|  |
| --- |
| AutoDesk Inventor 2017 HSM – EdingCNC Turning |
| Basic manual for Turning in Inventor HSM and USBCNC |
|  |
| This document is a quick tutorial in making a part in Inventor, setting up CAM operations and generating USBCNC compliant GCode. |
| Arjan Grootenboer June 2017 |

AutoDesk Inventor 2017 HSM – EdingCNC Turning

Basic manual for Turning in Inventor HSM and USBCNC

# Computer Aided Manufacturing

• • •

In the past CAM was quite complicated. It took weeks to understand a CAM package. Since Inventor launched HSM things became a lot easier. You can have your lathe turning the first parts in hours instead of weeks. This document will help you to get started with Inventor 2017 HSM and Eding USBCNC. Helping you to avoid some common mistakes and some pitfalls, giving you al smooth experience with these 2 excellent pieces of software, combined with the hardware from Eding.

# Getting Started

To get started we need some pre-requisites. These are: Inventor 2017 installed, the HSM module installed. EdingCNC installed. To be able to turn on a lathe you either need HSM, or the HSM professional version. HSM Express does not support turning. For both Inventor 2017 and HSM you can download a free 30 day trail at [www.autodesk.com](http://www.autodesk.com).

Out of the box however this won’t work. To work with EdingCNC you need GCode.

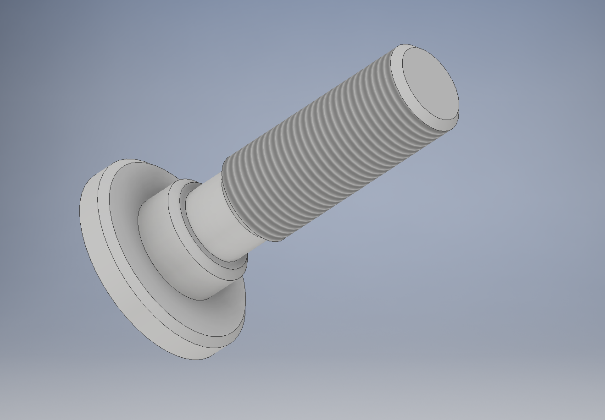
The workflow is the following: you draw a 3D object in Inventor and HSM converts that into machine readable GCode. There are however many flavors and dialects of GCode. To overcome this problem and to make sure your machine (or better your controller) does understand the GCode you need a Postprocessor. HSM generates general code, which can be finetuned for your setup with this postprocessor.

The postprocessor that works with EdingCNC can be found here: [www.edingcnc.com](http://www.edingcnc.com). If you install Inventor HSM you will get a PP (Postprocessor) but that one does not work with a Lathe as it is intended to be used for milling. And one of the best features offered by EdingCNC, which is threading with a G76 cycle is missing for example in the milling version.

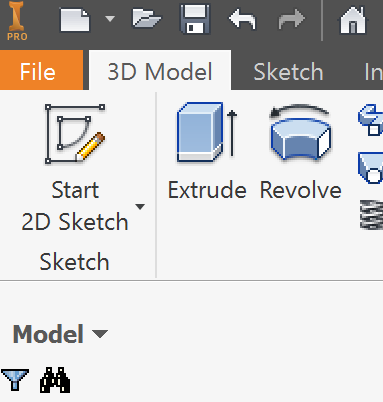
Ok, from this point on we assume you have the right version of Inventor installed, and HSM. The first thing we have to do is downloading the PP for Turning and placing that in the directory where HSM looks for the PP’s. The PP files have the extension .cps. On my computer the cps files are located in: C:\Users\Public\Documents\Autodesk\Inventor HSM\Posts. Copy your EdingCNC PP file to that location. Ok, we are all set!

