NCControl

Export PostPro XML File

The contents of this manual are relative to GO2cam version:

V6.12

August 2025

About GO2cam

Warning to the user: all the forms included in this document are given as examples only. GO2cam International cannot be responsible for any consequences of their use.

Table of contents

1 Introduction

AN xml file will be exported parallel to the NCfile generation. The file will have the same name as the NCfile but with extension “NCC” and in the same directory of NC output.

```
<?xml version="1.0" encoding="UTF-8"?>
<!--GO2cam NCcheck Information-->
<GO2camNCcheckData>
  <GO2camNCcheckInfo Date="15/12/2021" Time="09:44"/>
  <GO2camNCcheckNCfile Dir="C:\GO2cam_Intl\GO2camV609\Iso" Name="NCfile.H"/>
  <GO2camNCcheckController Controller="Undefined" Type="Undefined"/>
   <GO2camNCcheckTools Toolchange="Number">
      <Tool Number="1" Name="toolname1"/>
      <Tool Number="2" Name="toolname2"/>
      <Tool Number="3" Name="toolname3"/>
   </GO2camNCcheckTools>
   <GO2camNCcheckOrigin >
       <Origin Number="1" Name="$1_" NCcode="1"/>
       <Origin Number="2" Name="$2_" NCcode="2"/>
       <Origin Number="3" Name="$3_" NCcode="3"/>
   </GO2camNCcheckOrigin>
   <GO2camNCcheckParameters>
        <Parameter m128support="1"/>
        <Parameter cyc19support="1"/>
        <Parameter userend="0"/>
        <Parameter millturn="0"/>
        <Parameter maxangle="3"/>
        <Parameter wptype="0"/>
        <Parameter lcompnecessary="0"/>
        <Parameter langtype="0"/>
        <Parameter modangleflag="0"/>
        <Parameter tname="1"/>
        <Parameter writetime="1"/>
        <Parameter mbedangle="0"/>
        <Parameter scalefixax="1"/>
        <Parameter scalevarax="1"/>
        <Parameter fmaxvalue="10000"/>
        <Parameter fmax_x="10000"/>
        <Parameter fmax_y="10000"/>
        <Parameter fmax_z="10000"/>
        <Parameter toolchangetime="5"/>
        <Parameter inch="0"/>
        <Parameter machtype="0"/>
         <Parameter fiveaxisinterpolation="0"/>
         <Parameter specialtoolchange="50"/>
         <Parameter m6notnecessary="0"/>
         <Parameter contver="500"/>
         <Parameter dogleg="0"/>
   </GO2camNCcheckParameters>
</GO2camNCcheckData>
```

2 XML file description

2.1 Information

```
<GO2camNCcheckInfo Date="15/12/2021" Time="09:44"/>
```

Date = Date of generation of the NCfile.
Time = Hour of generation of the NCfile

2.2 File

```
<GO2camNCcheckNCfile Dir="C:\GO2cam_Intl\GO2camV609\Iso" Name="NCfile.H"/>
```

Dir = Directory where are generated the NCfile and the “NCC” file.
Name = Name of the NCfile.

2.3 Controller

```
<GO2camNCcheckController Controller="Undefined" Type="Undefined"/>
```

Controller = Name of NC controller (for example Fanuc / Sinumerik / Heidenhain)
Type = Type of NC controller (for example 30i / 810D / TNC640)

2.4 Tools

```
<GO2camNCcheckTools Toolchange="Number">
   <Tool Number="1" Name="toolname1"/>
   <Tool Number="2" Name="toolname2"/>
   <Tool Number="3" Name="toolname3"/>
</GO2camNCcheckTools>
```

Toolchange = Toolchange format *Number* or *Name*.
Number = Tool number.
Name = Name of tool defined in GO2cam.

2.5 Origins

```
<GO2camNCcheckOrigin>
   <Origin Number="1" Name="$1_" NCcode="1"/>
   <Origin Number="2" Name="$2_" NCcode="2"/>
   <Origin Number="3" Name="$3_" NCcode="3"/>
</GO2camNCcheckOrigin>
```

Number = Origin incremental number.
Name = Name of Origin defined in GO2cam (for example ***$1_*** on Heidenhain, ***$G54_*** on Fanuc, …).
NCcode = Code outputted in NC file (for example ***1*** on Heidenhain, ***G54*** on Fanuc, …).

