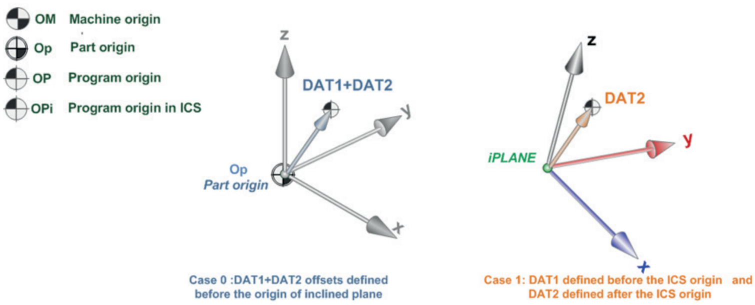
The opening of a second project is permitted and the user can customize in the two RTCP or iPlane functions in a same work session.

3.4.1 General Settings dialog



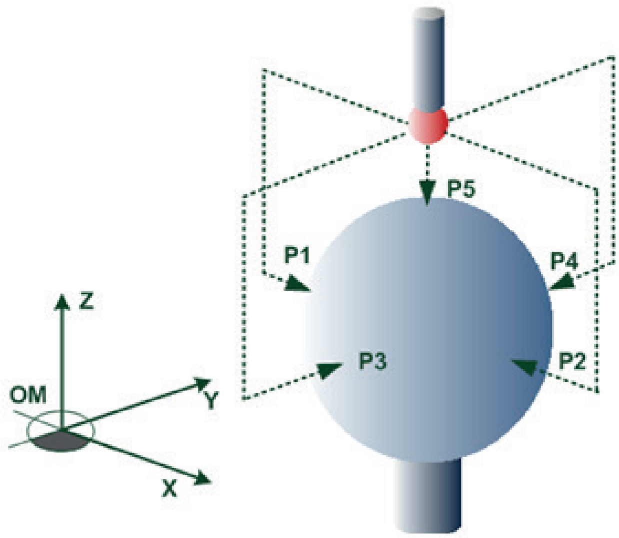
This dialog displays a list of common parameters of which are summarized in the tables below:

Variable	Description
Method of selection	Defines the method of selection of a kinematic configuration: **0:** Stands the choice is made by programming Tn in the call of RTCP function, with 'n' an integer from 1 and up to 8 customizable con figurations. **1:** Stands a selection made by an external parameter. The default value: 0
E parameter of kinematic selection	Specifies the address of E parameter of exchange array from PLC to the NC, assigned to the selection of kinematic configuration. This address is only used in case of a selection made by E parameter. Default value: 47124. Value stands that content of external parameter E47124 gives the number of the selected configuration (rank of configuration).
Data Segment	Memory segment allocated for saving the variables of the current context of RTCP function. This implies that a segment number has been specified and sized by the NC parameter P59Nx. The range of segment starts from 1 up to 32. A null value stands no segment allocated. Default value: 0  CAUTION **If the memory segment is empty (segment no sized by P59Nx) error 97 extension 11 is generated in the block containing G76+ instruction.** **If the flag writing protection of segment is enable then error 97 extension 14 occurs during the data writing process (protection of segments given by the external parameter E31003).**

E address for saving data	Start address for saving the current context of the RTCP function (25 data). Default value: 80170 stands the backup of the 25 variables of context begin at the external address E80170. **CAUTION** **The start address of E parameter should allow for saving the 25 variables, otherwise an error "Unrecognized E parameter" would be generated if the end address does not exist.** If a data segment is specified then only one external parameter is used, the recommended value is 80194.
Mode of datum programming	This variable specifies how DAT2 is applied when an inclined plane is associated with the RTCP function (see chapter 2.6.4)  0: stands DAT2 is applied before the declaration of iPLANE. 1: stands DAT2 is applied after the declaration of iPLANE. Default value: 0
M code for RTCP enable status	Number of M function sent to the PLC by RTCP function when RTCP is enabled. Default value: 252.
M code for RTCP disable status	Number of M function sent to the PLC by RTCP function when RTCP is disabled. Default value: 253
Customer program	Number of a user program called by RTCP function just before enabling. A null value no file. Default value: 0 **CAUTION** **If this macro file doesn't exist, error 25 is generated in the inclined plane definition block.**

File of calibrating data

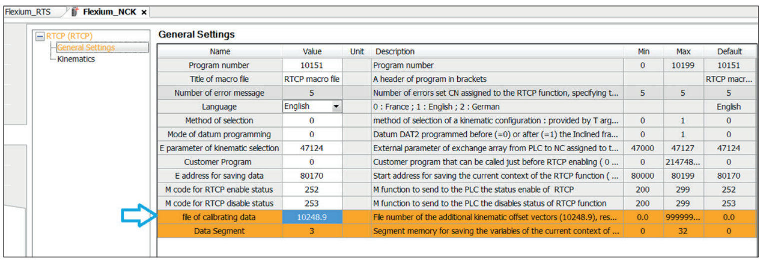
Optional calibration of Five-axes machine:



In order that RTCP function is now able to manage automatically the output results of a kinematic optimization cycle (Kinematics measuring Cycle248 with 3D touch probe).

This variable specifies the *Cycle248 Data File* number containing the Five-axes calibration results (the number of file: %10248.9).
Default value: 0 (no file)

CAUTION: If this file doesn't exist, error 25 is generated in the subroutine call block.

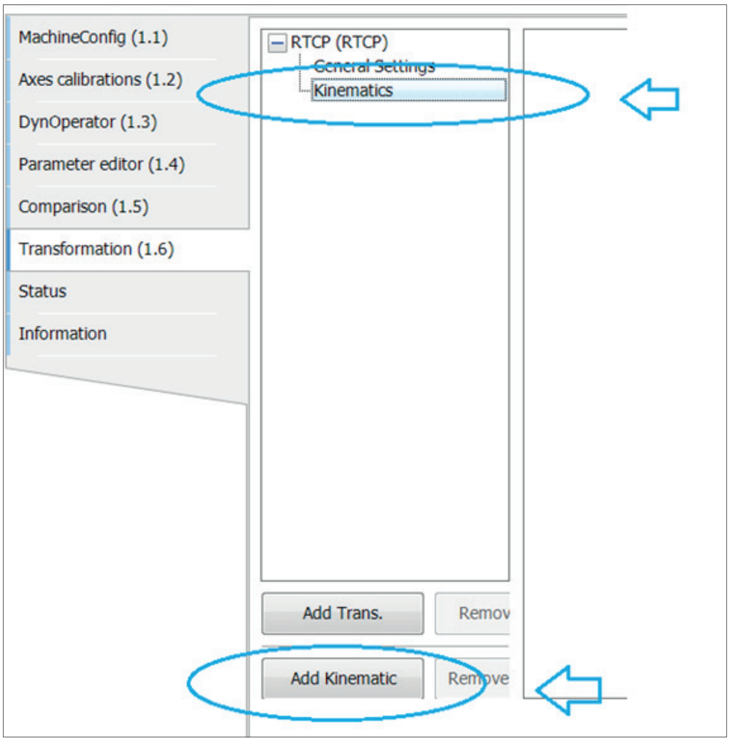


Name	Value	Unit	Description	Min	Max	Default
Program number	10151		Program number	0	10199	10151
Title of macro file	RTCP macro file		A header of program in brackets			RTCP macr...
Number of error message	5		Number of errors set CN assigned to the RTCP function, specifying t...	5	5	5
Language	English		0 : France ; 1 : English ; 2 : German			English
Method of selection	0		method of selection of a kinematic configuration : provided by T arg...	0	1	0
Mode of datum programming	0		Datum DAT2 programmed before (=0) or after (=1) the Inclined fra...	0	1	0
E parameter of kinematic selection	47124		External parameter of exchange array from PLC to NC assigned to t...	47000	47127	47124
Customer Program	0		Customer program that can be called just before RTCP enabling (0 ...	0	214748...	0
E address for saving data	80170		Start address for saving the current context of the RTCP function (...	80000	80199	80170
M code for RTCP enable status	252		M function to send to the PLC the status enable of RTCP	200	299	252
M code for RTCP disable status	253		M function to send to the PLC the disables status of RTCP function	200	299	253
file of calibrating data	10248.9		File number of the additional kinematic offset vectors (10248.9), res...	0.0	999999...	0.0
Data Segment	3		Segment memory for saving the variables of the current context of ...	0	32	0

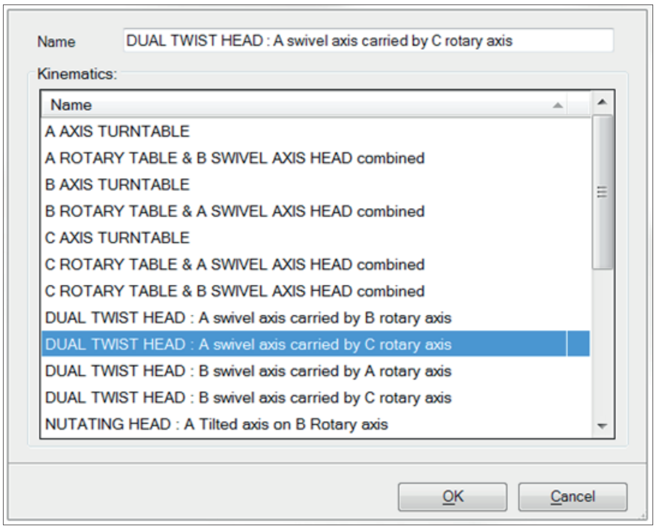
Notes.
Only the unshaded fields of value entry are modifiable in editing section.
A pop-up returns full text when the mouse is pointing a field which the content is partially hidden in the editing section.

3.4.2 Kinematic selection

- When the *"Kinematics"* folder is focused, then the button *"Add kinematic"* is released.
- An *“Add Trans”* command displays dialog containing a drop-down list of some kinematics.



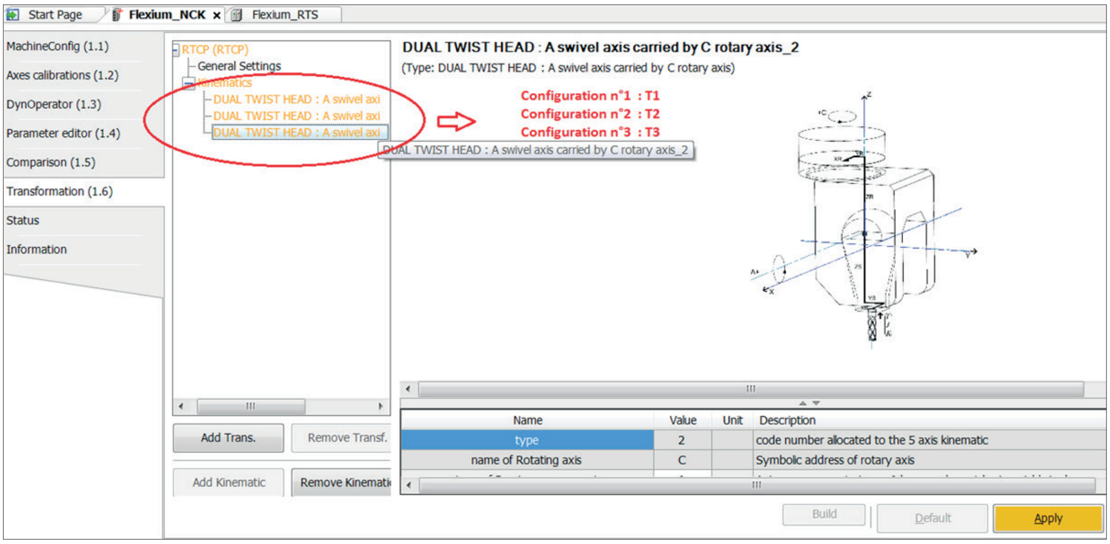
- Select an appropriate kinematic from a list of 23 items in alphabetical order:



3.4.3 Kinematic edition dialog

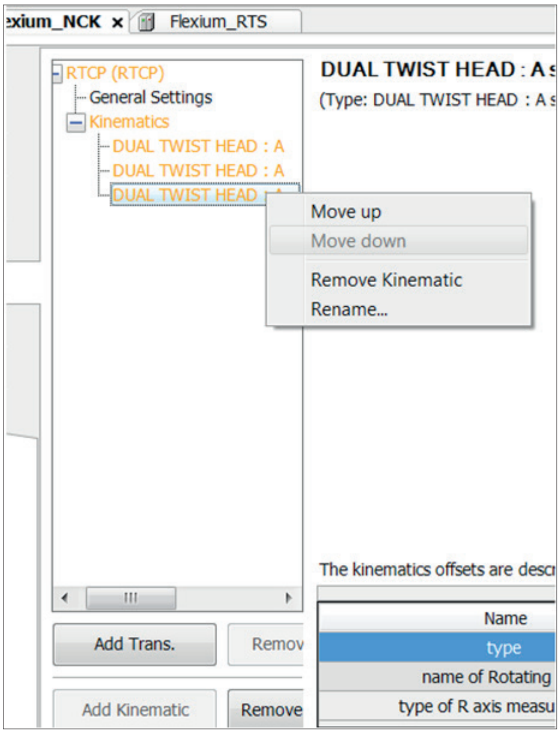
After selecting a kinematic, a new folder is added below the *"Kinematics"* root

The new kinematic folder is added to the previous one. The rank number in the *"Kinematics"* root implicitly defines the number of the configuration called by programming the RTCP function with T argument value.

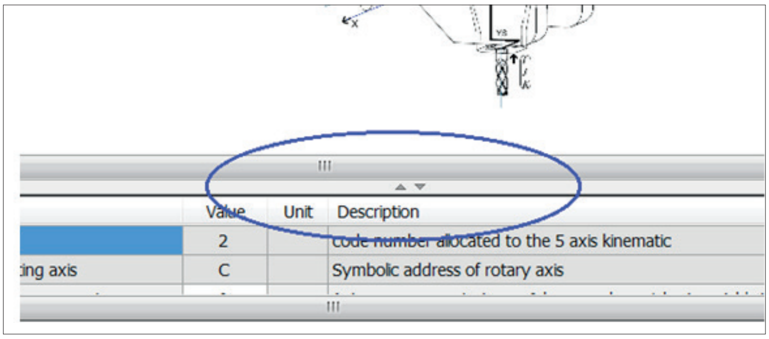


A right click on the selected kinematic folder allows:

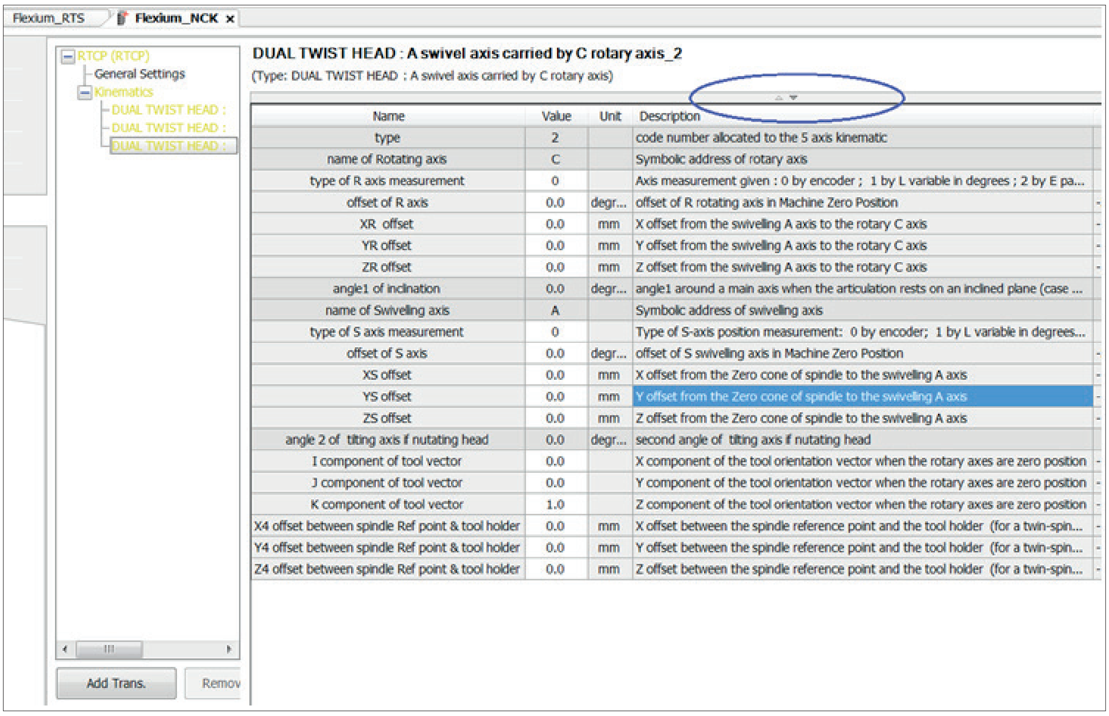
- To change the ranking number of the selected kinematic, with *"Move up"* or *"Move down"* commands.
- To rename the kinematic or its indexing and
- To remove it.



- Two lift up or down arrows ▲▼ can be used respectively to extend or to reduce the editing area.
- Acting on the bottom arrow ▼ shows the drawing of the kinematics at the expense of the editing area.



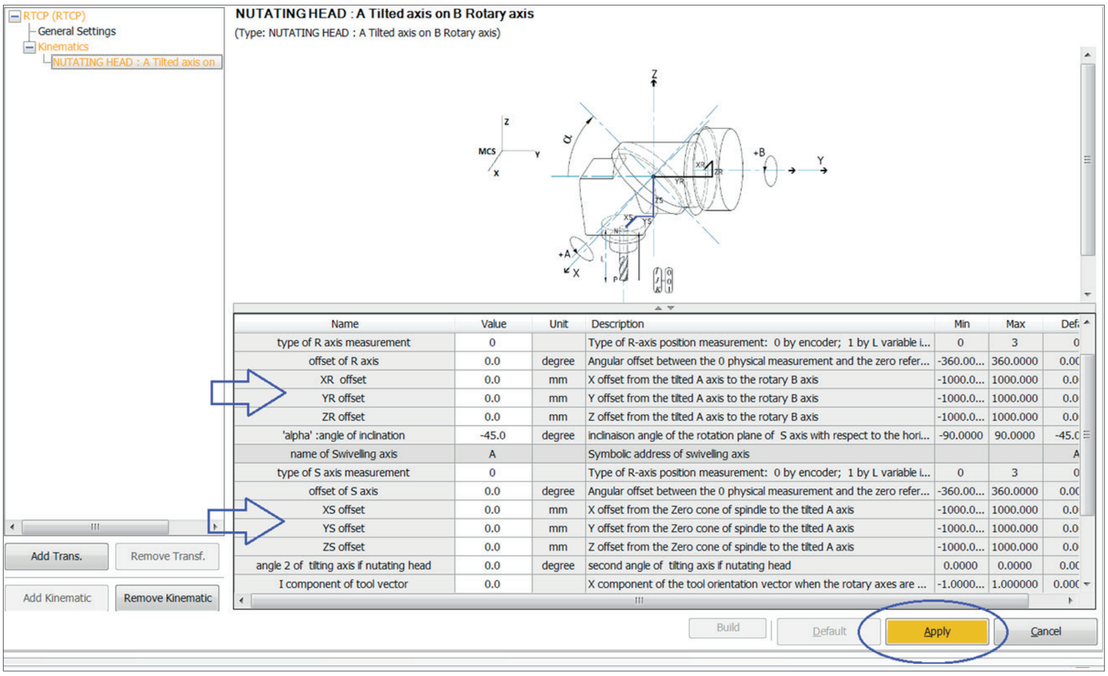
- The up arrow ▲ completely restores the edition area of the selected kinematics parameters.



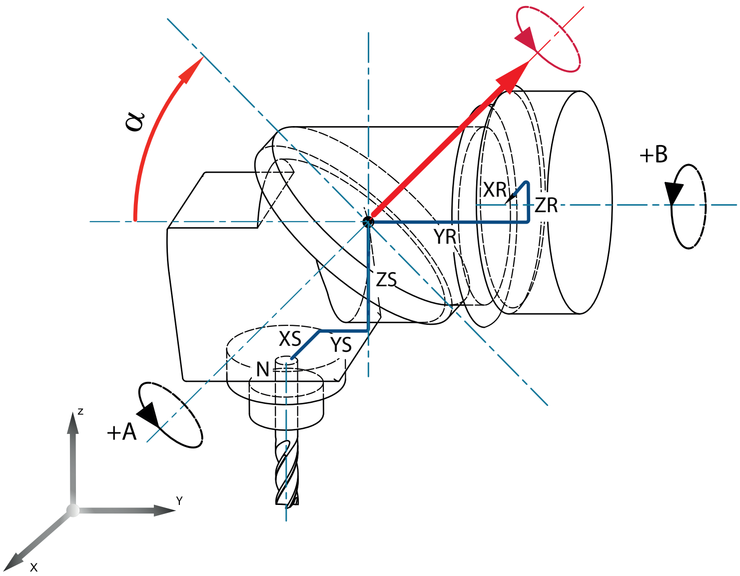
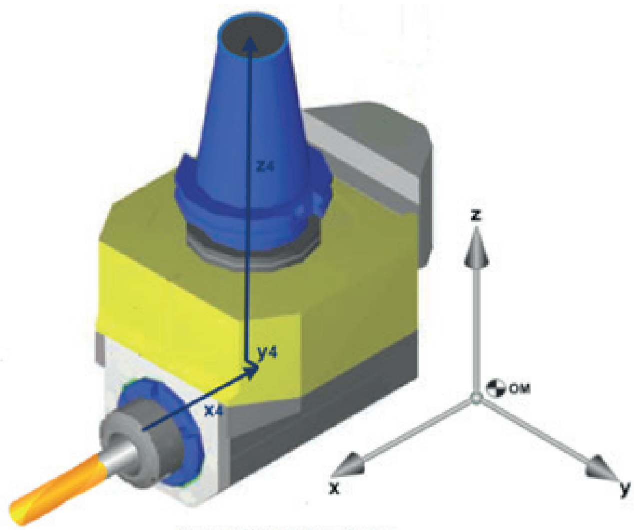
3.4.4 Kinematic setting

The nominal values of kinematics offset should be sufficiently accurate to limit the spatial error deviations of the tool center point in Five-axes transformation.

