3D Machining Considerations

Phil Dixon – CAD/CAM
Technical Instructor

Objectives

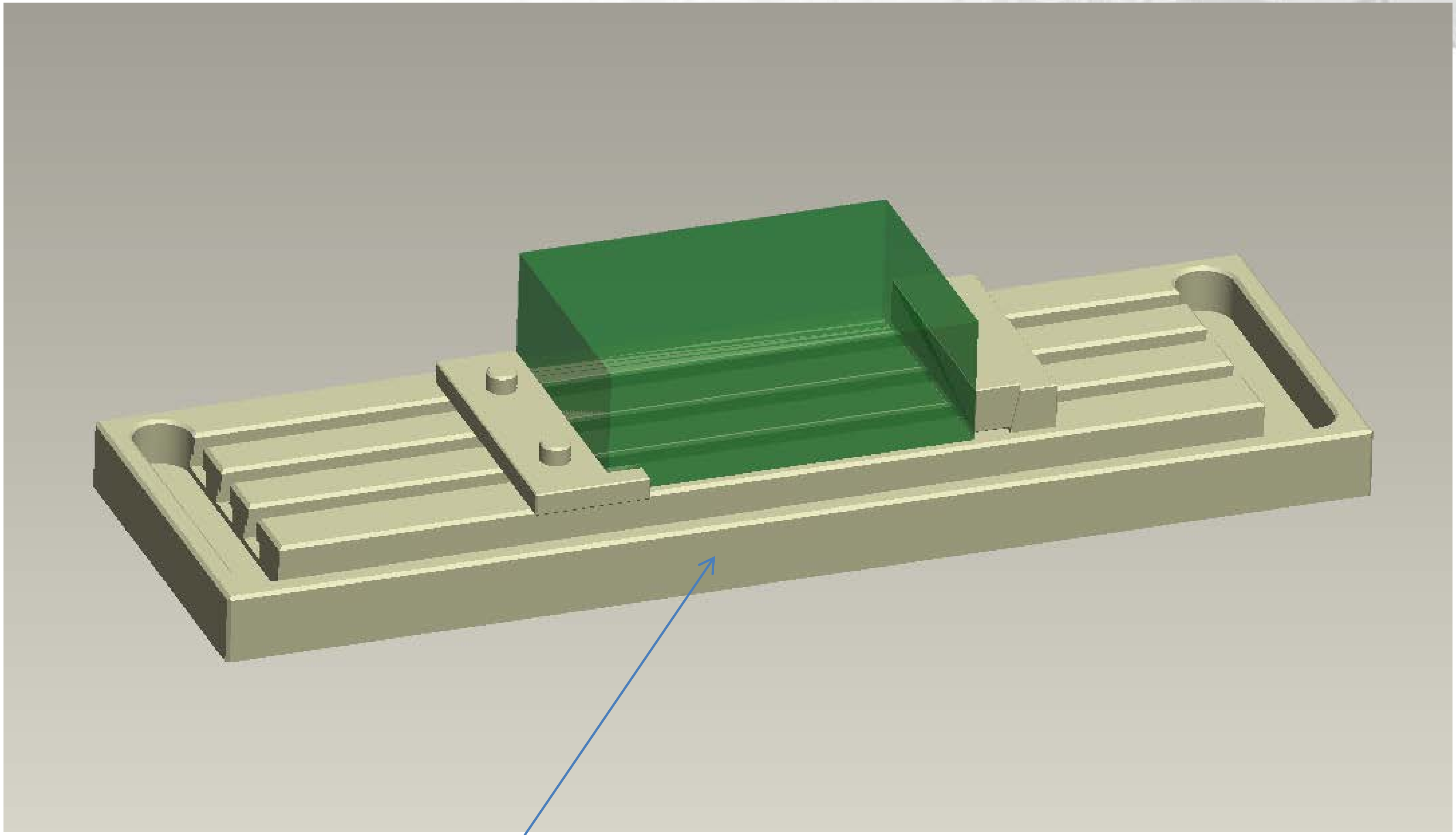
To enable the user to create and machine a 3D machined component from a part programmed file.

Examine the best strategy for machining.

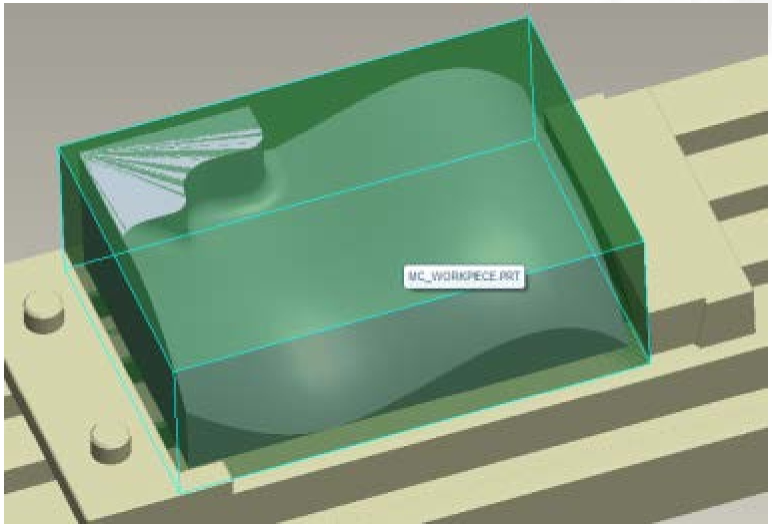
Look at the pitfalls usually encountered

