Computer Aided Manufacture and Robotics

CNC machining and software evaluation

Simon Hussey

6/1/2012

**Machining the gear change lever**

Part A

5) First the cad model of the gear level was created using Nx7.5. Once completed an stl file of the part was made and this was taken down to the labs and the CNC machine. The Program was written on the CNC machines computer before it was made into a physical model. The software on the computer was actually boxford 3D-geoCAM which is extremely similar to the denford quick cam application.

First step in the program was to select the material and size of billet. Axson lablite 109 modelling board came under 'plastics'. The model then needed to be resized so the machine knows size of the model that's going to be cut. The position of the part within the material needs to be defined before tool selection can take place.

The machine has a roughing operation and a finishing operation. They are shown as percentages. In this program Roughing makes up 90% and finishing 15%. The roughing is designated to an 8mm slot drill and the finishing tool is a 6mm ball-end.

Program needs to have cutting allowances and footprint control. This gives the machine information so it can successfully get to and cut all parts of the lever. You can also choose for the entire material to be roughed away which is what this program will do.

The tool path/CNC operations were chosen to be raster roughing and finishing as they gave the best finishes in the simulation. After simulating we check through the code to make sure that the deepest cut was within the billet depth so that no collisions with the CNC vice took place during machining.
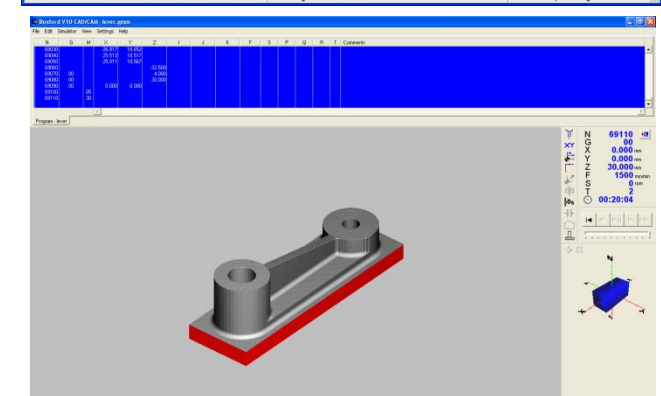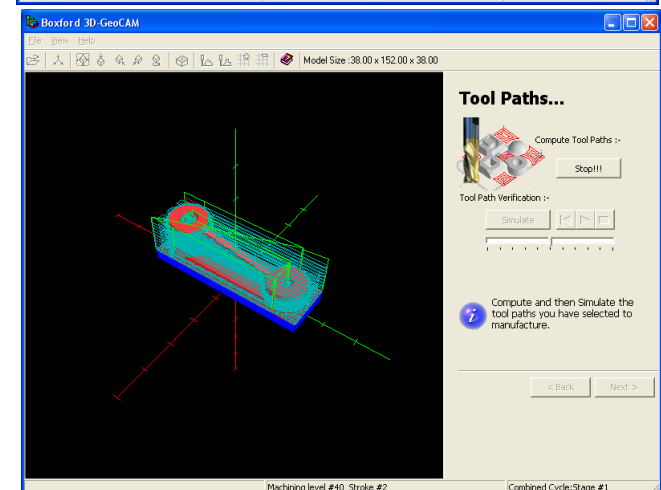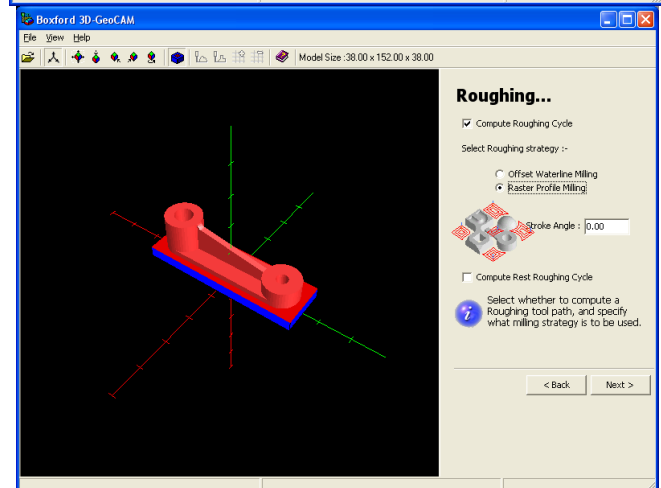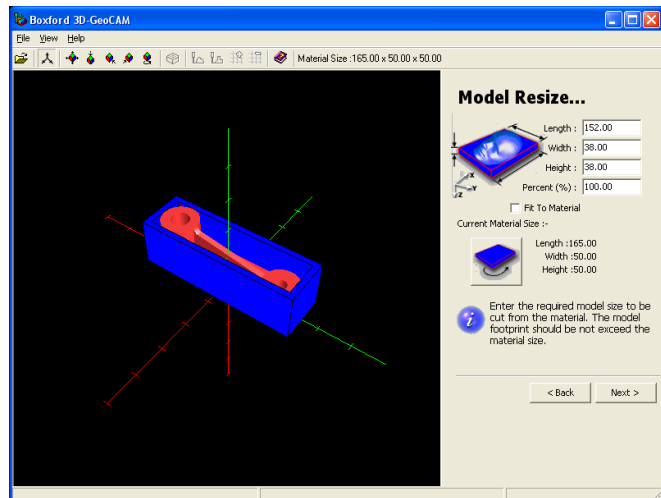
Part B Bracket