After setting, press to *"Apply"* button to save the current context of your opening project.



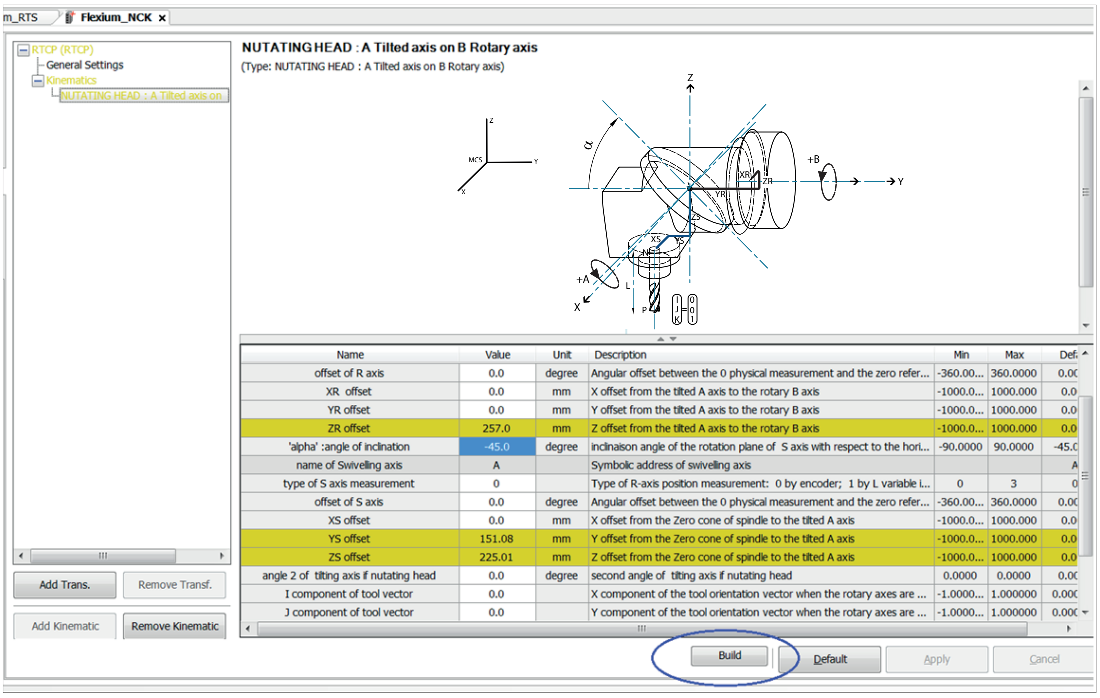
Variable	Description
Type of R axis measurement	Integer specifying the type of R-axis position measurement: 0 by encoder 1 by L variable in degrees 2 by E parameter in internal unit 3 no axis measurement
XR offset YR offset ZR offset	3 components of offset vector respectively along the X,Y and Z main axis direction, oriented from the first pivot axis (swivel axis of a milling head or tilting axis of a trunnion table) to the rotary axis (carrier axis of head or turntable axis). The offsets are expressed in mm.
Type of S axis measurement	Integer specifying the type of S-axis position measurement: 0 by encoder 1 by L variable in degrees 2 by E parameter in internal unit 3 no axis measurement
XS offset YS offset ZS offset	3 components of offset vector respectively along the X,Y and Z main axis direction, oriented from the zero cone position of the spindle to the swivel S axis (case of a milling head) or oriented from the machine origin to the tilting R axis (case of tilting turntable). The offsets are expressed in mm.

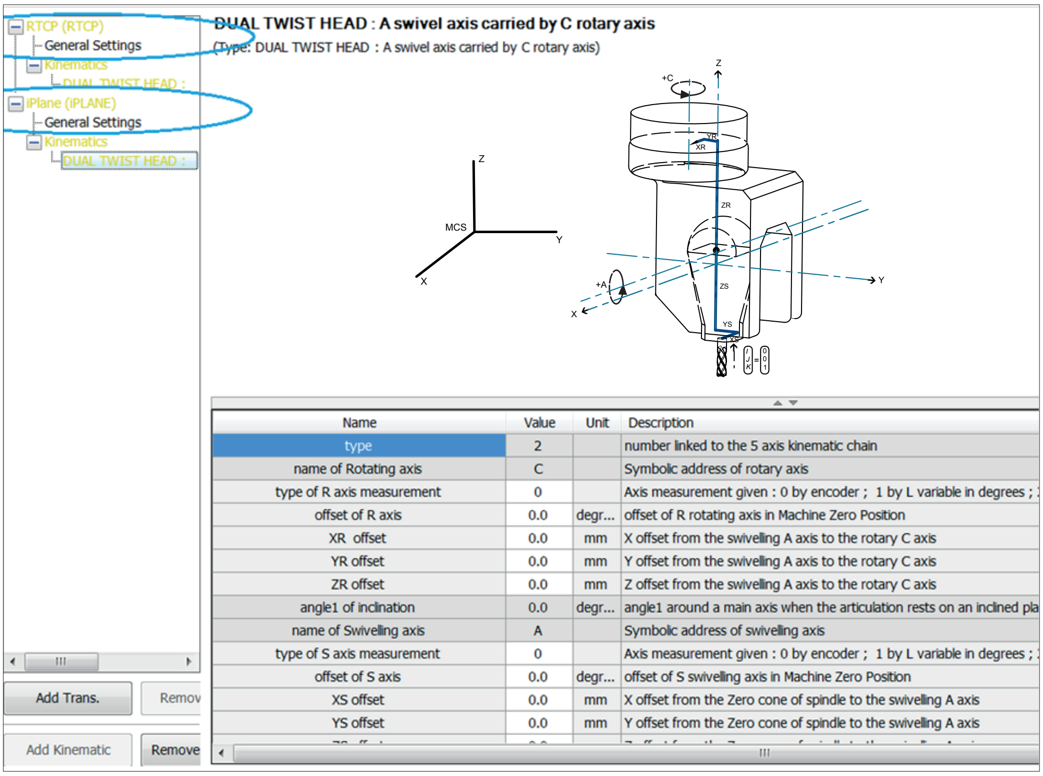
Angle of inclination	If the revolute joint is based on an inclined plane, this variable specifies the angle around a main axis parallel to the plane and its value is algebraic by the counterclockwise direction. The angle is expressed in degree.  *The figure shows an inclination angle α of the revolute A axis around X axis (-45°)*
I, J, K	3 components of the tool orientation vector when the rotary axes are zero position (reference pose of the kinematic). This unit vector is oriented from the tool tip to its attachment. Default value (0,0,1) a unit vector.
X4, Y4, Z4	3 components of additional offset vector to describe a shift starting from the spindle reference point to the tool holder. (These offsets are mainly used for a twin-spindle head or for an angle head) The offset unit is in mm. Default value; (0, 0, 0)  An angle machining head

3.4.5 Macro generation

A *"Build"* command compiles the current opening project and generates a loadable macro file in relating to the project (RTCP or iPLANE).

Let both RTCP project and iPLANE project folders present in the project context menu, the Build command generates the macro files of each project folder.



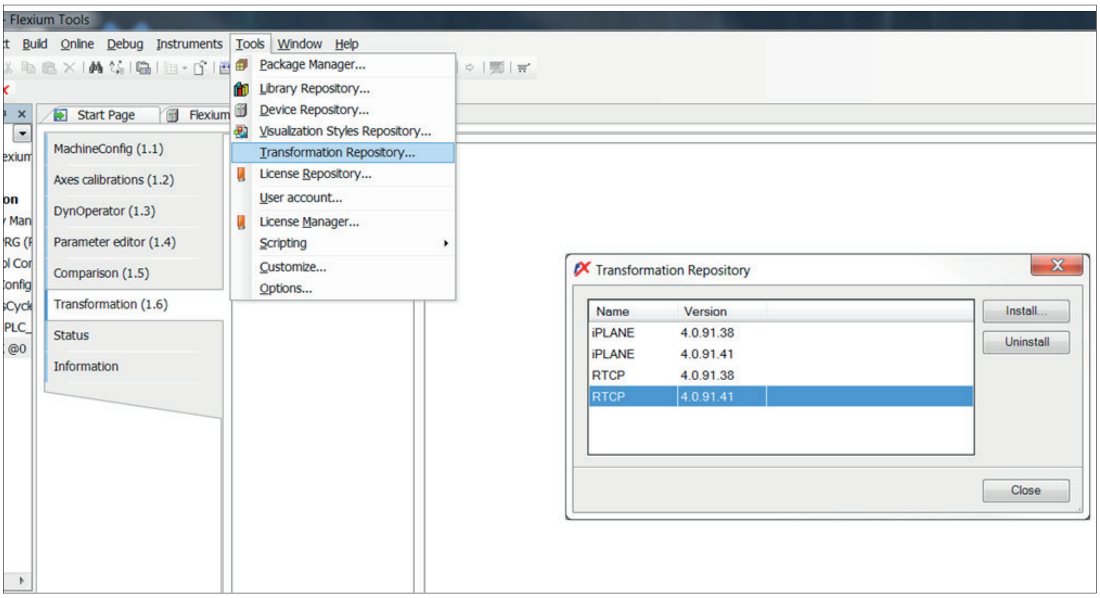


3.4.6 Upgrade a project file

From the tools menu, the command *"Transformation repository"* allows to load an upgrade of an iPLANE project or a RTCP project.

The project selection dialog displays all project versions already loaded:

- Press to *"Install"* button to add a new updated project in the list of project or
- Press to *"Uninstall"* button to suppress a selected project of the list.



- The *"Close"* button closes the *"Transformation repository"* dialog.

3.5 Programming RTCP

3.5.1 Overviews

The RTCP function allows, in case of a cutting tool of hemispherical shape, to keep the tool center point (TCP) on the programmed trajectory when the tool orientation changes.

It is possible to associate an inclined plane to the RTCP function in which the tool center path is programmed. The RTCP function takes into account the geometric offsets kinematic Five-axes but it also considers the characteristics of the tool and its dynamic tool offsets.

The speed and acceleration of each linear axis but also their travel limits are controlled in real time.

Six functions are provided:

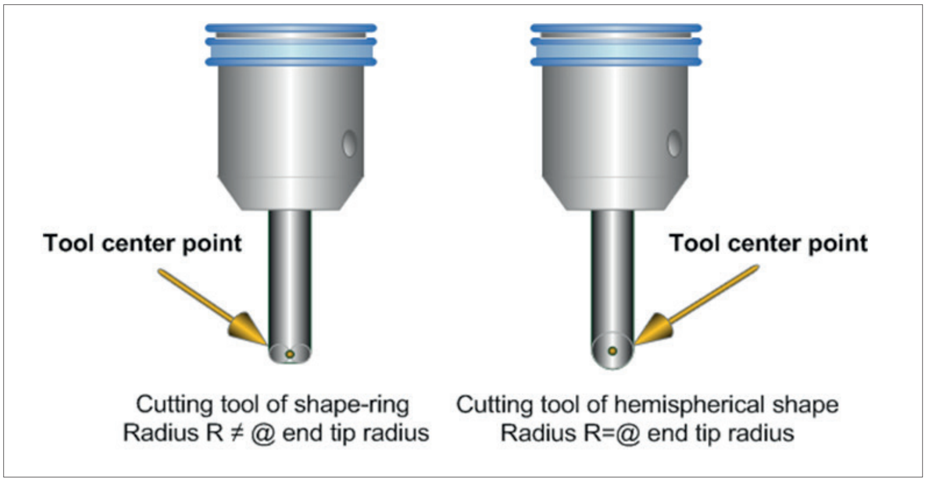
1. Selection of one kinematic configuration among eight configurations.
2. RTCP function enabling with specification of a inclined coordinate system with respect to the MCS.
3. RTCP function enabling with specification of an inclined coordinate system after a tool positioning in a known point of this inclined frame (enabling in teaching mode).
4. Canceling RTCP function and its inclined coordinate system.
5. Re-enabling RTCP function and its associated inclined coordinate system.
6. Re-enabling RTCP function without inclined coordinate system.

3.5.2 Convention of tool center point definition

The RTCP function is used to change the tool orientation without changing the path of the contact point between the tool and the piece.

This operation is effective only for a hemispheric tool (ball end tool, i.e. whose radius R is equal to the tool tip radius @).

For a tool ring (R≠@) the value of the tool tip radius is used to calculate the pivot point compensation.



3.5.3 Rule of an inclined plane description

We suppose the machine coordinate system (MCS: **0,x,y,z**) is an orthogonal and right hand system if only if the vector product of **x** by **y** vectors given **x** ∧ **y** = **z**.

A minimal representation of orientation can be obtained by using a set of three angles and the new origin of inclined frame can be defined by a translation vector with respect to the current frame (program reference frame).

The specification of a coordinate system in space based on a reference three-axes coordinate system describes the four steps of successive transformations as follow:

1st step: Translation vector (**EU, EV, EW**), between the origin of the reference coordinate system and the origin of inclined plane. **EU, EV** and **EW** are respectively the 3 translation components along X,Y and Z axis.

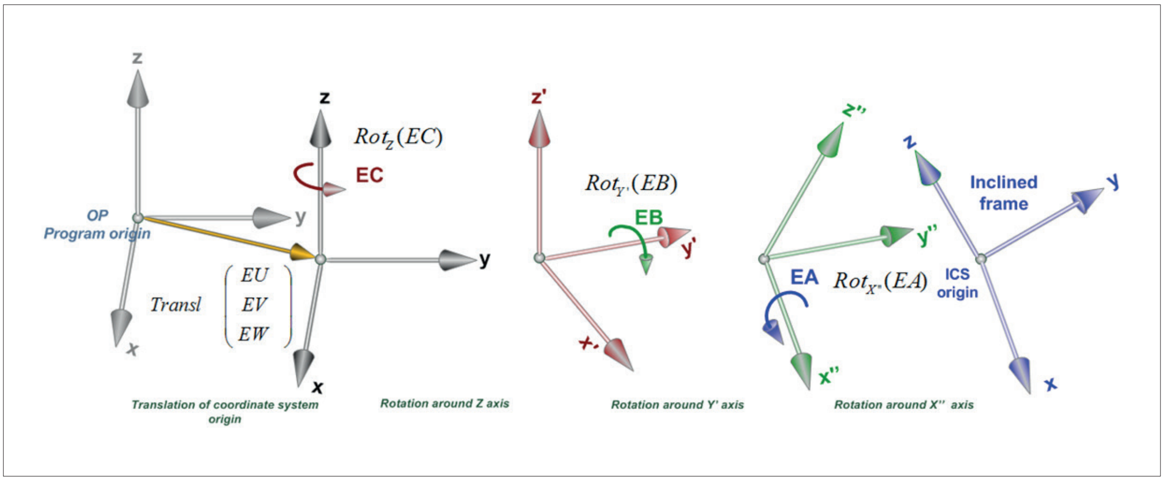
2nd step: Rotation of the previous shifted frame, *obtained in step 1st*, by an angle **EC** around axis z.

3rd step: Rotation of the current frame, *obtained in step 2nd*, by the angle **EB** around axis y'.

4th step: Rotation of the current frame, *obtained in step 3rd*, by the angle **EA** around axis x".

Explanation

- Each rotation is defined with respect to the preceding one. The frame with respect to which the rotation occurs is termed *current frame*.
- The composition of successive rotations is then obtained by post multiplication of the rotation matrices following the given order of rotations.
- The following figure shows successive rotations about axes of current frame.



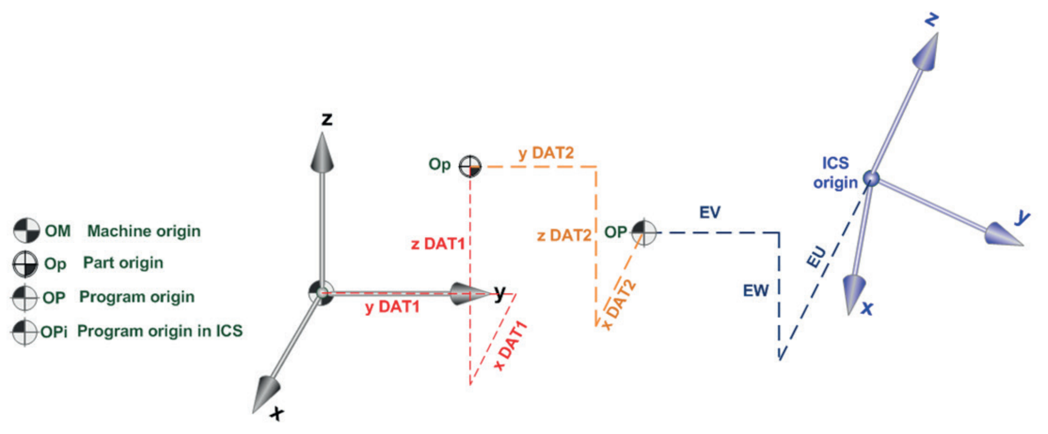
It is important to note that the inclined frame application operates correctly only with this definition of the successive transformations.

3.5.4 Application of DAT1 and DAT2

During the commissioning of RTCP, it can specify in two ways the part origin (DAT1) and the program origin (DAT2) when RTCP is associated with an inclined frame

3.5.4.1 Both DAT1 and DAT2 offsets applied before the ICS

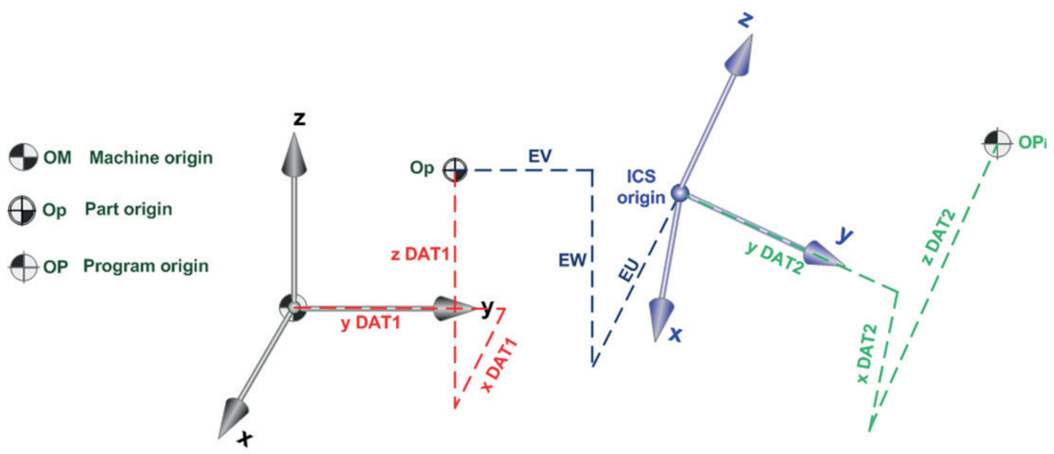
The figure below shows the different transformations to move from a machine coordinate system (OM origin) to the inclined coordinate system (ICS).



Both DAT1 and DAT2 offsets are applied before the Inclined Coordinate System ICS

3.5.4.2 Only DAT1 offsets applied before the ICS

The figure below shows the different transformations to move from a machine coordinate system (OM origin) to the inclined frame (OPi origin).