1: Part Alignment and Orientation

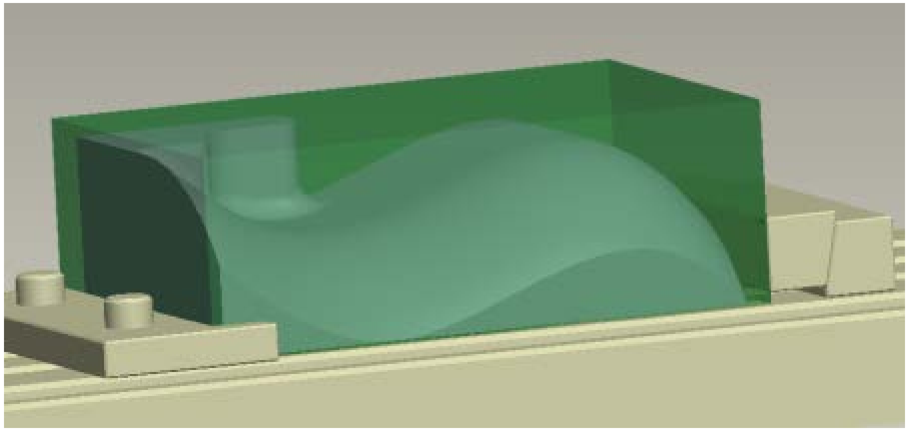
- The green workpiece should be representative of the material you are starting with. Our sample part measures 150mmx100mmx50mm when finished. We want all sides to be machined so our material is 160mmx110mmx55mm.



This is the front of the machine as you look at it. VMC table shown but you can also use it for the Boxford and Unimatic routers too. Ignore the clamps, we won't be using them.



This part is aligned to the top of the workpiece, this means the top of the block will not be machined and our block will remain at 55 thick. Our part is placed in from the edges by 5mm to allow machining all round.



This part is aligned to the bottom of the block, we will need to remove material from the top using a volume rough and our block will finish at 50mm, as before, the part is inset by 5mm in X&Y to allow machining all round.

Starting to machine out part.

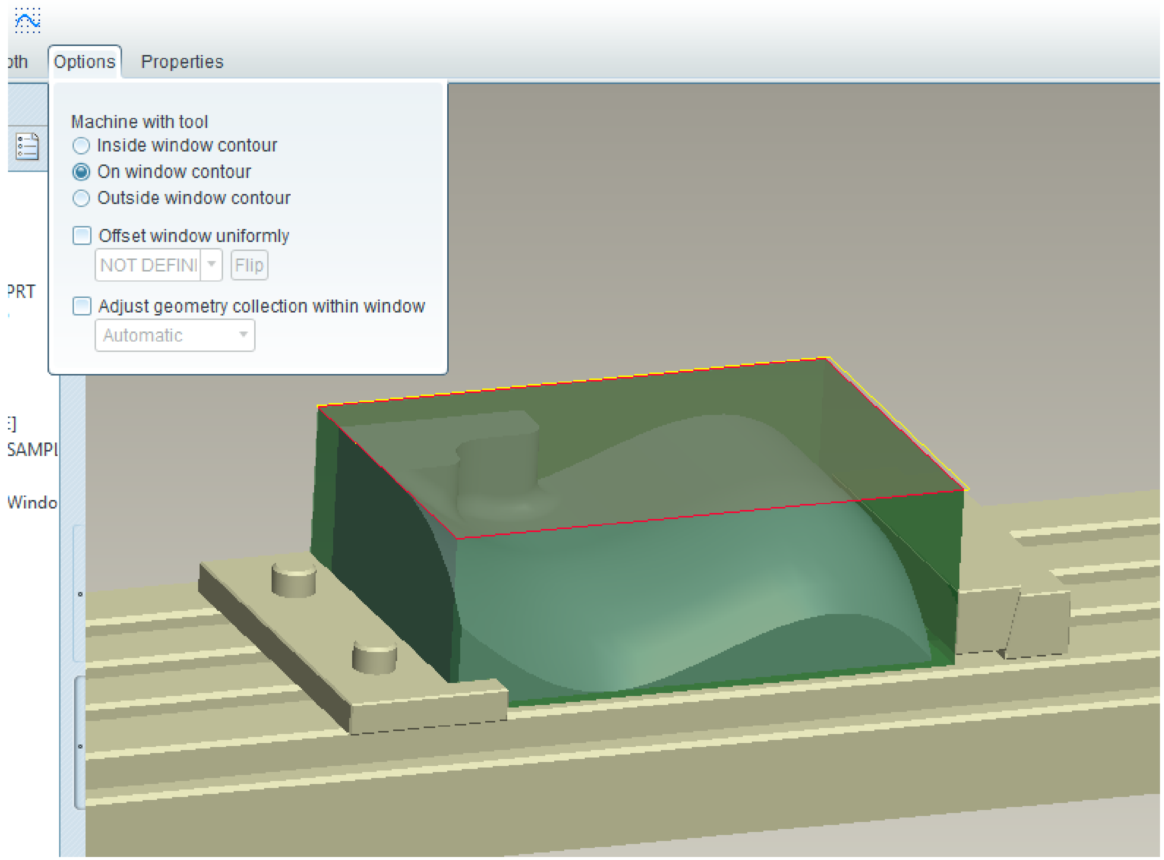
- Before we start we need to consider what cutters are available and their suitability.
- 1) Large areas should be machined with a large cutter
- 2) Areas with internal radii should be roughed with approximately the same diameter as the finishing tool.
- 3) Deep straight sides that are longer than the cutting flute of the cutter require machining with a cutter larger than the collet nut on the tool holder.

