Creo 3.0 G-code Tutorial

Irobotics
μTan(Clan)
# Table of Contents

1. Preface ................................................................................................................................. 2
2. CAD ............................................................................................................................................... 3
   A. Prepare the CAD .................................................................................................................... 3
   B. Define the Coordinate System .............................................................................................. 3
   C. Save the CAD ....................................................................................................................... 6
3. Create NC assembly .................................................................................................................. 6
   A. Create File ............................................................................................................................ 6
   B. Place reference model .......................................................................................................... 7
   C. Create Mill ............................................................................................................................ 8
   D. Create Operation ................................................................................................................. 11
4. Drilling ...................................................................................................................................... 13
   A. Standard Drilling / Pecking ................................................................................................. 13
5. Roughing .................................................................................................................................. 18
   A. Roughing .............................................................................................................................. 18
   B. Volume Roughing ................................................................................................................ 22
   C. Profile Milling ...................................................................................................................... 29
   D. Trajectory Milling ............................................................................................................... 33
   E. Finishing ............................................................................................................................... 37
6. Export the G-Code ................................................................................................................... 39
   A. Single Operation .................................................................................................................. 39
   B. Multiple Operation .............................................................................................................. 44
1. Preface

This guide is written to help creating g-code file (.tap file) using Creo 3.0 NC machining extension. This feature is not included in Creo student version. In the guide, couple of basic 3-axis NC/CNC sequences are introduced, but choosing appropriate operation is completely depend on user preference and design. Brief explanation of each operations are included.

Before reading this guide or using CNC machine, I suggest you to go over how much you know about 3-axis milling operation, such as what you can do with 3-axis mill, how to calculate adequate RPM or IPM feed-rate, what tool should be used for application, how to set zero, how to maintain or operate the machine, etc. I am writing the guide assuming you know how to do these.

If you have any problem question, or suggestion, feel free to contact me.

skim449@illinois.edu
2. CAD

A. Prepare the CAD

   i. It can be made in Creo 3.0, or imported in .step/.stp file.

B. Define the Coordinate System

   i. It is generally recommended to use upper left corner of the part as the origin.

   ii. Click the top surface
iii. Drag the two green circles to reference top and left surface.

iv. Set the offsets to zero
v. Go to orientation tap, and flip/change the coordinate so that the part fits in positive-x and negative-y.
vi. Click Ok (middle button on the mouse)

vii. Change the coordinate name to G54

![](image)

Note: G54 to G59 is work coordinate system in G-code.

C. **Save the CAD**

   i. Save the CAD to .prt file, which is the default file extension for Creo part file.

3. **Create NC assembly**

   A. **Create File**

      i. New ➔ Manufacturing ➔ NC assembly
Note: This feature is not included in Creo student version. It only work on EWS computer.
Note: Default template uses inches.

B. **Place reference model**

   i. Click ‘Reference Model’

   ➔ open the part file

   ➔ place it (use default or fix option)
→ click Ok

C. Create Mill

i. Check the Number of Axes to be 3

ii. Set the tools – go to tool tab → tools...
1. Create the list of tools
   a. Type name → Select Type → Enter the dimension of the tool
b. Must enter the diameter of the blade. (usually, this is only dimension needed)
   i. Length of the blade and tool is unnecessary, if the operation is single tool operation.
   ii. If the operation is multi-tool operation, the length of the tool is necessary to adjust z-coordinate.

c. Click ‘Apply’ to add on the tool list

2. After adding the tools, click Ok

   iii. Set the maximum RPM – go to parameters tab → type the number
   (This step is optional, but good practice for safety. Personally recommend for 5000 RPM max.)
iv. Click Ok

D. **Create Operation**

i. Select the Mill you just created (default - MILL01)

![Select Mill](image)

ii. Set G54 as the Coordinate – expand the part file in the model tree → Click G54
iii. Set the default clearance – Open ‘Clearance Tab’ → set the type as plane → Select the top surface → set Value (between 0.15 and 0.5 is usually enough)
iv. Click Ok

4. Drilling

Drilling is operation to cut or enlarge the hole. It consist of only vertical movement (z-axis), and moving in other xy-plane direction is forbidden. It is important to find the right rotation speed (RPM) for the operation, which depend on the diameter of the tool, material of the tool, material of the stock, and feed-rate.

