# QuickCAM 2D Design

Covers:

> Defining Billet
> Setting up Grid
> Creating rectangles and circles
> Creating single thickness text
> Creating machining plans: Follow, Offset inside and Offset outside
> Checking and output of CNC program

This tutorial follows the process of designing a simple key-fob design in QuickCAM 2D Design. It will also show how to create toolpaths and a CNC program for manufacturing the part.
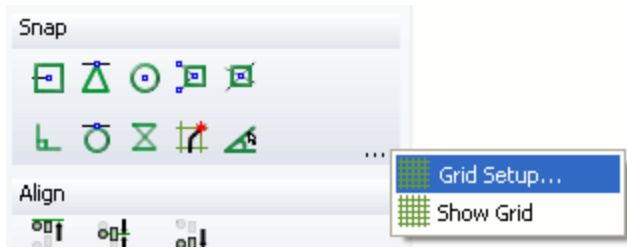
Define working area (billet)
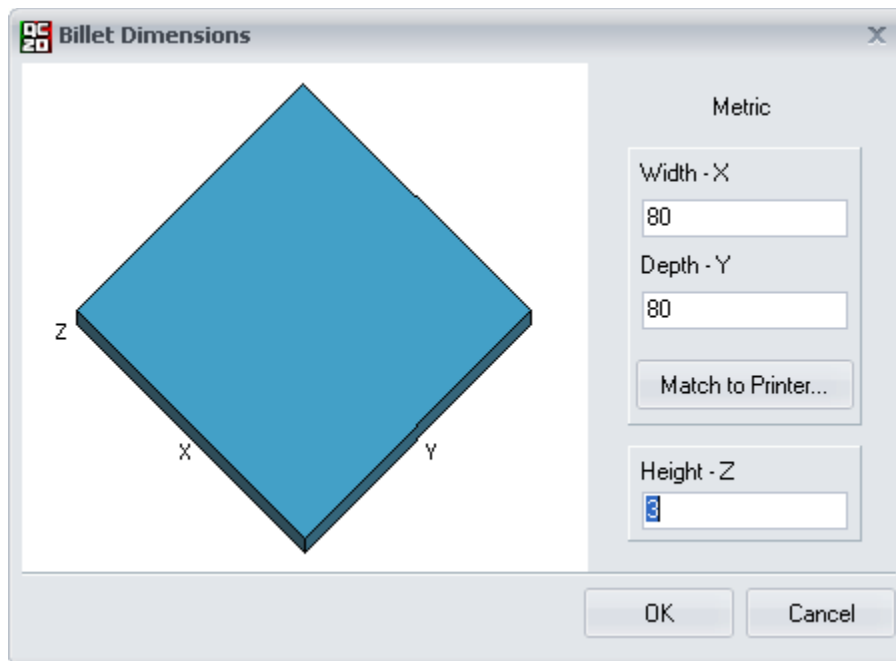
Define the size of the billet needed to create the component (X 80mm by Y80mm).

This can be done as the software starts, or by clicking the resize material icon:

Set Grid Size to 10mm:

Select grid options from the options menu on the the Snap toolbar, or by right mouse

clicking the Snap toolbar:

Now enter the **grid size** as 10mm
Set the step size (cursor nudge) value to 1 at the same time
Once the grid is setup, you can turn the snap to grid on and off quickly by pressing the
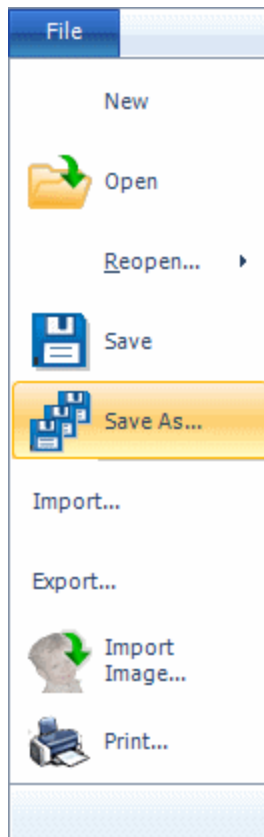

toolbar button:

With grid snap on, all shapes created will snap to the nearest multiple of the grid (10) ,
eg X10 Y10 or X40 Y20

Name the design

It is a good idea to save the design before starting so that the CAM output knows what
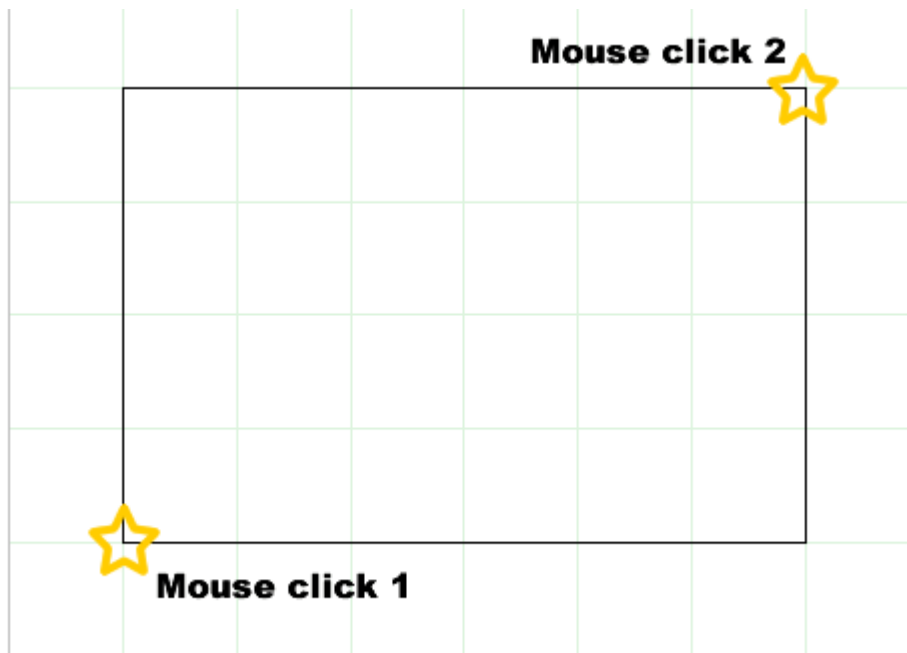to call the CNC file, and you can keep saving throughout the design process.

From the top File menu, select Save As... Choose where to save your file, and enter a suitable name eg, "KeyFob.MCM"

Draw the Key Fob

The key fob design will be made up of three basic shapes: Rounded rectangle, Circle and Text:
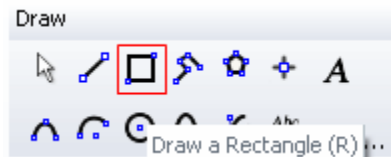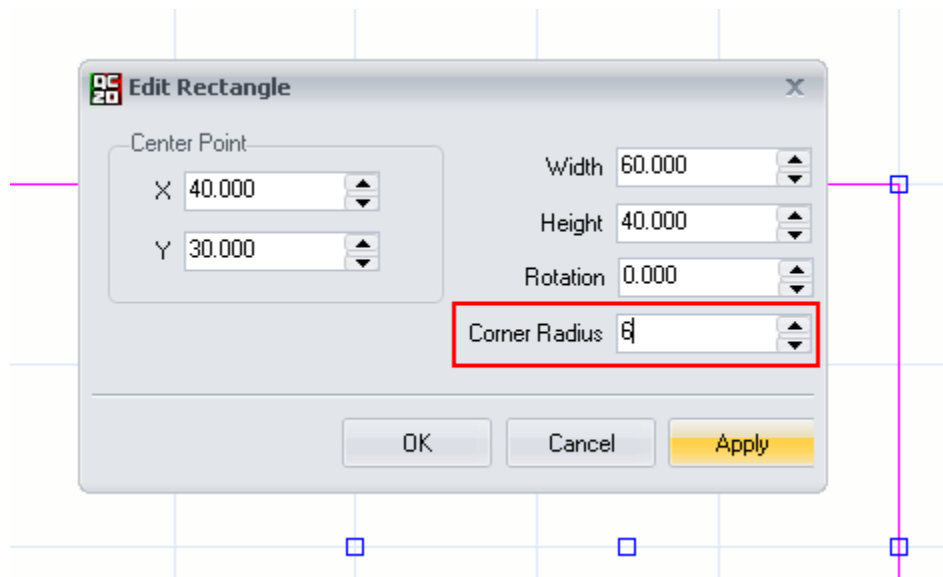