Machining processes

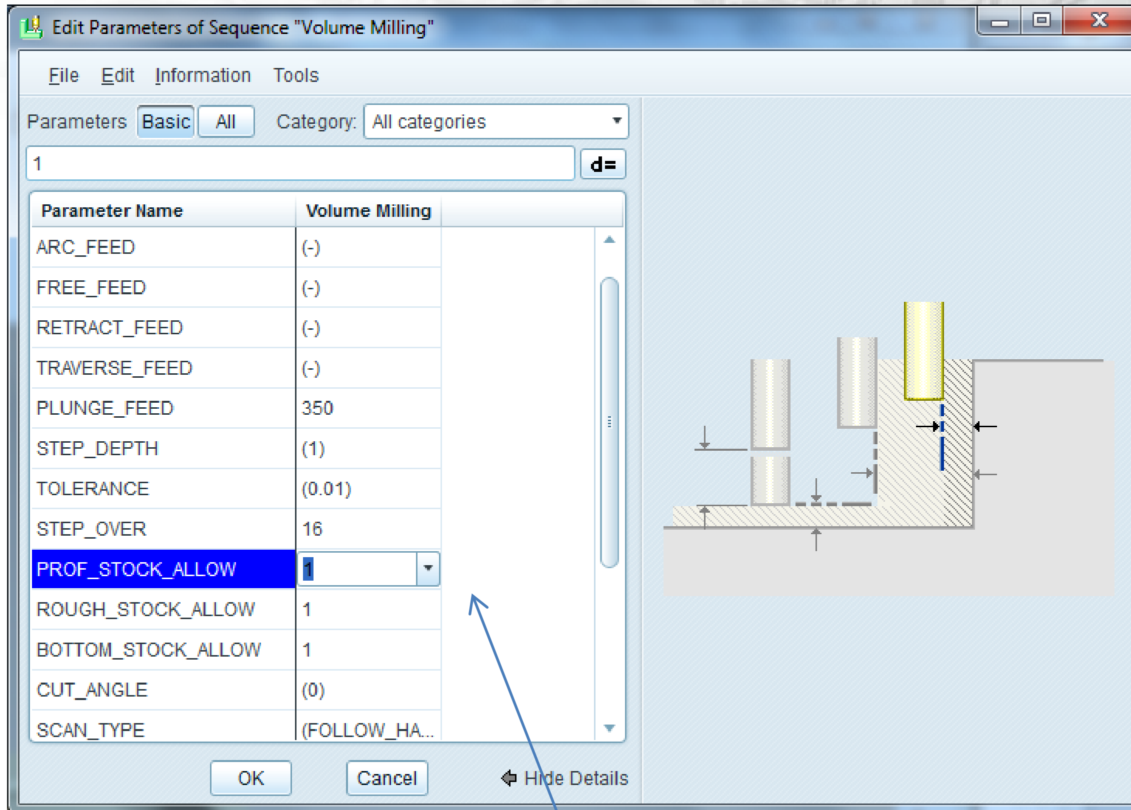
- The three main processes will be using are:
- Volume Rough – to remove material to the rough shape of the part
- Surface Milling – to machine the profile
- Trajectory Milling – to finish the straight sides.
- **There is no drilling cycle on the routers**

Volume rough by window

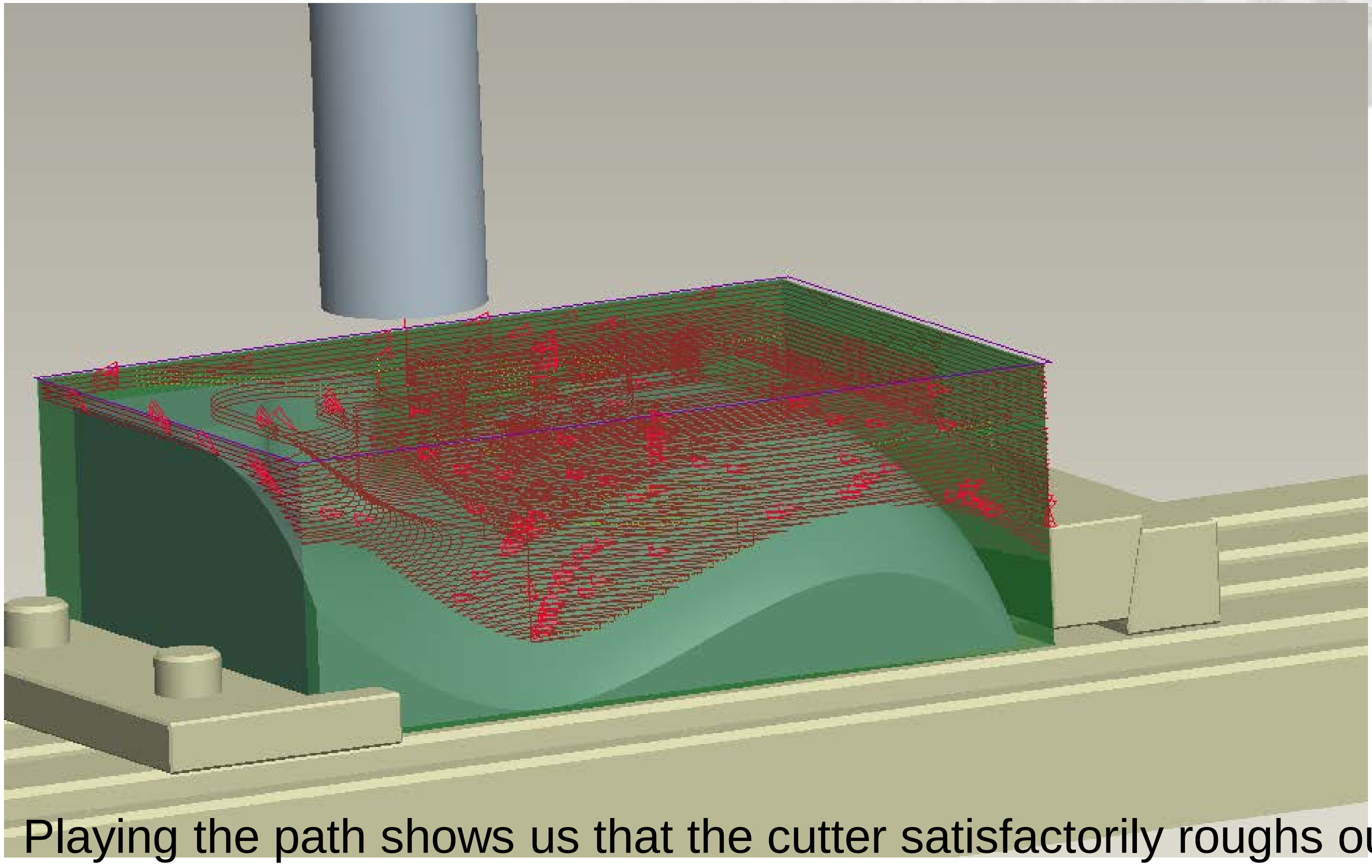
Define a window that covers all of the workpiece. Make sure you define it from the top of the workpiece NOT THE TOP OF THE PART! If you define from the top of the part the cutter will smash into the top of the block. To make sure we machine the entire block we tell Creo to machine out to the window contour. I.E. it will machine past the window to the centre line of the cutter.