A. Standard Drilling / Pecking

i. Go to Mill tab

ii. Select Standard or Pecking
Note: Standard drilling is single motion drilling through the depth, and pecking has multiple motion to remove chips as it drill. Pecking is recommended if the material is thick (generally pecking is used when drilling the depth more than twice the diameter of the drill bit).

iii. Select the tool

iv. Click holes or multiple holes
v. Set Parameters – Go to parameters tab → click to see the detail of parameter.
i. Cut_Feed – Speed in Z. For drill bit, IPR (inches per revolution) can be found in the table. (it depend on the material and diameter of the drill bit) Feed rate = RPM x IPR
   I used 7 – 10 IPM when using ¼ inch drill bit to drill aluminum 6061.
ii. Free_Feed – The traveling speed in X and Y. It is not mandatory to fill (if it is left empty, the default value is equal to the Cut_Feed), but it is recommended to put value to reduce the total operation time. Generally, number between 35-45 is enough.
iii. Peck_Depth – Only appears for pecking. It depends on the chip clearance and coolant delivery. Usually, 1.5 to 3 times the diameter of the tool is used.
iv. Clear_Distance – Clear distance above the surface. If the number is large, it will take longer time to retract the tool. If the number is too small, the tool might contact the surface while traveling. (0.15 – 0.3 is recommended for flat surface)
v. Pullout_Distance – Distance above the top surface when traveling in X and Y. If the value is not given, default is Clear_Distance.
vi. Spindle Speed – Tool RPM. Use table or other reference to determine this number. Usually, running too slow RPM is major cause of breaking tool, especially when cutting soft (relatively) material like aluminum. It depend on the material of the drill bit, material of the cutting part, and the diameter of the tool.

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Drilling 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>CUT_FEED</td>
<td>8</td>
</tr>
<tr>
<td>FREE_FEED</td>
<td>35</td>
</tr>
<tr>
<td>TOLERANCE</td>
<td>0.001</td>
</tr>
<tr>
<td>BREAKOUTDISTANCE</td>
<td>0</td>
</tr>
<tr>
<td>SCAN_TYPE</td>
<td>SHORTTEST</td>
</tr>
<tr>
<td>CLEAR_DIST</td>
<td>0.15</td>
</tr>
<tr>
<td>PULLCUTDIST</td>
<td>-</td>
</tr>
<tr>
<td>SPINDLE_SPEED</td>
<td>2000</td>
</tr>
<tr>
<td>COOLANT_OPTION</td>
<td>OFF</td>
</tr>
</tbody>
</table>

(Standard Parameters)

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Drilling 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>CUT_FEED</td>
<td>8</td>
</tr>
<tr>
<td>FREE_FEED</td>
<td>35</td>
</tr>
<tr>
<td>TOLERANCE</td>
<td>0.001</td>
</tr>
<tr>
<td>PECK_DEPTH</td>
<td>0.375</td>
</tr>
<tr>
<td>SCAN_TYPE</td>
<td>SHORTTEST</td>
</tr>
<tr>
<td>CLEAR_DIST</td>
<td>0.15</td>
</tr>
<tr>
<td>PULLCUTDIST</td>
<td>-</td>
</tr>
<tr>
<td>SPINDLE_SPEED</td>
<td>2000</td>
</tr>
<tr>
<td>COOLANT.Option</td>
<td>OFF</td>
</tr>
</tbody>
</table>

(Pecking Parameters)

vii. Click OK

viii. Note: By clicking , these parameters can be reused when creating other operation.

ix. Note: In the process tab, you can calculate the operation time. The number is meaningless in terms of unit, because every CNC or NC machine has slightly different running speed. However, the number can be used to compare with other operations. In my experience, the actual operation time takes 2 to 4 times the calculated time in minute.
vi. Click Ok to create drilling operation.