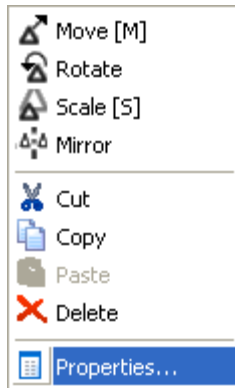Select rectangle tool from the toolbar, or press **R** on the keyboard

Click the first (bottom left) corner of the rectangle this will be X10 Y10, then the next top right which will be X70 Y50.
Notice that we have created a rectangle 60 wide and 40 high, starting at 10,10 - **count the number of grid lines.**

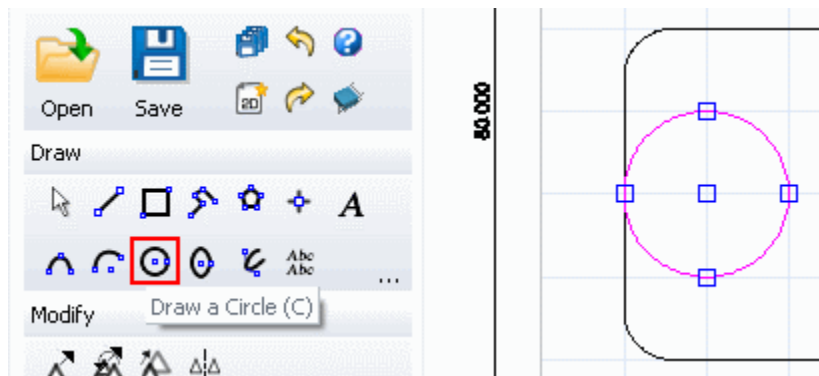Now we want to turn the rectangle into a rounded rectangle

To edit the properties of any shape, you can double-click on the shape,
or, select the shape (high-lighted in magenta) and right mouse click to show the popup

menu:

Alter the corner radius property to 6mm and press OK

Select the circle tool from the toolbar, or press **C** on the keyboard.

First mouse click where the centre should be, then click the nearest grid point for any point on the circumference of the circle.

