Mill Turning on a Tormach PCNC440 with Fusion 360

Introduction

I am basically a lazy person. My father and my grandfather both impressed on me the adage that 'if a machine could do it then a human should not be doing it'. In my working life while running a business I was always keen to get in the latest piece of kit to make my team's work easier and more to the point quicker. As a result, for a small company, we had a disproportionately large investment in fantastic facilities and systems.

Being retired puts a different focus on time. How much more time have you got left and what do you want to do with this unknown period. Well I like making things and clocks in particular interest me.   I have made William Smith's Gearless Gravity Arm design and enjoyed it and the next project is going to be something more conventional.   Bill's design convinced me that I don't want to spend large tranches of time cutting and spoking wheels, marking out and drilling plates and turning support pillars so that all look identical.   There has to be an easier way.   The purists will howl with derision that I am undermining the skill and prestige of 'making it all by hand' but I would argue that there is a prestige and skill in having the vision to see and use evolving processes

On my blog are details of using the Sherline CNC indexing table to make cutting wheels on the lathe simpler and more accurate than conventional indexing plates.   I am also now much more comfortable with CNC having purchased the Tormach PCNC440 CNC milling machine.   On my site I also detail the use of Gearwheel Designer for direct CNC cutting of clock wheels on the CNC mill. Finally with a CNC mill it is possible to easily and accurately cut clock plates to support the mechanism.

What is missing from this accumulated list of modern approaches is the ability to produce the pillars that rigidly space and support the clock plates.   More than anything else these pillars need to be identical in length and visually of the same shape. The conventional means of construction is to turn these on a lathe and try to achieve these important characteristics by skill of hand.

My first reaction is that I need to invest in a CNC lathe but frankly I am running out of space in my workshop and at my time of life you find that money does not grow on trees. Having said that the Sherline CNC lathe products would probably do the job.

Tormach make a device called the Mill Turn that is a lathe style motor and chuck that sits on the CNC mill table and gives a turning facility.   This is fully integrated with the PathPilot machine controller software.   The problem is that this facility is not available on my smaller 440 version.

I considered trying to mimic this facility using the Sherline motor and spindle assembly that I already have but couldn't clearly see how I might achieve this.

A few hours (quite a few in fact) browsing on YouTube lead me to the conclusion that a technique called Mill Turning might solve my problem for the limited number of occasions I would need this facility. In a nutshell Mill Turning is the technique whereby the material to be machined (let's say a brass rod) is mounted in the mill spindle and the cutting tools are fixed on the milling table.   The milling software thinks the brass rod is the cutting tool and presents it to the fixed cutters in a way that faces, parts, shapes and drills the material.   I hope with this write up will help to clear the mists on how this is implemented.

Overview

The process of Mill Turning can be broken down into the following steps : -

1   Design the item to be turned in CAD software (e. g. Fusion 360).

2   Create a CAM GCode program from the CAD design and load into the CNC mill.

3   Mount the material to be machined in the mill spindle.

4   Mount the cutting tools required on the milling table.

5   Program the offsets (WCS) for each tool so the program knows which tool to position the material against.

6   Hit start, get a cup of coffee and watch the action.

Well that sounds easy …. But there is a lot of detail involved.

Design in CAD

I will reference using Fusion 360 as this is my preferred program.   This is also the easiest part of the process (apart from sitting back with your coffee).

1   Think about the turned shape you want as the result of the machining operation.   Cut a vertical slice through the diameter and then regard the result as two symmetrical halves. You now need to draw either the left or right hand half as you want it to be and then revolve extrude.   A few pictures will help clarify this.

2   Having got the solid model you might want to add axial tapped holes or stubs at the ends and these can be extruded directly on the model and Mill Turned where practical.

