CRAFT CENTRE FOR RESEARCH AND
APPLICATIONS IN FLUIDIC TECHNOLOGIES

SOP: HEIDELBERG uPG501 Mask Writer

Contacts:		
Emergency: Dan Voicu	Dan.Voicu@uotoronto.ca	(416)-978-8473
Training: Kayla Soon	k.soon@uotoronto.ca	(416)-573-9921

Equipment Overview

Notes before use:

1. If the software appears to stall/hang and does not appear to respond to any input, reboot the computer (not just the software) and proceed as per the instructions and recommended wait times.
2. If unsure about material, mask size, and design, contact the trainer. DO NOT load masks in the writer if uncertain about the writing process.
3. Check the design in AutoCAD (Only)
   a. Open the design file in AutoCAD
   b. For 3D Solids, flatten all the designs for the mask into one plane by typing **flatten** in command line, press **enter**, **select** all the features and press **enter**
   c. For multilayered designs, ensure that the origins are the same (preferably center of the mask) for all the masks
   d. All lines should be closed, If there are gaps, close them by typing **pedit** then **close**
   e. Select all the features forming each design and type **pedit** at the command line then press **enter**, then type **join** and press enter
   f. Explode any designs drawn using array functions.
   g. Note down the measurement units (mm or microns etc)
   h. Save the file as a .dxf file (2013 compatible) and note down the name of the layer and file name. Contact staff if you would like to use other file types like CIF or GDS2 etc.

Supplies and Reagents

Provided by CRAFT
- 3, 4, 5 inch photomasks
- Developer reagents (AZ400K and CEP200)

Method

1) Setup:

1. Make sure the Mask writer is turned ON. Check for the green LED
2. Check to see if upg501 software is open. If not, open it by double clicking the icon. (opening the software twice will require a system reboot)

3. If the system was rebooted, **wait for 1 min**, before opening the upg501 software. The drivers take a while to boot – at least 1 min.
4. Click **OK** on the access denied pop up message
5. **Wait for 30 sec** (if not, the software will hang, and would need a system reboot)

2) Procedure:

Mask file conversion:

   a. Copy or save the .dxf design file to C:\HIMT\designs\dxf. Saving the file in a sub-folder may lead to errors when converting.
   b. On the main welcome page for the upg501, click next. If the upg501 software was already open ensure that it is at the welcome page by clicking **back** multiple times until it reaches the welcome page (the back button would be greyed out at that point), then click **Next**
   c. This would present an exposure set up page.
   d. The first step is to convert the dxf file into a writer readable HIMT file. Click on 'start conversion interface'
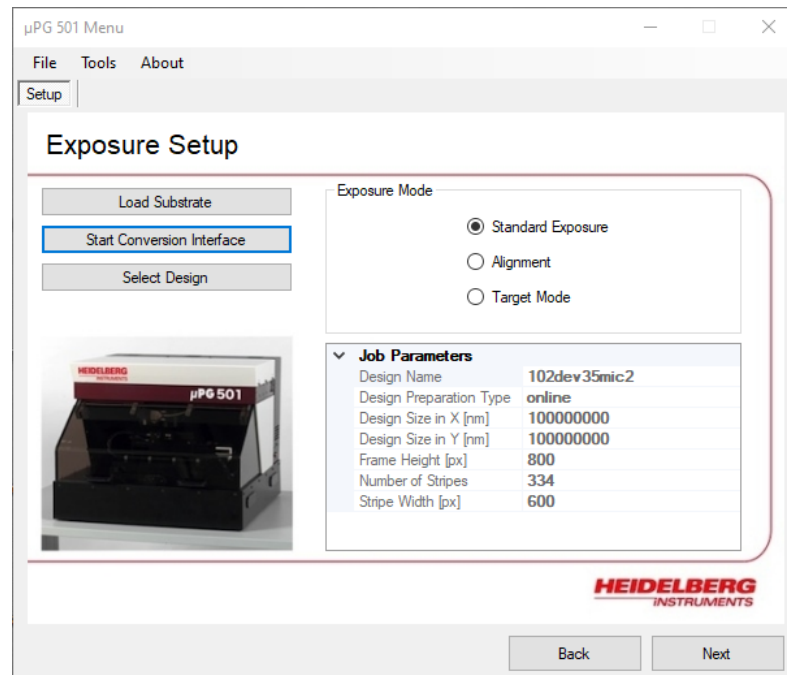

