5-axis circular pocket-hole milling

SINUMERIK 840D sl

Warranty and liability

Note

The application examples are not binding and do not claim to be complete regarding the circuits shown, equipping and any eventuality. The application examples do not represent specific customer solutions; they are intended only as support for typical tasks. You are responsible for ensuring that the products described are used correctly. These application examples do not relieve you of the responsibility in safely and professionally using, installing, operating and servicing equipment. By using these application examples, you agree that Siemens cannot be made liable for possible damage beyond the liability clause described. We reserve the right to make changes to these application examples at any time and without prior notice. If there are any differences between the suggestions made in these application examples and other Siemens publications such as catalogs, the contents of the other document(s) take priority.

We give no guarantee that the information contained in this document is complete, accurate, or up-to-date.

We accept no liability for any damage or loss caused by the examples, information, programs, planning data or performance data described in this application example, irrespective of the legal basis for claims arising from such damage or loss, unless liability is mandatory. For example, according to the product liability law, in cases of malfeasance, gross negligence, due to endangerment of life, body or health, due to assumption of a guarantee for a product's characteristics of state, due to malicious concealment of a defect or due to violation of basic contractual obligations. Any compensation for violation of basic contractual obligations, however, shall be limited to the foreseeable damage or loss which is typically envisaged in contracts unless there has been gross negligence or unless liability is mandatory due to endangerment of life, body, or health. Any change to the burden of proof to your disadvantage is not covered hereby.

Any form of duplication of these application examples or excerpts hereof is not permitted without the express consent of Siemens Industry Sector.

Security notes

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

To ensure that Siemens products and solutions are operated securely, suitable preventive measures (e.g. cell protection concept) must be taken and each component must be integrated into a state-of-the-art holistic industrial security concept. Third-party products that may be in use should also be considered. For more information about industrial security, visit http://www.siemens.com/industrialsecurity

To receive information about product updates on a regular basis, register for our product newsletter. For more information, visit http://support.automation.siemens.com
# Table of Contents

Warranty and liability ................................................................................................... 2

1 5-axis machining ........................................................................................................ 4
   1.1 Machining a hole with helical motion .............................................................. 4

2 Kinematic transformation ............................................................................................. 5
   2.1 5-axis transformation ......................................................................................... 5
   2.1.1 Precondition .................................................................................................. 5
   2.1.2 Tool orientation ............................................................................................ 6
       Machine-related orientation ............................................................................. 6
       Workpiece-related orientation ..................................................................... 6
   2.1.3 Kinematic motion ......................................................................................... 7
   2.2 TRAORI ............................................................................................................. 7
   2.2.1 Tasks of TRAORI ......................................................................................... 7
   2.2.2 Example ....................................................................................................... 8
   2.2.3 Programming 5-axis transformation ............................................................ 9
   2.2.4 Program syntax ............................................................................................. 10
   2.3 Tool orientation ................................................................................................. 11
   2.3.1 Programming linear and rotary axes .......................................................... 11
   2.3.2 Programming directions vectors .............................................................. 12

3 Helical cycle for roughing ............................................................................................ 13
   3.1.1 Overview ..................................................................................................... 13
   3.1.2 Angle functions ............................................................................................ 13
   3.1.3 Setting the angle and vectors ...................................................................... 14
   3.1.4 Breaking down into increments .................................................................. 15

4 NC programs used ...................................................................................................... 16
   4.1 Main program 5-axis circular pocket hole milling .......................................... 16
   4.2 Subprogram 5-axis circular pocket hole milling .............................................. 17

5 Contact person ........................................................................................................... 19

6 History ......................................................................................................................... 19
1 5-axis machining

1.1 Machining a hole with helical motion

With multi-axis machining, the main objectives are to achieve perfect surface quality, precision, and speed without any need for remachining (reworking). Specifically in tool and mold making as well as in the aerospace domain, complex surface geometries must be machined with a high degree of precision in a short time. A machine tool is required, which in addition to linear axes X, Y and Z, has at least two additional rotary axes (rotation around the linear axes). The rotary axes must be able to be simultaneously controlled.