3   The finished model is now ready to move into CAM.
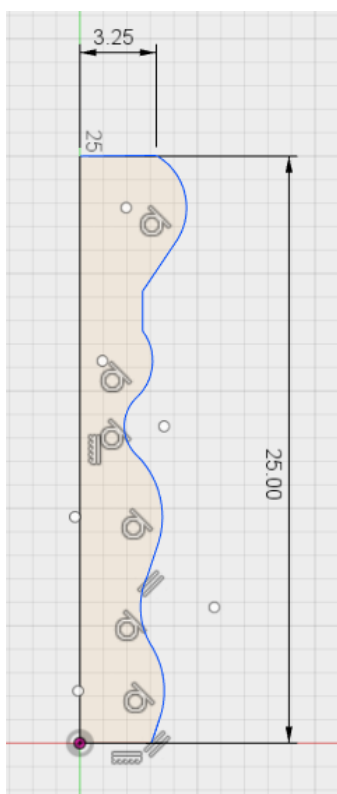
Half section starting Profile
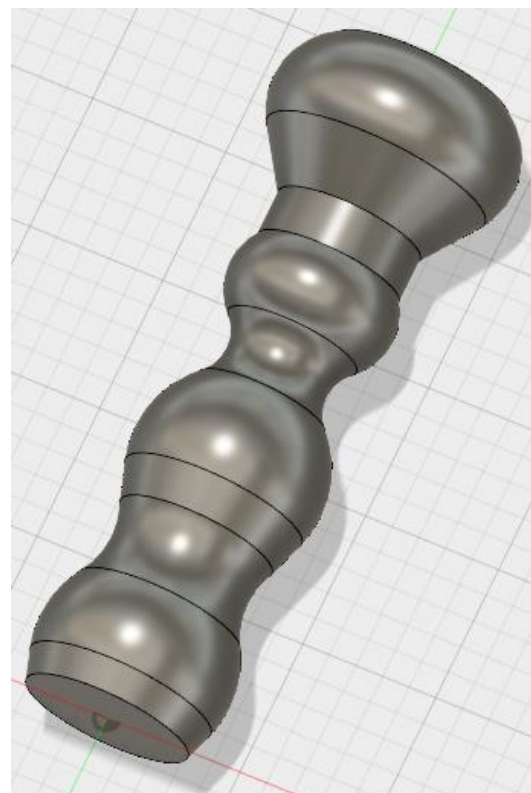
Rotated Extrusion

CAM Creation

If you are also using Fusion 360 you need to make a change to the post processor code to enable mill turning.   Rather than try to explain this I will refer you to an excellent YouTube video by Tom Zelickman.

https://www.youtube.com/watch?v=hIiGqXM8iu0

You might also want to view three videos by Jason Hughes which got me up to speed albeit with a few gaps. The first one is his first attempt at Mill Turning and is bit disjointed in not giving the full picture.   The second and third are a revisit by Jason that have been cleaned up and are very good.

https://www.youtube.com/watch?v=EUmf3ARA-DA

https://www.youtube.com/watch?v=qQOpOq1Ce8o&lc=z234xhhrlkz4x10wj04t1aokggqmz0b30fljqo em1so4bk0h00410&feature=em-comments
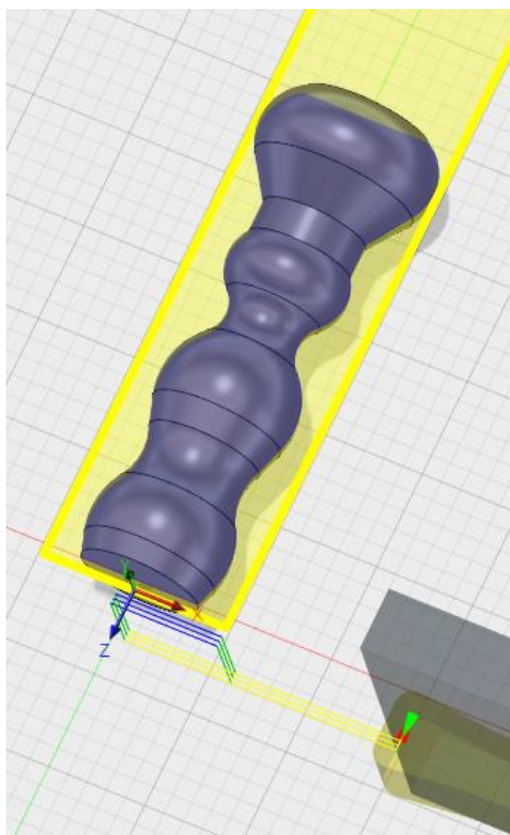
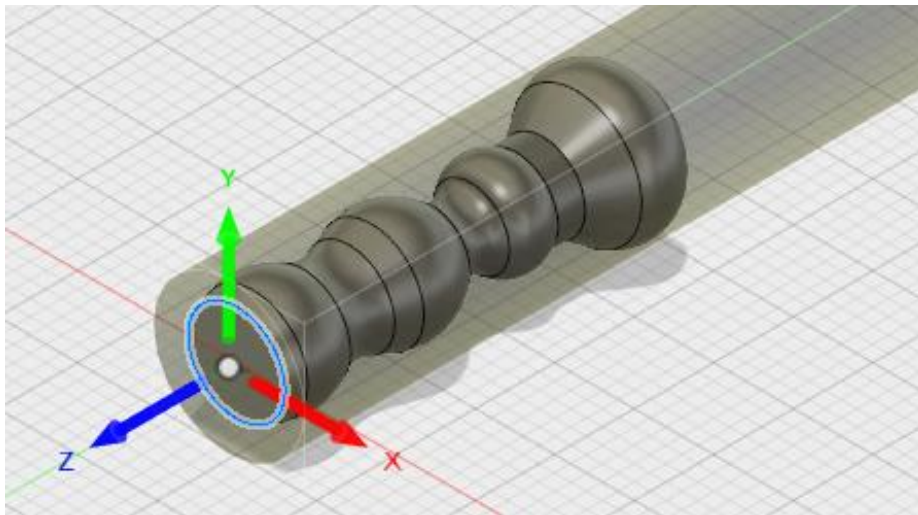https://www.youtube.com/watch?v=8xV16yVqocY