2) The bottom part will be machined first so that the holes will already be drilled and the fixture can be added and used for cutting out the top.

After loading the stl file into denford the orient of the model must be changed by 180 degrees so that it lies face down in the program. The next step of setting up the model is to set the cut depth. This tells the program the greatest depth that will be cut and for this model it needs to be set at -20 (the height of the bracket wall.

The billet size is done automatically but to make sure that the cutter does not go too far to either side of the bottom 2mm is added to each axis. The model size does not need to be changed and the model position should be central to the billet already.

Now the part is set up in the program the tools need to be chosen. There needs to be an end mill for the roughing which is in the program but only had one 20mm one. Slot drills and ball nose are the other main cutters in the program. I have selected a 6mm slot drill for the holes, 20mm end mill for clearance, and a 6mm ball nose for the rounds. Once selected the machining plans must be defined. Typically on the program you use a roughing plan, finishing plan and a fine finish plan. Raster roughing is the first machining method using the 20mm end mill. After that a constant step over works from the middle outwards with a 6mm slot drill, this method cuts the holes into the bracket. The finishing the method is pencil milling with a ball nose that finishes the rounds and smoothes of the edges and finishes holes.

The finished product has bits of the billet still left on the side because the cutter is too big to go round the edge and it would cut into the clamp. This can be fixed by either increasing the billet size or using a smaller cutter. To make sure that the CNC clamp is not damaged I will increase the billet size so that the slot drill which is smaller than the end mill can take away the edges. This means less tool changes.

Another problem is that the bottom hole is deeper than the 3 at the top of the bracket so the cutting depth must be changed to allow it t be cut. If the bottom cannot be clamped down when machining he top of the bracket the part may rise up when It is cnc machine cuts. The cut depth has been change to the bottom of the apart. The cutter may also need t0 be changed if the flute is not long enough. If it is not then there may be a collision. The billet
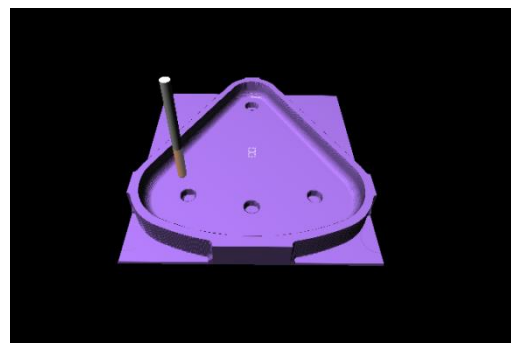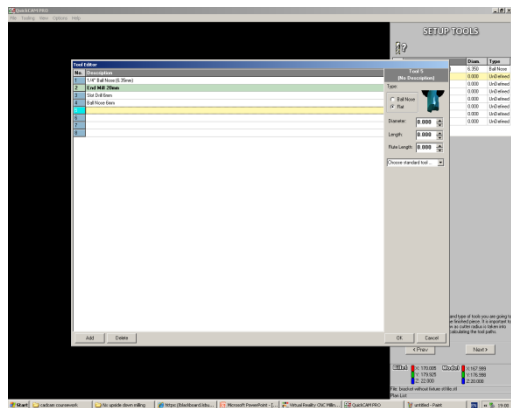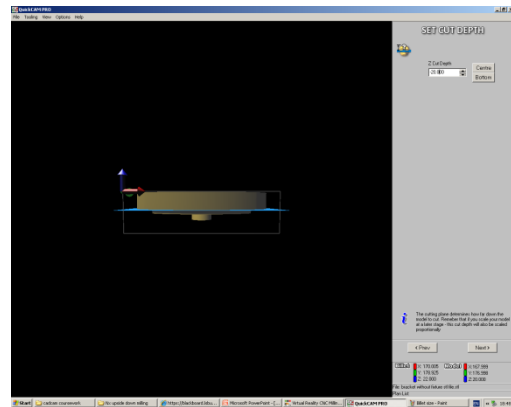
has been increased by 5mm either side so a total of 7mm each side. Still can't cut around the widest edges and bottom so billet size is increased again.