In most cases, selection criteria as to which machine is most suitable for production is dependent on the size and weight of the workpiece, as well as the traversing range of the rotary axes required. 5-axis transformation is necessary to be able to efficiently machine workpieces.

A hole is to be milled in a cylinder using 5-axis transformation. Drilling motion is helical, the milling tool position is positioned at a specific angle and tilted backward when milling. The TRAORI function with orientation vectors A3, B3, C3 is used in order that the milling tool can be optimally used, and/or the diameter of the milling tool (for generally used milling tool sizes) is independent of the hole diameter. The 5-axis helical solution can be applied for holes and circular pockets.

Overview of the automation task

The following screenshot provides an overview of machining a hole using an inclined milling tool that has been "tilted backward".

Fig. 1-1 Machining a hole with helical motion of the milling tool
2 Kinematic transformation

The kinematic transformation represents a dynamic transformation, where up to 6 axes can be simultaneously moved. 5-axis transformation is the most frequently used type of transformation.

2.1 5-axis transformation

The task of 5-axis transformation is to compensate movements of the tool nose/tip, which result from changes in orientation while machining. This is realized using the appropriate compensating motion of the geometry axes. The orientation movement is decoupled from the motion of the workpiece contour. This package thus allows an axially symmetrical tool (milling tool) to be oriented in any desired relation to the workpiece in every point of the machining space. This is realized using the TRAORI function.

At the same time, the traversing motion of the milling tool used is described using the mathematical angle function in the "Helix08" subprogram. As a consequence, this ensures helical motion of the milling tool when machining. Combining the two listed functions (TRAORI and description of the angle functions), represents a simple and efficient solution to machine holes or circular pockets, especially in very hard materials. For instance, recutting of the milling tool can be eliminated, and as a consequence, also the chatter marks that occurred in the past.

Advantages of the solution presented here

The solution presented here involving 5-axis transformation offers the following advantages:

- Fast programming at the machine for holes and circular pockets
- Can be used for very hard materials
- Chatter marks are avoided, as recutting of the milling tool is not required
- This results in cost and time savings
- It can be used on any machine, as the workpiece-related orientation is independent of the machine kinematics

2.1.1 Precondition

The precondition for 5-axis transformation is a machine tool, which in addition to three linear axes X, Y and Z, also has two additional rotary axes (rotation around the linear axes), which can be simultaneously controlled. The kinematic transformation requires data about the design of the machine kinematics, which is saved in a machine data.
2.1.2 Tool orientation

Tool orientation can be specified in two ways:

Machine-related orientation
The machine-related orientation depends on the machine kinematics, and is programmed via direct rotary axis positions.

Workpiece-related orientation
The workpiece-related orientation is independent of the machine kinematics and is programmed with:
- Euler angle
- RPY angle
- Vector components

The direction of the tool is described in the workpiece coordinate system in the tool-related orientation. This version is used to subsequently implement the application. It is possible to program a specific component of the tool in its orientation to the workpiece. In most cases it is realized using the longitudinal axis of the tool, the tool center point (TCP). TCP programming represents an alternative term. Tool motion is always programmed in the right-angled workpiece coordinate system.

Programmed or set FRAMES rotate and shift this system to the basic system. Tool length correction is taken into account when machining. The kinematic transformation converts this information into motion instructions for the real machine axes. Programming path and path velocity corresponds to the procedure applied for 3-axis programming.

The tool orientation is additionally programmed in the motion blocks. The real-time transformation performs the calculation of the resulting motion of all 5 axes. The generated machining programs are therefore not machine-specific.
2.1.3 Kinematic motion

The swiveling rotary table is the machine kinematics used (Fig. 2-1). The workpiece is orientated using two axes in the table (type P). The table can also be used, rotated in the XY plane through 90°.

Fig. 2.1 Kinematic solution used

A transformation with the following properties is required in order to be able to take into account all of the effects in the NC program:

- Programming must be independent of the machine kinematics.
- Automatic tool length correction must be incorporated.
- When the tool orientation is changed, compensation is automatically realized.

All of this is implemented using the TRAORI function.