- Next we define our roughing cycle, we need to leave material for finishing so we add 1mm to our stock allow parameters, this will machine the block and leave material for our finishing passes.



Add 1mm to these 3 boxes. If you are working on the routers, there is no need to program the spindle speed, it’s done manually.



Playing the path shows us that the cutter satisfactorily roughs out our part. Note that the outside has not been machined in this instance and there is 1mm left on the top surface that needs to be removed. The display shows us the centre line movement of the cutter only with rapid moves being shown as a yellow dotted line. We can see that the cutter spends all of it's time machining and very little time making rapid moves, this shows that an efficient cutting process has been defined.

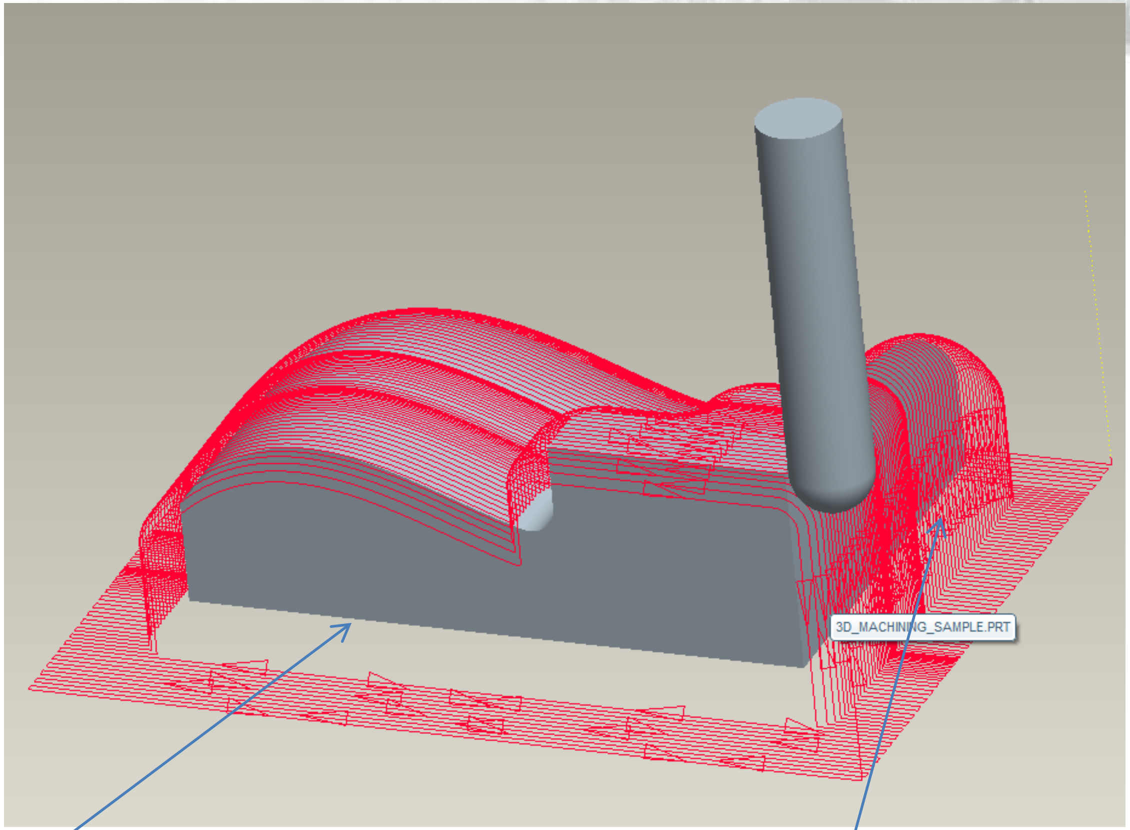
Finishing our part.

- Machining considerations:
- **Straight sides should not be machined using a surface milling cycle.** Trajectory milling is a more efficient strategy.
- Ball nose cutters will plunge past the bottom of the part, you need to

  a) allow for this in your fixturing (so don't use 10mm thick base board for a Ø20mm cutter, it will mill it all away!