After changing the machining methods I use raster roughing with a20mm end mill, raster + waterline with a 6mm slot drill and pencil pass machining for rounds and edges. These methods gave the best finish but could still not get clear the right hand side from the billet. The hole at the bottom of the bracket is not easy to drill either. The last screen shot was taking before most of the changes took place. Unfortunately I did not take a photo of the final part before closing and the file cannot be re-opened. This problem with the program as you cannot go back and make changes once it has been saved. You must start again.

Top face of bracket

After loading the stl file of the bracket with fixture to the denford program. The cut depth must be set. Z axis is set to -26 so that just the top profile face is showing. The rest has already been cut out in the pervious program. The billet length is increased by 5mm each side t leave room for error in case there is one when using the real machine. I will be using the same drills and cutters as the previous program. Another cutter may need to be added. Raster roughing this time did not give the desired clearance cut and left a lot over. Area clearance which is a Z-level roughing method cleared the face leaving material where the 20mm end mill is too big and not designed for rounds. Raster finishing is almost good enough to leave without a fine finish, but parts do need an extra method for better finish. Pencil pass machining method, I was going to u a smaller 2mm ball nose cutter but found that the boundary needed to be increased so that the billet would be cut fully away from the bracket. This means I can use a 4mm ball nose which cuts more efficiently than the 2mm cutter for this part.
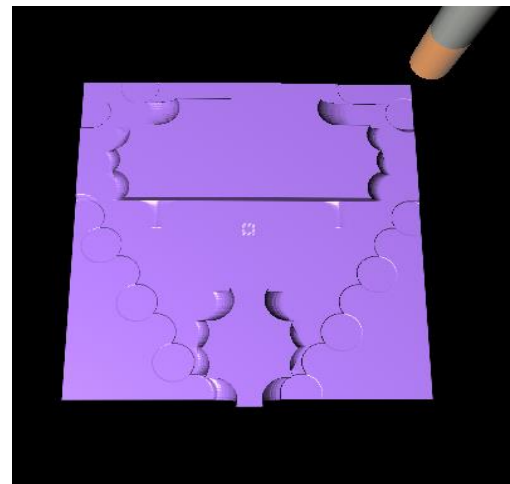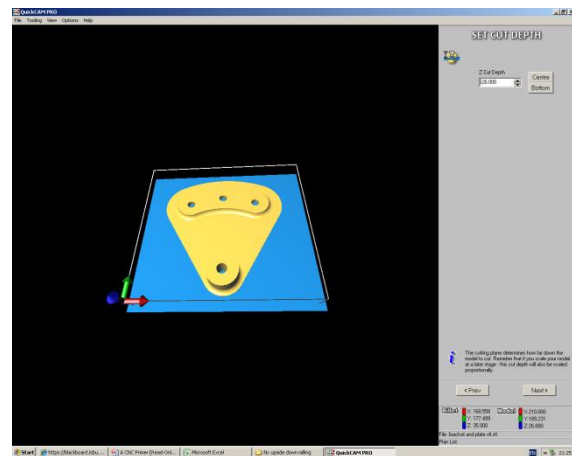
After finding out that the boundary was the issue with the program for the back of the part, that would be one of the only this that needs changing in that program.