2.2 TRAORI

2.2.1 Tasks of TRAORI

- Compensation motion in the X, Y, and Z axes is automatically calculated when the tool orientation changes. In so doing, the tool center point (tool tip) maintains its position. As a consequence, CNC programs become more compact and data management is reduced.
- Active tool length and programmable FRAMES, with rotation with respect to the basic frame, are also included in the calculation. This means that tool length and work offset can be directly changed at the control. This change is immediately and directly taken into account in the program execution.
- The programmed feedrate refers to the tool center point (TCP); as a consequence, this eliminates the real-time calculation for the rotary axes.
- In this case, the real-time transformation performs the calculation of the resulting motion of all 5 axes.
2 Kinematic transformation

Advantage

- The CNC programs generated are independent of any particular machine. This permits a higher degree of flexibility when machining 5-axis workpieces.
- In this particular case, kinematic-specific post-processors are not used for 5-axis machining operations with TRAORI.

2.2.2 Example

TRAORI is not used

The controller does not take into account the tool length. It rotates around the axis pivot point (1). The tool center point moves out of its position, and therefore does not remain fixed in space. Compensation values for tool length from the tool center point to the swivel point must be calculated by the CAM system. As a consequence, tool length cannot be corrected at the control.

Fig. 2-2 Machining a workpiece without TRAORI

Using TRAORI

The control only changes the tool orientation, the position of the tool center point therefore remains fixed in space. The compensation motion of the linear axes (2) required is automatically calculated. When TRAORI is active, the control automatically compensates the change in the tool length and the work offset.
2 Kinematic transformation

2.2.3 Programming 5-axis transformation

CNC programming independent of a machine

CNC programming independent of the machine can, when compared to machine-dependent versions, be applied to various machine kinematics. NC programs are always generated referred to the workpiece. This means that all tool positions are referred to the workpiece coordinate system (WCS). This involves workpiece-related programming.

In order that an NC program can be executed at the machine, the positions must be transformed into axis motion, i.e. converted into the machine coordinate system (MCS). The TRAORI function is responsible for this.

Generally, TRAORI is called in the CNC program, which is output by the CAM system. The CNC program then simply contains:

- The coordinates of the point to be approached in X, Y, Z
- The tool orientation with regard to this point in the form of a direction vector $A_3=, B_3=, C_3=$ or
- Tool orientation with RPY angle $A_2=, B_2=, C_2=$

When transformation is activated, the positional data (X, Y, Z) always refers to the tool center point, TCP. Changing the position of the rotary axes involved in the transformation causes compensating movements of the remaining machine axes so that the position of the tool center point remains unchanged.

Fig. 2-3 Machining a workpiece with TRAORI
2.2.4 Program syntax

TRAORI(<n>, <X>, <Y>, <Z>, <A>, <B>, <C>)

<table>
<thead>
<tr>
<th>NC command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRAORI</td>
<td>Activates the 1st programmed 5-axis transformation.</td>
</tr>
<tr>
<td>TRAORI(&lt;n&gt;, &lt;X&gt;, &lt;Y&gt;, &lt;Z&gt;, &lt;A&gt;, &lt;B&gt;, &lt;C&gt;)</td>
<td>Activates the 5-axis transformation specified under parameter &quot;n&quot;.</td>
</tr>
<tr>
<td>&lt;n&gt;</td>
<td>Number of the 5-axis transformation (n = 1, 2, 3, 4). TRAORI activates the 1st programmed 5-axis transformation. The number can be ignored if it involves the first five-axis transformation.</td>
</tr>
<tr>
<td>&lt;X&gt;, &lt;Y&gt;, &lt;Z&gt;:</td>
<td>Directional normal vector (direction reference) in which the tool tip points. The components of the directional normal vector are defined using parameters 2, 3 and 4. If the number of the first 5-axis transformation is not programmed, then instead of the transformation number, a space must be inserted, e.g. TRAORI (),0,1,2), in order to guarantee that parameters are correctly identified when specifying a directional normal vector. Direction vector data are absolute, and are not modified by an active frame. The absolute value of the vector is not decisive, but instead, only the direction. Non-programmed vector components can be set to zero.</td>
</tr>
<tr>
<td>&lt;A&gt;, &lt;B&gt;, &lt;C&gt;:</td>
<td>Programmable rotary axis offset. The rotary axis offset is specified in parameters 5, 6 and 7 for the first, second and third rotary axis (this depends on the axis configuration). When 5-axis transmission is active, an additional offset of the rotary axes can be directly programmed here. Preceding parameters do not have to be specified if the correct sequence was maintained when programming, e.g. TRAORI (), ,A,B,C) – or if an offset is only required for one rotary axis, for example TRAORI ( ), , , ,C). As an alternative to the rotary axis offset, the programmable rotary axis offset can be used in the work offset that can be saved, and is added to the active work offset for the rotary axes involved in the transformation. Accepting data from the work offset is configured using MD24590 $MC_TRAFO5_ROT_OFFSET_FROM_FR_1.</td>
</tr>
</tbody>
</table>