2.6 Parameters

```
<GO2camNCcheckParameters>
   <Parameter m128support="1"/>
   <Parameter mcyc19support="1"/>
   <Parameter userend="0"/>
   <Parameter millturn="0"/>
   <Parameter maxangle="3"/>
   <Parameter wptype="0"/>
   <Parameter lcompnecessary="0"/>
   <Parameter langtype="0"/>
   <Parameter modeangleflag="0"/>
   <Parameter tname="1"/>
   <Parameter writetime="1"/>
   <Parameter mbedangle="0"/>
   <Parameter scalefixax="1"/>
   <Parameter scalevarax="1"/>
   <Parameter fmaxvalue="10000"/>
   <Parameter fmax_x="10000"/>
   <Parameter fmax_y="10000"/>
   <Parameter fmax_z="10000"/>
   <Parameter toolchangetime="5"/>
   <Parameter inch="0"/>
   <Parameter machtype="0"/>
   <Parameter fiveaxisinterpolation="0"/>
   <Parameter specialtoolchange="50"/>
   <Parameter m6notnecessary="0"/>
   <Parameter contver="500"/>
   <Parameter dogleg="0"/>
</GO2camNCcheckParameters>
```

m128support = RTCP support
- 0 = No
- 1 = Yes
- 2 = without C-axis calculation (for ex. Integrex)

mcyc19support = manage tilted planes (for ex. PLANE, CYCL DEF 19, ROT, cycle800, G68.2, ...)

userend = for customer specific case.

millturn = TurnMill activ
- 0 = No
- 1 = Yes

maxangle = maximal angle step for interpolation

wptype = Calculation type for workplane
- 0 = Workplane coordinates
- 1 = Absolute workpiece coordinates
- 2 = Z axis in spindle

lcompnecessary = Tool-length compensation
- 0 = Automatically activ (like on Heidenhain)
- 1 = Will be separately called (like on Siemens)

langtype = Type of controller
- 0 = Heidenhain klartext
- 2 = MillPlus
- 3 = Fanuc
- 4 = Maka-BWO
- 6 = Siemens840D

11 = Roeders

modeangleflag = Rotary axis shortest way (that means angle output only between 0 and 360 degrees
0 = No
1 = Yes

tname = Tool call (Optional)
0 = Call with tool number
1 = Call with tool name

writetime = Unterstuetzung zeitbasierter Modus in Moduleworkssimulation (Optional)
0 = No (default)
1 = Yes

mbedangle = Machine bed angle on turning machine (Optional)

scalefixax = Direction of fix axis / tilt axis (Optional)
1 = matematical (default)
-1 = invers

scalevarax = Direction of variable axis / round axis (Optional)
1 = matematical (default)
-1 = invers

fmaxvalue = Rapid movement feed (Optional)

fmax_x = Maximal feed on X-axis for Haas with "Dogleg" in rapid (Optional)

fmax_y = Maximal feed on Y-axis for Haas with "Dogleg" in rapid (Optional)

fmax_z = Maximal feed on Z-axis for Haas with "Dogleg" in rapid (Optional)

toolchangetime = Toolchange time in Seconds for time calculation (Optional)

inch = Unit (Optional)
0 = metric (default)
1 = inch

machtype = Machine type (Optional)
0 = (default)
31 = Fanuc Turning
33 = Integrex angled machine bed
34 = Integrex I-serie
51 = MonfortsIPT
52 = Variaxis

fiveaxisinterpolation = Interpolation for simultaneous milling (Optional)
0 = with machine axis (default)
1 = with verctors

specialtoolchange = optional: automatic turn back of tilt axis on big tilting angle (Q parameter on Heidenhain)

m6notnecessary = M6 for toolchange (Optional)
0 = M6 compulsory
1 = M6 not compulsory (for ex. On Siemens)

contver = Controller version
4xx = TNC530
5xx = TNC640 (for now only CYCLE 208 with/without bore milling)

dogleg = Dogleg management
0 = No
1 = Yes