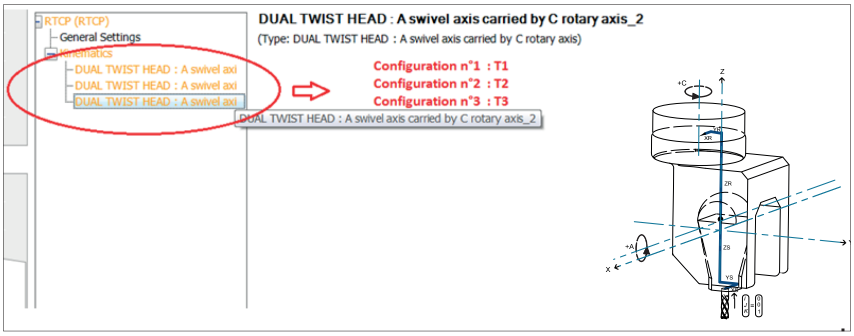
Only DAT1 offset is applied before the Inclined Coordinate System ICS

3.6 Syntax of programming

3.6.1 RTCP enabling with specification of an inclined coordinate system (ICS)

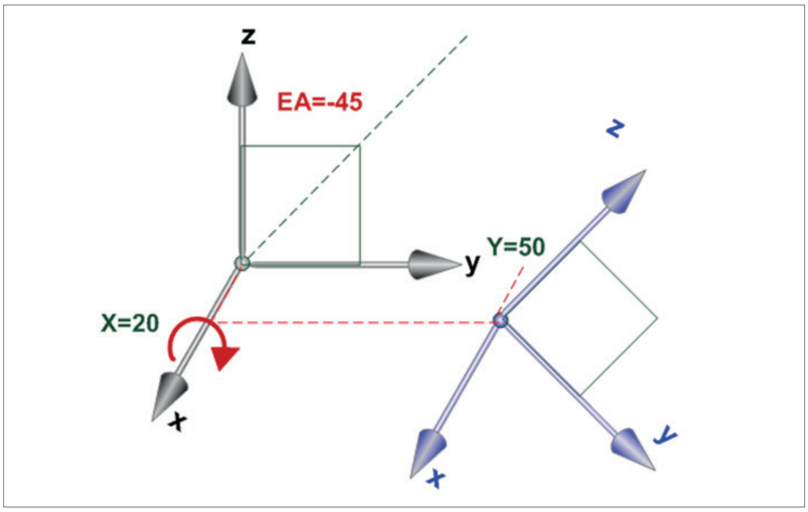
Syntax

G151 [EA…EB…EC…][1] EU…EV…EW…[T]

G151	**G function of RTCP.**
EA..., EB..., EC... (optional)	Specify the three Euler angles of orientation of the new coordinate system, (with EA angle around X axis, EB angle around Y axis and EC angle around Z axis). If an angle is not specified, a zero value is assigned to it (optional programing). The angles are expression in degrees.
EU..., EV..., EW...	Specify the position of inclined coordinate system origin with respect to the part program origin. EU, EV and EW are respectively the offsets of origin along the X, Y and Z axis direction. At least one of these three arguments must be specified. These parameters are expressed in mm.
T (optional)	Code number of the kinematic configuration among eight max configurations.  The setup number is in related to the rank of kinematic declaration. If T argument is omitted, the function chooses the previously programmed otherwise the first setup will be selected.

[1] An optional argument is written in squared brackets.

RTCP function computes the 5 axis transformation with the current tool offset (L length, R radius and @ tool tip radius). This implies that a tool offset corrector has been programmed before activating the RTCP function.

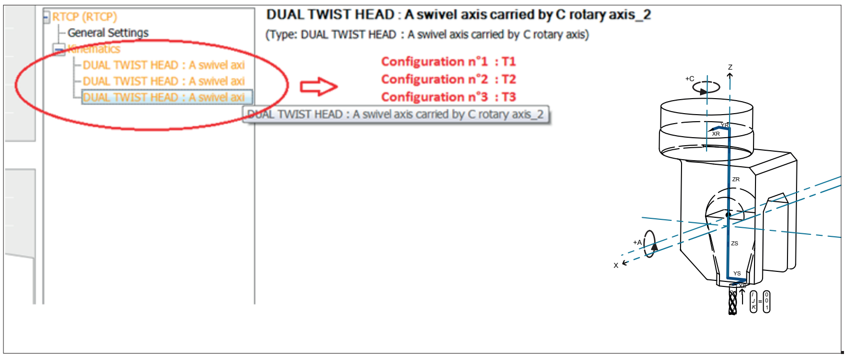
Example	**G151 EA-45 EU20 EV50 T1** This statement enables the RTCP function, set with the first kinematic configuration (T1) and defines an inclined coordinate system whose origin is located at 20 mm along X and at 50 mm along Y from the initial workpiece origin. The new frame (ICS) orientation is given by a rotation of -45 degrees around the X axis. 

3.6.2 RTCP enabled with specification of an ICS after positioning TCP in a known point of the ICS (RTCP in teaching mode)

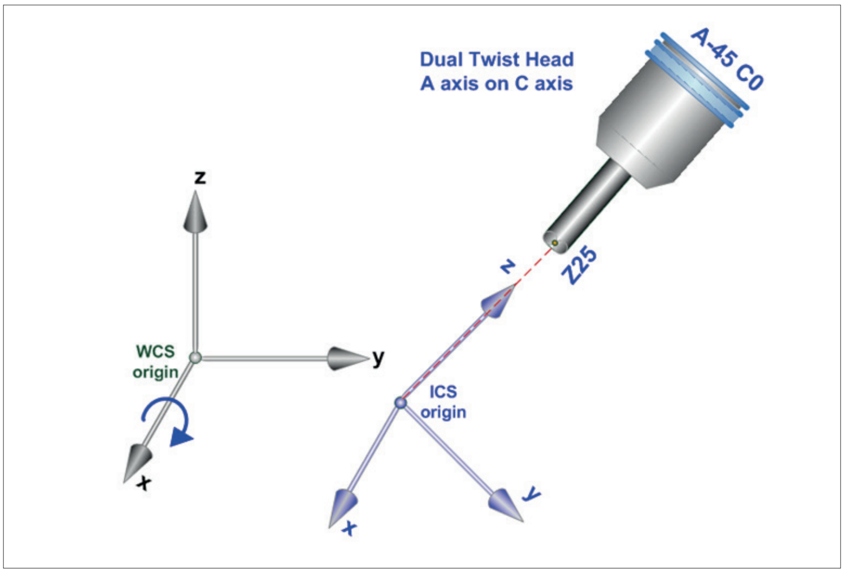
Syntax

G151 X…Y…Z…[EA…EB…EC…] D…[T]

<table>
<tr><th>G151</th><th>G function of RTCP.</th></tr>
<tr><td>X..., Y..., Z...</td><td>Specify the position of the TCP with respect to the inclined coordinate system (ICS) If one of coordinates is not programmed, a zero value is assigned to it.</td></tr>
<tr><td>EA..., EB..., EC...</td><td>Specify the three Euler angles of orientation of the new coordinate system.
The presence of one of these three angles is mandatory. If an angle is not specified, a zero value is assigned to it. The angles are expression in degrees.</td></tr>
<tr><td>D...</td><td>Tool offset assigned to the current tool. This argument must be specified.</td></tr>
<tr><td>T (optional)</td><td>Code number of the kinematic configuration among eight max configurations.



The setup number is in related to the rank of kinematic declaration.
If T argument is omitted, the function chooses the previously programmed (saved in the L919 variable equivalent to addresses T) otherwise the first setup will be selected.</td></tr>
</table>

Example	Consider a dual twist of which the swiveling axis is positioned at -45 degrees as shown in opposite figure. **G151 Z25 EA-45 D7** This block specifies an inclined coordinate system in which the TCP is located at 0mm in X 0mm in Y and 25mm with respect to the ICS. The origin position of ICS is computed knowing the current tool and its tool offset given by D7. The new ICS orientation is given by a rotation of -45 degrees around the X axis. The kinematic setup considered by the RTCP function will be the recently programmed. 

3.6.3 Cancelling the RTCP function and its associated ICS

Syntax

G151 S0

G151	**G function of RTCP.**
S0	This function cancels the RTCP function and its associated inclined coordinate system.

The RTCP function and its associated inclined plane are canceled after a RESET button or an end of program M02.

3.6.4 Re-enabling the RTCP function without its associated ICS

Syntax

G151 S1

G151	**G function of RTCP.**
S1	Use this function to reactivate alone the RTCP function and to revoke its previous inclined coordinate system. The programmed movement of TCP is in the workpiece coordinate system.

3.6.5 Re-enabling the RTCP function and its associated ICS

Syntax

G151 S2

G151	**G function of RTCP.**
S2	Use this function to re-enable the RTCP function and to restore the last context of the active inclined coordinate system (ICS). The last ICS was specified by the Syntaxs : G151 [EA…EB…EC…] EU…EV…EW…[T] or by the Syntaxs G151 X…Y…Z…[EA…EB…EC…] D…[T] and cancelled by G151 S0 or after a NC reset or a CNC initialization (see the notes).

Example

Assume a RTCP function with its inclined plane is enabling and a tool change is requested.

The changer requires positioning and movements on axes with fixed positions with respect to the measurement origin.

`...`	
`D...`	
`X... Z...`	
`G151 EA..EU T2`	RTCP and inclined coordinate system enabled, with configuration n°2
`Z...`	
`X... Y...`	Machining with respect to the ICS
`...`	
`G151S0`	Cancelling RTCP and its inclined plane
`G G52 Z17`	Positioning for a tool change
`T..D..M6`	Positioning for a tool change
`G151 S2`	Reactivating RTCP and its inclined plane with configuration n°2

3.6.6 Enabling a TOOL ICS from the current tool orientation

Syntax

G151 R1 D…

G151	**G function of RTCP.**
R1	From the current tool orientation, this function computes an inclined coordinate system (ICS) whose the origin coincides with the tool tip and whose the Z axis is directed along the tool axis direction. In general, the tool is oriented along Z+ when the 5 axis kinematics is in reference position. This Five-axes function allows programming an axial retraction or a lateral motion of the tool according to the frame TOOL ICS, in JOG mode or in MDI programming. Its use is restricted to Five-axes kinematics having a single-twist head or a dual-twist head. Otherwise, the following error message is set (Error 515 extension 2): “Tool frame computation doesn’t support this Five-axes kinematics”.
D...	Tool offset assigned to the current tool. This argument must be specified.

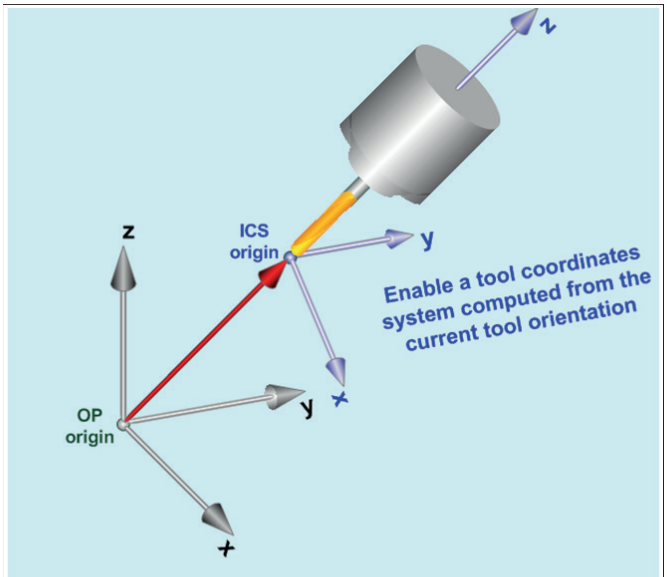
From the current orientation of the tool, the Five-axes function **G151 R1 D…**, calculates a set of three Euler angles defining the frame TOOL ICS:

- EA angle around the X axis,
- EB angle around the Y axis and
- EC angle around the Z axis,

and activates this frame Tool ICS whose origin coincides with the tool tip.

It is possible to activate the tool frame, associated with a targeted kinematic configuration, by adding the additional argument T as following:

G151 R1 D10 T2



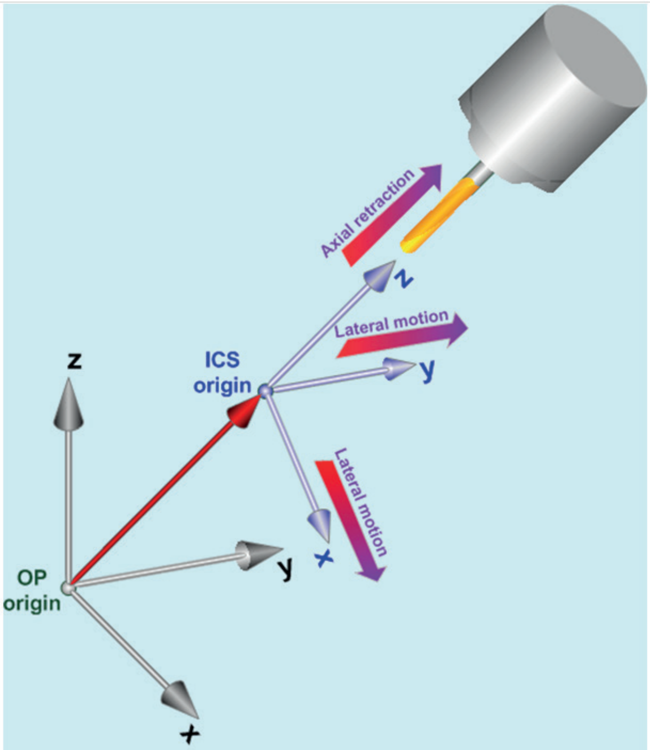
G151 R1 : Tool frame activation for a tool retraction

For example, a frequent use would be a retraction after a tool breakage by moving the tool along the Z axis direction or would be to carry out an axial clearance movement.

For other uses, it is possible to program a tool movement in both lateral directions in TOOL ICS frame, but also to change the tool orientation around its TCP.

To know exactly the orientation of lateral axes of TOOL ICS frame, we can imagine a tool frame, parallel to the MCS frame whose origin is glued on the TCP when the twist head axes are in zero position.

The orientation of the ICS frame axes is deduced from the head axis positions.



Axial or lateral retraction of the tool
with respect the tool frame

Executing this function does not change the data of previous programming of RTCP function. Then G151 S2 re-enable the RTCP function and re-activates the previous context.

Example

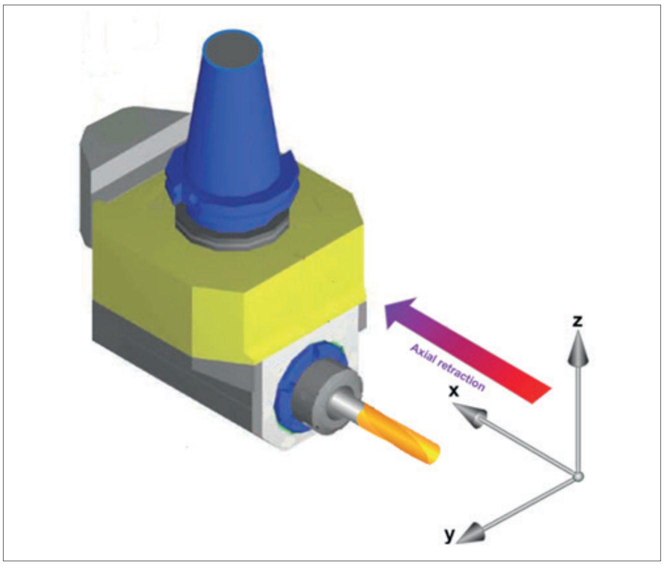
`G90 G1 F`	
`D...`	Tool corrector
`X... Z...`	
`G151 EC17.5 EU0 T1`	Configuration T1 : Dual Twist Head A on C RTCP and inclined coordinate system enabled
`A...C...Z...X...Y...`	
`A...C...Z...X...Y...`	Machining with respect to the ICS
`...`	
`G151 R1 D…`	Compute a frame TOOL ICS and activate the RTCP
`G91 Z10 F200`	Retraction in incremental mode along the Z axis direction of the frame TOOL ICS. G91 programming is optional because the axis motion is expressed in the frame TOOL ICS.
`G90 F5000`	
`A...C...`	Change the tool orientation
`G151 R1 D…`	Compute and enable a new frame TOOL ICS
`X10 F200`	Lateral motion

If an angle machining head is mounted on spindle of a twist head and configured to have a right-angle tool rotation such as shown in the figure opposite, then the X axis of TOOL ICS frame would be the retraction axis.

Example of an angle head mounted on the dual twist head (B swiveling axis carried by a rotary C axis). The tool is oriented along X+.

For reminding, the tool orientation vector IJK, in the RTCP commissioning file, specifies the tool axis direction when the rotary axes are zero position (reference pose of the kinematic).

The vector components are [I,J,K] = [1.,0.,0.].



Example

Let the T2 configuration number defining a dual twist head B on C axis with mounting of an angle head

`G90 G1 F`	
`D...`	
`X...Y...Z...`	
`G151 EU… T2`	
`B...C...Z...X...Y...`	
`...`	
`G151 R1 D1`	Tool ICS enable for retraction
`G91 X20 F400`	Retraction along X axis
`...`	

3.6.7 Tool length correction

In case of a twist head, the length correction is included in the kinematic features of the head with its orientation given by IJK vector at the reference pose.

Therefore, the declaration and the dimension must be assigned to the tool correction before activating RTCP.

3.6.8 Tool wear offset

Such remark of the previous chapter can be also applied for the tool length wear offsets less than 100µ (0.1 mm), immediately taken into account and included in the twist movements with the orientation given by IJK vector at the reference pose.

The total length of tool used in the RTCP transformation is: **Ltool + Ltool_wear - @tip_radius**.

3.6.9 Tridimensional Tool correction (G29)

When the RTCP function is used with three-axes tool correction (i.e. no tool orientation vector IJK in the G29 blocks), the CNC can only process spherical tools.

When the RTCP function is used with Five-axes tool correction (i.e. presence of tool orientation vector IJK in the G29 blocks), the system is able to process the case of the toroidal tool.

3.6.10 Table off-centering (DAT3)

The offsets related to an off-centered turntable (DAT3) are no longer processed as such when the RTCP function is enabled.

Since the position of the turntable rotary axis is known, this correction is automatically included in the RTCP function.

3.6.11 Program Origin Offset (G59)

A function [G90/G91] **G59** X.. Y.. Z..[I.. J.. (K).. ED..], associated with one or more arguments, axes and values, shifts the programing origin.

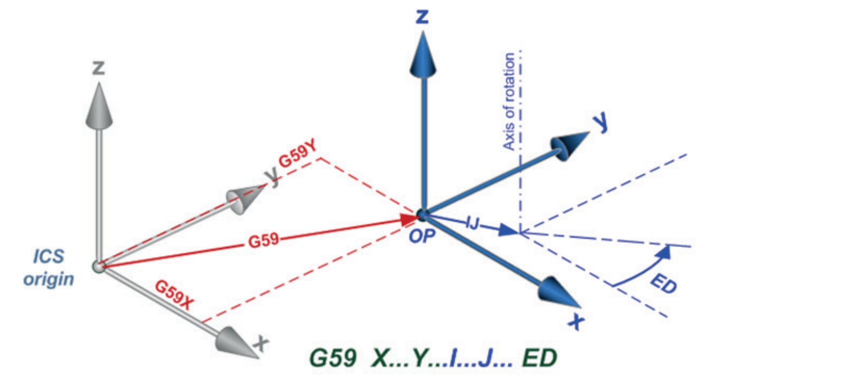
If RTCP function is enabled with specifying an inclined coordinate system (ICS), then the program origin offset is performed with respect to the ICS.

The optional IJ or IK or JK arguments define the center of rotation of an angular shift programmed with ED, respectively in a G17/ G18 / G19 plane with respect to ICS frame.

Example n°1: angular offset programmed in a G17 plane with respect to an ICS origin

RTCP with definition of an inclined coordinate system:

- G151 EA..EB..EC..EU..T2
- Programming in absolute dimensions, **G17** plane G90G17XY
- Program origin offset with definition of a center of rotation of an angular shift ED in the **G17** plane (rotation around Z axis with respect to the inclined frame)
- G59 X..Y..Z..I..J..ED30

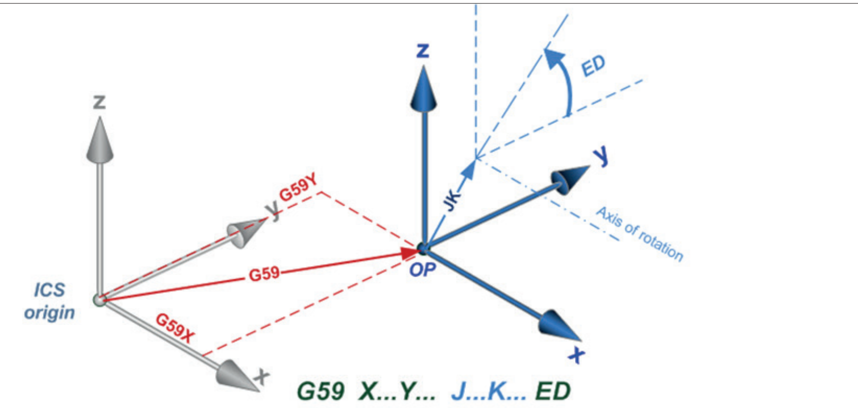


Machining programmed with respect to the ICS origin is shifted and oriented according to ED. XY plane (of ICS) defines the rotation plane

Example n°2: angular offset programmed in a G19 plane with respect to an ICS origin

RTCP enabling with its previously programmed ICS:

- G151 S2
- Programming in absolute dimensions, **G19** plane: G90 G19YZ
- Program origin offset with definition of a center of rotation of an angular shift ED in the **G19** plane (rotation around X axis with respect to the inclined frame)
- G59 X..Y..Z..J..K..ED30



Machining programmed with respect to the ICS origin is shifted and oriented according to ED. YZ plane (of ICS) defines the rotation plane

Important notes about the RTCP function and its associated inclined plane

- The RTCP function and its associated inclined plane are canceled after a RESET button or after an end of program M02.
- A tool offset change when the RTCP function is valid causes an error message 16.
- When the RTCP function is enabled, homing on an axis from the keyboard is prohibited and homing by the axis jogs generates error message 24
- Any change of the DAT1 or DAT2 datum shifts, either directly or by programming of the corresponding external parameters (E60xxx and E61xxx), is prohibited when the inclined plane is enabled.
- Programming of movements in G52, i.e. in absolute coordinate referenced to the measurement origin, is prohibited when an inclined plane is valid.
- If the bit6 of P7 word N1 is set to 0, programming an inclined plane axis in G52 returns error 24. If this bit is set to 1, programming an inclined plane axis in G52 is enabled.
- Programming of a movement in G53 (DAT1 or DAT2 datum offsets cancelled) is prohibited if the RTCP is enabled.
- The rotary axes of the RTCP function have to be homed. Error message 159 is generated.