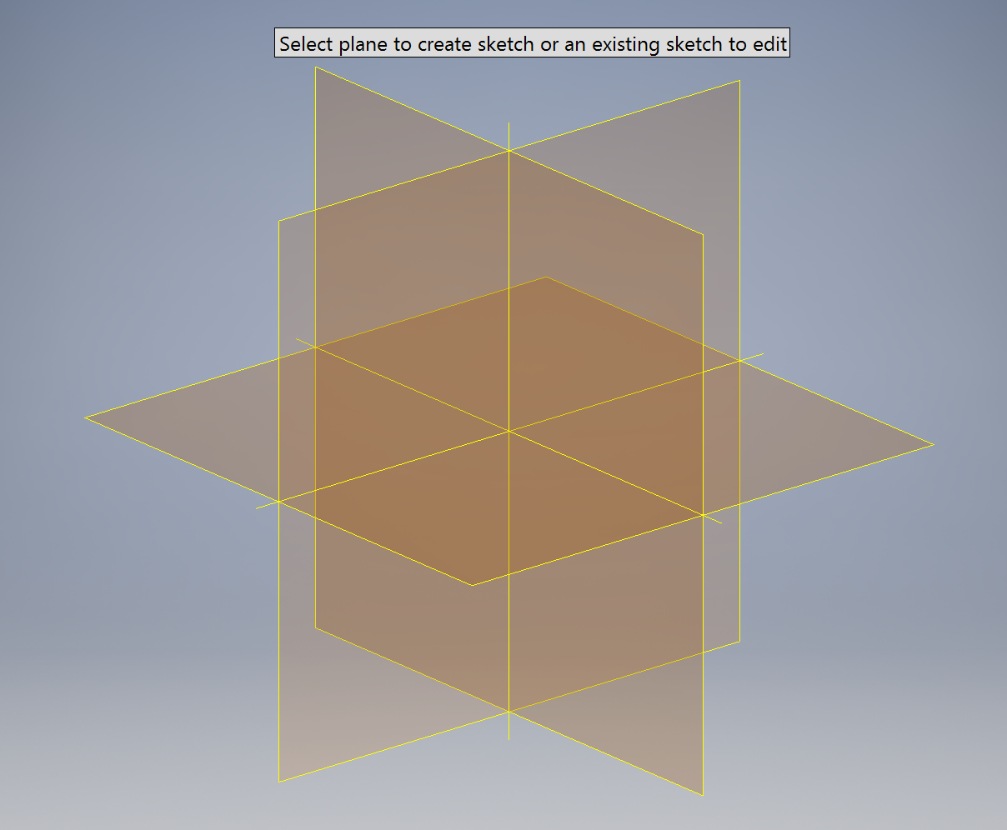
# Drawing a part in Inventor

I will leave out all the things you don’t really need or will be discovering yourself in the future. There are however some very important things to understand from the start. This will save you a lot of trouble, and perhaps even your Turret going full into your Jaws, I speak from experience. That’s really not funny. So let’s draw our first part.

It’s good practice to organize your projects in Inventor. For that we will start a new project, we’ll call it Screw. Go to Projects on the button bar top left, in the popup screen choose the button new in the lower left corner. Choose Single User, and click next. In the “Project name” textbox type Screw and click finished, then click done. We have now created a new project called Screw.

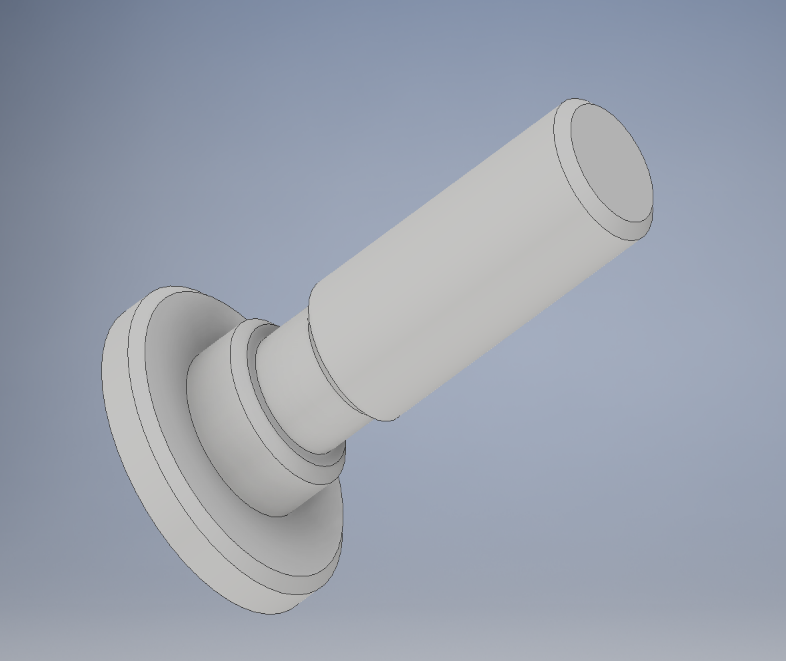
Now we have our project we need to create a part. In the top left corner click on the New icon. If you have setup AI in the metric system you will get a metric part. If you choose the New icon slightly below the new icon top left you can choose many more types of parts, if you prefer this route, please make sure to select a Metric part.

We have an empty part now. The next thing is sketching the part. Click on Start 2D Sketch. Now you see some planes to start your sketch. Please try to understand this part carefully. Coordinate systems are very important in CAD/CAM. We are going to create a screw. We can do this by starting a circle with a D=4 mm and extrude that. Then select an end of the screw, start a new sketch on that. Make a circle for the head of the screw, extrude this again. If you want to go this route, than select the XY plane.

If you need more flexibility and have a more complicated part, the XY Plane won’t work very well, than select the XZ plane. The XZ plane is the same as the Z axis / X axis on your lathe. Where the Z is the axis coming out of your Chuck, and the X the Tool. In the XZ plane you can draw in the same plane as your lathe will be later working on. The centerline in your sketch should be a centerline, when finishing the drawing use the Revolve command to generate the part.

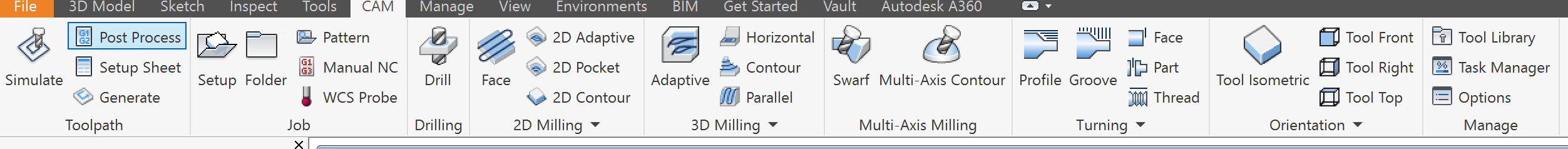
Now start a sketch in the ZX plane. If you are done, click on finish sketch. Back in the 3D model tab click on Revolve. Select the profile and the centerline if AI has not found that automatically yet. If all went well you have ended up with a basic “special” screw. To finish the whole thing we can do some chamfering and put a Thread on the first 6 mm diameter part. The part should now look like this. Next save the part and name it for example “screw.ipj”.