3
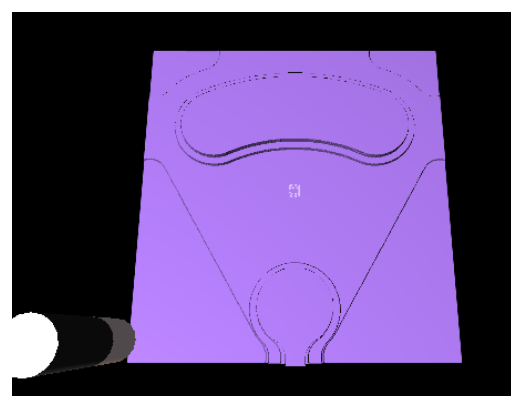
## Computer Aided Manufacture and Robotics

Part B 3)

To start the Nx7.5 CNC simulation for the bracket, the bracket must be turned over and the bottom side of the bracket will be cut out first on the program and on the real CNC machine. This is because the billet can be clamped from either side in the CNC machine, the bottom part can be hollowed and still be held in place by the clamps. The holes can be drilled in this stage then when the piece if flipped over it can be held in place by screws and a fixture. The fixture is held by the clamps.

The first part of this machining operation is to view and set up the die moulding operation and setting up the template selecting the geometry of the workpiece and specifying the material which is aluminium.

Using a cavity mill operation to start cutting away the billet and the back of the bracket. Tool size 20mm end mill using rough milling. This operation at first did not take out the entire billet to give the arcs around the bracket. IT cut down to the rounds but took nothing away from the outside.
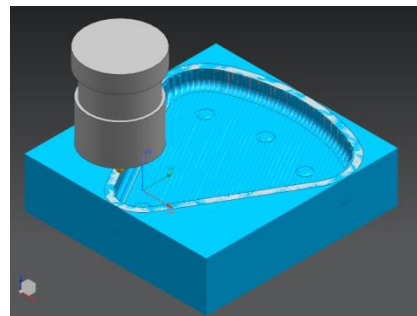
by changing the geometry and cutting area to involve everything apart from the rounds inside. This allows the end mill to cut away the sides and the inside leaving a block of material like a step around where the rounds need to be cut out.
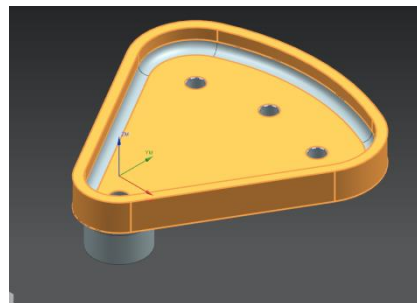

To machine the rounds I started to use streamline area mill to cut out the rounds but this did not leave a good finish. First a 6mill ball mill and then an 8mm ball mill. This still did not cut down to the part. I changed the operation to a Z-level profile which follows the z axis. To save time I've used the same cutter as the pervious operation which is the 20mm end mill tool. The finish was also changed from Mill-semi-finish to mill-finish. Now that outside has been cut away the Z-level was not used for the rounds but to go over the cuts made around the outside of the bracket and on the inside above the rounds. This is an operation that was missed out when first attempted.

Originally I went on to use plunge mill the holes using a crari drill. This too did not give a good enough finish and material had not been fully removed from the base around the holes. So the plunge mill was changed to face-milling-area operation. This operation removes the layer of material from the rim edge and the bottom of the bracket, but still leaves material by the rounds. Again using the 20mm end mill so no tool change is necessary and the cut pattern is follow periphery.
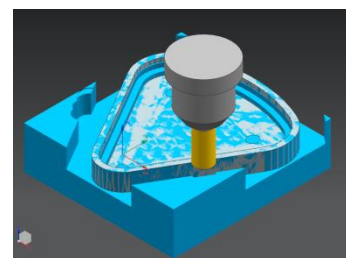
Now the streamline method can be used to cut the rounds using a 6mm ball end cutter and Mill-Finish. Originally this did not give a good finish but after adding steps before the operations it works well. The last thing for the back is to use a Z-profile with standard 6mm carbide drill to drill the holes. This operation makes the cutter drill down into the hole and then moves in circular motion to cut the hole out.