Warning about the restore of the last context of RTCP function[1]

In case of power failure, the data stored in a memory segment are saved; accordingly the last context of RTCP can be restored after enabling RTCP and its associated inclined frame.
But if the restore of RTCP context is set by saved external parameter so the last context of RTCP can be restored after enabling RTCP and its associated inclined frame and the under the strict condition that homing is completed on rotary axes otherwise error 159 occurs.

[1)] The next NC firmware version will allow a backup of RTCP context in saved memory segment. These segments are reserved by NUM. So in case of power failure, the restore of data will be guaranteed and it will be possible to re-enable RTCP function with its last inclined frame under condition homing made on the rotary axes.

Page deliberately empty

1
2
3
4
A

Summary

Page deliberately empty

4 iPLANE transformation

4.1 General

The inclined plane machining function, named iPLANE, is designed to allow machining in a tool coordinate system that is not aligned with the physical axes of the machine while keeping programming simple.

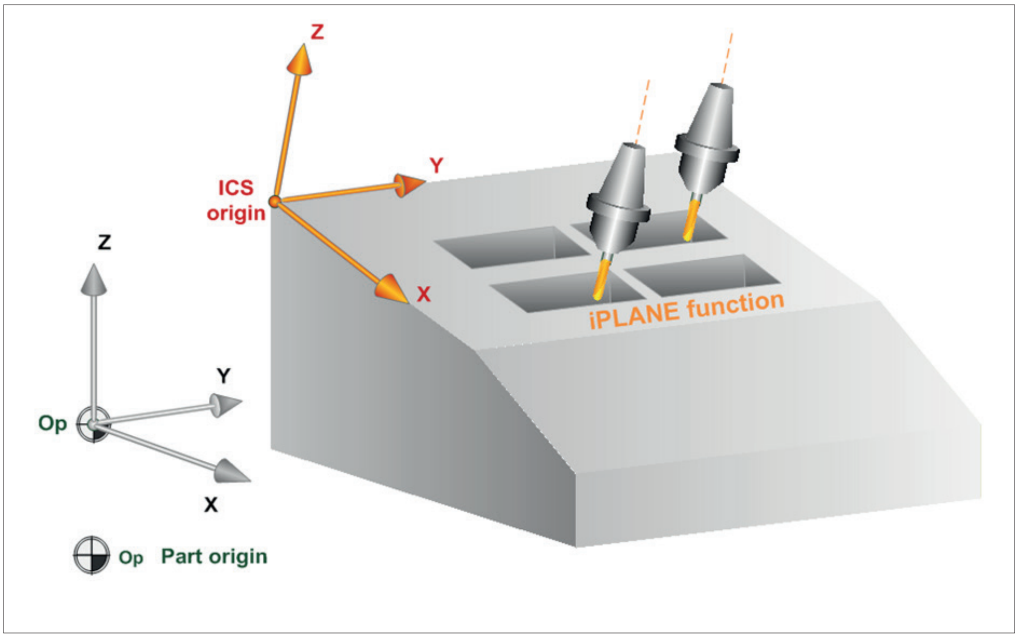
All five-axes machine structures are supported with their kinematic offsets:

1. Machine-head with two rotary axes with or without angle.
2. Machine-head with one rotary axis and turntable with one rotary axis.
3. Turntable with two rotary axes.

iPLANE function is parameterized using Flexium tools.

Six functionalities are supported:

- Selection of one out of a maximum of eight kinematic configurations.
- Selection of the primary axes (X,Y or Z) or secondary axis (U,V or W) used in the inclined plane.
- Enabling the iPLANE function after positioning the tool in a known position with respect to the new frame origin.
- Enabling of iPLANE by programming its origin.
- Temporally interruption of iPLANE
  (return to the machine coordinate system for instance for a tool change).
- Reactivation of the last programmed iPLANE.



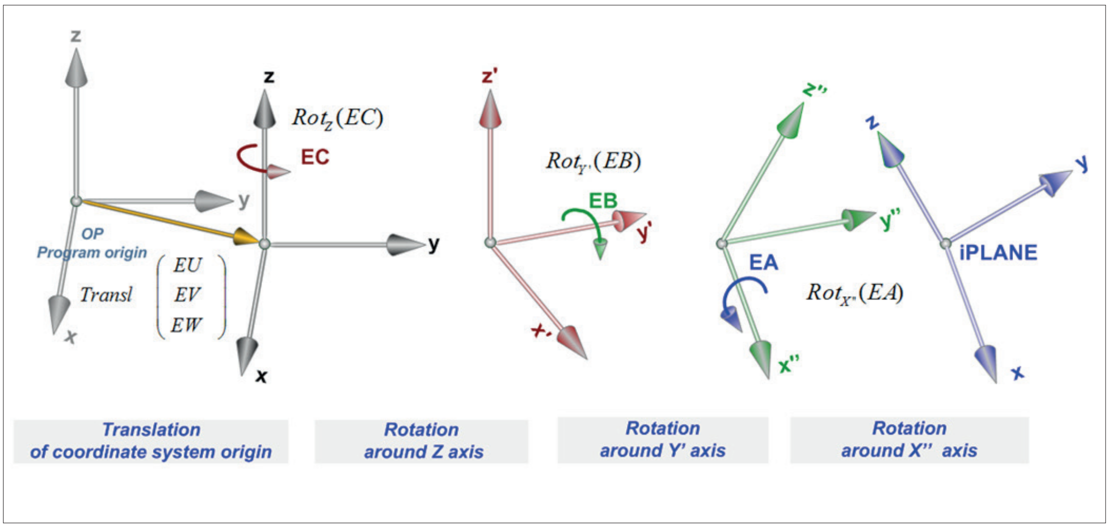
4.2 iPLANE definition

The rules are the same as those applied to an inclined plane associated with the RTCP function, namely:

- We suppose the machine coordinate system (MCS: **0,x,y,z**) is an orthogonal and right hand system if only if the vector product of **x** by **y** vectors given **x** ∧ **y** = **z.**
- A minimal representation of orientation can be obtained by using a set of three angles and the new origin of inclined frame can be defined by a translation vector with respect to the current frame (program reference frame).

The specification of a coordinate system in space based on a reference three-axes coordinate system describes the four steps of successive transformations as follow:

- **1st step**: Translation vector (**EU,EV,EW**), between the origin of the reference coordinate system and the origin of inclined plane**. EU, EV** and **EW** are respectively the 3 translation components along X,Y and Z axis.
- **2nd step**: Rotation of the previous shifted frame, *obtained in step 1st*, by an angle **EC** around axis z.
- **3rd step**: Rotation of the *current frame, obtained in step 2nd*, by the angle **EB** around axis y'.
- **4th step**: Rotation of the *current frame, obtained in step 3rd*, by the angle **EA** around axis x".



It is important to note that iPLANE function operates correctly only with this definition of the successive transformations (translation of origin, rotation around Z axis, then around Y axis, then around X axis).

4.3 Prerequisites

iPLANE function requires having:

- The option of validating Inclined plane machining **(FXSO000914)**

The data of kinematic structure, linked to the iPLANE function, have to describe accurately the actual Five-axes machine.

The values of kinematic offsets should be very close to the exact dimensions to guaranty the Five-axes machining accuracy.

A kinematic measuring cycle248 with 3D touch probe is available in option for calibrating the Five-axes kinematics supported by RTCP function.

4.4 Interface for design of iPLANE commissioning

Flexium Tools offers a new design for customizing data managed by iPLANE function.

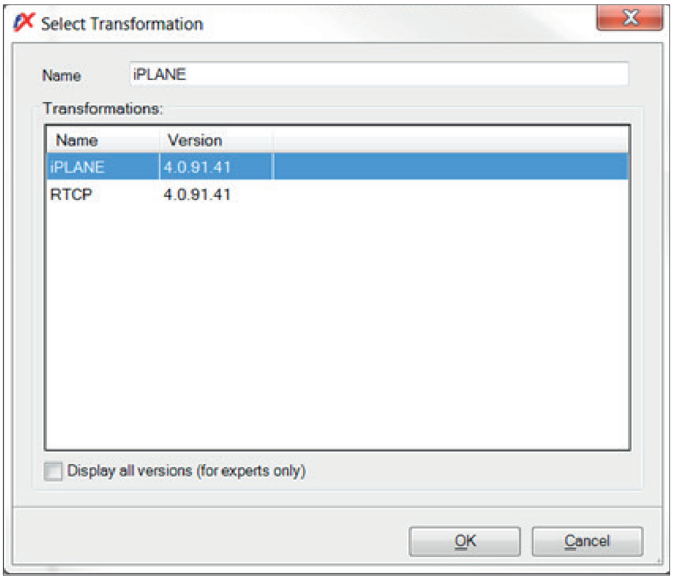
The *"Transformation"* tab provides access to the tool of both iPLANE customization and RTCP customization projects.

The *"Add Trans"* button opens a library of configurations linked to a iPLANE customization project.

Project selection dialog

By default, two most current projects are displayed in this dialog:

- iPLANE project dedicated for customization of an inclined plane function.
- RTCP project dedicated for customization of a RTCP function.

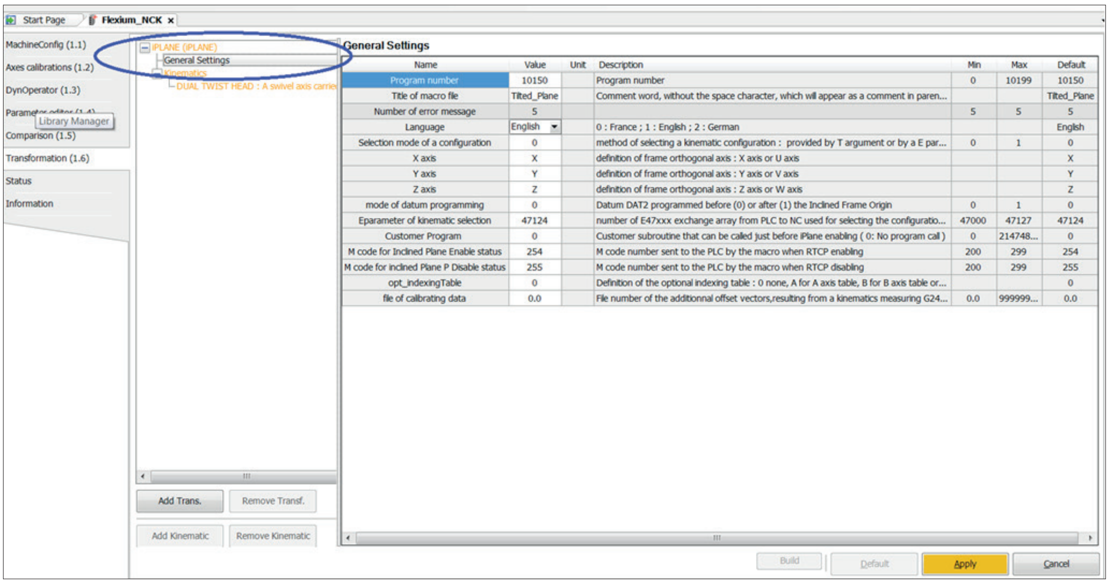


After selecting a iPLANE project, two folders are displayed under the root of iPLANE tree:

- *"General Settings"* folder for editing the common parameters of the project .
- *"Kinematics"* folder from which you can add some kinematic configurations.

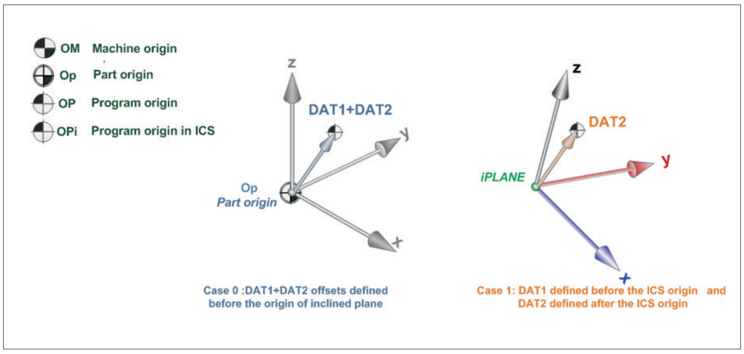
The opening of a second project is permitted and the user can customize in the two RTCP or iPlane functions in a same work session.

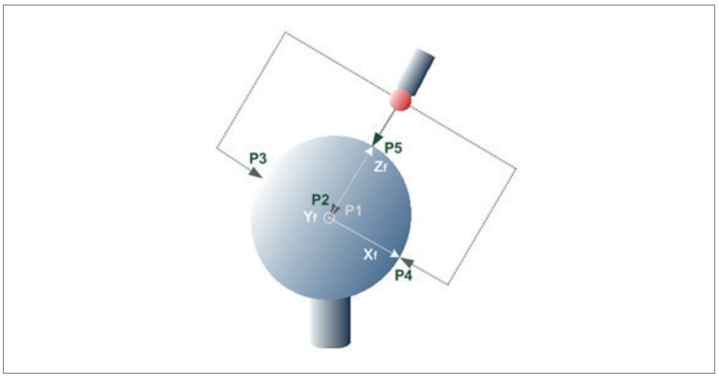
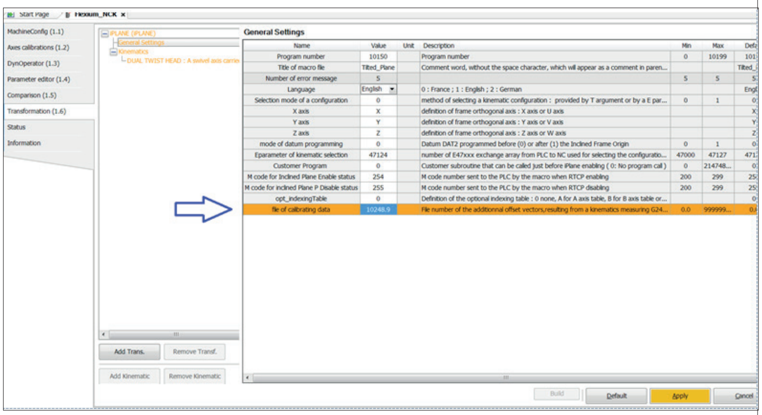
4.4.1 General Settings dialog



This dialog displays a list of common parameters of which are summarized in the tables below:

Variable	Description
Method of selection	Defines the method of selection of a kinematic configuration: **0** : Stands the choice is made by programming Tn in the call of iPLANE function, with ‘n’ an integer from 1 and up to 8 customizable configurations. **1** : Stands a selection made by the PLC with external parameter. Default value: 0
E parameter of kinematic selection	Specifies the address of E parameter of exchange array from PLC to the NC, assigned to the selection of kinematic configuration. This address is only used in case of a selection made by E parameter. Default value: 47124. Value stands that content of external parameter E47124 gives the number of the selected configuration (rank of configuration).
X axis	Specifies the name of the first axis of the Cartesian coordinate system (CCS) in which the inclined plane is activated. Two axes are proposed: X or U. Default value: X.
Y axis	Specifies the name of the second axis of CCS in which the inclined plane is activated. Two axes are proposed: Y or V. Default value: Y.
Z axis	Specifies the name of the third axis of CCS in which the inclined plane is activated. Two axes are proposed: Z or W. Default value : Z.

Mode of datum programming	This variable specifies how DAT2 is applied when an inclined plane is associated with the RTCP function (see chapter 2.6.4).  0: stands DAT2 is applied before the declaration of iPLANE. 1: stands DAT2 is applied after the declaration of iPLANE. Default value: 0
M code for iPLANE enable status	Number of M function sent to the PLC by the application when iPLANE is enabled. Default value: 254.
M code for iPLANE disable status	Number of M function sent to the PLC by the application when iPLANE is disabled. Default value: 255.
Customer program	Number of an user program called by iPLANE function just before enabling the inclined coordinate system. A null value means no file Default value: 0 (none) **CAUTION. If this macro file doesn’t exist, error 25 is generated in the inclined plane definition block.**

File of calibrating data	Optional calibration of Five-axes machine:  In order that iPLANE function is now able to manage automatically the output results of a kinematic optimization cycle (Kinematics measuring *Cycle248* with 3D touch probe). This variable specifies the *Cycle248 Data File* number containing the Five-axes calibration results (the number of file: %10248.9). Default value: 0 (no file) **CAUTION: If this file doesn't exist, error 25 is generated in the subroutine call block.** 
Optional indexing table	This variable specifies the symbolic name of an optional table.  Possible choices: 0 (none), A , B or C. Default value: 0 (none).

4.4.2 Kinematic edition dialog

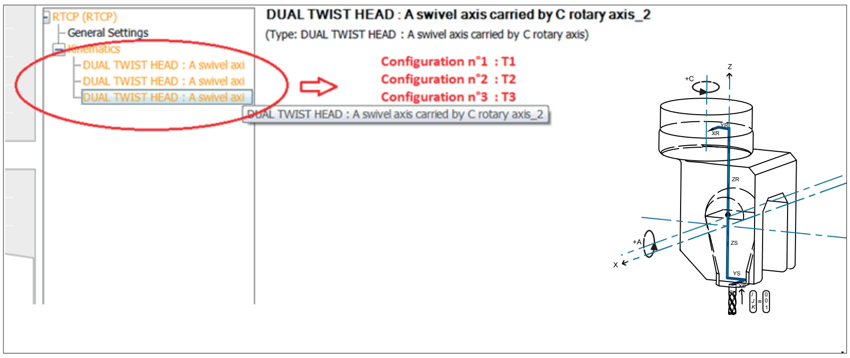
See the chapter 3.4.3.

4.5 Programming iPLANE

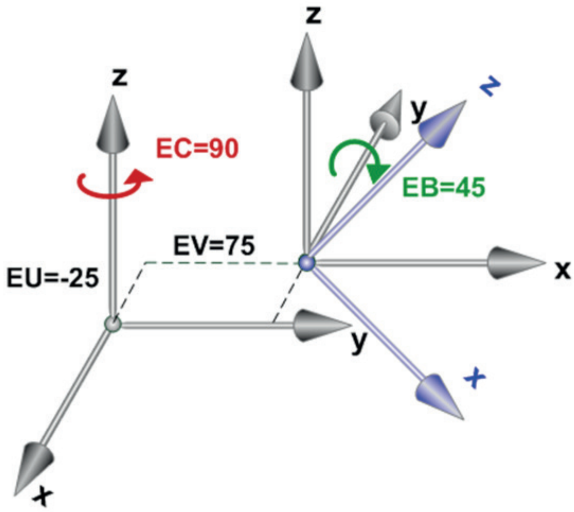
4.5.1 Specification of a iPLANE with respect to the MCS

Syntax

G150 EA…EB…EC… EU…EV…EW…[T]

<table>
<tr><th>G150</th><th>G function of iPLANE.</th></tr>
<tr><td>EA..., EB..., EC...</td><td> Specify the three Euler angles of orientation of the new coordinate system. The presence of one of these three angles is mandatory. If an angle is not specified, a zero value is assigned to it. The angles are expression in degrees. </td></tr>
<tr><td>EU..., EV..., EW...</td><td> Specify the position of inclined coordinate system origin with respect to the part program origin. EU, EV and EW are respectively the offsets of origin along the X, Y and Z axis direction. At least one of these three arguments must be specified. These parameters are expressed in mm. </td></tr>
<tr><td>T (optional)</td><td> Code number of the kinematic configuration among eight max configurations.


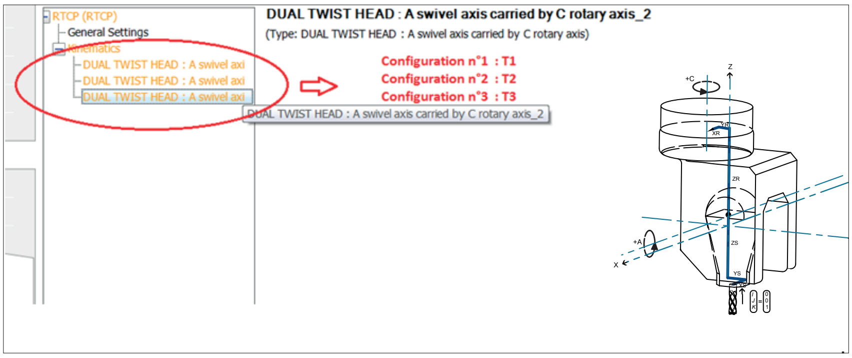
The setup number is in related to the rank of kinematic declaration. If T argument is omitted, the function chooses the previously programmed otherwise the first setup will be selected. </td></tr>
</table>

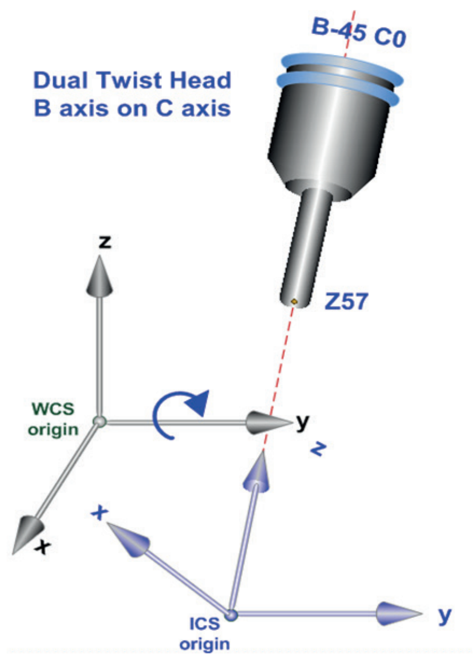
Example	**G150 EC90 EB45 EU-25 EV75 T2** This statement enables an iPLANE function, set with the second kinematic configuration (T2) and defines a inclined coordinate system whose origin is located at -25 mm along X and at 75 mm along Y from the initial workpiece origin. The new frame (ICS) orientation is given by two successive rotations: the first rotation of 90 degrees around the Z axis followed by a rotation of 45 degrees around Y' axis. 