vii. Play the path and check
5. Roughing

A. **Roughing**

Roughing operation is used to remove the excess material from the stock and roughly cut to the final shape. In general, the result of roughing cut is not very accurate, so finishing operation is recommended after roughing. (When roughing any large hole, the size of the hole is slightly smaller than the actual size, which can be used for press-fit the bearing.) In Creo, this sequence uses Mill Window, and try to remove any material that is not projected by the specified window.

i. The roughing operation requires Mill Window.
   a) Click Mill Window
   b) Click the top surface
   c) If necessary, specify the depth
d) Click Ok

ii. Select Roughing

iii. Select the tool

iv. Select References – go to ‘References’ page → Select Mill Window in model tree

v. Set Parameters (for detail, click )
1. **Cut_Feed** – Cutting speed. It depend on material, RPM, and diameter. For aluminum 6061, 10-16 IPM was used for 3/8 inch end mill operation.

2. **Free_Feed / Retract_Feed** – They are not required but recommended to reduce operation time. In general, machine in ESPL can handle about 35-45 IPM.

3. **Step_Over** – 1/3 of the diameter of the tool or less is generally recommended.

4. **Max_Step_Depth** – It depend on the material of the tool and number of flute. The operation time largely depend on this parameter
vi. Click Ok to create operation

vii. Play the path to check
B. **Volume Roughing**

Volume roughing is used for the same purpose as ‘Roughing,’ but it uses Mill Volume instead. Simply, it creates the path to remove the mill volume as much as possible. It is simple and easy to use, so it is recommended for general purpose machining.

i. **Volume Roughing uses Mill Volume**

a. Create a volume using extrude, revolve, sweep, or other operation.

(b) Trim the volume using referenced model – Click ‘Trim’

→ Click Referenced Model (not the mill volume)
→ modify using extrude or other tools to remove unnecessary volumes
c. Click Ok
   Note: To redefine or edit the mill volume, right-click the extrude and click 'redefine mill volume'

ii. Click Volume Rough

   ![Image of Volume Rough tool]
   
   Volume Rough
   Insert a volume roughing step.

iii. Select the Tool

iv. Select reference – Go to ‘Reference’ tab → click ‘Select items’

   ![Image of Reference tab]
   
   → Click Mill Volume on the screen
(Make sure, Machining Reference start with ‘Quilt’)

(If it is set to extrude, left-click and remove the reference and click the mill volume on the screen)

v. Set Parameters
1. **Cut_Feed** – Cutting speed. It depend on material, RPM, and diameter. For aluminum 6061, 10-16 IPM was used for 3/8 inch end mill operation.

2. **Arc_Feed** – If the value is not given, default is Cut_Feed.

3. **Free_Feed / Retract_Feed / Traverse_Feed** – They are not required but recommended to reduce operation time. In general, machine in ESPL can handle about 35-45 IPM.

   **Plunge_Feed** – The rate of plunging tool into the material. It depend on the material, material of the tool, and the number of flute. In general, it is lower than Cut_Feed.

4. **Step_Over** – 1/3 of the diameter of the tool or less is generally recommended.

5. **Max_Step_Depth** – It depend on the material of the tool and number of flute. The operation time largely depend on this parameter.

---

### Table

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>CUT_FEED</td>
<td>13</td>
</tr>
<tr>
<td>ARC_FEED</td>
<td>-</td>
</tr>
<tr>
<td>FREE_FEED</td>
<td>35</td>
</tr>
<tr>
<td>RETRACT_FEED</td>
<td>40</td>
</tr>
<tr>
<td>TRAVERSE_FEED</td>
<td>40</td>
</tr>
<tr>
<td>PLUNGE_FEED</td>
<td>8</td>
</tr>
<tr>
<td>TOLERANCE</td>
<td>0.001</td>
</tr>
<tr>
<td>STEP_OVER</td>
<td>0.0625</td>
</tr>
<tr>
<td>PROF_STOCK_ALLOW</td>
<td>0</td>
</tr>
<tr>
<td>ROUGH_STOCK_ALLOW</td>
<td>0</td>
</tr>
<tr>
<td>BOTTOM_STOCK_ALLOW</td>
<td>-</td>
</tr>
<tr>
<td>CUT_ANGLE</td>
<td>0</td>
</tr>
<tr>
<td>MAX_STEP_DEPTH</td>
<td>0.0625</td>
</tr>
<tr>
<td>SCAN_TYPE</td>
<td>TYPE_3</td>
</tr>
<tr>
<td>CUT_TYPE</td>
<td>CLIMB</td>
</tr>
<tr>
<td>ROUGH_OPTION</td>
<td>ROUGH_AND_PROF</td>
</tr>
<tr>
<td>CLEAR_DIST</td>
<td>0.15</td>
</tr>
<tr>
<td>SPINDLE_SPEED</td>
<td>2666</td>
</tr>
<tr>
<td>COOLANT_OPTION</td>
<td>OFF</td>
</tr>
</tbody>
</table>