TRAFOOF Deactivates all transformations.
2 Kinematic transformation

**Directional normal vector for the reference of the tool center point**

- `TRAORI (1,0,0,1)` – activates the first 5-axis transformation, with a Directional normal vector of the tool center point in the Z axis
- `TRAORI (1,0,1,0)` – activates the first 5-axis transformation, with a vector normal to the direction of the tool center point in the Y axis
- `TRAORI (1,1,0,0)` – activates the first 5-axis transformation, with a vector normal to the direction of the tool center point in the X axis
- `TRAORI (2,0,1,1)` – activates the second 5-axis transformation, with a Direction vector

**Note**
The directional normal vector of the tool center point does not lead to any change in the tool orientation itself, but only represents the direction reference of the tool center point.

2.3 Tool orientation

**2.3.1 Programming linear and rotary axes**

To accommodate machining scenarios involving tools set at an angle or in order to mill geometries located anywhere in space, the three linear axes X, Y and Z are required along with two of the rotary axes A, B or C. The axes must be able to be simultaneously controlled. The tool position is approached in space using linear axes X, Y and Z. This enables the tool center point to adopt any position. For 3-axis machining by programming the three linear axes. The contour is milled line by line by moving the 3 linear axes.

Rotary axes will also be required if the tool also has to be set at an angle. Using three linear axes and two rotary axes, theoretically any point in space can be approached with any tool orientation.
2.3.2 Programming directions vectors

The components of direction vector 1 are programmed with addresses A3, B3, and C3. The vector points in the direction of the toolholder. The vector length is of no significance. Non-programmed vector components are set to zero.

Fig. 2-4 Programming a direction vector

Table 2-2 Syntax of the direction vectors

<table>
<thead>
<tr>
<th>NC commands</th>
<th>Direction vector</th>
</tr>
</thead>
<tbody>
<tr>
<td>G1 X, Y, Z, A3=.. B3=.. C3=..</td>
<td>Programming direction vectors</td>
</tr>
</tbody>
</table>

In the example (Fig. 2-4), the tool is at position (X0, Y0, Z0) as diagonal of a cube of 35.26° in the resulting XY plane.

N10 TRAORI
N20 G54 D1
N30 G1 X0 Y0 Z0 A3=1 B3=1 C3=2 F1000
....
3 Helical cycle for roughing

3.1.1 Overview

The cycle subsequently presented is used to machine holes or circular pocket. A hole is machined in the cylindrical unmachined part (blank); the milling tool used performs helical motion. The milling tool moves at an angle and "tilted backward" while the program is being executed. The cycle comprises a main program and sub program. The TRAORI and cycle 832 functions, already explained in Chapter 2, are used in the main program of the cycle. The complete helical motion of the milling tool is the content of the Helix08 subprogram.

3.1.2 Angle functions

The helical motion of the milling tool can be explained using trigonometric functions in right-angled triangles. The sinusoidal and/or cosinusoidal function is defined as follows.

![Definition of angle functions](image)

\[ \cos \alpha = \frac{x}{c} \quad (3.1) \]

\[ \sin \alpha = \frac{y}{c} \quad (3.2) \]

The X coordinate can be expressed using equation (3.1) as follows using this example.

\[ \cos \alpha \times c = x \quad (3.3) \]
Equation (3.4) is obtained for the value of the X coordinate in subprogram Helix08 (N190).

\[ X = \_APOSX + \cos \alpha \times \_DURCH \]  

(3.4)

This is displayed as follows in the Helix08 program.