4.5.2 Specification of an iPLANE after positioning the tool in a known point of the ICS

Syntax

G150 X…Y…Z…EA…EB…EC…D…[T]

G150	G function of iPLANE.
X..., Y..., Z...	Specify the position of the tool tip position with respect to the inclined coordinate system (ICS) If one of coordinates is not programmed, a zero value is assigned to it. The OEM selects the axes of the default Cartesian coordinate system that will be used for the iPLANE function (XYZ, UYZ, XVZ,…). The axes can be changed by programming. Examples: if the XVZ Cartesian coordinate system is chosen as default during the iPLANE integration. **G150 EA90 EU10** block defines an inclined plane on the default coordinate system, i.e. on X,V and Z. **G150 U..Y..EA90 EU10** block defines the same inclined plane on axes U,Y and the third axis of the default coordinates, i.e. Z.
EA..., EB..., EC...	Specify the three Euler angles of orientation of the new coordinate system. The presence of one of these three angles is mandatory. If an angle is not specified, a zero value is assigned to it. The angles are expression in degrees.
D...	Tool offset assigned to the current tool. This argument must be specified.
T (optional)	Code number of the kinematic configuration among eight max configurations. If T argument is omitted, the first configuration is chosen (T1).  The setup number is in related to the rank of kinematic declaration. If T argument is omitted, the function chooses the previously programmed (saved in the L919 variable equivalent to addresses T) otherwise the first setup will be selected.

Example	Consider a dual twist of which the swiveling axis is positioned at -45 degrees as shown in opposite figure. **G150 Z57 EB-45 D17 T2** This statement specifies an inclined coordinate system in which the tool tip is located at 0mm in X, 0mm in Y and 57mm in Z with respect to the ICS. The origin position of iPLANE is computed knowing the current tool and its tool offset given by D17.The new ICS orientation is given by a rotation of -45 degrees around the Y axis. The kinematic setup considered by the iPLANE function is programmed by T2. 

4.5.3 Cancelling the iPLANE function

Syntax

G150 S0

G150	**G function of iPLANE.**
S0	This function cancels the inclined plane function. Subsequently the programmed movements of axes in the machine coordinate system (MCS).

4.5.4 Re-enabling the iPLANE function

Syntax

G150 S1

G150	**G function of iPLANE.**
S1	This function reactivates the last active inclined plane (previously specified by following Syntaxs G150 EA EB EC EU EV EW T or G150 EA EB EC D T) and cancelled by G150 S0, a reset or.

4.5.5 Enabling a TOOL ICS from the current tool orientation

Syntax

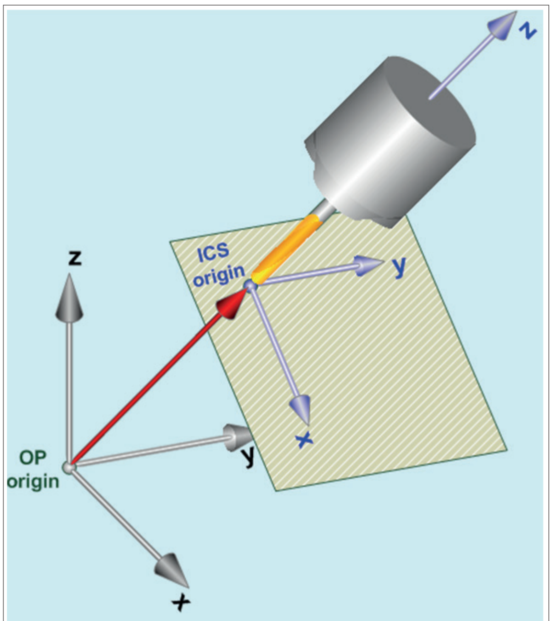
G150 R1 D…

G150	**G function of iPLANE.**
R1	From the current tool orientation, this function computes an inclined coordinate system (ICS) whose the origin coincides with the tool tip and whose the Z axis is directed along the tool axis direction. This Five-axes function allows programming an axial retraction or a lateral motion of the tool according to the frame TOOL ICS, in JOG mode or in MDI programming. Its use is restricted to 5-axis kinematics having a single-twist head or a dual-twist head. Otherwise, the following error message is set (Error 515 extension 2): *“Tool frame computation doesn’t support this Five-axes kinematics”.*
D...	Tool offset assigned to the current tool. This argument must be specified.

From the current orientation of the tool, the Five-axes function.

G150 R1 D…, calculates a set of three Euler angles defining the frame TOOL ICS:

- EA angle around the X axis,
- EB angle around the Y axis and
- EC angle around the Z axis, and activates this frame Tool ICS whose origin coincides with the tool tip.



G150 R1 D : Tool frame activation from the current tool orientation

Unlike the retraction function of RTCP (by a G151 R1 D statement), the activation of the tool frame by means G150 R1 D statement saves the current context. In other words, a recall of previous inclined plane features G150 S1 programmed after an axial retraction restores the retraction frame.

Example

`GG90 G1 F G16R+`	
`D...`	Tool corrector
`X... Y...Z...`	
`A30 C0`	
`G150 EA30 EU10.5 T1`	
`A...C...Z...X...Y...`	
`A...C...Z...X...Y...`	Machining with respect to the ICS
`...`	
`G150 R1 D…`	Compute a frame TOOL ICS and activate the RTCP
`Z10 F200`	Retraction in incremental mode along the Z axis direction of the frame
`...`	TOOL ICS. G91 programming is optional because the axis motion is
`...`	expressed in the frame TOOL ICS.
`G150 S0`	Reset iPLANE
`X...Y...Z...`	
`A...C...`	
`G150 R1 D…`	Compute and enable a new frame TOOL ICS
`X10 F200`	Lateral motion

Important notes about iPLANE function

- When the iPLANE is enabled, the Z axis of the new ICS has to be parallel to the tool axis direction.
- Any change of tool orientation when iPLANE is enabled is not permitted at the risk of causing machining geometry errors.
- An active iPLANE remains enabled even after a reset or a program end M02. It can only be cancelled by programming G150 S0 or reinitializing the CNC.
- When power is restored after a power failure, it is possible to activate an inclined plane or to re-enable the last activated iPLANE function before homing is completed on the axes (for instance, to retract a tool in the same orientation after a power failure).
- When the iPLANE is enabled, homing on an axis is prohibited .
- Any change of the DAT1 or DAT2 datum shifts, either directly or by programming of the corresponding external parameters (E60xxx and E61xxx), is prohibited when the inclined plane is enabled.
- Programming of movements in G52, i.e. in absolute coordinate referenced to the measurement origin, is prohibited if the iPLANE is enabled, except if parameter P7N1 Bit6 =1 (move in G52 according to the MCS when in iPLANE is active).
- Programming of a movement in G53 (DAT1 or DAT2 datum offsets cancelled) is prohibited if the iPLANE is enabled.

Page deliberately empty

Page deliberately empty

Summary

1
2
3
4
A

Page deliberately empty

5 Kinematic measuring cycle248 with 3D touch probe

5.1 General

In Five-axes machining, the Flexium+ Numerical controller uses kinematic transformations to give the positioning accuracy and the desired orientation of the tool center along the toolpath.

The kinematic model, involved in this transformation, is described by two ideal pivot-axes (two revolute joints) and two offset vectors to locate it.

It is therefore necessary to identify precisely the location of pivot points of theses rotary axes.

The kinematics measuring cycle248 enables the measurements[1] of spatial deviations for indexing positions of rotary axes involved in the 5 axis transformation and optimizes the component values of geometric offset vectors. The corrections of pivot point parameters are automatically saved in the Flexium controller.

The benefits are:

- An improvement rotary axis performance by eliminating the positioning and orientation errors coming from the 5 axis transformation.
- A cost effective application for checking the precision of the rotary axes.

This cycle requires a calibrated 3D touch probe and a single calibration sphere mounted on the work-piece table.

A calibration sphere, clamped on the work-piece table, is used as a reference feature for measurements

A calibrated 3D touch probe mounted in spindle

The cycle measures the characteristics of the sphere by probing its surface for various angular positions of the rotary axes.

After the measuring step, the cycle computes the corrections of kinematic offset components.

(1) The cycle248 is an indirect calibration method using artefacts to estimate the geometric errors and assuming these errors are mainly due to an inaccurate modeling of TCP control and assuming the main axes of the machine are previously calibrated.
The error components are not directly measured but computed through inverse kinematics model.

5.2 Prerequisites

The measuring cycle248 is an option and is available from version 4.1 of the Flexium system including the new generation of RTCP macro file (Rotate Tool Center Point) or iPlane macro file (Inclined Frame function).

The data of the configuration file linked to the RTCP function, accurately describe the actual kinematics of the 5-axis machine. The nominal values of kinematic offsets are approximatively known.

- The 3D touch probe must be calibrated before starting the measuring cycle.
- The calibration sphere is clamped on the machine table. Its diameter is known with high precision.
- The geometry of Cartesian axes is assumed near perfect, i.e. the linear error motions, the angular error motions, the straightness errors and the squareness errors have been previously compensated through a calibrating procedure by laser interferometer.

5.3 Application range

Cycle248 is intended to measure the deviations between the kinematic model involving rotary axes and the kinematic characteristics of the Five-axes machine but also to adjust the data of the kinematic modeling relevant to Five-axes transformation.

The new settings are automatically taken over by the RCTP function or by the inclined frame function.

Kinematic managed by this cycle of measures are those managed by the RTCP function and Inclined Plane function. A list of allowed kinematic is given in annex A.

It requires having also:

- A Flexium+ system from version 4.1.0
- The RTCP transformation option **(FXSO000154)**
- The recording current parameters values option **(FXSO000511)**
- The on the fly measurement/probing option **(FXSO000520)**
- The structured and symbolic programming option **(FXSO000535)**

This cycle determines the geometric errors offsets from nominal values previously set during commissioning. This cycle provides is an optimizing method of kinematic offsets of a Five-axes machine.

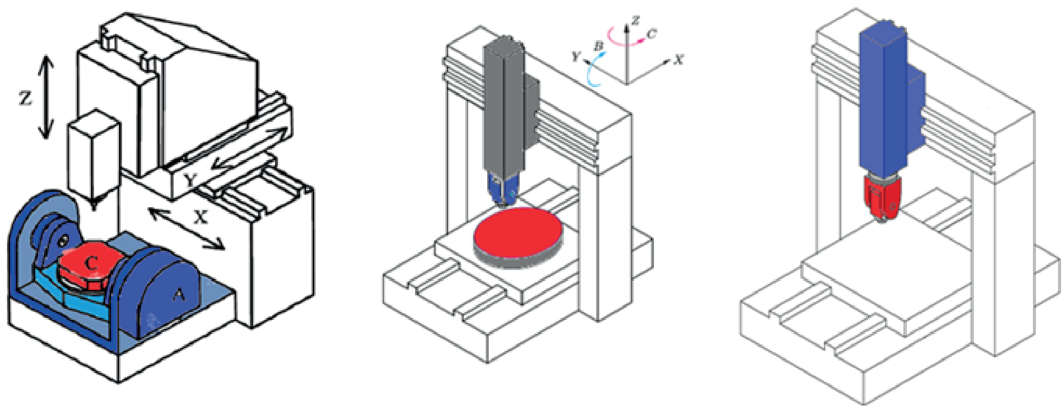
5.4 Targeted kinematic chains

The cycle is dedicated to Five-axes machines with the following configurations:

- A dual twist head but also a head with nutating[(1)] spindle
- A tilting turntable
- A swivel head and a rotary table combined.

(1) The word of "nutating" meaning an oscillatory motion or a slow spinning about an axis of a rotating body, a nutating head has a rotating pivot axis in an inclined plane.

The cycle is also suitable for conventional 4-axes machine, equipped with either a single turntable or with a single swiveling axis head.



Example of configuration

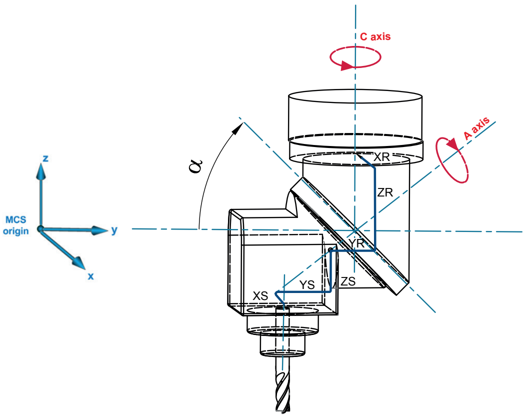
The description rule for a Five-axes kinematic is critical and has been changed in respect to previous controller versions, it starts from the spindle tool holder to the machine base and from the machine base to the workpiece table.

1
2
3
4
A

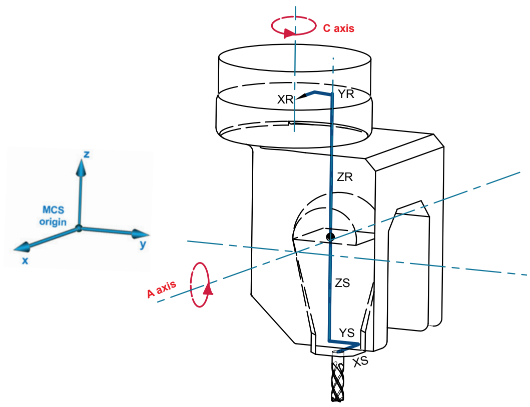
The first vector of kinematic offsets features either a swiveling axis, from the tool holder to the pivot axis or *a tilting table from the machine frame to the pivot axis. (XS, YS and ZS are its spatial components along xyz direction).*

The second vector describes the kinematic offsets between the two pivot axes (XR, YR and ZR are its spatial components along xyz direction).

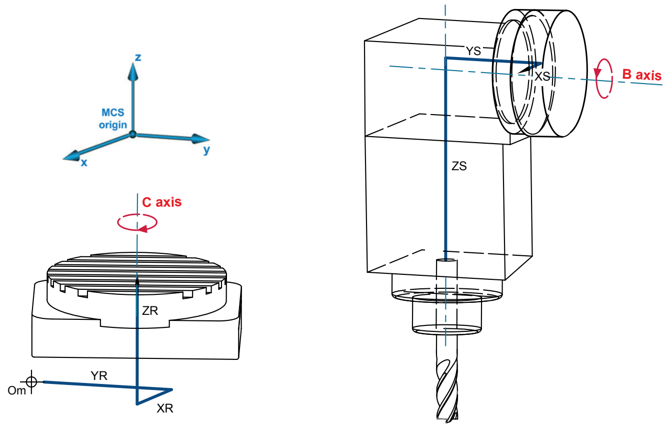
These two vectors are expressed with respect to machine frame O-xyz.x.



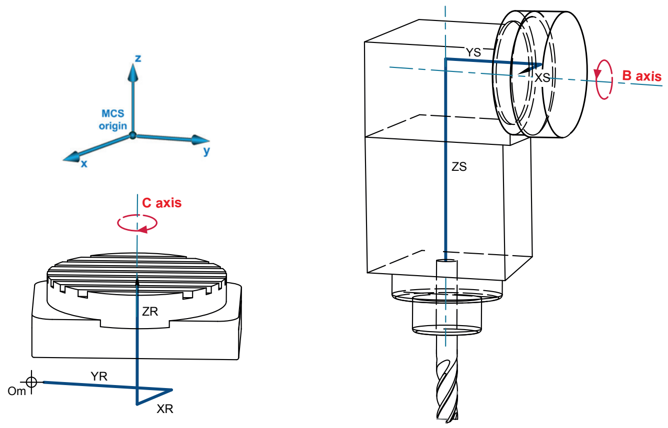
Shift vectors of a nutating head



Shift vectors of a dual twist head



Shift vectors of a turntable and single head combined



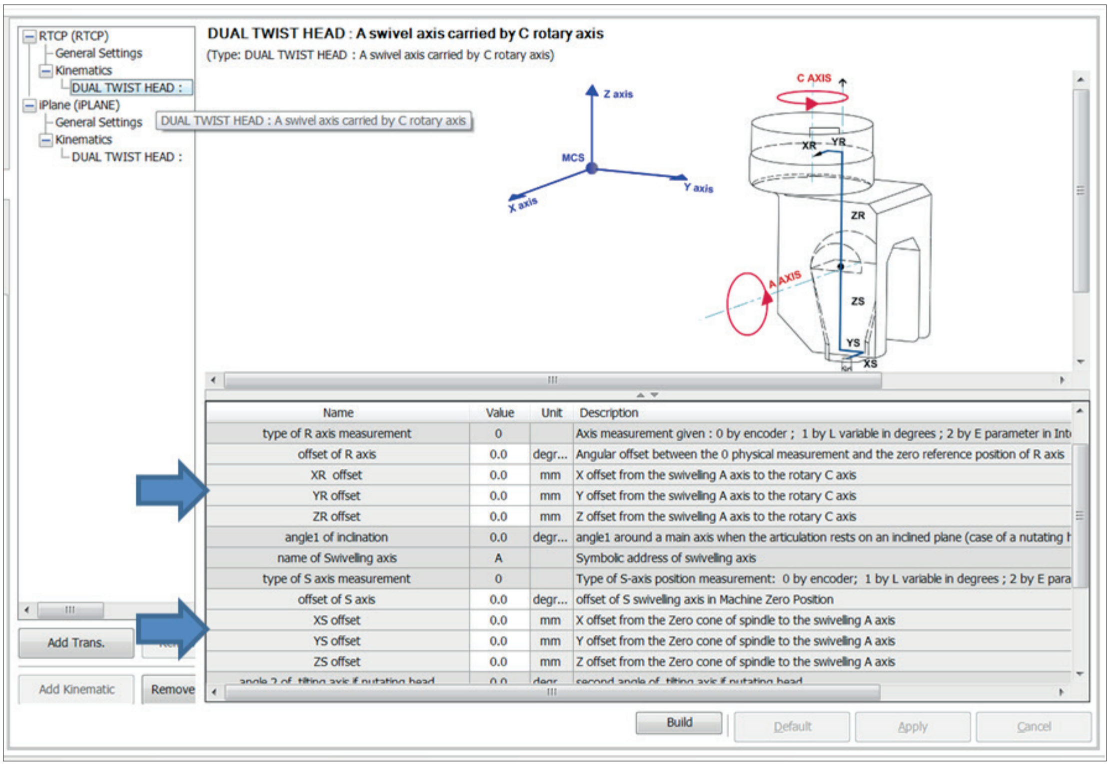
Shift vectors of a turntable and single head combined

The different configurations accepted by Cycle248 are those managed by the RTCP function and described in annex A.

5.5 Design of RTCP commissioning tool

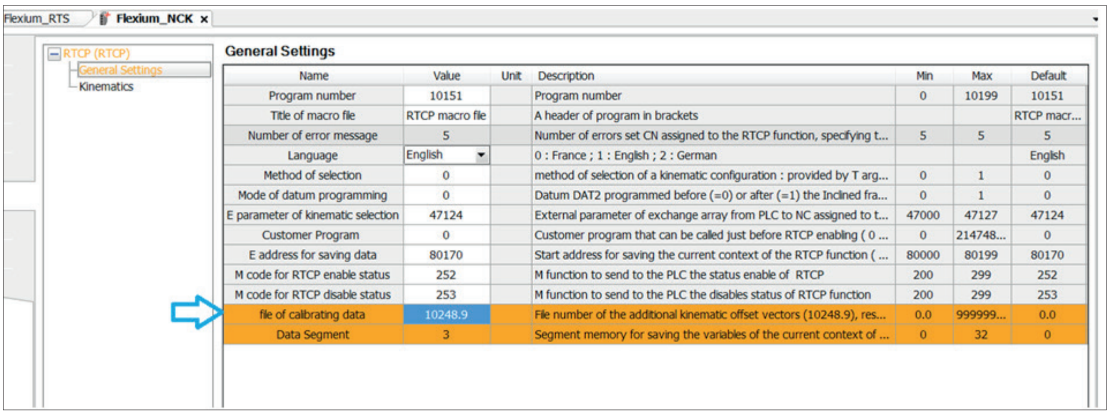
FXTools offers a new design for customizing data managed by the RTCP function.

Edition of the kinematic offsets nominal values is managed by this instrument.



In order that RTCP function is now able to manage automatically the output results of a kinematic optimization cycle (Kinematics measuring *Cycle248 with 3D touch probe*).

This variable specifies the *Cycle248 Data File* number containing the Five-axes calibration results (the number of file: %10248.9).