---

vi. Click Ok.

vii. Play the path and check
Note: The path is more visible if you hide the mill volume.
C. **Profile Milling**

Profile milling is generally used to create outer shape/profile from the stock material. When using this operation, you should be careful on the set-up and the path, so that the tool does not run into the vice or the platform. This operation is great to create large hole, enlarge any hole, or profile inner arc/shape.

i. Click Profile Milling

![Profile Milling](image)

ii. Select the tool

iii. Select the reference surfaces

Ctrl – Click multiple surfaces

![Profile Milling Diagram](image)

iv. Set Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>CUT_FEED</td>
<td>✓</td>
</tr>
<tr>
<td>ARC_FEED</td>
<td>-</td>
</tr>
<tr>
<td>FREE_FEED</td>
<td>-</td>
</tr>
<tr>
<td>RETRACT_FEED</td>
<td>-</td>
</tr>
<tr>
<td>PLUNGE_FEED</td>
<td>-</td>
</tr>
<tr>
<td>STEP_DEPTH</td>
<td>✓</td>
</tr>
<tr>
<td>TOLERANCE</td>
<td>0.001</td>
</tr>
<tr>
<td>PROF_STOCK_ALLOW</td>
<td>0</td>
</tr>
<tr>
<td>CHK_SRF_STOCK_ALLOW</td>
<td>-</td>
</tr>
<tr>
<td>WALL_SCALLOP_HGT</td>
<td>0</td>
</tr>
<tr>
<td>CUT_TYPE</td>
<td>CLIMB</td>
</tr>
<tr>
<td>CLEAR_DIST</td>
<td>✓</td>
</tr>
<tr>
<td>SPINDLE_SPEED</td>
<td>✓</td>
</tr>
<tr>
<td>COOLANT_OPTION</td>
<td>OFF</td>
</tr>
</tbody>
</table>
1. **Cut_Feed** – Cutting speed. It depend on material, RPM, and diameter. For aluminum 6061, 10-16 IPM was used for 3/8 inch end mill operation.

2. **Arc_Feed** – If the value is not given, default is Cut_Feed.

3. **Free_Feed / Retract_Feed** – They are not required but recommended to reduce operation time. In general, machine in ESPL can handle about 35-45 IPM.

   **Plunge_Feed** – The rate of plunging tool into the material. It depend on the material, material of the tool, and the number of flute. In general, it is lower than Cut_Feed.

4. **Step_Depth** – It depend on the material of the tool and number of flute. The operation time largely depend on this parameter

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>CUT_FEED</td>
<td>13</td>
</tr>
<tr>
<td>ARC_FEED</td>
<td>-</td>
</tr>
<tr>
<td>FREE_FEED</td>
<td>35</td>
</tr>
<tr>
<td>RETRACT_FEED</td>
<td>40</td>
</tr>
<tr>
<td>PLUNGE_FEED</td>
<td>8</td>
</tr>
<tr>
<td>STEP_DEPTH</td>
<td>0.0625</td>
</tr>
<tr>
<td>TOLERANCE</td>
<td>0.001</td>
</tr>
<tr>
<td>PROF_STOCK_ALLOW</td>
<td>0</td>
</tr>
<tr>
<td>CHK_SRF_STOCK_ALLOW</td>
<td>-</td>
</tr>
<tr>
<td>WALL_SCALLOP_HGT</td>
<td>0</td>
</tr>
<tr>
<td>CUT_TYPE</td>
<td>CLIMB</td>
</tr>
<tr>
<td>CLEAR_DIST</td>
<td>0.15</td>
</tr>
<tr>
<td>SPINDLE_SPEED</td>
<td>2566</td>
</tr>
<tr>
<td>COOLANT_OPTION</td>
<td>OFF</td>
</tr>
</tbody>
</table>