Edit the properties of the circle (double click) and change the diameter to 7mm - press OK
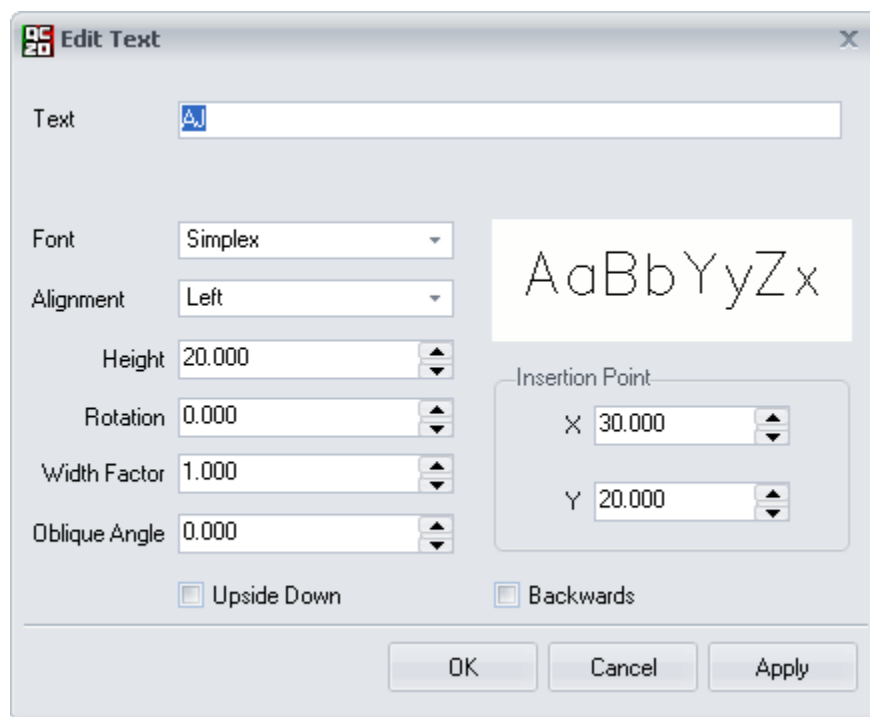
Now to add some text. QuickCAM Design is able to import any TrueType fonts, but the standard font set also includes single thickness fonts which are ideal for following with a small cutter.
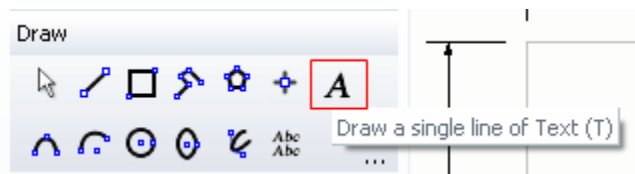
Select the text tool.
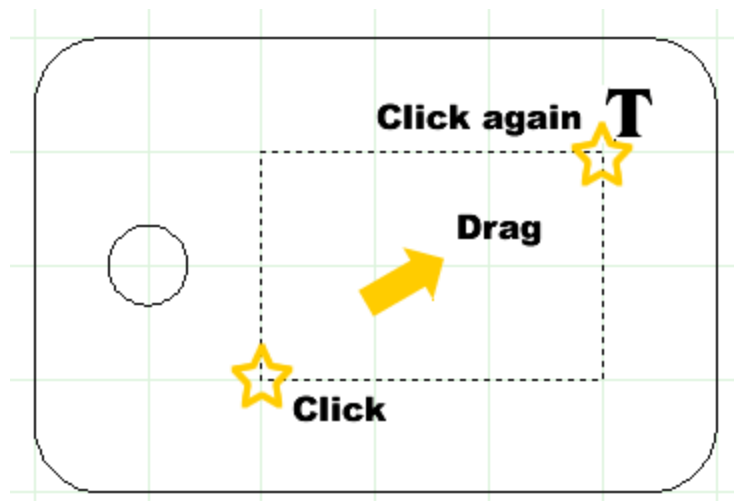
To add text you must define the text start point and height by drawing a text box (note that the width of box does not constrain the text at all)

Now, enter the text you require (AJ in this case)
Choose the single thickness font (Simplex)
Click the Apply button to preview how the text will look
Click OK when happy

Save the design now by clicking the quick save toolbar button:

Create the toolpaths
Now the design is complete, we need to run the CAM wizard in order to create a CNC