This finishes the boring part, now we move to the interesting part CAM.



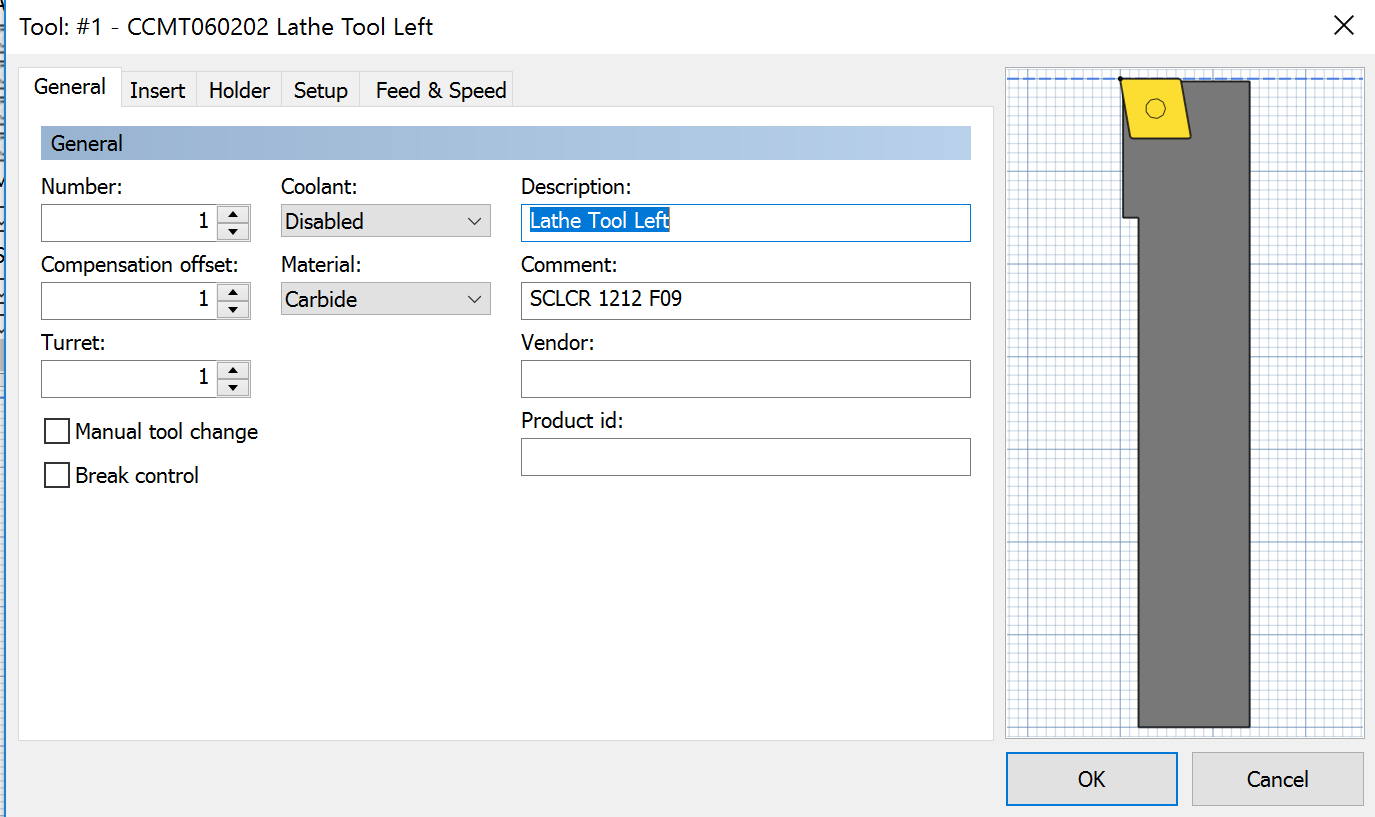
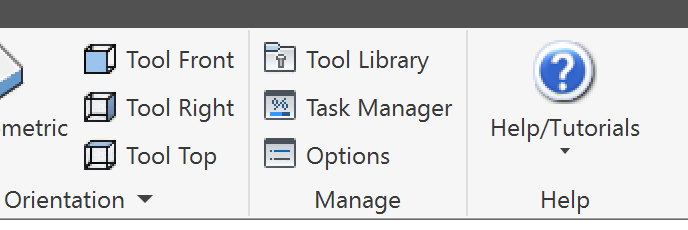
# C:\Users\gbr0.SPRINGERNATURE\AppData\Local\Microsoft\Windows\INetCache\Content.Word\CAM -1.png