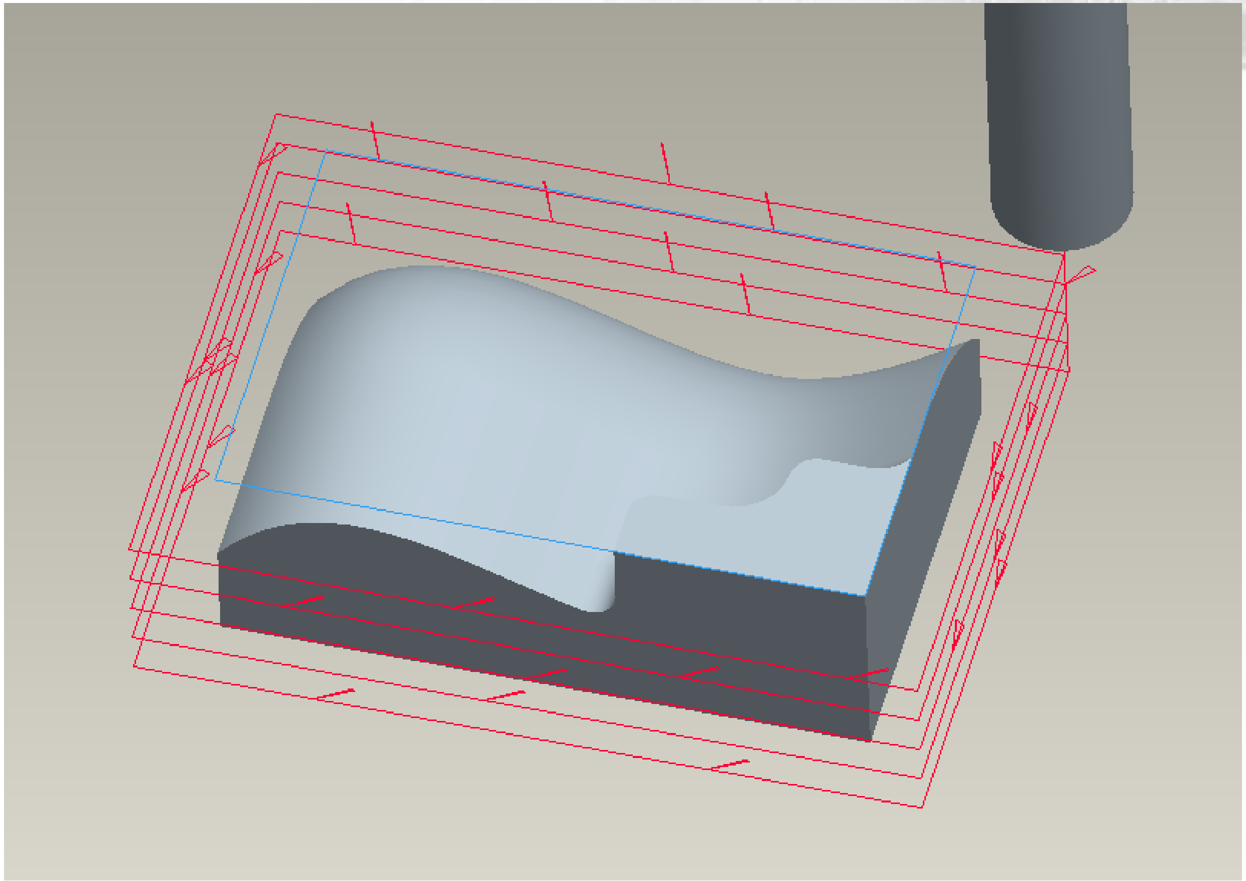
  b) allow for additional length when choosing a cutter (so if a Ø20mm cutter will go 100mm deep it will actually plunge to 110mm so 90mm deep max is achievable.

Trying to machine parallel straight sides using a profile mill will force the cutter to contact the part along the entire length of the cutter. If the diameter is the same or smaller than the cutting diameter you will either cause the part to move, snap the cutter or leave gouges in the finished part