g-code program.
For this example we have setup a 3.175mm slot mill and material settings (feed / speed) for Acrylic.
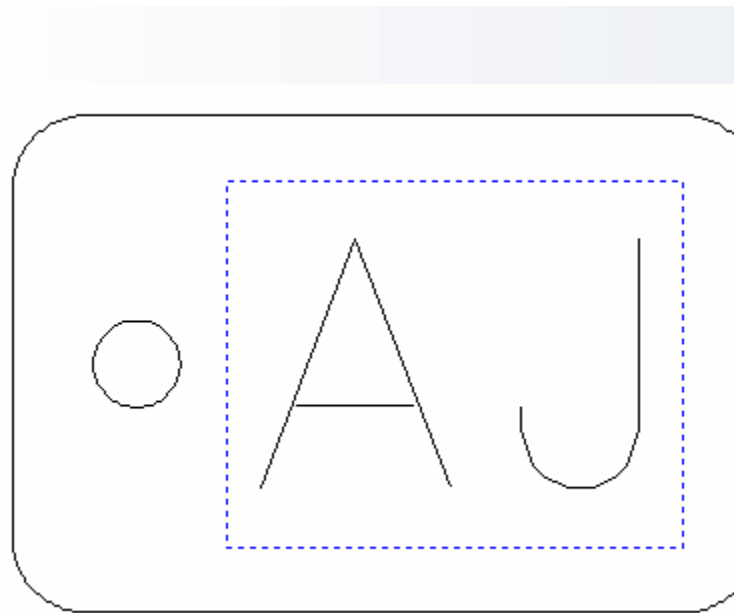Setting up tools and materials is covered in another tutorial.

Click the toolbar button:

Choose the correct material as this will affect the speed, feed and tool step-down depth in the final CNC program
Click Next >

Click the **Follow** button from the list of available machining plans
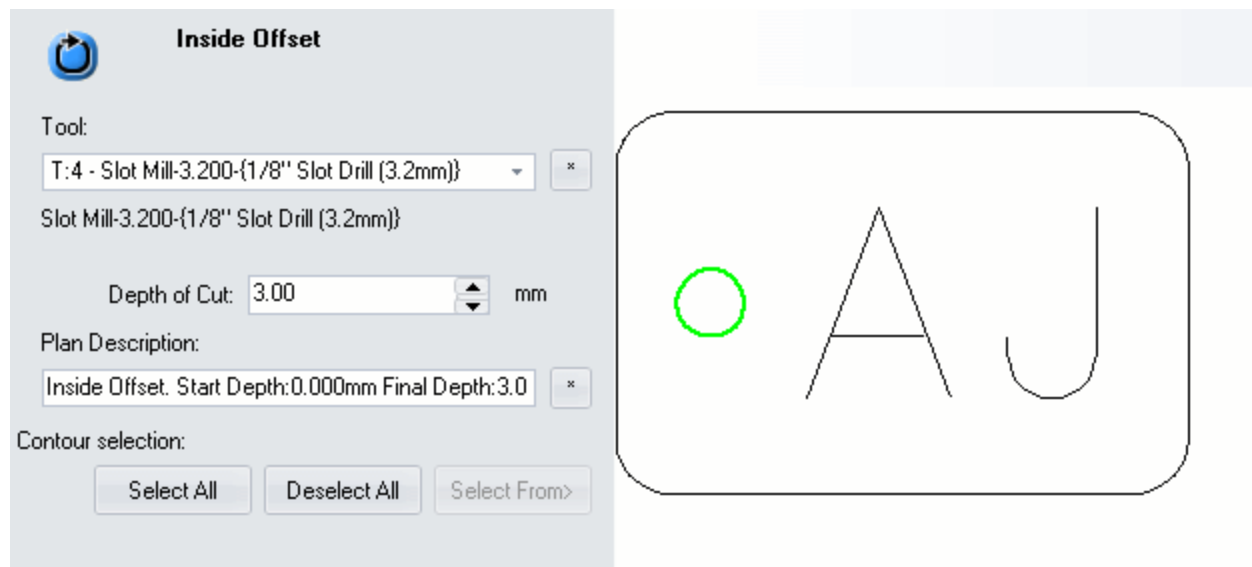