5.6 Procedure of kinematic measurement

The procedure of kinematic measurement begins by the swiveling/tilting axis and finishes by the second rotary axis. These sequences must be respected.

In case of one axis, the cycle performs the measurements according to the characteristics of this axis (swiveling/tilting axis or rotary axis).

We recommend repeating the measurement cycle:

- The first step consists on measuring the calibration sphere and calculating the corrections of kinematic offsets, available for the next step.
- The second step measures the residual errors of offset vectors when the RTCP function uses the optimized offsets obtained in step n°1. This last step is useful to compare the residual offsets with a tolerance value and to check the assumptions(1) about the accuracy of the structure of three orthogonal linear carrier axes (XYZ).

(1) The deviations caused by the linear carrier axes, such as the linear positioning errors, the straightness deviations or the squareness errors between axes should be compensated before starting the cycle of calibrating of 5 axis kinematics.

5.7 Recommendation of range indexing positions

It must be ensured that in the range of indexing positions of the R axis (rotary) and the range of the S axis (swiveling) no collision can occur during the movements of the calibration sphere to measure it in five points (four in a equatorial plane and one on the pole).

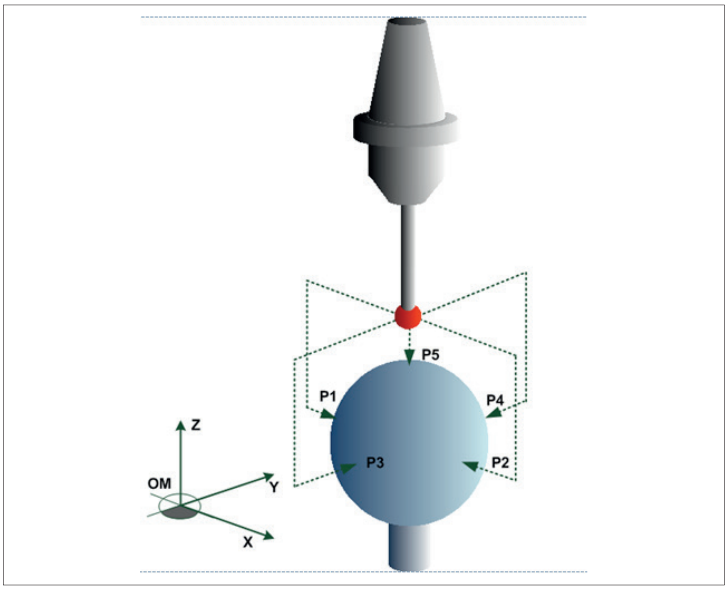
If the sphere is mounted on a work-piece rotary table, we advise to move away its location from the center of rotation in order to increase the sensitivity of deflection measurement.

5.8 Principe of measuring cycle248

The cycle measures 5 points on a calibrated spherical surface for a given tool orientation/inclination.

The 5 points are distributed on the spherical surface as following:

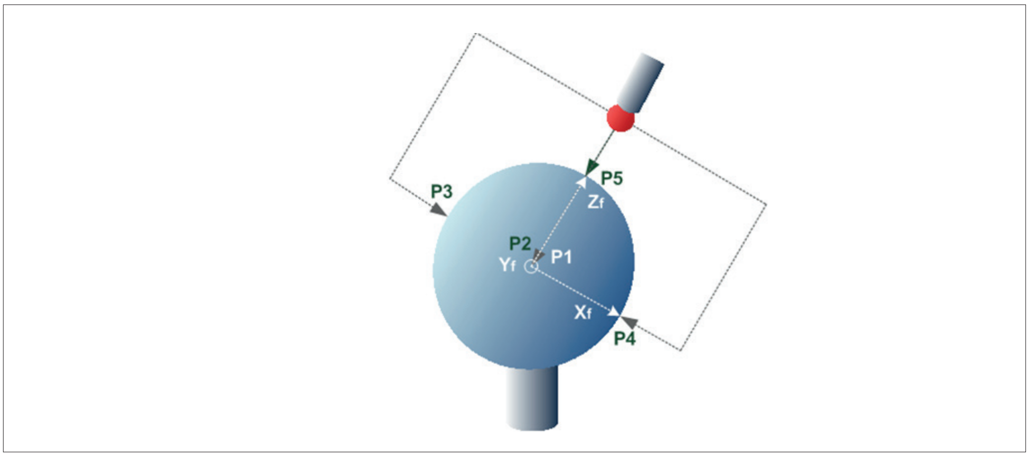
- 4 points in an inclined[(1)] "equatorial" plane.
- 1 point at a "top" of surface for latitude given by the tool axis direction.



[(1)] The equatorial plane through the center of the sphere can be tilted so that its perpendicular is collinear with the direction of the probe.

All movements are made according to the axes of a tilted frame coordinate system whose origin coincides with the sphere center and the z axis is parallel to the axial direction of probe.

After computing the sphere center at each indexed position (R,S) of rotating axes, the cycle estimates its spatial deviations as following : $deviation_{(R,S)} = center_{(R,S)} - center_{(0,0)}$



The next step estimates the geometric corrections to add to the nominal values of kinematics offset, with the assumption that the measured deviations are mainly linked to the kinematics modelling errors.

5.9 Pre-positioning of the probe center position

The probe, mounted on the milling spindle, is pre-positioned over the sphere before starting the cycle.

This starting position is recorded and interpreted as a retract position in each measurement cycle related to the swiveling axis (S axis) and to the rotary axis (R axis).

5.10 Approximated location of calibration sphere

The approach movement of the probe into the sphere requires to approximately knowing the rough location of the sphere center.

These coordinates are transmitted by programming either in call arguments X..Y..Z.. or either by reading them in the customization file linked to the measuring cycle.

These coordinate values are expressed with respect to the machine coordinate system.

5.11 Customization file Cycle248 Data File (10248.9)

This file contains the input data of the measuring cycle248.

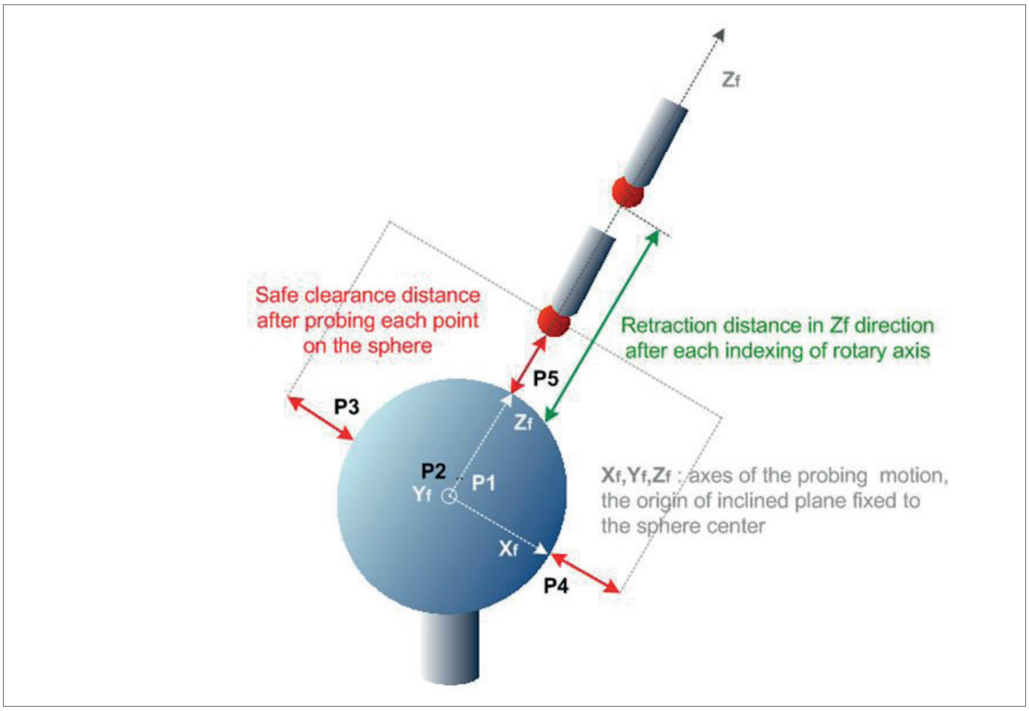
These data are user editable.

5.11.1 Specific data of the probing cycle

This file contains the input data of the measuring cycle248.

These data are user editable.

Probe n°1	Description
[diam_Ball]=25	Diameter of calibrated ball expressed in mm
[M_ON]=150	M function of probe activation
[Sp_rpm]=100	S spindle speed for indexing in rpm
[EC_ind] = 0	Start indexing angle of spindle in degrees
[Feed]=200	Low feed-rate of measuring in mm/min
[SafeClrD]=10	Safe clearance distance after probing each point on the sphere
[RetractDzf]=50	Retraction distance in Zf direction after each indexing of rotary axis
[axial_ovt]=0	Overtravel distance in axial direction of probe in mm
[ovt_OK]=1	Compensate the stylus overtravel distance : 0/1 value
[ctre_OK]=1	Swap the sphere center by its computed coordinates: 0/1 value



Two data sets of probing cycle are available. The selection of a parameter set is made by programming (with P argument).

5.11.2 Center sphere

The calibration sphere, clamped on the work-piece table, is used as a reference feature for measurements. Its location is given approximatively by its center for the approach motion.

Sphere center	Description
X=+277.7777	X coordinate value of the sphere center expressed in mm
Y=+107.7575	Y coordinate value
Z=+235.2245	Z coordinate value
[Xc/Om]=L923 **[Yc/Om]=L924** **[Zc/Om]=L925**	Allocation of the coordinates of sphere center in specific symbols

In the absence of programming of three arguments XYZ, giving the coordinates of the sphere center, the cycle reads them in the *Cycle248 Data File (%10248.9)*.

5.11.3 Programming of pivot axes

The sequences of rotary axis positioning (A, B or C) are programmed with specific block numbers.

As example for a kinematic having an A swivel axis carried by C rotary axis,

Block numbered N20 has to describe the sequence of indexing positions for the swiveling axis, the rotary axis being at zero reference position (i.e. C0):

N20 [nbS]=9	Indexing positions of swivel axis
A0.0 C0 **A22.5** **A45.0** **A67.5** **A90** **A-22.5** **A-45.0** **A-67.5** **A-90**	

And block numbered N30 haves to describe the sequence of indexing positions for the rotary axis, the swiveling axis being at zero reference position (i.e. A0):

N30 [nbR]=8	Indexing positions of the rotary axis
C45 A0 **C90** **C135** **C179** **C225** **C270** **C315** **C345**	

How choosing a number of measuring points?
The number of experiments affects the estimation of kinematic offsets and it reduces the bias error. A fine optimization will recommend having an equidistant distribution of indexing positions.

5.11.4 Results of kinematic optimizing

The results of kinematics measuring are written in *Cycle248 Data File*.

STEP1 : Results of the first measuring cycle	
N100	**Vector of offset corrections for Rotary R axis**
X=+0.0	Correction dXR of kinematics offset XR_mm
Y=+0.0	Correction dYR of kinematics offset YR_mm
Z=+0.0	Correction dZR of kinematics offset ZR_mm
[dXR]=L923 [dYR]=L924	
[dZR]=L925	
N101	**Vector of offset corrections for Swiveling S axis**
X=+0.0	Correction dXS of kinematics offset XS_mm
Y=+0.0	Correction dYS of kinematics offset YS_mm
Z=+0.0	Correction dZS of kinematics offset ZS_mm
[dXS]=L923 [dYS]=L924	
[dZS]=L925	
N102	

STEP2 : 2nd Cycle of measuring of residual errors	
N200	**Vector of offset corrections for Rotary R axis**
X=+0.0	Correction dXR of kinematics offset XR_mm
Y=+0.0	Correction dYR of kinematics offset YR_mm
Z=+0.0	Correction dZR of kinematics offset ZR_mm
[dXR]=L923 [dYR]=L924	
[dZR]=L925	
N201	**Vector of offset corrections for Swiveling S axis**
X=+0.0	Correction dXS of kinematics offset XS_mm
Y=+0.0	Correction dYS of kinematics offset YS_mm
Z=+0.0	Correction dZS of kinematics offset ZS_mm
[dXS]=L923 [dYS]=L924	
[dZS]=L925	
N202	

Auto-backup of 1st STEP results	
N300	**Backup vector of offset corrections for Rotary R axis**
X=+0.0	Correction dXR of kinematics offset XR_mm
Y=+0.0	Correction dYR of kinematics offset YR_mm
Z=+0.0	Correction dZR of kinematics offset ZR_mm
[dXR]=L923 [dYR]=L924	
[dZR]=L925	
N301	**Backup vector of offset corrections for Swiveling S axis**
X=+0.0	Correction dXS of kinematics offset XS_mm
Y=+0.0	Correction dYS of kinematics offset YS_mm
Z=+0.0	Correction dZS of kinematics offset ZS_mm
[dXS]=L923 [dYS]=L924	
[dZS]=L925	
N302	

5.12 Programming cycle248

The kinematic measurement cycle248 requires:

- The definition of the kinematic structure
- Choice between two sets of probe data.

5.12.1 Syntax of programming

The following syntax to start a measuring cycle is:

Syntax

G248 T.. EI.. [XYZ] [5] [P] [4] [F..] [4] [D..] [4]

G248	G function of volumetric error compensation.
T	Code number of the kinematic structure the code number is that given by the macro RTCP (see a second annex).
EI...	optional for an iterative method of kinematic optimizing in two steps: • 0: Compute the corrections of kinematic offset in the first step of measuring • 1: Compute the residual errors of kinematic offset in the second step of measuring • 2: Transfer the sum of error components of both steps in the *Cycle248 Data File*. This last step is optional and can only be achieved if the residual errors of kinematic offset are obtained in the desired tolerance.
[XYZ]	Coordinates of the sphere center/OM used for the approach motion (optional argument , an alternative is a reading of the sphere center position defined in the *Cycle248 Data File*.
[P]	Number of the set of probing data (1 or 2). The default value is 1.
[F]	Rapid feed rate for preposition of probe and for a safety retraction.
[D]	Tool corrector number given for the 3D touch probe.

5.12.2 Examples

- **Call 1st measuring cycle : measuring & estimation of kinematic offset errors**

G248 EI0 X100Y150Z255 P1 T1

With programming of sphere center coordinates with respect to the machine coordinate system (MCS). The cycle uses the data set of the first kinematic configuration (T1).
The rapid feed rate is that previously programmed.

G248 EI0 P1 T1 F5000 D100

The coordinates of sphere center are read from the *Cycle248 Data File*. The cycle uses the first kinematic configuration data set (T1). Rapid feed rate is given for the prepositioning and the safety retraction movements.

- **Call 2nd measuring cycle : measures/estimation of residual errors, the RTCP function is set with the new kinematic offset corrections from the 1st cycle**

G248 EI1 X100Y150Z255 P1 T2 D100

With programming of sphere center coordinates with respect to the machine frame. The cycle uses the data set of the second kinematic configuration (T2).

G248 EI1 P1 T1

The coordinates of sphere center are read from the data file. The cycle uses the data set of the first kinematic configuration (T1).

Page deliberately empty

Page deliberately empty

Summary

1

2

3

4

A

Page deliberately empty

6 Tool Vector Programming and 3D Workpiece Positioning Compensation

6.1 General

When a workpiece is installed on a machine, it can be very difficult to align it perfectly due to weight, structure, previous machining operation or any other reason. Any tilt quickly becomes a problem by generating a change in the incidence of the tool. A common solution is, after identifying offsets and tilt angles, to rework the part program to take into account misalignments, but this obviously takes some time.

To handle this issue, NUM provides the 3D WPC function (3D Work Piece Positioning Compensation).

The goal of this function is to compensate the position and orientation of a part to machine with respect to its theoretical values without having to reprocess the program by defining a Balanced Coordinate System (BCS) of the part with respect to an ideally positioned one.

This function currently only applies when RTCP is enabled.

In addition, the tool orientation defined in the part program should be constantly adapted during machining to maintain the desired tool orientation with respect to the ideally positioned part.

For this the function “Tool vector programming” later explained in this chapter should be used.

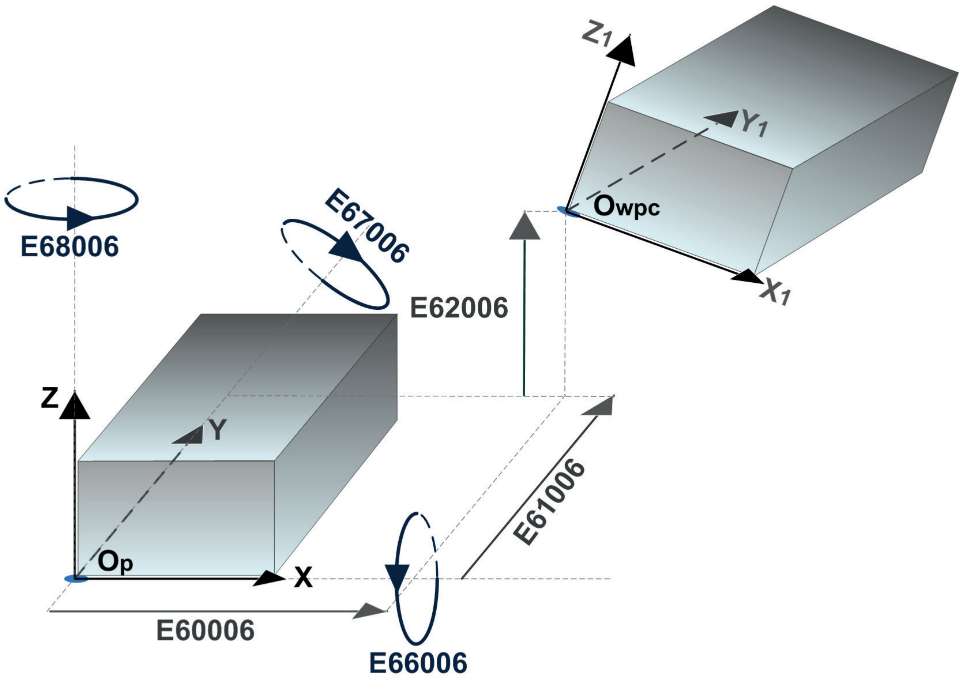
6.2 Implementation

The 3D WPC function only relies on six E parameters to define the part malposition

Shifts theoretical part referential : E60006 – E60001 – E62002

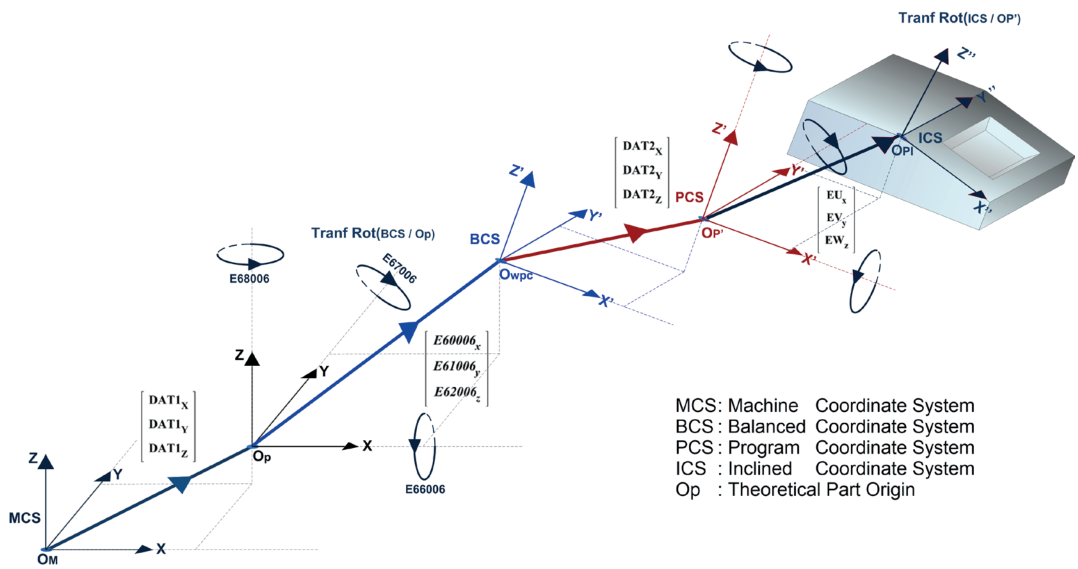
Angles (Pitch, Roll and Yaw) : E66006 – E67006 – E68006

These parameters are read write and can be entered by programming or on the HMI. They define the position and orientation of the BCS (Balanced Coordinate System) with respect to the theoretical Part Coordinate System.



The figure below represents the link between the different referential systems.

The referential Op(XYZ) represents the theoretical part referential along which the 3DWPC parameters are applied.



6.3 Tool orientation

We have seen that to correctly machine an off-positioned part it is necessary to adapt the tool orientation to maintain its theoretical orientation with respect to the part.

To do this the CNC must be aware of the tool orientation either explicitly or by recalculation as well as the mechanical structure of the machine.