What totally passed me by (and is probably a reflection of my inexperience in CNC) is the concept of Work Coordinate System (WCS).   Let me digress to try to explain my understanding.
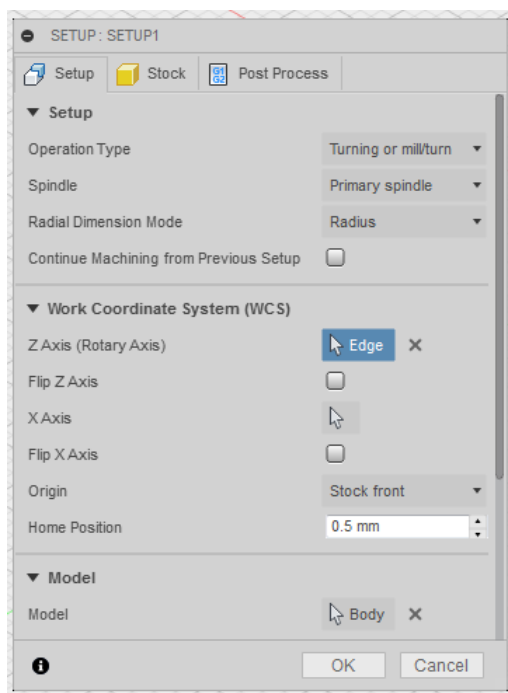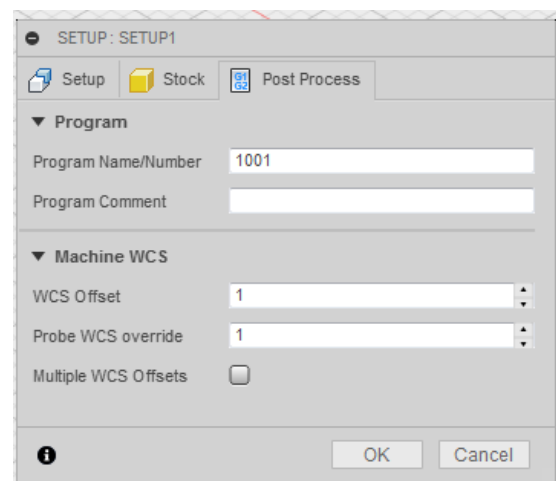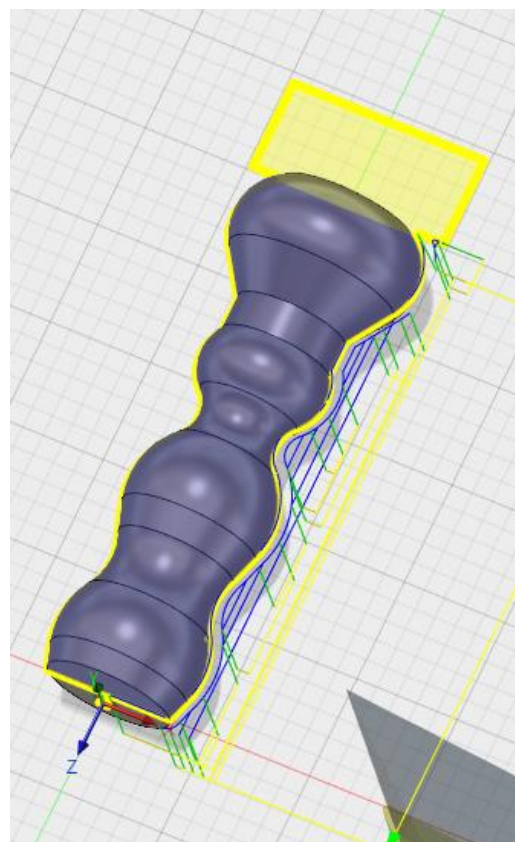
Setup First Tab	Setup WCS