N190 G1 X=\_APOSX+COS(0)*\_DURCH Y=\_APOSY+SIN(0)*\_DURCH

Table 3-1 Helical cycle commands used

<table>
<thead>
<tr>
<th>Commands used</th>
<th>Significance</th>
</tr>
</thead>
<tbody>
<tr>
<td>_APOSX</td>
<td>Actual value of the milling tool in the WCS</td>
</tr>
<tr>
<td>( \cos \alpha )</td>
<td>( \alpha = 0^\circ )</td>
</tr>
<tr>
<td>_DURCH</td>
<td>Specifies the radius of the hole in, as _DURCH=_DURCH/2</td>
</tr>
</tbody>
</table>

The Y coordinate is calculated analogously using a sinusoidal function.

3.1.3 Setting the angle and vectors

The radius of the active tool must be defined, as otherwise it is possible that the milling tool would exceed its tolerances and violate the contour. Direction vectors A3/B3/C3 are specified as follows. As the milling tool makes a type of helical motion, when specifying the circular motion, the opposite side must also be taken into account. This is the reason that 180° must be used, as otherwise only a semicircular machining operation will be performed. A second aspect is the fact that the milling tool is tilted backward. As a result, there is the risk that the edge of the unmachined part will be violated by the milling tool when it performs its helical motion. A correction factor TAN_SCHRAE is introduced to prevent this happening. C3 actually corresponds to the cosinusoidal value of the angled position of the milling tool. This value was rounded off and set to a value of 1.

N150 TRAORI
N160 \_DURCH=\_DURCH/2-$P\_TOOLR;
N170 \_KPOSZ=\_KPOSZ+sin(\_SCHRAE)*$P\_TOOLR
N170 \_KPOSZ=\_KPOSZ+sin(\_SCHRAE)*$P\_TOOLR
N190 G1 X=\_APOSX+COS(0)*\_DURCH Y=\_APOSY+SIN(0)*\_DURCH
N200 G0 Z=\_KPOSZ A3=(COS(\_WINK+180-_VORWI)*TAN(\_SCHRAE))
B3=(SIN(\_WINK+180-_VORWI)*TAN(\_SCHRAE)) C3=1
3 Helical cycle for roughing

3.1.4 Breaking down into increments

Mark 1 checks the condition of the helical motion. The circle is split up into identical increments after splitting up the circle of the helical motion into angular segments. This checks as to whether, when executing the program, the actual milling tool position coincides with the required final position. The program has been completed once this condition has been fulfilled.

```
N210 MARK1:
N220 _WINK= _WINK+(360/(6.28*_DURCH*2))
N230 _KPOSZ= _KPOSZ-(_STEP/(360/(360/(6.28*_DURCH*2))))
N240 IF _KPOSZ<_KENDZ
N250 _KPOSZ= _KENDZ
```
4 NC programs used

4.1 Main program 5-axis circular pocket hole milling

```
G17 G90 G64 G54 G71 ; Starting point
WORKPIECE(,"C",,"CYLINDER",0,0,-50,-80,60) ; Workpiece definition
T="D8R1" M6 ; Tool call and change
S8000 M3 D1
TOFFL=0
CYCLE832(0.05,_ORI_ROUGH,1) ;Tolerance value 0.05 mm, "roughing", multiaxis machining
F2000
TRAFOOF
; Deactivate 5-axis transformation
N100 G0 A0 C0
N110 G0 Z100
N120 G0 X0 Y0 ; Retract
N130 TRAORI ; Activate 5-axis transformation
N140 G0 X0 Y0 A3=0 B3=0 C3=1 ; Align tool parallel to the Z axis
FGROUP(C)
FGREF[C]=21 ; Feedrate calculation
Axis orientation
N150 G1 Z0 F3000 ; Selecting the feedrate
N170 HELIX08(26,20,15,1); Diameter, depth, angular position, infeed per revolution
N190 M30 ; End of program
```
4 NC programs used