The following mutually exclusive G codes have been created for this purpose:

G code	Function	Condition	Status after start-up or reset	Specific additional arguments?
G30	The rotary axes are programmed by their physical values. This is the classic current mode; it is created in order to cancel the other G32, G34, G34- when necessary.	No one	Active by default	No
G32	The tool orientation is programmed directly via an extended syntax to define the tool vector. The CNC will compute the physical position of the Tool head axes taking into account this vector and the kinematic head structure. The other rotary axes remain programmed by their dimensions.	RTCP enabled with kinematic number argument	Inactive by default	No. The expression of the tool vector uses a new programming syntax.
G34-	The rotary axes are programmed by their physical dimensions. Considering the Tool orientation axes, the NCK will constantly calculate the tool orientation from these coordinates and compensate it according to the part balancing parameters before moving these axes to their new position.	RTCP enabled with kinematic number argument	Inactive by default	No
G34	Programming the rotary axes by their physical dimensions. Considering the Tool orientation axes, the NCK will constantly calculate the tool orientation from these coordinates and compensate it according to the part balancing parameters and an inclined plane before moving these axes to their new position.	RTCP enabled with kinematic number argument	Inactive by default	No

We have seen that in order to compute the rotary axes dimensions (from the tool vector in G32 or recalculated in G34 / G34-) the NCK needs to be aware of the machine kinematic.

Therefore, a new parameter (N) defining this kinematic has been added to the G26 code. Like always, G26 is handled by the %10151 macro which must be recompiled using Flexium Tools 4.1.2 or above

Currently the above functions (excluding G30) are available for the two following kinematics:

Type 2: Dual twist A on C

Type 4: Dual twist B on C

6.4 Details on G codes

G30

G30 is intended to program the RTCP involved rotary axes by their **physical dimensions**.

It is the standard programming mode. This G code has been created for compatibility and to leave the possibility of cancelling the new G32 / G34 codes.

G32

G32 is intended to define the **tool vector directly** (abstract programming). From the expression of the tool vector, the NCK firmware will compute the dimensions of the RTCP rotary axes.

Expression of the Tool vector

The tool vector is programmed by its projections along the X, Y, Z axes of the balanced coordinate system or inclined coordinate system.

The programming addresses for the tool projections are respectively:

A_0 = …,

B_0 = ….,

C_0 = …

The values following the sign “=” can be expressed as usual by a numerical value, an L variable, an E parameter, a symbolic variable or an expression between {..}.

The three projections are not necessary: any missing projection is considered as null; but at least one projection must be programmed in order to get a tool vector orientation.

There is no need to norm the vector at a specific value, the NCK firmware norming it internally to 1.

Example

```
G30 A0 C0
G32
G1 X.. Y.. Z.. L0 =1/_SQRT3 A_0=-L0 B_0=-L0 C_0=L0
```

G34 -

G34 – is intended for programming the tool vector by the **physical dimensions** of the RTCP involved axes (like G30) but in this mode the NCK will adapt these values by computing in real time the tool orientation and adjust it taking into account the part balancing parameters.

G34

G34 is intended for programming the tool vector by the **physical dimensions** of the RTCP involved axes (like G30) but in this mode the NCK will adapt these values by computing in real time the tool orientation and adjust it taking into account the part balancing parameters and an inclined plane.

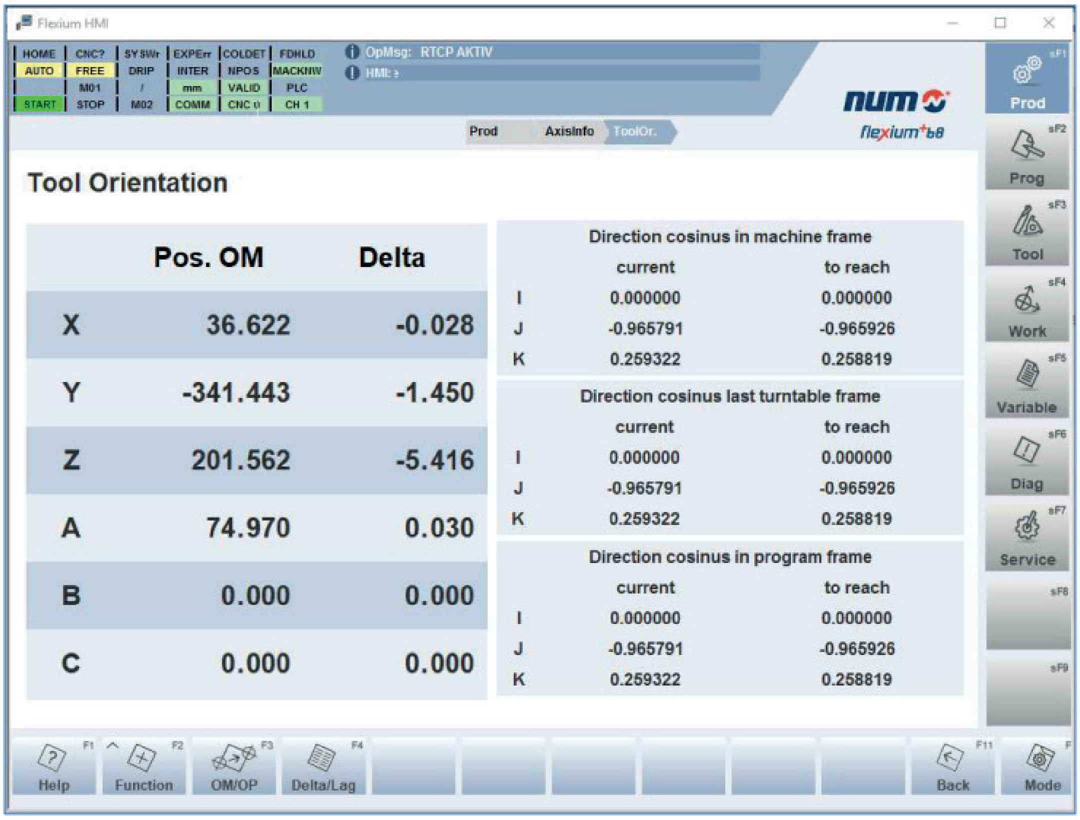
6.5 Complement

The current tool vector data is available as E parameters (read only):

- E790020, E79021, 79022 (in Machine coordinate system respectively X, Y and Z projections).
- E790023, E79024, 79025 (in Part coordinate system respectively X, Y and Z projections).
- E790026, E79027, 79028 (in Balanced coordinate system respectively X, Y and Z projections).

The display of current tool vector is available even outside of Tool vector programming – tool direction compensation. It is available as soon as a twist is present in the RTCP kinematic description. On the other hand, the display of the tool vector to reach at end of the block requires the programming of G32, G34-, or G34.

The tool orientation display is available on the HMI.



6.6 Precautions and limits of usage

3D Workpiece compensation and Tool Vector Programming is available as an option on the CNC

In the case of Tool vector programming, the tool direction compensation supposes small and gradual changes of the orientation programmed. Of course large changes are allowed for example to move to a start position with a particular orientation. In this case it should be handled by the CAM system at a safe distance from the part to avoid collision.

The strategy is to work by choosing the minimal relative path to travel for the RTCP rotary axes. In case of RTCP modulo rotary axes, the NCK forces the minimal relative path.

➔Important: If there is RTCP modulo rotary axis, when switching from {G32, G34-, G34} to G30 it is strongly recommended to set E11010 to 0 at the beginning of the part program in order force the minimal relative path continuously in G30.

Whatever the type of RTCP rotary axes, it is necessary to program at least once the RTCP rotary axes in absolute programming G90 before using G32, G34- or G34. The reason is that the previous dimensions programmed of these rotary axes should be available to calculate the possible relative paths.

Polynomial programming is allowed but not for RTCP rotary axes.

Examples

Using 3D WPC is done in five simple steps:

- Determine the part misalignment shifts and rotation by probing or any other mean
- Enter those value in the part program in G30 mode (standard mode)
- Make sure the rotary axes are programmed at least once in G90 G30 mode.
- Activates RTCP
- Activate G32 : Programming rotary axes via tool orientation

  or G34 / G34- in case the rotary axes are programmed by their physical coordinates

  (G34 allows for defining an additional inclined plane with ICS referential)

Programming in G32

```
%100(ROT_X45_Y45_Z45 NUM G34)()

(Define 3DWPC data)
E60006=0/U1 E61006=0/U1 E62006=0/U1 E63006=0/U1 E64006=0/U1 E65006=0/U1 E66006=0/
M1 E67006=0/M1 E68006=0/M1

G0 G30 M6 T1 D1
M3 S3000
G0 XYZAC

(Activates RTCP)
G151 T1 EU0 EV0 EW0 EA45 EB45 EC45

(Activates G32 - programming tool orientation)
G32

G00 X4.954 Y26.029 Z300.000 C_0=1
Y30.742                  B_0=.00099483  C_0=.99999950
Y31.269  Z179.958 B_0=.01539319  C_0=.99988151
Y32.314  Z179.898 B_0=.04722848  C_0=.99888411
…
X1.954  Y37.355  Z178.725 B_0=.22692328  C_0=.97391263
Y37.832  Z178.531 B_0=.24655926  C_0=.96912771
...
G151 S0 G30
M2
```

Programming in G34

```
%100(ROT_X45_Y45_Z45 G34)()

(Defines 3DWPC data)
E60006=0/U1 E61006=0/U1 E62006=0/U1 E63006=0/U1 E64006=0/U1 E65006=0/U1 E66006=0/
M1 E67006=0/M1 E68006=0/M1

G0 G30 M6 T1 D1
M3 S3000
G0 XYZAC

(Activates RTCP)
G151 T1 EU0 EV0 EW0 EA45 EB45 EC45

(Activates G34 – programming via rotary axes coordinates)
G34

(Programming physical axes coordinates)
G00 X4.954 Y26.029 Z300.000 A0.000 C0.000
G01 Z179.966
X4.660  Y26.044
        Y30.742       A-0.057
...
X1.953  Y32.834  Z179.845 A-3.651
...
G151 S0 G30
M2
```

Page deliberately empty

Page deliberately empty

1
2
3
4
A

Summary

Page deliberately empty

A Appendix

A.1 Kinematic structures managed

A.1.1 Kinematic structures managed by the RTCP and iPLANE functions

This list is not exhaustive and may evolve with the version of RTCP and iPLANE functions.

Format	Name	G2xx cycle	Unit or value
kin n°01	DUAL TWIST HEAD A ON B	R = B axis	S = A axis
kin n°02	DUAL TWIST HEAD A ON C	R = C axis	S = A axis
kin n°03	DUAL TWIST HEAD B ON A	R = A axis	S = B axis
kin n°04	DUAL TWIST HEAD B ON C	R = C axis	S = B axis
kin n°05	NUTATING ROTARY HEAD TILTED A ON B	R = B axis	S = A axis
kin n°06	NUTATING ROTARY HEAD TILTED A ON C	R = C axis	S = A axis
kin n°07	NUTATING ROTARY HEAD TILTED B ON C	R = C axis	S = B axis
kin n°08	NUTATING ROTARY HEAD B ON TILTED A	R = A axis	S = B axis
kin n°09	B ROTARY TABLE & A AXIS HEAD	R = B axis	S = A axis
kin n°10	C ROTARY TABLE & A AXIS HEAD	R = C axis	S = A axis
kin n°11	A ROTARY TABLE & B AXIS HEAD	R = A axis	S = B axis
kin n°12	C ROTARY TABLE & B AXIS HEAD	R = C axis	S = B axis
kin n°13	TILTING ROTARY TABLES A ON B	R = A axis	S = B axis
kin n°14	TILTING ROTARY TABLES A ON C	R = A axis	S = C axis
kin n°15	TILTING ROTARY TABLES B ON A	R = B axis	S = A axis
kin n°16	TILTING ROTARY TABLES B ON C	R = B axis	S = C axis
kin n°17	TILTING ROTARY TABLES C ON A	R = C axis	S = A axis
kin n°18	TILTING ROTARY TABLES C ON B	R = C axis	S = B axis
kin n°19	SWIVEL A AXIS HEAD		S = A axis
kin n°20	SWIVEL B AXIS HEAD		S = B axis
kin n°21	A ROTARY TABLE	R = A axis	
kin n°22	B ROTARY TABLE	R = B axis	
kin n°23	C ROTARY TABLE	R = C axis	

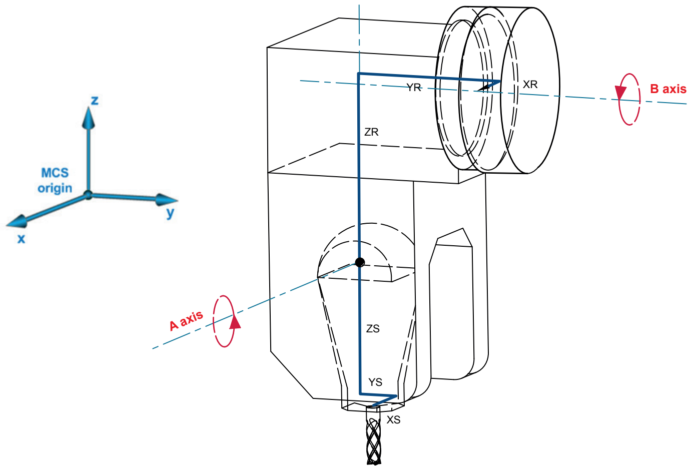
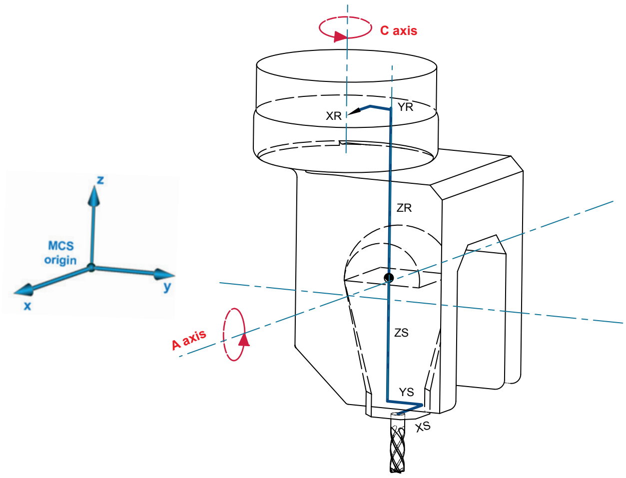
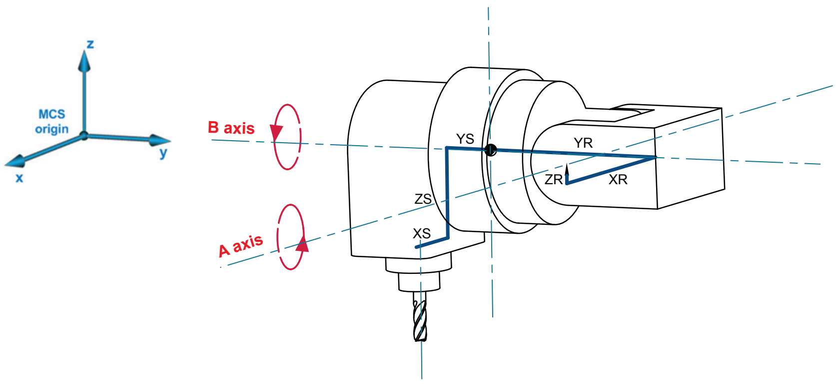
A.1.2 Kinematic structures managed by Cycle248

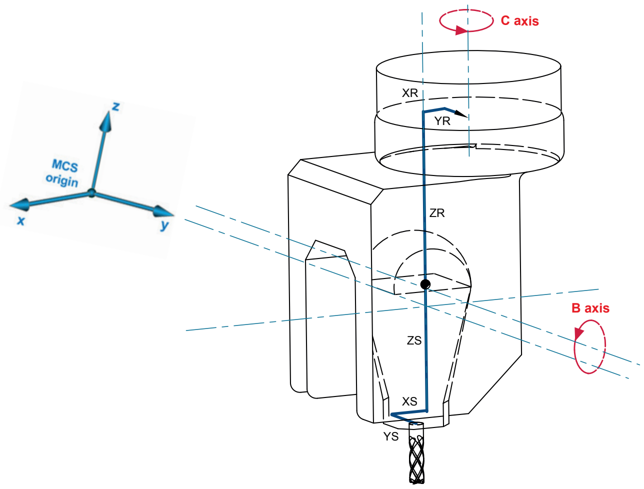
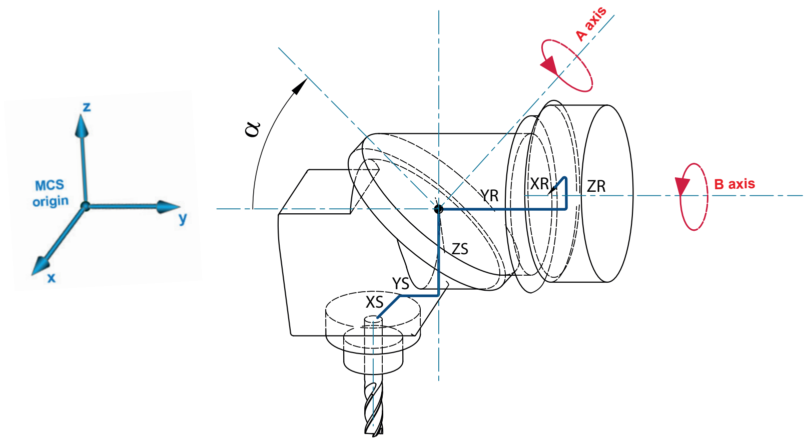
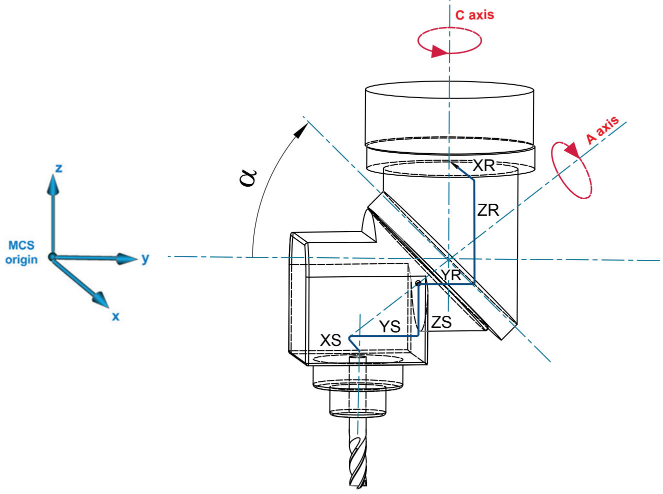
This list is not exhaustive and may evolve with the version of RTCP function.