Facing Tool Path

Profile Toolpath

Work Coordinate System

When you switch the mill on to begin working on it, the first thing that you do is Reset and then you home the X,Y and Z axis.   This moves the milling table and spindle to the extremes of the available travel. These are absolute limits and have X Y Z coordinates which are stored in a register in the controller software (PathPilot).   The data for these absolute references can be recalled in a machining program by using the GCode G53.

Further digression follows.

Assume you had a piece of material on the mill table that was exactly the size of the limits of travel of the X and Y axis.   If you wanted to machine the edges of this plate it would be impossible because you cannot physically get a tool in the spindle to sit outside of the X and Y limits.

The material therefore must be smaller in size that the absolute size of the milling envelope available to you for doing operations.   Once the material is smaller in size you can move it around within the milling envelope to best suit the machining operations.   The problem is that the machine no longer knows where the material is within its envelope.

To solve this problem you decide on some point on the material that will be your reference point from which all your machining operations will be measured.   Locating and defining this reference can be done by various methods.   The most common is to use a Taster or Haimer device. This has a dial indicator type display and you move the X,Y and Z axis to bring the reading to zero at a certain point on the workpiece.   At this position the centre of the milling spindle is directly over the reference point and the depth of the material is also now known. (There are some complications in setting Z but leave this for the moment).

As you have moved the spindle to each zero position you press the Zero X, Y and Z buttons on the display readout and this information is stored into the machine register in the same way as the absolute readings have already been done.   You now have two XYZ positions stored.   One is an absolute and one is a relative position. Unless you tell the software otherwise the variable zero position will be stored in the register accessed by GCode G54.   The machine now knows through simple calculations where the material zero position is (G54) relative to its machining envelope as stored in G53.

There are a number of these workpiece zero position storage registers available in the software. This is helpful when you have multiple operations to perform on the workpiece and one reference point would not serve all.   A good example of this is top and bottom side operations.

Tormach's PathPilot has G54, G55, G56, G57, G58, G59, G59.1, G59.2 and G59.3 as nine readily accessible work zero storage locations.   These are known as WCS locations.

For most simple machining jobs you will probably only use one WCS and by default this will be G54. If you are making two items at once on a jig you would also use two WCS registers so once the first part was completed, the mill could automatically repeat the processes on the next part 'moved along' the milling table. Watch John Saunders video for an example of this.

https://www.youtube.com/watch?v=-Qtvj5iM_VI

There is also a video by Kevin Ellison also on John Saunders NYC CNC website that goes into this and how to make this match on your CAM model.

https://www.nyccnc.com/fusion-360-cam-set-wcs-work-coordinate-system/

Why is this relevant to mill turning ?

Remember what I said earlier that mill turning puts the material in the spindle and the tools in fixed positions on the milling table.   Normally the tool in the milling spindle is moved to locations on the material based on a single WCS.   In Mill Turning the material is held in the spindle and you now use multiple WCS locations to tell the spindle where the tools to be used are located.   More of this later.

It is important to make sure that the tools that you are going to specify for each operation are physically the right shape to be able to 'get in' and machine the material shape you have defined. You need to think this through in your head.   If you use standard example tools stored in the Tool Library in Fusion this gets you along the road.

Some points emphasised in Jason's videos.