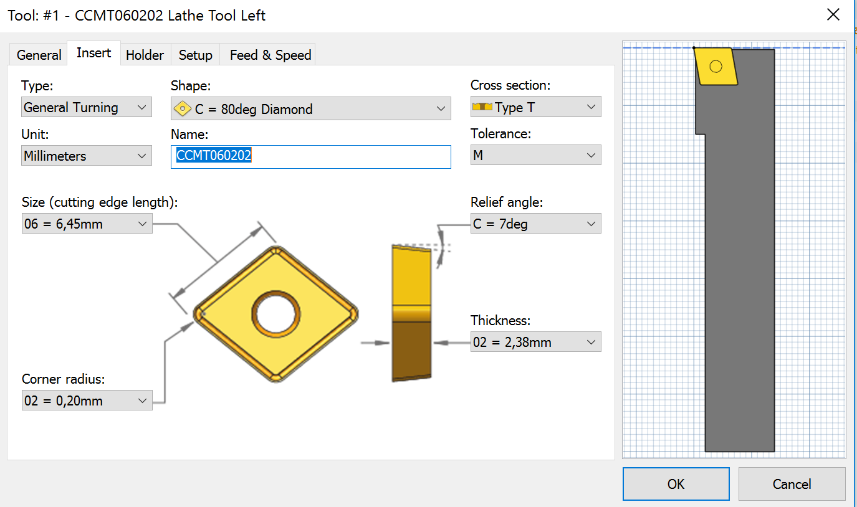
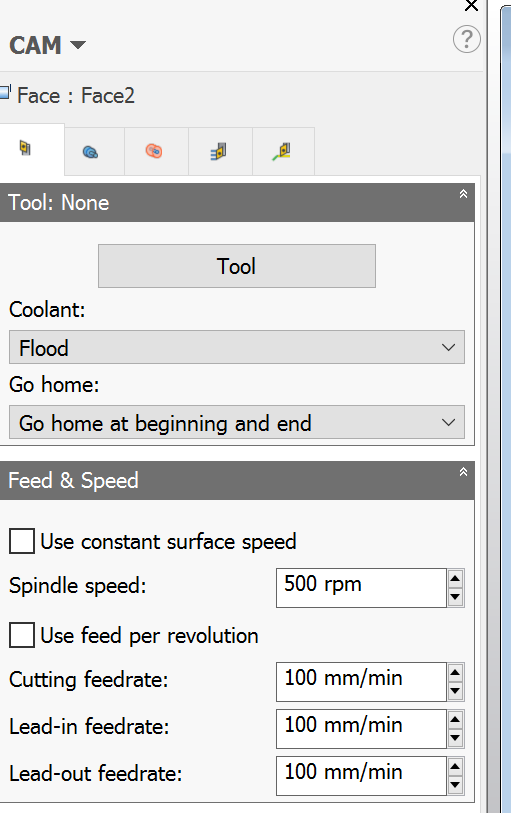
# HSM CAM

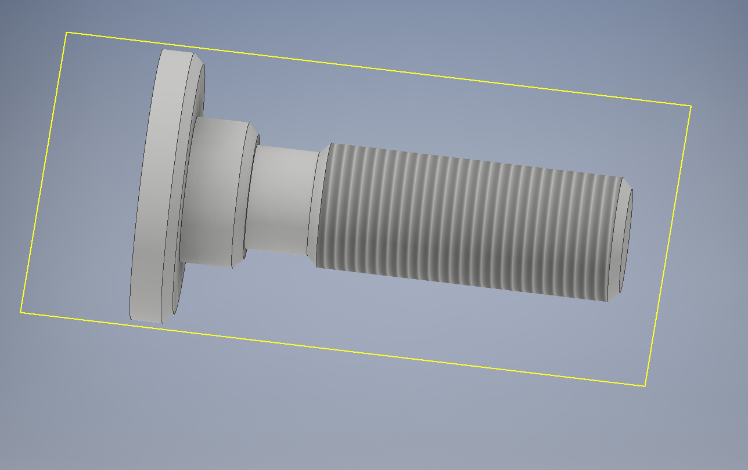
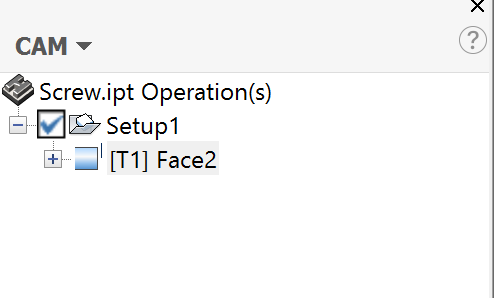
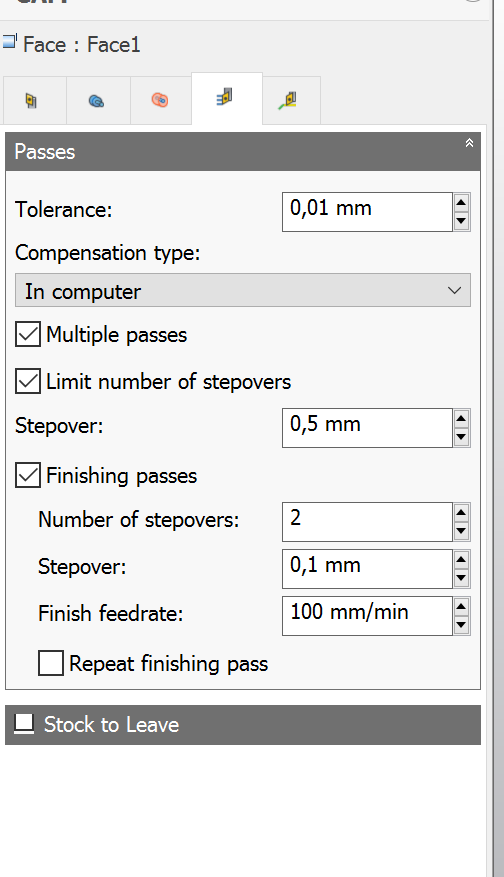
We have our part now, the screw. Let’s fire up the CAM module. Click on CAM in the ribbon bar on the upper part of the screen. 