Material is left on the inside wall (above where rounds would be) of the bracket but after analysing the thickness the left over material is only approximately 0.03mm.
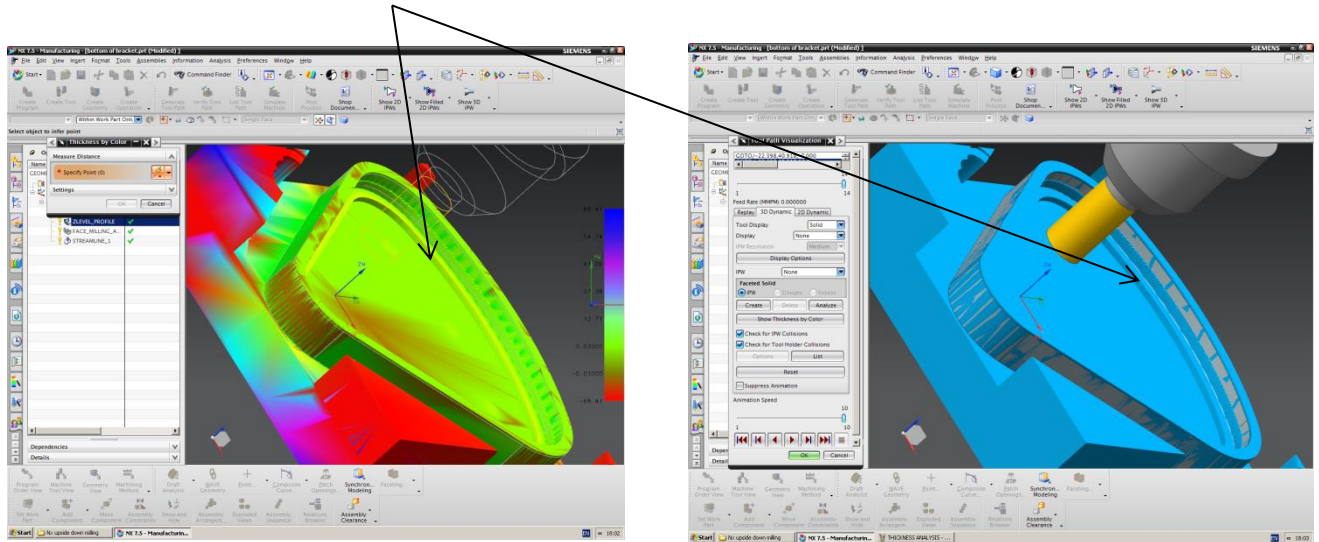

The next step is to do the front part of the bracket. The holes have already been drilled so now a blank plate fixture with holes in the same position of those on the bracket can be screwed in place and the fixture can be held securely in the vice of the CNC machine.

The first thing is to set the billet size which is set on the surface of the bracket because the sides have already been cut out from the previous program. Specify check has been highlighted as the holes in the bracket and the fixture it's screwed onto.

I started with a cavity mill using a 20mm end mill highlighting everything but the rounds as the cutting area. At first was unsure as to why material was not being cut at a fast speed and not that much material had been removed. After examining the operation Rough-semi-Mill was selected instead of Mill-Rough-finish. This removed more material. I then used another cavity mill with semi finish to remove left over material. To cut the rounds contour area was used but problems with the cutter colliding with part where the raised hole is. Switching to a 10mm ball end still collided. End mill did not collide but did not leave a smooth finish at all.

I went back through the program to change and edit the operations as only one of them was working properly. The first cavity mill was changed, cut pattern follow part to follow periphery.
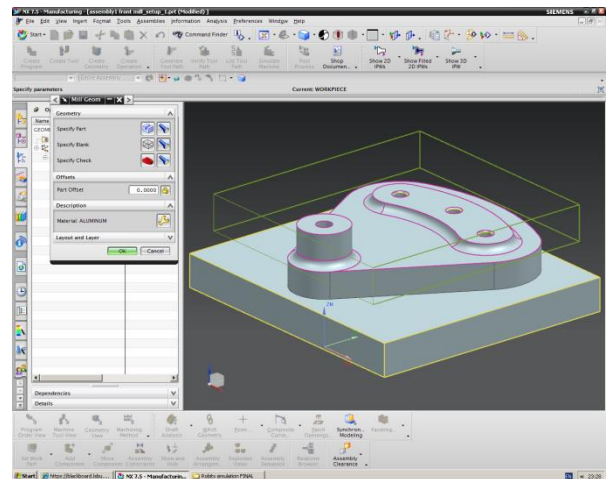
Changed the second cavity mill to face-milling with a 10mm end mill cutter to clear the top areas where the holes are and the top of the bracket but leaving material over the rounds. At first the cutter was too big for the bottom raised hole but after changing the cut pattern to follow periphery it did not collide.