You need a different Fusion CAM Setup for each tool that will be used. Note that one tool can undertake a number of processes under the same setup.

You need to use the same Tool Number for each tool being used on the project and each tool will have its own setup in the Fusion Tool Library even though there are other tools with the same number.

You need to ensure that the offset from the stock front face is the same for all processes.

You need to change the WCS reference number to match the setups associated with each tool and the first reference must be '1' and not '0'. Fusion comes up with '0' as standard.   WCS #1 will be G54.

Write down which WCS number you have allocated to each tool setup and which tool this refers to on the milling table.

Setting up the Mill

You need to mount the cutting tools to be used in a secure manner on the mill table.   If it is a simple turning job you can use the machine vice to hold a parting tool and a shaping tool. Make sure there is enough space between the tools to allow the spindle and hence workpiece access. Note that the tools mount horizontally with the cutting face looking forward when on the left hand side of the spindle.

Having got the tools mounted, put a sewing needle in a collet and mount this in the mill spindle. Jog the milling table and spindle such as to bring the point of the needle immediately centred on the point on the first cutting tool edge where it will do the cutting. Make sure you have the common tool number you have chosen to use displayed on the PathPilot control panel.   Check which WCS is displayed and if it does not match your setup in Fusion change it on the MDI interface by typing G5x and return. Once the tool and WCS match, zero out X Y and Z with the needle point as close as you dare on the tip of the cutting tool.   This process is the same whether for a cutting tool or a drill bit etc.

Repeat for each tool in use on the job and put some form of marking on each tool so you can remember which WCS each one is referenced to.

Put the workpiece stock blank in the spindle and grip it tightly.   Try to minimise the amount sticking out to help rigidity but bear in mind the depth of the model that is needed from the collet to the cutting tool and that it clears the tool holders and each other.   This completes the WCS allocation and referencing.

Chose one of the tools and bring the workpiece cut end down as close as you can to the cutting tip of one of the tools.   Select the correct WCS for this tool using the MDI line.   In PathPilot chose the Offsets tab and then Tool. Click on Touch Z box.

This has now globally set all tool tips to Z0.   You can test this by going to each tool in turn, select its associated WCS via the MDI entry line and bring the spindle down to touch the cutting tip. You should see that the Z readout showing zero.

See Erik Colvin's video for more details on this process.

https://www.youtube.com/watch?v=7EOENs38kCo

All is now ready to run the job.   If you are like me and ultra cautious before running a new progam you can cut air by not bringing the material fully down onto the tip of the tool before setting the Z0. This will allow you to see if the material/spindle moves to the correct tool for each step of the program.

Conclusion

This process is not a replacement for a CNC lathe. Depending on the project it could well entail a long piece of material in the spindle that is unsupported at its free end. There is no ability to implement the equivalent of a tailstock to support it. It is important therefore to be very conservative in the depth of cut taken by each pass of the turning process.   Axial drilling does not have such a problem.

You will have noticed that most of the videos I have given references show the mill spindle having a conventional lathe chuck fitted.   These are not in a TTS collet but integrated onto a R8 spindle fitment.   This is a large amount of spinning mass and appropriate safety measures need to be considered.

From my point of view most of the items I would chose to make using Mill Turning are less than 12mm in diameter.   Anything over this diameter would almost certainly be undertaken in my manual lathe.   A 12mm maximum diameter is readily accommodated in an ER32 collet (20mm maximum) and Tormach offer an ER32 collet holder in their TTS range of tool holding products.   This is a less unwieldy mass to have in the spindle and less likely to lead to disaster scenarios.

Likewise it is possible buy ER collet holders in shaft extension styles and an ER16 size would accommodate drill bits up to 10mm.   The shaft extension style collet holders have a truncated body that will allow mounting in a jig on the mill table. (See across).   This gives flexibility on drill selection for axial drilling.   Two of these ER16 holders on the jig should satisfy most needs. Add a centre drill having a separate fixed mount position and also a boring bar. Erik Colvin's video gives an overview of this type of jig albeit with fixed drill bits rather than flexible collet mounts.

Closing Comment

This write up describes how I got to where I am now with Mill Turning and the references and resources I tapped into. Hopefully this will save you some time and will help you follow my route to my present state of progress. It may not fully paint the picture but hopefully will help clear some of the mists.