# C:\Users\gbr0.SPRINGERNATURE\AppData\Local\Microsoft\Windows\INetCache\Content.Word\CAM -2.pngFor now we will concentrate on the Turning part. If you have an automatic turret with for example a drill you also need the Drill option under drilling to drill a hole in the part. The first thing we need to do is to setup HSM. Click on the Setup icon. On the left you will see the CAM explorer, with the basic setup. Change the operation type to Turning or mill/turn. This will result in the stock appearing round instead of square. The radius of the biggest part of our screw is 8 mm, so put in (2 x 8) + 2 mm = 18 mm for the diameter of the stock. Make sure the coordinate system is starting at the threaded part of the screw, where the Z axis points away from the screw and the X axis, C:\Users\gbr0.SPRINGERNATURE\AppData\Local\Microsoft\Windows\INetCache\Content.Word\AI Sketch Screw.pngsee image. On Home position choose “from stock front” and fill out 2 mm to have some material to face. In the next tab, stock fill out the stock details, 18 mm diameter and 30 mm long. Model position 2 mm offset from front. This concludes the setup, click Ok in the left bottom.

# First operation: Facing

After the setup we are ready for the first operation. This will be facing. For this we will be using an SCLCR 1212 F09 Tool with an CCMT060202 insert. It will save you a lot of hassle if you right away start your own Tool Library. For that click on the Tool Library Icon and right click under the My Library list item. If you place your tools in your own Library you can just select them for future operation, avoiding to redefine your tools again and again. Give your tools a number, and if you are using a turret, fill out the correct turret position, this will generate automatically the right M6 Tx in the GCode.

Select Face in the top ribbon bar. On the left CAM menu press the Tool button and select the right tool. In my case that’s a SCLCR 1212. Adjust the Feed rates and Spindle speed according to your machine’s specs. In my case the cutting federate is 100 mm/min and the speed is 1000 RPM. Setting are not perfect now, but let’s see what happens when we simulate this. Press the Ok button in the face operation. Your screen on the left should now look like this, having a first setup and a first operation.

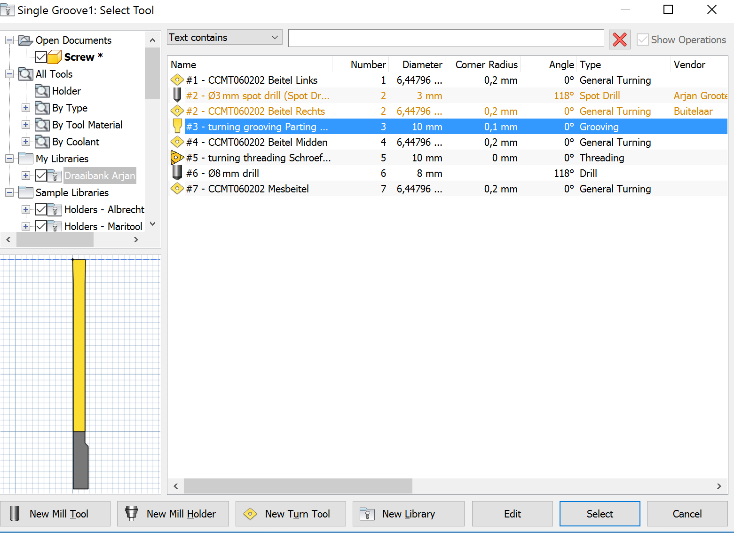
Now select Setup 1 on the left and press the Simulate button. You will see a tool appearing, and on the bottom a timeline. In the left menu click on small icon left to Stock, to see your stock and how it looks before we do our first operation. Click Play in the bottom of the screen and watch how 2 mm gets turned off from your stock in one go with 1000 RPM and at a feed of 1000 mm/min. This is way too much for a smaller CNC Lathe. To correct this Right click on [T1] Face 1 in the left menu and select Edit. Now you are back in the menu to edit the operation. Change the cutting federate to 100 mm/min in the first tab. Then click on the Passes tab icon, and select multiple passes, Limit number of stepovers, and select Finishing passes, set Stepover at 0,5 mm and Stepover in the Finishing passes at 0,1 mm. This will give a smooth finish, some roughing passes taking of 0,5 mm and finishing off passes taking off 0,1 mm of stock. This concludes our first operation.

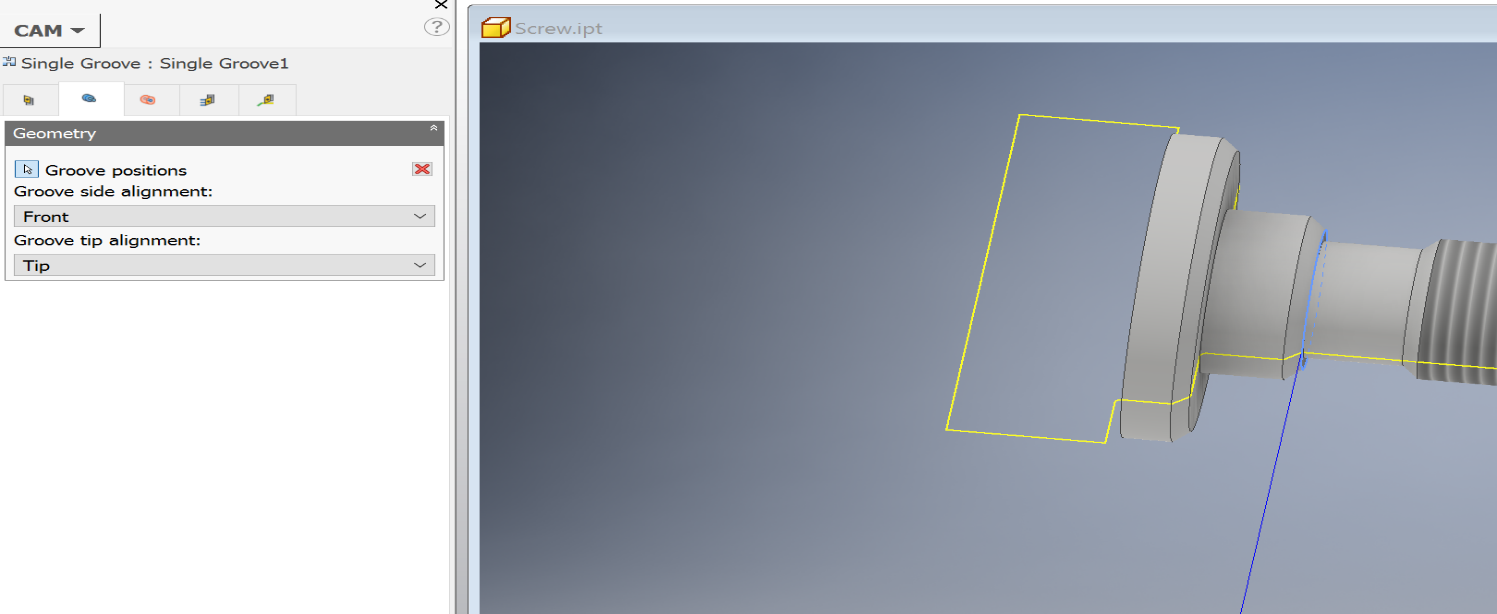
# C:\Users\gbr0.SPRINGERNATURE\AppData\Local\Microsoft\Windows\INetCache\Content.Word\Groove -2.pngSecond operation; Profiling