v. Click Ok

vi. Play the path and check
(In general, the above operation is impossible since the piece must be held by the vice, but the tool will run into the vice. In real application, you must set the depth and make a space for vice grip.)

Note: When selecting the reference, it is possible to select boundary by using shift when clicking the surface.
D. **Trajectory Milling**

Trajectory milling simply create a path along the chosen trajectory. In this guide, trajectory milling does not cut/remove any material, but it can be generally used to create chamfer or round feature.

i. Click Trajectory Milling

![Image of Trajectory Milling](image)

ii. Select the tool

iii. Set Parameters

iv. Click the top surface of the part

![Image of part with top surface selected](image)

v. Select Tool Motions

   a. Go to Tool Motion tab

   ![Tool Motion tab with options](image)

   b. Click the type of cut (Curve Cut, Drive Surface Cut, Follow Curve) (example is Curve Cut)
c. Select the trajectory

d. To make a chain, click detail

e. Using Ctrl, click chain of trajectory
6. Click Ok

f. To add more trajectory, click ‘insert here,’ and repeat

g. Click Ok

vi. Play the path and check
E. **Finishing**

Finishing is an accurate operation to finalize the piece with lowest tolerance available. It is important to remember this operation is not meant to remove the large amount of material. Instead, it runs through all the surface and tries to remove any rough surface. It is generally used at the end of any end-mill operation.

i. The finishing operation requires Mill Window.

ii. Click Finishing

![Finishing icon](image)

iii. Select the tool

![Tool selection](image)

iv. Select References – go to ‘References’ page → Select Mill Window in model tree

![Model tree](image)

v. Set Parameters (for detail, click ![link](image))
vi. Click Ok to create operation

vii. Play the path to check
6. Export the G-Code

A. Single Operation

i. Go back to Manufacturing tab

![Manufacturing tab](image)

ii. Select Save a CL File

![Save a CL File](image)

iii. Click ‘NC Sequence’ from menu

![Menu Manager](image)
iv. Click the operation that you want to export (4. Volume Milling 1 for example)

v. Click ‘File’

vi. Check ‘MCD File’
vii. Click ‘Done’ and save

viii. Click ‘Done’
ix. Click UNCX01.P12

x. Close the log file, and click Done Output

xi. Go to Working Directory
xii. The g-code file extension is .tap file. Open .tap file in Notepad. ('volume_milling_1.tap' for example)

![Image of g-code file]

xiii. Bring this file to CNC machine.

xiv. Note: Before running the code, the machine will stop at N25. This is because T1 M06 means change the tool to tool #1 (T1), and pause the machine until the tool-change is done (M06). This is used for CNC with ATC (Automatic Tool Change). Otherwise, machine will just stop until start button is pressed again, so that people can manually change the tool. In this case, it is ok to delete the line N25 before starting the operation.
B. Multiple Operation

i. Click ‘Save CL File for a Set’

ii. Click Create in the menu

iii. Enter the set name, and click the checkmark

iv. Check necessary operations in a set

v. Click ‘Done Sel’ (Done Selection)

vi. Note: You can also modify the set or create different set.
vii. Select the Set (Set01)

![Menu Manager](image)

viii. Click Display and Done to generate the path. Check the path before output.

![Path Manager](image)

ix. Click ‘File’ and check ‘MCD File’

![Menu Manager](image)

x. Click ‘Done’

xi. Save the set
xii. Click ‘Done’

xiii. Click UNCX01.P12

xiv. Close the log file, and click ‘Done Output’

xv. Go to working directory, and find .tap file for the set
Bring this file to CNC machine.