For the rounds contour area was the next operation but this was changed to a streamline operation which provides a better finish with a 6mm ball nose. This operation is just for the rounds around the 3 top holes of the bracket.

Another contour area was originally used for the rounds and sides of the extruded hole at the bottom of the bracket. Not only did the tool collide with the high sides of the extrusion but after changing the previous contour area to streamline I did the same. I also used a Z-level operation just before because the cuts need to be made on the Z axis.
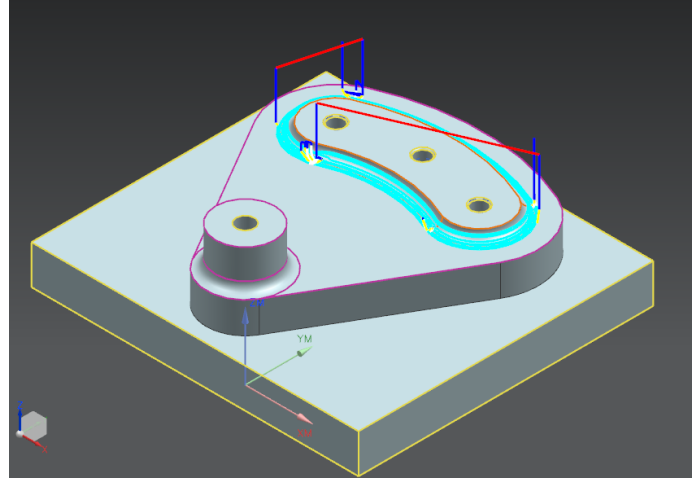
For both operations I used the 6mm ball nose which collided with the extrusion even after changing the tool to an 8mm and then 10mm. I made my own tool because none where big enough in the tool library. The cutter has 8mm diameter with a 25mm flute length. This was big enough not to collide with the hole.

As the last streamline and the z-level need to use the tool with extended flute length I went back into the 3rd operation in the program (streamline) and changed the tool to the new flute length cutter. This means it saves a tool change and it doesn't affect the operation. Now only 2 tool changes are need I the program.

4)

Comparing Nx7.5 to denford quick cam pro, Nx has much wider ranch of options to use with its manufacturing program. Denford is simply quick and easy to use. Where as Nx there is a lot more to do just to set up the work piece. But the more options that the program has the more control you

have over the CNC program. When you are in the manufacturing part of Nx you can click and go back to the cad model to check the dimensions, see what you have previously done and correct any mistakes made with the model. Obviously it is a much bigger program and is also a cad system but these features make it much better to use. Nx has a much larger tool library and one of the biggest differences is that in denford you cannot select certain geometry and features for a specific
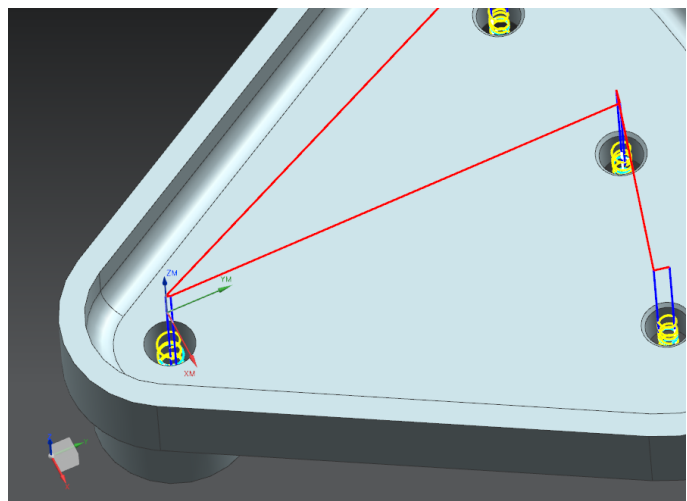
operations. When doing the top face of the bracket in denford, the wholes are drilled again, where as in Nx you can simply specify them not to get cut. The holes are a good example of this. Using denford I was not able to specify the holes did not need to be cut when working on the top face of the bracket.

The visual aids on nx are also better. For instance the thickness by colour option with gives you an indication of the thickness of the material by colouring it in and giving a colour scaled chart.

In denford there is 12 choices of operation, in nx there is 11. But these are called operation subtypes. There are then drive methods which give you further control over what the machine does. For simple parts and quick prototypes denford is perfect. But if you are looking for highly accurate more controlled cncing then Nx is the best program.