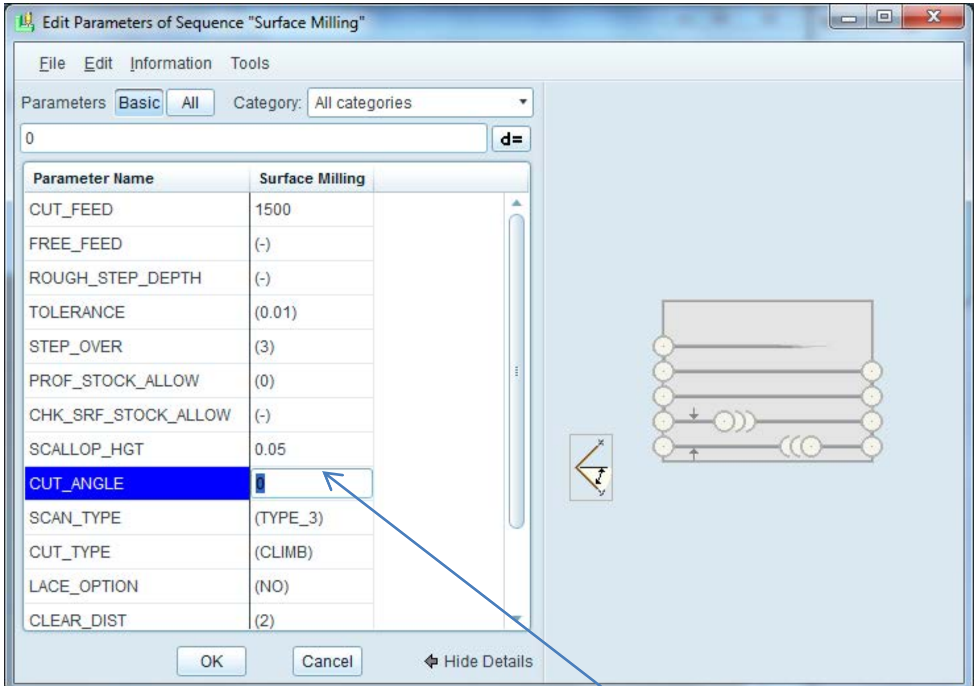
This area has not machined

Cutter has impacted the part along the length of the tool



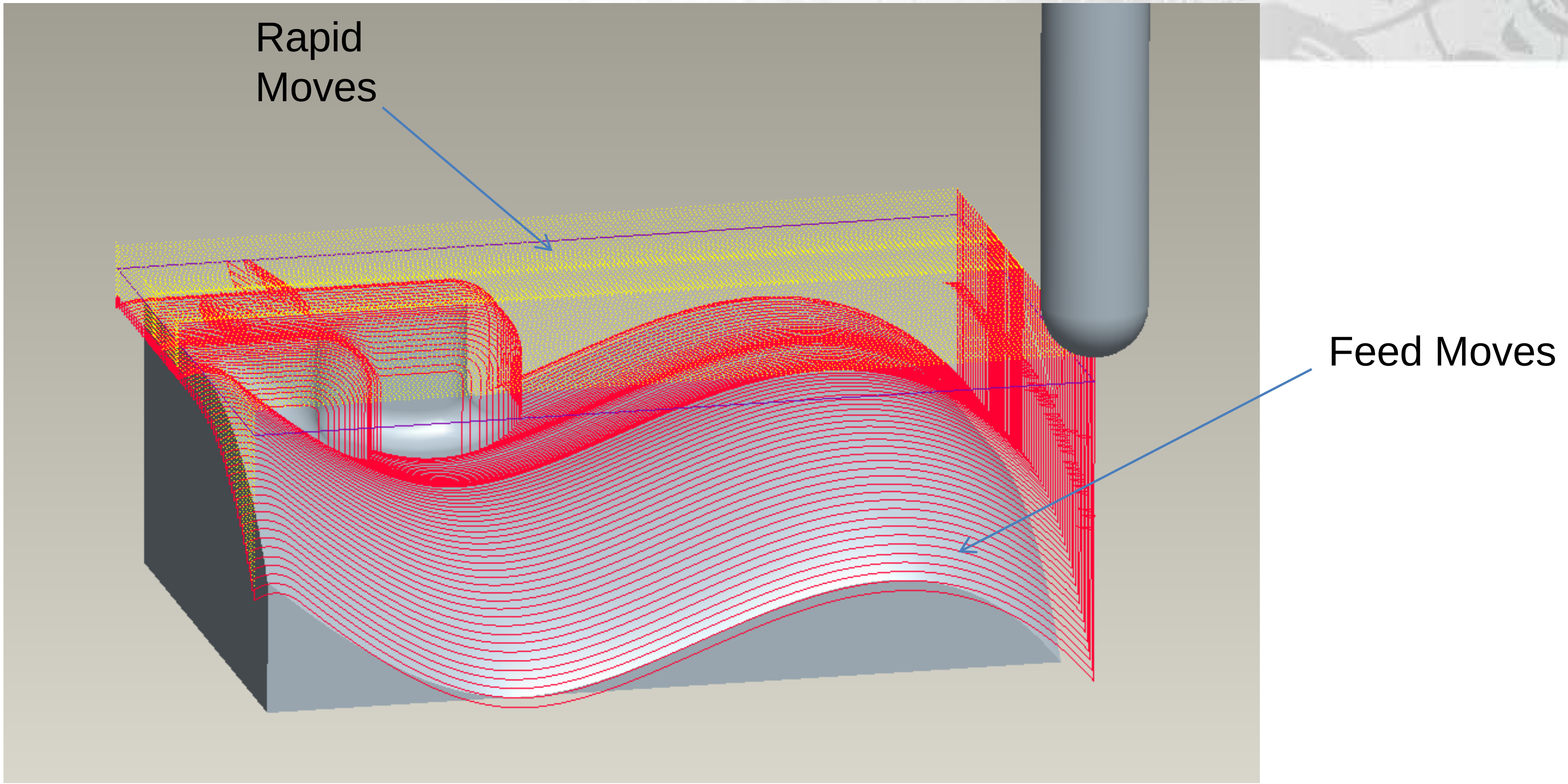
Using a trajectory mill cycle allows us to machine the straight sides, it's also faster. Don't forget to set the stock allow back to 0 in the parameters!
It makes sense to use the same tool that we used for roughing to save on tool changes, you could also finish the top flat surface but we're going to profile mill that at the same time as the contour to ensure dimensional accuracy.