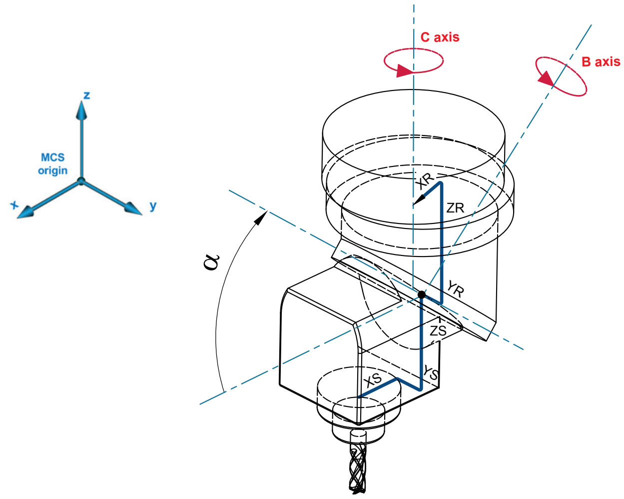
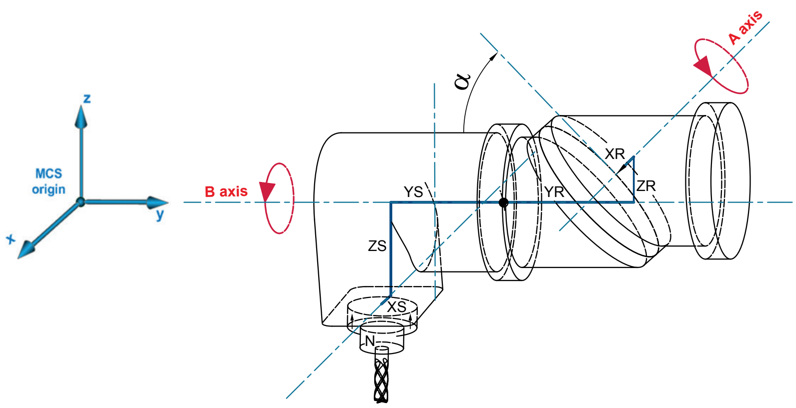
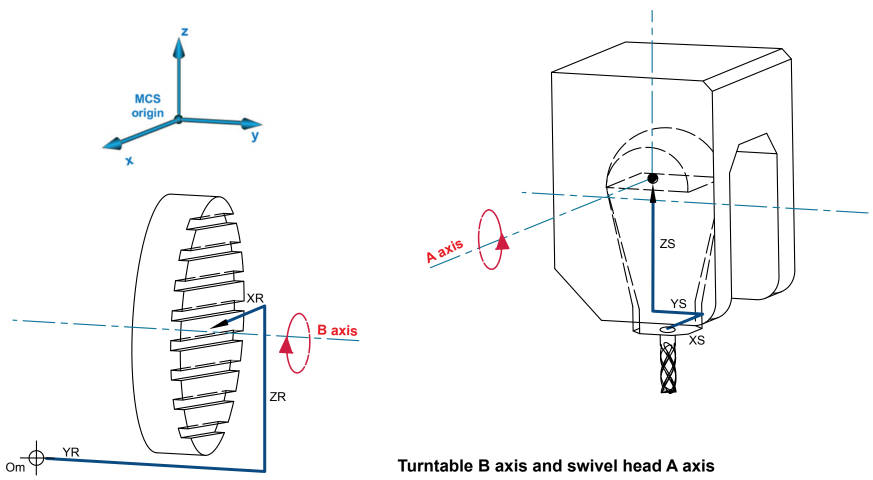
Format	Name	G2xx cycle	Unit or value
kin n°01	DUAL TWIST HEAD A ON B	R = B axis	S = A axis
kin n°02	DUAL TWIST HEAD A ON C	R = C axis	S = A axis
kin n°03	DUAL TWIST HEAD B ON A	R = A axis	S = B axis
kin n°04	DUAL TWIST HEAD B ON C	R = C axis	S = B axis
kin n°05	NUTATING ROTARY HEAD TILTED A ON B	R = B axis	S = A axis
kin n°06	NUTATING ROTARY HEAD TILTED A ON C	R = C axis	S = A axis
kin n°07	NUTATING ROTARY HEAD TILTED B ON C	R = C axis	S = B axis
kin n°08	NUTATING ROTARY HEAD B ON TILTED A	R = A axis	S = B axis
kin n°09	B ROTARY TABLE & A AXIS HEAD	R = B axis	S = A axis
kin n°10	C ROTARY TABLE & A AXIS HEAD	R = C axis	S = A axis
kin n°11	A ROTARY TABLE & B AXIS HEAD	R = A axis	S = B axis
kin n°12	C ROTARY TABLE & B AXIS HEAD	R = C axis	S = B axis
kin n°13	TILTING ROTARY TABLES A ON B	R = A axis	S = B axis
kin n°14	TILTING ROTARY TABLES A ON C	R = A axis	S = C axis
kin n°15	TILTING ROTARY TABLES B ON A	R = B axis	S = A axis
kin n°16	TILTING ROTARY TABLES B ON C	R = B axis	S = C axis
kin n°17	TILTING ROTARY TABLES C ON A	R = C axis	S = A axis
kin n°18	TILTING ROTARY TABLES C ON B	R = C axis	S = B axis
kin n°19	SWIVEL A AXIS HEAD		S = A axis
kin n°20	SWIVEL B AXIS HEAD		S = B axis
kin n°21	A ROTARY TABLE	R = A axis	
kin n°22	B ROTARY TABLE	R = B axis	
kin n°23	C ROTARY TABLE	R = C axis	

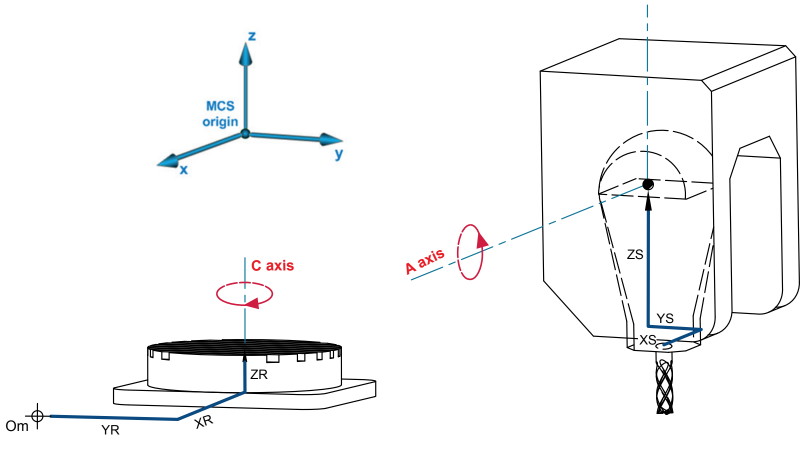
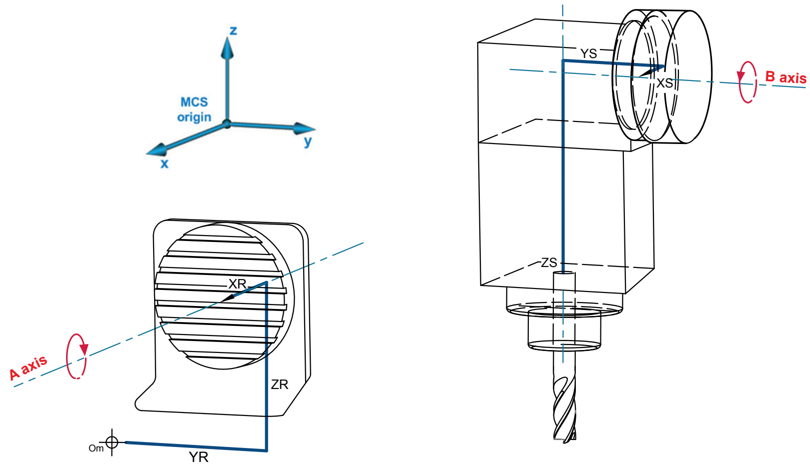
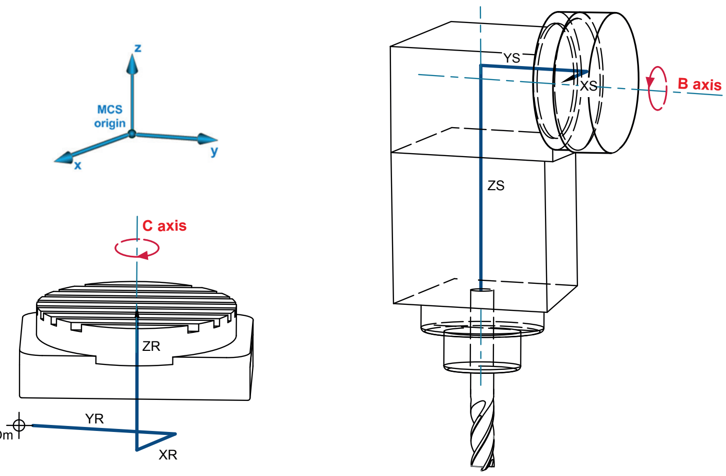
A.2 Summary of the reference postures of Five-axes kinematics

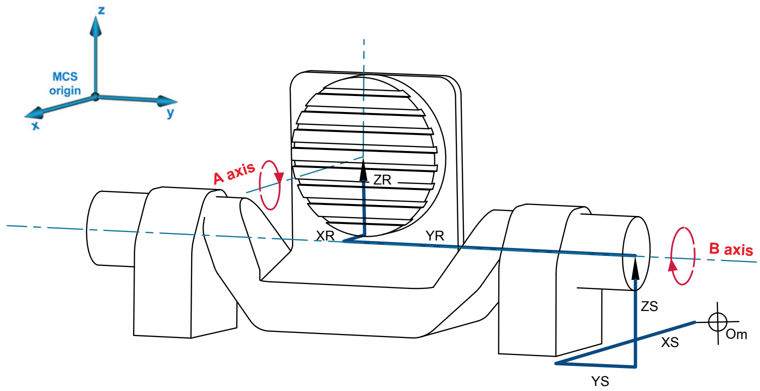
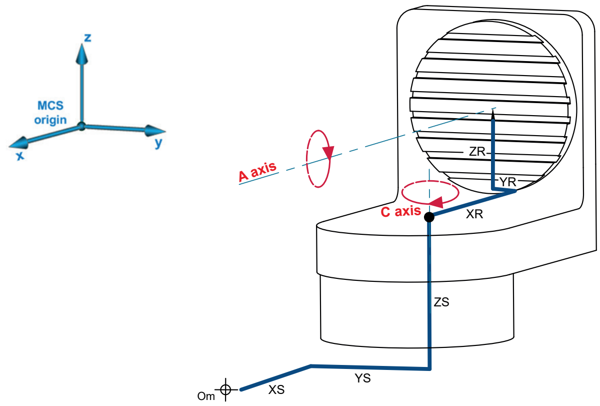
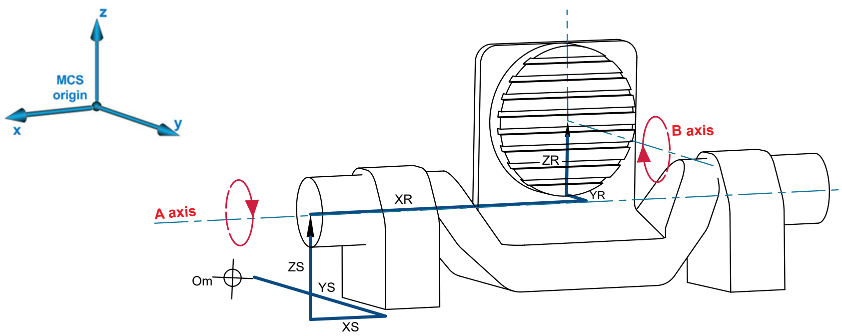
A reference position of a Five-axes kinematic defines its attitude when the position of each revolute axis is zero.

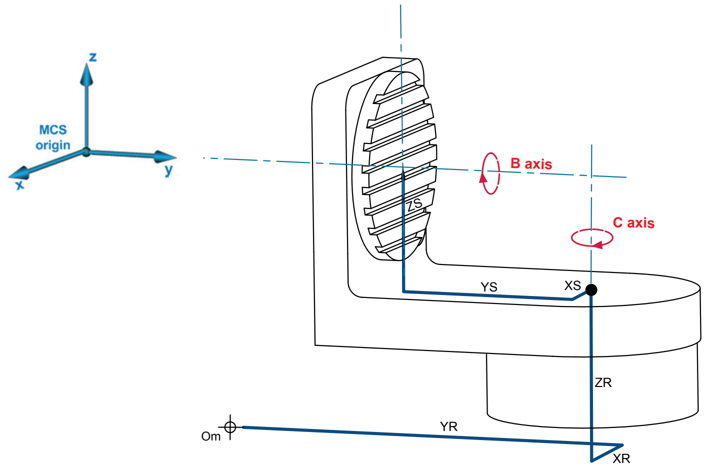
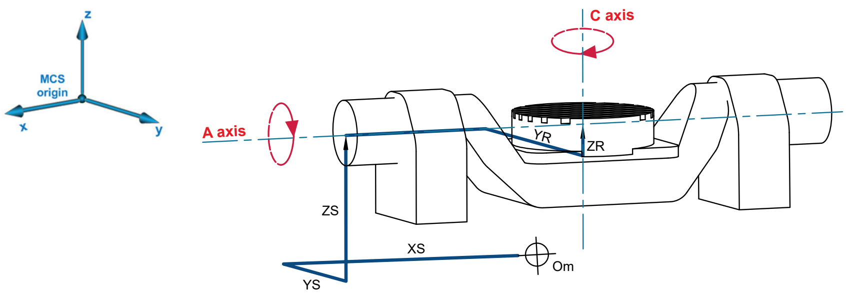
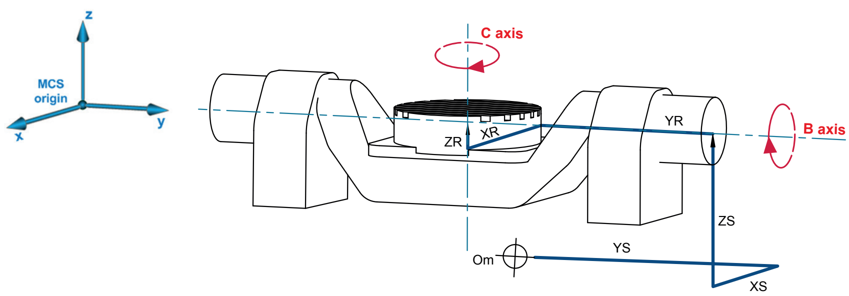
Type of kinematics	Representation of kinematic
Type 1	 Dual twist head: A axis on B axis
Type 2	 Dual twist head: A axis on C axis
Type 3	 Dual twist head: B axis on A axis

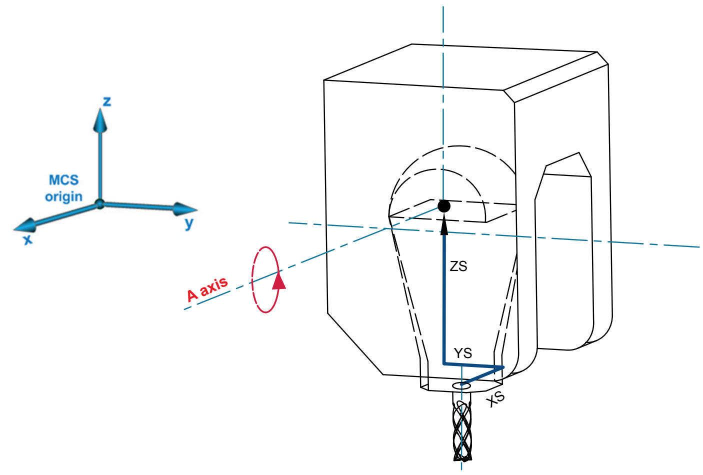
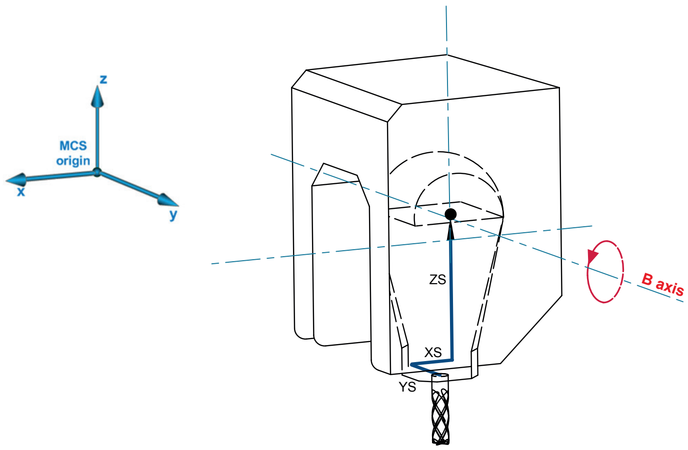
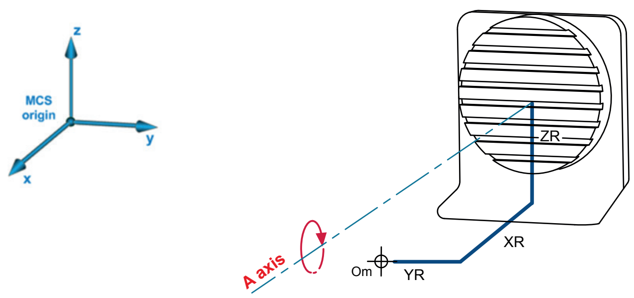
Type of kinematics	Representation of kinematic
Type 4	 Dual twist head: B axis on C axis
Type 5	 Dual twist head: A inclined on B axis
Type 6	 Dual twist head: A inclined axis on C axis

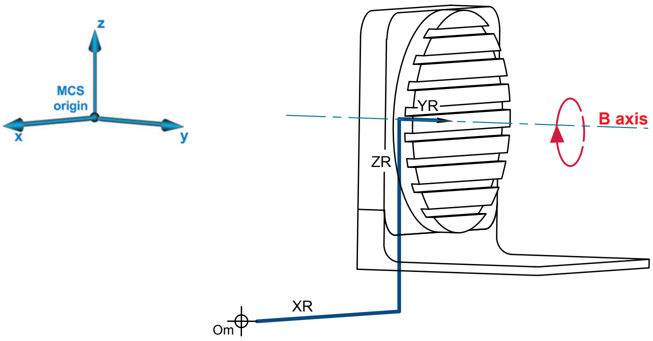
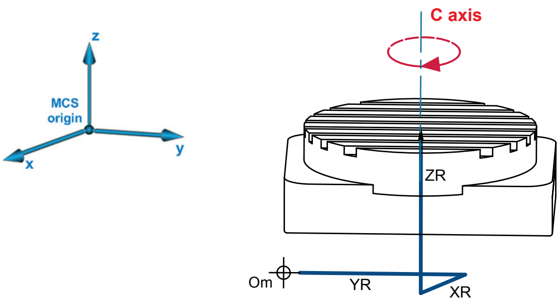
Type of kinematics	Representation of kinematic
Type 7	 Dual twist head: B inclined axis on C axis
Type 8	 Dual twist head: B axis on A inclined axis
Type 9	 Turntable B axis and swivel head A axis

Type of kinematics	Representation of kinematic
Type 10	 Turntable C axis and swivel head A axis
Type 11	 Turntable A axis and swivel head B axis
Type 12	 Turntable C axis and swivel head B axis

Type of kinematics	Representation of kinematic
Type 13	 Tilting turntable: A rotary table carried by B axis
Type 14	 Tilting turntable: A rotary table carried by C axis
Type 15	 Tilting turntable: B rotary table carried by A axis

Type of kinematics	Representation of kinematic
Type 16	 **Tilting turntable: B rotary table carried by C axis**
Type 17	 **Tilting turntable: C rotary table carried on A axis**
Type 18	 **Tilting turntable: C rotary table carried by B axis**

Type of kinematics	Representation of kinematic
Type 19	 **Simple twist head: A axis**
Type 20	 **Simple twist head: B axis**
Type 21	 **Turntable A axis**

Type of kinematics	Representation of kinematic
Type 22	 **Turntable B axis**
Type 23	 **Turntable C axis**

A.3 E80x parameters used by RTCP and by iPLANE functions

The RTCP function uses a series of 25 E Parameters for saving the current context of RTCP internal calculated variables.

If no memory segment is allocated, then the backup address begins at E80170 but can be customized by the machine manufacturer.

E parameter	Function	Description
E80170 to E80194 **(1)**	RTCP	Reserved by the RTCP function in case of an usual customization (see the customization program %10151.9).
E80194 **(1)**	RTCP	Reserved by the RTCP function for saving the kinematic configuration Tn.
(1) If no memory segment is allocated.		

The iPLANE function uses 4 E parameters. The default addresses are:

E parameter	Function	Description
E80196	iPLANE	Used for saving the kinematic configuration Tn.
E80197	iPLANE	Used to define the coordinate system of the inclined plane function.
E80198	iPLANE	Containing the position measurement of the rotary axis and according to the type of the R rotary axis measurement.
E80199	iPLANE	Containing the position measurement of the swivel axis and according to the type of the S swivel axis measurement.

A.4 Errors list

A.4.1 Errors list handled by the RTCP & iPLANE functions

Error messages of series 500 for RTCP or Inclined Frame control.

Error 500	iPLANE configuration: Invalid option FXSO000914
Extension	**Meaning of the error**
-	Not present extension

Error 501	Invalid T argument given the 5 axes kinematic configuration
Extension	**Meaning of the error**
-	Not present extension

Error 502	T arg.: positive index of setting table of RTCP function : T1 up to T8
Extension	**Meaning of the error**
-	Not present extension

Error 503	T arg.: index limited to 8 configurations
Extension	**Meaning of the error**
-	Not present extension

Error 504	Unauthorized secondary axis
Extension	**Meaning of the error**
-	Not present extension

Error 510	RTCP configuration: Invalid option FXSO000154
Extension	**Meaning of the error**
-	Not present extension

Error 511	Syntax error in the current block G151Sn
Extension	**Meaning of the error**
-	Not present extension

Error 512	Angular measurement by encoder is required by RTCP control
Extension	**Meaning of the error**
-	Not present extension

Error 513	Error in the call conditions of RTCP by G77 statement: see the process in state G999
Extension	**Meaning of the error**
-	Not present extension

Error 514	Programming error of RTCP function : the value of an argument is not allowed
Extension	**Meaning of the error**
-	Not present extension

Error 515	Programming error of tool retraction
Extension	**Meaning of the error**
1	R argument value is erroneous (allowed R1)
2	Tool frame computation doesn't support the 5 axis kinematics

A.4.2 Errors list handled of kinematic measuring cycle248

The Cycle248 messages are in the common range 517-519 with the following extension.

Error 517	Programming Error in the call of cycle248
Extension	**Meaning of the error**
1	Current CNC mode prohibited in a touch-probing cycle
2	T argument : positive index of setting table of RTCP function : T1 up to T8
3	T argument : index limited to 8 configuration table : T1 up to T8
4	T argument is missing: Add the argument T to select the table of settings for your machine kinematics
5	Not used
6	No tool corrector programmed : program a tool corrector conforming to the 3D probe of the customization file
7	The 5-axis machine not managed by the current version of kinematics measuring/calibrating cycle
8	Exceeding the allowed size of swiveling S axis: nbS <=36 indexing positions
9	Exceeding the allowed size of rotary R axis: nbR <=36 indexing positions
10	The kinematic measuring cycle must be done separately on one rotary axis only, the other axis being at zero position
11	The customization file of RTCP function is incompatible with the kinematic measuring cycle248

Error 518	Error occurred while execution of kinematic measurement cycle
Extension	**Meaning of the error**
1	IT error : no interrupt is detected on a descent movement towards the top of the ball
2	The measuring cycle doesn't currently support this kinematics
3	Invalid value of EI argument the allowed values are: 0 for kinematics measuring cycle STEP 0 / 1 for measuring of residual errors of kinematic offsets.
4	Error domain of arc tangent computation : the both arguments sin/cos are null

Error 519	Sphere solving error
Extension	**Meaning of the error**
1	ill conditioned problem in the sphere solving subroutine

Page deliberately empty

Published by NUM Spa

Via F. Somma 62
20012 Cuggiono (MI) (Italy)
Tel. : +39 02 979 691
website: www.num.com

Printed March 2021

CNC HighEnd Applications