Low Merit

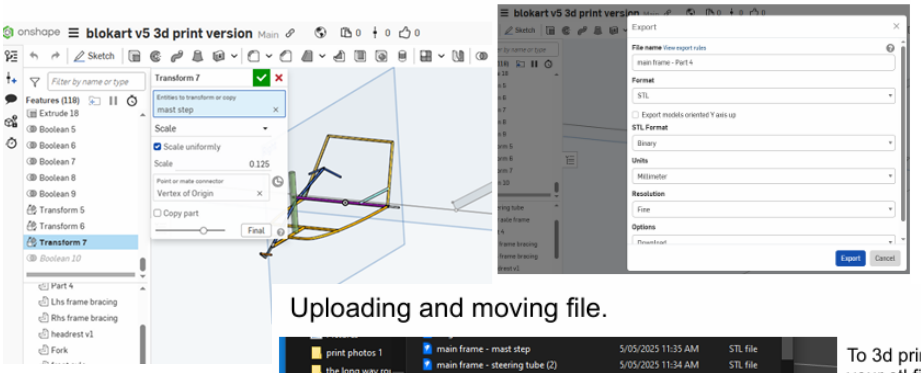
NZQA Intended for teacher use only

CNC

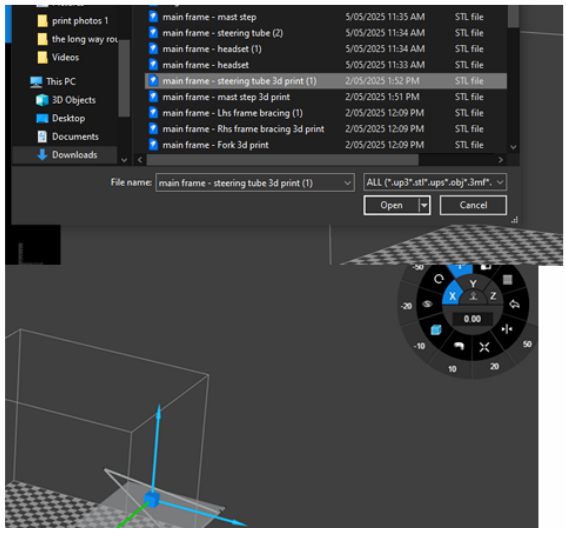
Making a graphical representation using the limits of the cnc machine.

To create a graphical representation suitable for the 3d printer I first took my onshape workspace (with the 3d cad model) and copied this. I scaled this version down using the transform tool so the largest part was under 200mm by 200mm by 80mm (the bed size of the big 3d printers at school). In the end I made it a bit smaller than this to save on time and printer filament. This is important as it minimises the scale of errors if they do happen ensuring efficient use of material. Next I needed to export each part as an stl file which is what the 3d printer requires. To do this you can simply press export part and then select stl file on onshape.

1



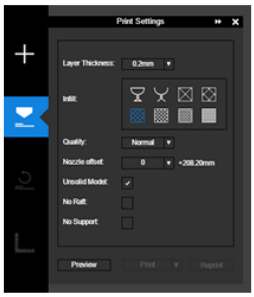
Uploading and moving file.



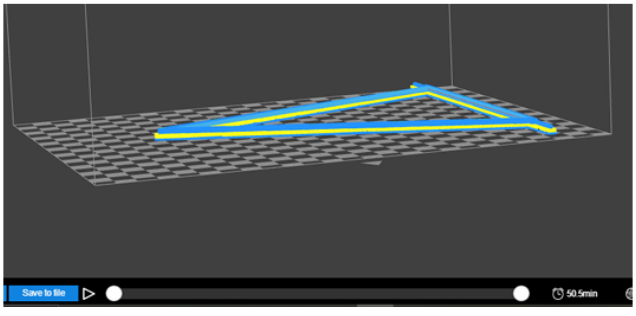
To 3d print something you first need to upload your stl file to the slicing software accepted by your 3d printer, in this case UP studio. Once your part is uploaded then most of the time it needs moving and rotating so it prints the fastest, and uses less filament as it requires less support. This is important as it helps keep printing within the timeframe not requiring teachers to be present at odd hours so the 3d printer can keep printing. This also minimises waste material .

2

Preview of print and settings



Once you are happy with the position of the print you can press print. This pops up a window showing the print settings which for my purposes default was fine. However if you were making a mold or something requiring extra strength this is where you can change infill which determines how solid the model is. You can also change the quality, layer thickness, infill, infill type, the amount of support and nozzle offset. Here is what each of those settings do: ***Quality :*** changes layer thickness and width which affects the smoothness and level of detail also changes speed of printing time, ***layer thickness :*** this changes the quality (detail, smoothness) of the print far more precisely than the quality setting as it allows your to change the layer height values rather than just have 4 setting of quality when changing the quality setting. ***Infill :*** This is how solid the model is. For example if you are just making a prototype, the default setting is ok as it doesn't have to be especially strong and you can economise of time and filament. If you are making gears for example that are to be used in a product then having more infill would be beneficial. ***Infill type :*** This controls the pattern of the infill which affects the density. Similar to the percent of infill this controls how strong the print will be and should be used in conjunction with it. Making sure you get the settings right is important to ensure the outcome/print is of a high quality standard.