Profile milling the contour



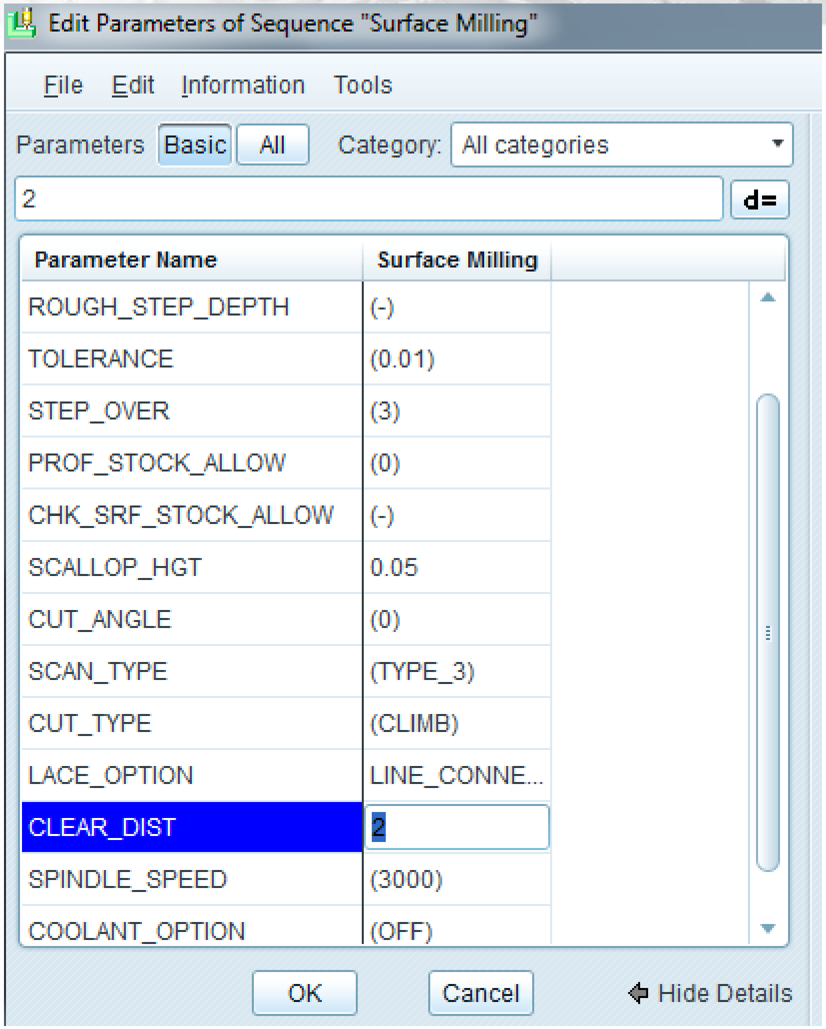
By default you would usually use a ball nose cutter for profiling. If you have a shape where you've machined away a fair proportion on the outside shape it may be beneficial to define a window that more closely follows the finished shape, you can save a lot of time this way.

We can either define the step over (which we would do if we were roughing out the contour) or define how rough we want the surface. We define this in scallop height, accept the default of 0.05 but make sure it's not in brackets.

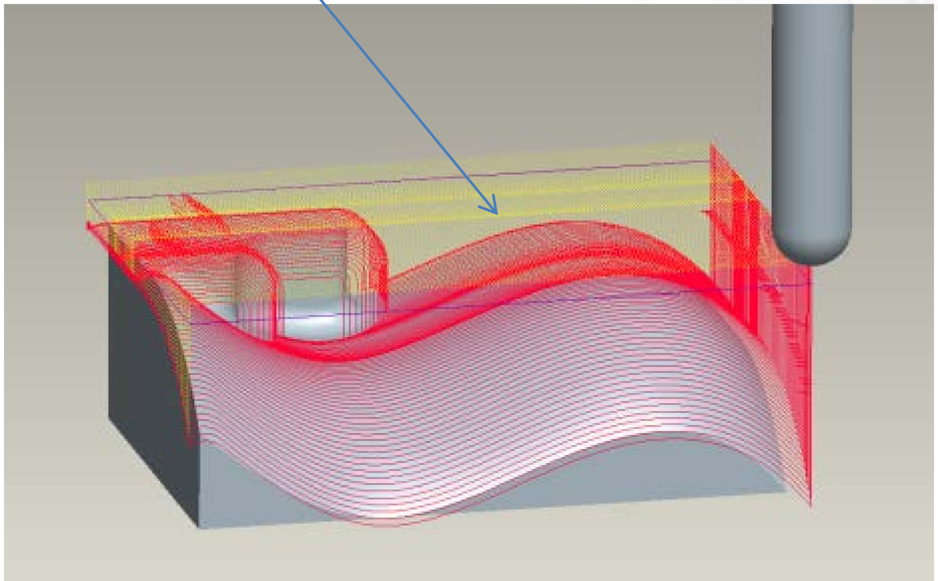


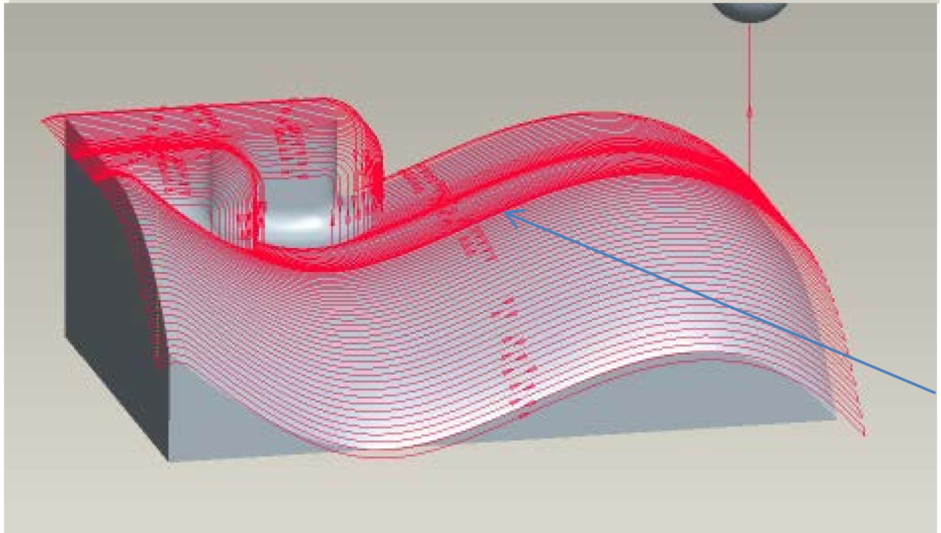
Playing the path shows us the machining cycle. You can see that there is a lot of rapid moves, almost as many as feed moves (in red). This is not an efficient machining cycle.