4.2 Subprogram 5-axis circular pocket hole milling

```
N10 PROC HELIX08(REAL _DURCH,REAL _TIEF, REAL _SCHRAE, REAL _STEP); SAVE SBLOF DISPLOF
N20 DEF REAL _APOSX
N30 DEF REAL _APOSY
N40 DEF REAL _APOSZ
N50 DEF REAL _KPOSZ
N60 DEF REAL _KENDZ
N70 DEF REAL _WINK
N80 DEF REAL _VORWI
N90 _APOSX=$AA_IW[X] ; Reads the actual values of the X axis in WCS
N100 _APOSY=$AA_IW[Y] ; Reads the actual values of the Y axis in WCS
N110 _APOSZ=$AA_IW[Z] ; Reads the actual values of the Z axis in WCS
N120 _KPOSZ=_APOSZ
N130 _WINK=0
N140 _VORWI=_SCHRAE

N150 TRAORI
N160 _DURCH=_DURCH/2-$P_TOOLR; ; Radius of the active tool
N170 _KPOSZ=_KPOSZ+$P_TOOLR*sin(_SCHRAE) ; Actual position of the milling tool with radius correction
N180 _KENDZ=_APOSZ+(-1)*_TIEF+sin(_SCHRAE)*$P_TOOLR ; Final position of the milling tool with radius correction
N190 G1 X=_APOSX+COS(0)*_DURCH ; Definition of the X coordinate
    Y=_APOSY+SIN(0)*_DURCH ; Definition of the Y coordinate
N200 G0 Z=_KPOSZ
    A3=(COS(_WINK+180-_VORWI)*TAN(_SCHRAE))
    B3=(SIN(_WINK+180-_VORWI)*TAN(_SCHRAE))
    C3=1 ; Definition of direction vectors A3, B3, C3
N210 MARK1:
N220 _WINK=_WINK+(360/(6.28*_DURCH*2)) ; Breaking down the circle into angular segments using the formula 360°/2 π r
N230 _KPOSZ=_KPOSZ-(_STEP/(360/(360/(6.28*_DURCH*2)))) ; Breaking down the circle into identical segments
N240 IF _KPOSZ<_KENDZ
N250 _KPOSZ=_KENDZ ; End of mark1 when reaching the final position
N260 ENDIF
N270 G1 X=_APOSX+COS(_WINK)*_DURCH Y=_APOSY+SIN(_WINK)*_DURCH
    Z=_KPOSZ A3=(COS(_WINK+180-_VORWI)*tan(_SCHRAE))
    B3=(SIN(_WINK+180-_VORWI)*tan(_SCHRAE))
    C3=1
```

5-axis circular pocket hole milling
N280 IF (_WINK+(360/(6.28*_DURCH*2)))<360 ; Condition to repeat the mark
N290  GOTOB MARK1
N300 ENDIF

N310 IF _KPOSZ>_KENDZ ; Condition to repeat the mark
N320  _WINK=0
N330  GOTOB MARK1
N340 ENDIF

N350 SPRUNG:
N360  _WINK= _WINK+(360/(6.28*_DURCH*2))
N370; MSG("STEP="<<(360/(6.28*_DURCH*2)))
N380  G1 X=_APOSX+COS(_WINK)*_DURCH Y=_APOSY+SIN(_WINK)*_DURCH
Z=_KENDZ A3=(COS(_WINK+180-_VORWI)*tan(_SCHRAE)) B3=(SIN(_WINK+180-
_VORWI)*TAN(_SCHRAE)) C3=1
N390 IF (_WINK)<360 ; Condition to repeat the jump
N400  GOTOB SPRUNG
N410 ENDIF

N420  G1 Z=_APOSZ
N430  G1 X=_APOSX Y=_APOSY A3=0 B3=0 C3=1 ; Align tool parallel to the
Z axis
N440 M17  End of program
5 Contact person

Siemens AG
Industry Sector
I DT MC MTS APC
Frauenauracher Straße 80
D - 91056 Erlangen
mailto: MC-MTS-APC-Tech-Team.i-dt@siemens.com

6 History

Table 6-1

<table>
<thead>
<tr>
<th>Version</th>
<th>Date</th>
<th>Revision</th>
</tr>
</thead>
<tbody>
<tr>
<td>V1.0</td>
<td>07/2014</td>
<td>First Edition</td>
</tr>
</tbody>
</table>