Once the settings and position of your model are satisfactory, you need to initialise the printer. This means the printer ensures it knows it's coordinates and is ready to print. Following this you preview your print which shows you the tool paths and where there will be fill. It also shows you how long the print will take and how much filament, which is helpful to ensure not using excessive resources. If you aren't happy with the preview, for example it will take too long, then go back and check the position and the settings.

3

Printing procedure and troubleshooting.

Once the print settings and preview are as desired the printer is ready to print. Before pressing print make sure there is enough filament (abs in this case) to avoid only getting halfway through a print. Once you have pressed print, the printer bed will take a while to heat up so the print doesn't warp off of it. This is indicated by the white temperature icon. The nozzle also has to heat up which is indicated by the red temperature icon. After this it will start printing and doesn't require constant attention and you can unplug your computer from the 3d printer. However it is good to check on your print every so often incase it has warped off the bed or become unstuck so you avoid wasting time and filament. Some of the causes of this is the bed not being fitted correctly or not being clean. These are things you should check if your print has failed before trying again after considering your settings again. Once your print has finished you can take the bed out of the printer and scrape/lever your print off the bed. Following this you can separate the part from the support material. Now you can measure your part or test fit it with other parts to make sure it is as desired. The key things for an efficient 3d printing process is only making your print quite small and making sure it is oriented so that it has the least support required to keep it up whilst having a large area on the bed to reduce the chance of warping and wasting a print. Well our printers are pretty safe as we are working with safer filaments it is important to be aware of the risks involved such as getting burned by the hot bed, inhaling to much fumes (although our printing filament is relatively safe fume wise) and the fire risk posed. Some strategies to mitigate these risks are: waiting for the printer to cool before getting your print out and making sure the ventilation is on (fans in the printing room) along with minimising exposure by checking on a print through the front window instead of opening the top lid. To manage the fire risk you should not leave the printer completely unattended, while you don't have to watch it print, someone should be in the ptec block when the 3d printer is running e.g make sure you don't leave it running over night..

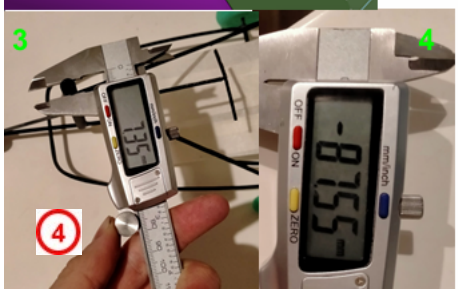
30

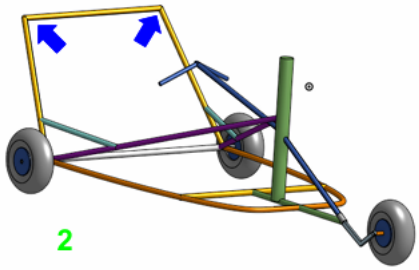
5

Assessment against specification.

Comparing image one to image two, I would say my cnc model is an accurate representation of my graphical representation. Along with this I could use tape to model what will be the fabric seat of my blokart which I was unable to model on CAD. To check my cnc product aligned with my second cnc specification; that it must be a consistent scale, I measured multiple components. As the scale is 8.3 to one, I then multiplied this measurement by 8.3 and checked it against my cad model. Doing this for my measurement of the mast step shown in image three, I found that this lead to the calculated size being 61mm which is 1mm off the 60mm it should be shown in image 5. This could have been off because the 3d printer printed it too large but I think in reality I just needed to push the calipers together with more force as the measurement only needed to be 0.12 of a mm smaller to give the right calculated diameter. To make sure the scale was consistent throughout I also measured along the top bar indicated by the two blue arrows on image 2. This was quite tricky to line up so I found it easier to get the calipers on the corners and then press zero. After this I could return it to zero and find the difference and thus the length. This measurement as shown in image 4 once multiplied by 8.3 gives 726.66mm which I would say is close enough to the 727mm it should be. Due to these measurements I believe my cnc model meets the specification of being a consistent scale.

6





31

xxxx you have given this a good go but I think you are missing some depth in areas. For example, you don't discuss What temperature the bed and nozzle need to be preheated to, you don't discuss initializing the printer and what this does And I'm not sure if the 3D Printer has default settings, just what someone previous used. I would have loved to see a 3D printing specification that you would then evaluate against. You have discussed enough of the relevant points to get an achieved.

7