For the second operation we will do some profiling our our screw. Close the simulation and select Profile in the top ribbon bar. Adjust the cutting federate and Spindle speed, and then continue to change the Roughing passes a and the Finishing passes in the 4th tab. If all settings are correct for your machine click the Ok button.

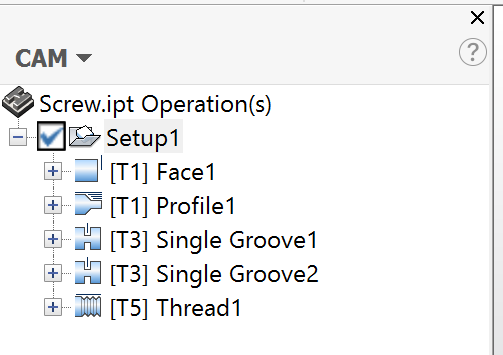
# Third operation; Single Groove

# C:\Users\gbr0.SPRINGERNATURE\AppData\Local\Microsoft\Windows\INetCache\Content.Word\Groove -3.png

 In the 3rd operation we will be making the groove just below the head of the screw. This acts as a relief for the thread we are going to make later. Select the Single Groove operation from the top ribbon. Please note it’s Single Groove, and not the Groove operation. To select this option you have to expand the menu by clicking on Turning with the arrow down at the left.

Select a parting tool in the library of tools, and set the correct Spindle Speed and Feed rate’s. the relief for the thread is 3 mm in length and the width of our parting tool is 2,27 mm. This means we have to come in twice, once on the left and once on the right. We will start with the left part of the groove, the side close to the head of the screw. On the second Geometry tab in the operation select the bottom of the groove close to the head of the screw, and set the Groove side alignment to Front. Groove tip alignment can stay at Tip.

# Fourth operation; Single Groove

Repeat the steps in the previous operation, but now select the bottom of the other side of the groove, close to the thread, and set Groove side alignment to Back. Now run the whole simulation to check. Do this by selecting Setup 1 in the CAM explorer and click Simulate.

# C:\Users\gbr0.SPRINGERNATURE\AppData\Local\Microsoft\Windows\INetCache\Content.Word\Threading -`1.pngFifth Operation: Threading