Set the depth of cut for this follow plan to 0.5mm
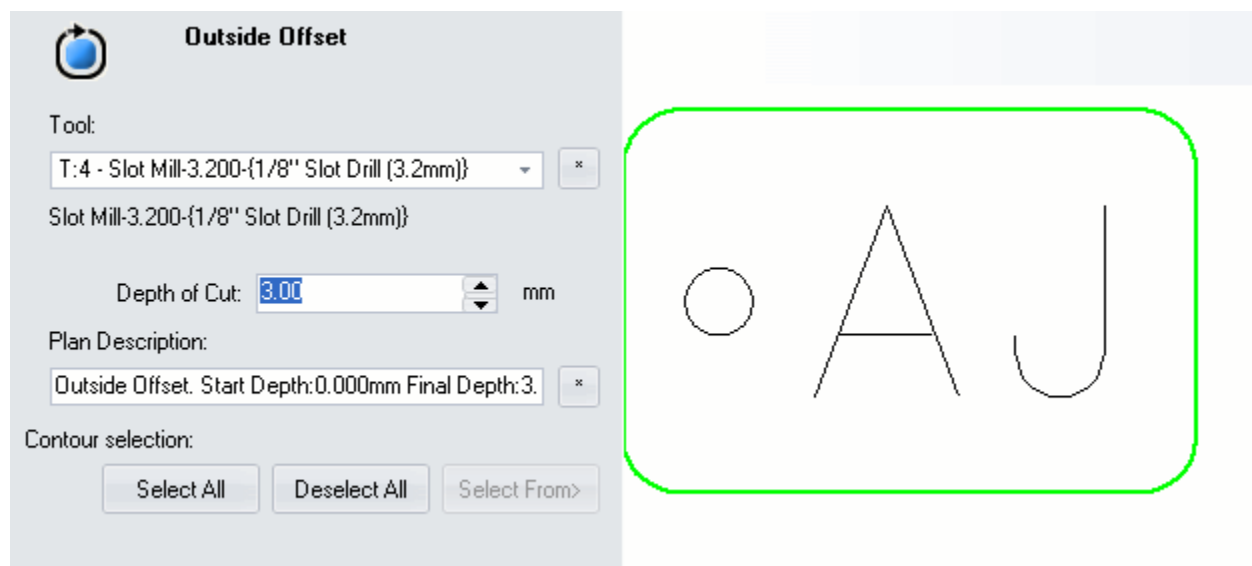Click and drag to select the text shapes that we want to follow
Click OK

Now select **Inside Offset** plan

Set the cut depth to 3mm - this is the thickness of the Acrylic we are using.
Select the circle for the Inside offset cut by clicking on it - it will turn green when selected.
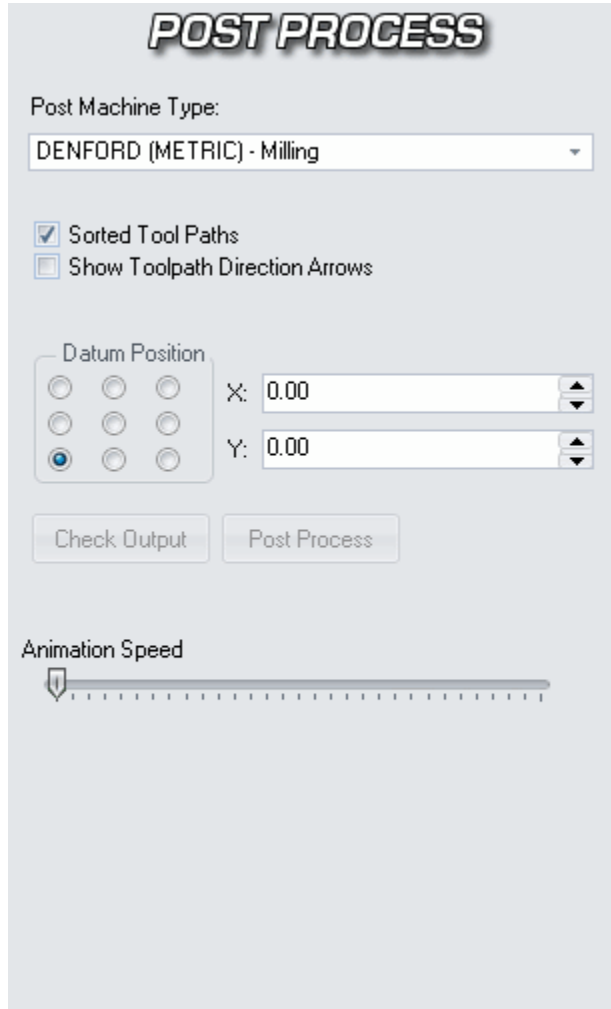Press OK