We can reduce the rapid moves by defining how the cutter moves from the end of one machining pass to the start of the next. At the moment lace is set to NO, we select line connect and it will cut in a reciprocal motion. Think of it as defining how you lace your shoes.



No Lace Option



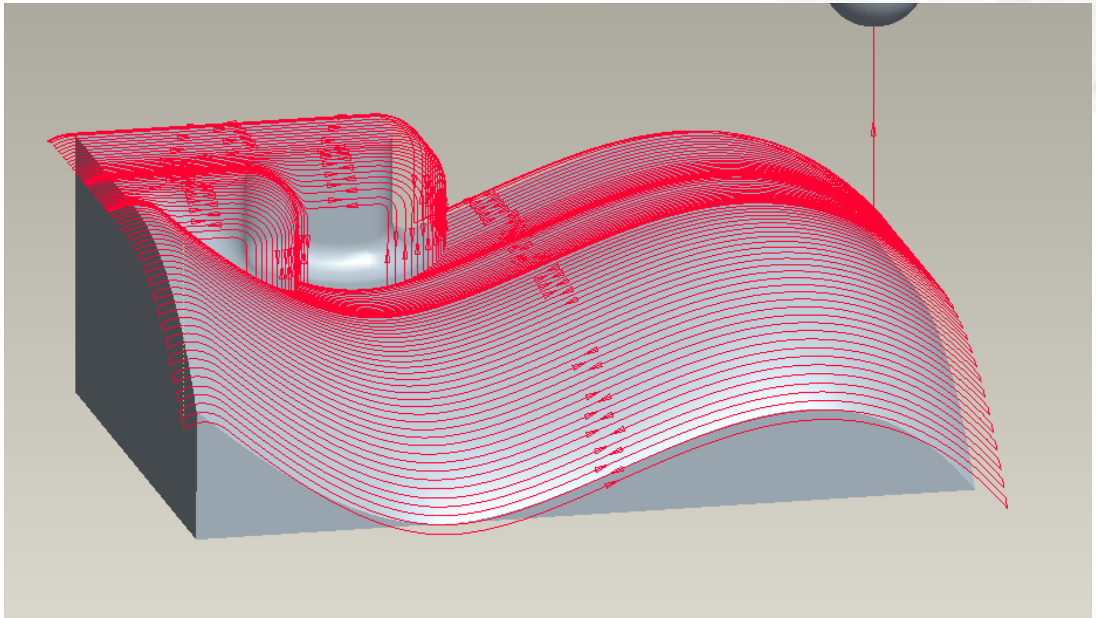


Lace option: Line Connect.

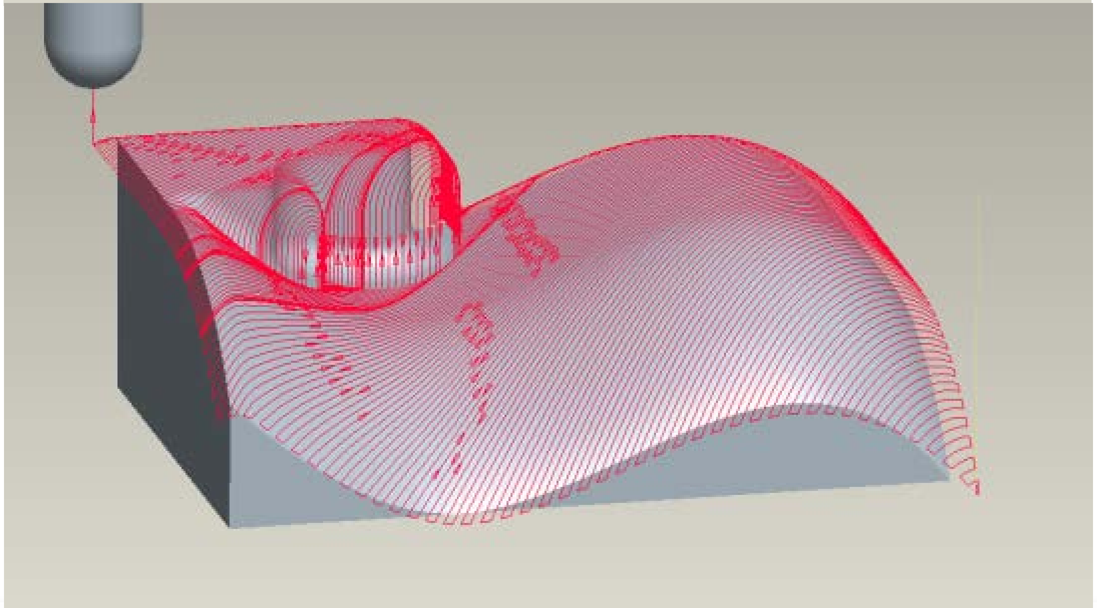
If we compare this to the previous cycle you can see there are now virtually no rapid moves. Much more efficient!

DO NOT CUT IN ONE AXIS ONLY!

Defining our cut to only machine in one axis causes one motor to do all of the work. We can “spread the load” by defining the cutting angle to 45 degrees. This is kinder to the machine and the reduced speed the motor runs at allows more time for the motor to properly position, this gives a better contour.



Defined cut feed: 1000mm/Min. X motor has to run at 1000mm/min, Y motor does nothing.



Defined cut feed: 1000mm/Min.
X motor runs at 707 mm/min
Y motor runs at 707 mm/min
Load is spread across both motors.
This is more important the higher we define the feedrate.

To recap:

We can machine a better part by taking into consideration:

How we align our part in the part program

Using the right cycle for the right application

Examining the played path to determine if the cycle is efficient or not.