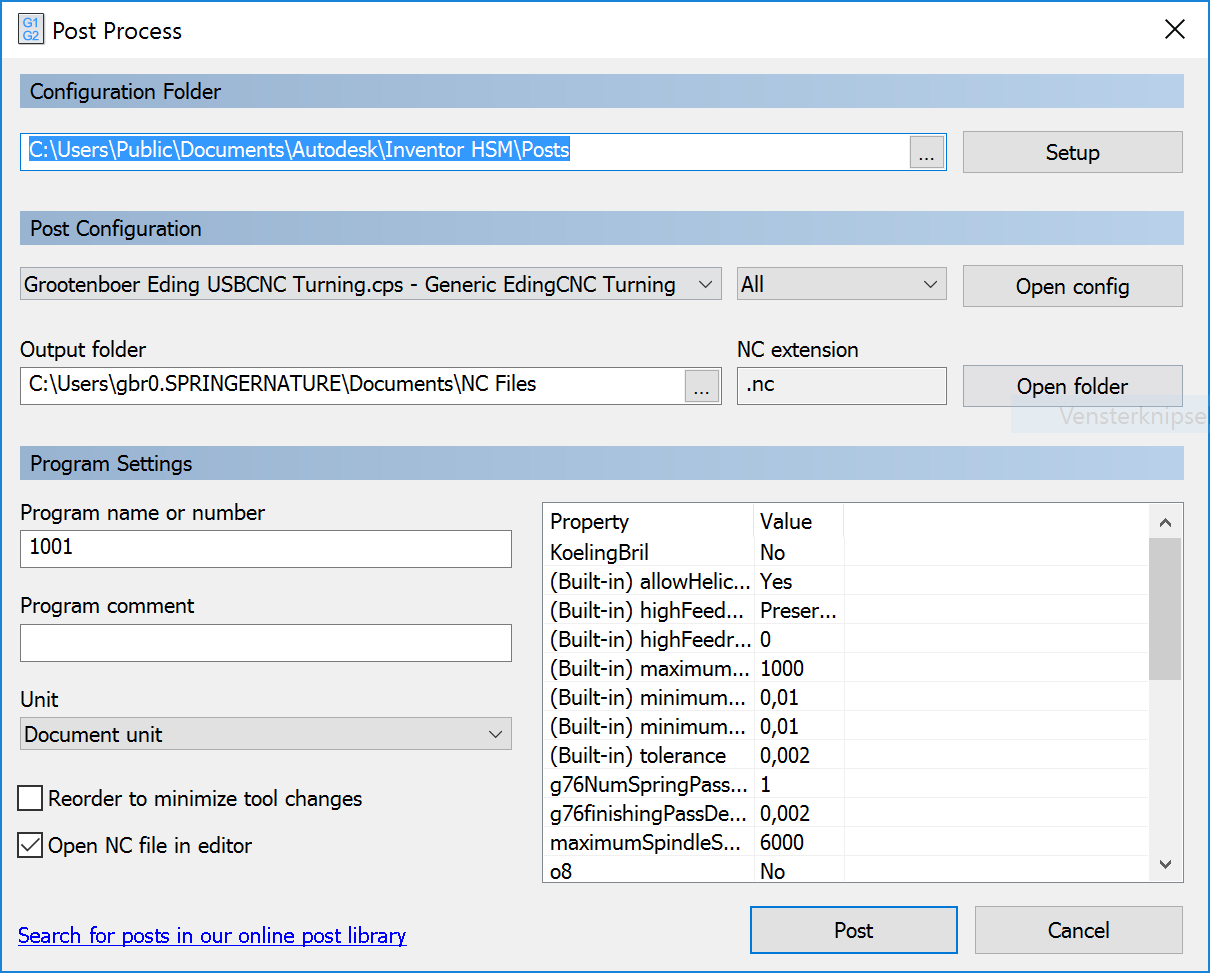
Select Thread for our last operation in the ribbon bar. Select a tool to cut threads, set the spindle speed and all the feeds. Then select in the second tab the Thread faces. Frontside stock offset at 2,5 mm and Backside stock offset at 0 mm. In the 4th tab set Thread depth to 0,5 mm and thread pitch to 0,5 mm (for M6 x 0,5). Click under Constant infeed on Use cycle. This is very important for the Postprocessor later.

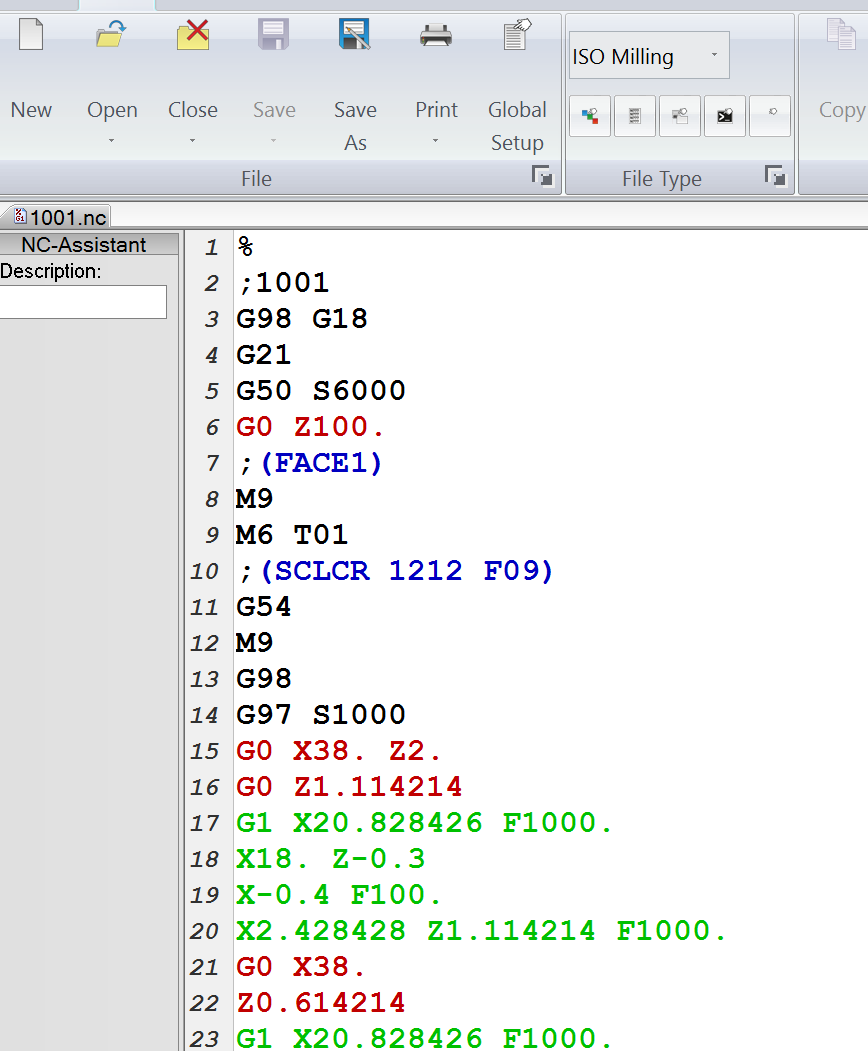
Run the whole simulation and look if everything went ok. You can speed up the simulation with the small dot under the timeline, shift it all the way to the right.