Finally, click the **Outside Offset** plan button

The cut depth from the last plan is remembered so will not need changing.
Select the rounded rectangle shape to create an outside offset path from.
Click OK

The three plans are now in the correct order for machining ie, the final cutout of the outline is last. However, it is possible, from the machining plan screen, to alter the order in which the plans are run when manufacturing on the CNC.
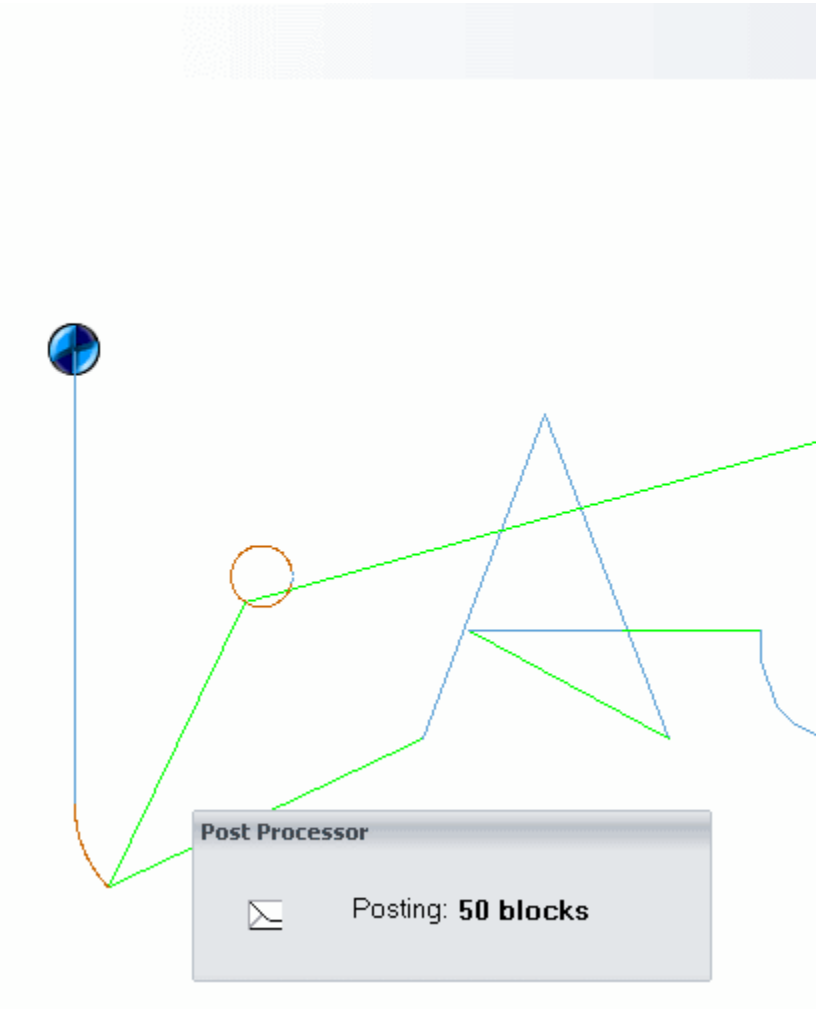
Click Next >

From the Post screen,
Select the Post machine type from the drop down list Denford (Metric). This will determine how the CNC program is constructed.
By default the destination for the CNC program is **\My Documents\ Denford\CNC Files** if you want to save it elsewhere then click the browse [...] button and choose your own filename and location.
Either click the Post Output button to create the file, or the Check Output which will animate the cutter path whilst posting.
Press OK to return to the design screen.

Now you can use the CNC program to manufacture your key fob:

Hopefully you can make a more interesting design !