Done! The only thing we have to do now is running the postprocessor.

# Postprocessing and generating GCode

Now we have all the operations, its time to generate some GCode. In the CAM explorer select our first operation, Face 1. Click on Post Process in the upper left corner, you will see the PP window. In the Post Process window select the Post Configuration. You will find the PP you have downloaded over there, it’s called Grootenboer Eding USBCNC Turning.cps.



Select a number for your program, 1001.NC in this case and click on Post. Now you will see the NC editor popping up, showing you the generated CNC code for our facing operation. Please note also a correct M6 T01 is in. Selecting the right tool from our turret.

# Running our first GCode

Before running this program in USBCNC, make sure the following things are ok:

1. Your machine has valid limits, so no accidents will happen with tool going where they shouldn’t.
2. Home all axis before you begin. This can also be set in the EdingCNC configuration, mandatory homing before program start.
3. If you use a turret, make sure the software has the same tool in mind as the one actually in the turret.

IMPORTANT

Now mount the 18 mm stock in your lathe and set the WCS, The work coordinate system. This can be done by moving the tool you are using with the Jog panel to beginning of our stock. Set this position to Z=0 with the following command:

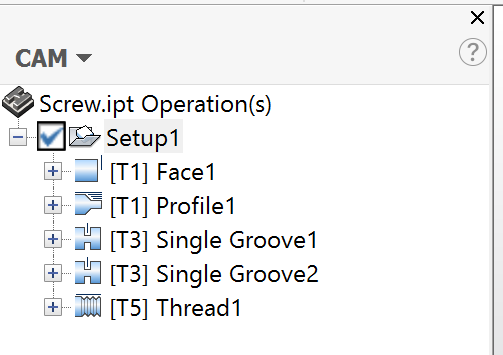
G92 Z0.00

Now move your tool in the X axis and do this until the tool just scratches the stock. If you have 18mm stock this means tool 1 is exactly at position X = 18. Enter the following command, setting the WSC sytem:

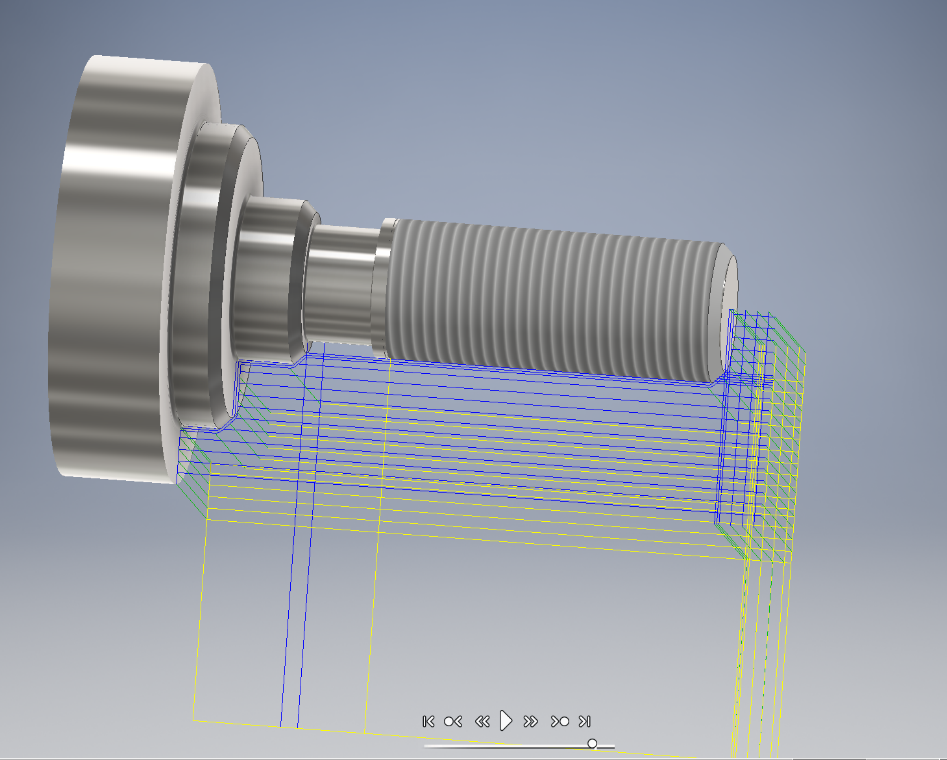
G92 X18.00

Now the coordinate system of our machine and the code we just generated matches. A minus Z value moves the tool into the stock, and all X below 18.00 also moves into the stock.

Be very careful with G28 moves. This moves the Tool into Z and X direction according to your G28 settings. This can go really wrong depending on the tool orientation. Better to avoid these moves. In the 0.2 Beta version of our PP, all G28’s have been removed.

Ok. This all set, go to Auto in the EdingCNC menu and select Load. Load the GCode file for facing we have just made. Run the code and see the part being faced. If you are not really sure on how this is going to work it’s a good idea to practice first without stock in the machine, of some PVC or Plastic.

If this works we can generate the whole program. Select Setup 1, and click on Post Process. This will generate the whole GCode for making our Screw, including the G76 cycle for Threading.



Arjan Grootenboer.

Please Note:

Although the author has made every effort to ensure that the information in this document is correct at the time of publication, the author does not assume and hereby disclaim any liability to any party for any loss, damage, or disruption caused by errors or omissions, whether such errors or omissions result from negligence, accident, or any other cause.