   e. This button will pop up a new window called '**GUI HIMT convert'**. In this window, **file →New Job** and give it a name. [Recommendation : filename_maskname or filename_masknumber]. Ensure that all characters are lower case and without special characters. Use ' ' for spaces.
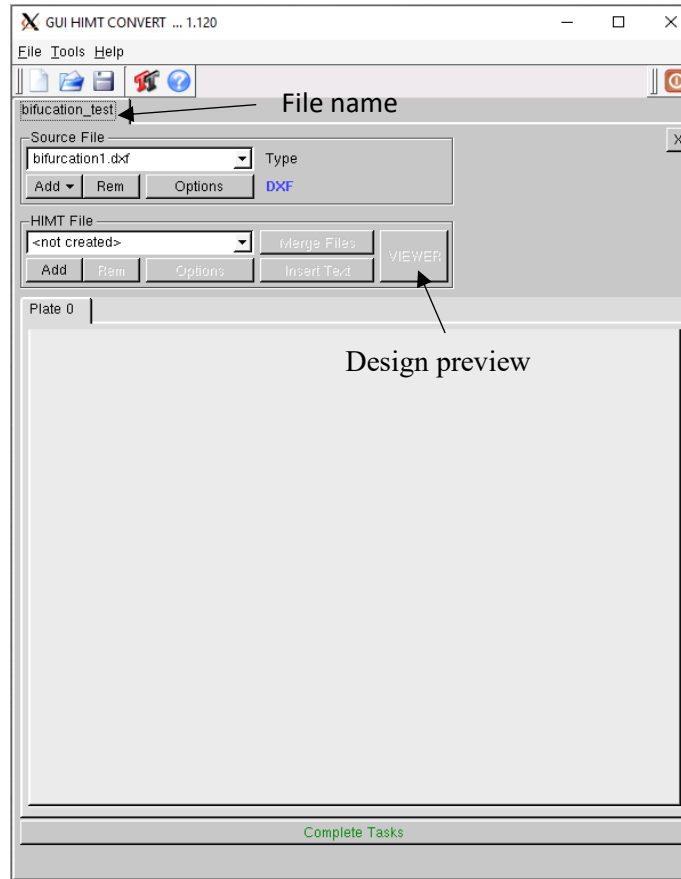
f. This will update the GUI HMT convert interface with more options
g. Click **add file → DXF**
h. Llocate the .dxf design file from the desktop
i. Once selected, a pop up titled 'dxf options' is presented to identify the units and scaling. For dxf files with millimeter as the design units, DXF units [nm] = 1000000. If designed in centimeters, DXF units [nm] = 10000000 and if designed in microns, DXF units [nm] = 1000.

Last Edited: May 28, 2023
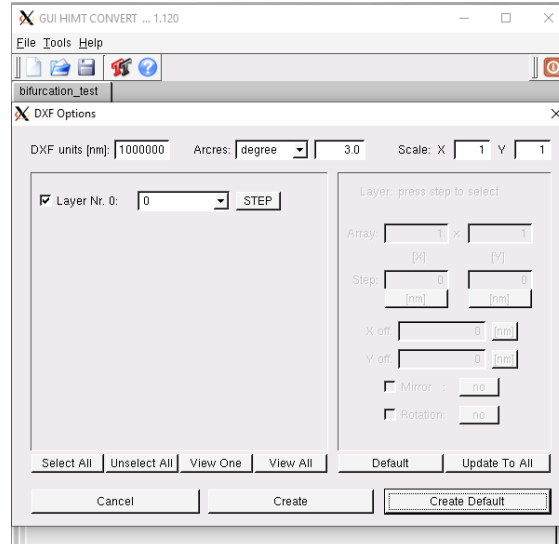
j. Click on 'Create" to convert to HIMT format
k. Check the main status popup window for any errors and/or warnings (Some warnings may be ok to proceed with. Any errors could result in bad/no conversion and printing)
l. You may check the converted file by clicking the viewer
m. In order to identify the regions for printing, check options under mode in the HIMT Viewer pop up window. Select XOR only if you have bounded regions (feature within a feature, like posts within a channel). Any preview settings in the HIMT viewer will NOT set the exposure settings. The HIMT viewer is intended to help you make decisions about the exposure.
n. Confirm the design size. If the design size is an abnormal value, there is likely a problem with the design.
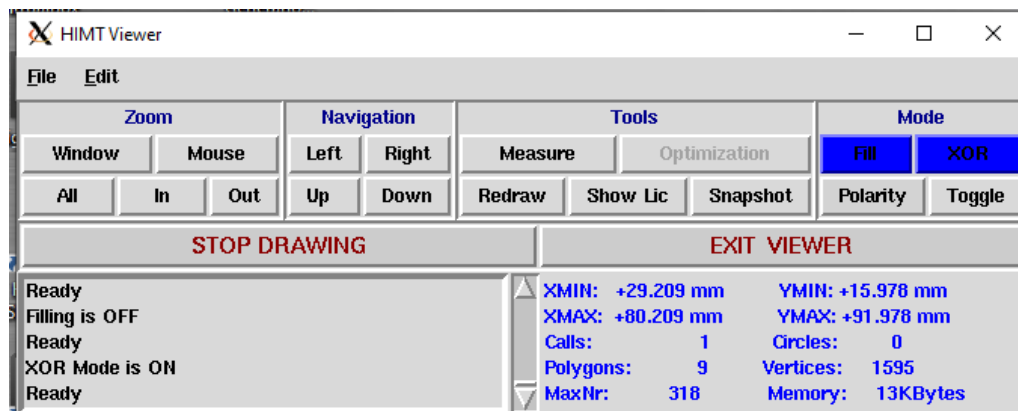
o. To define exposure, click on 'expose options' under the file exposure details.
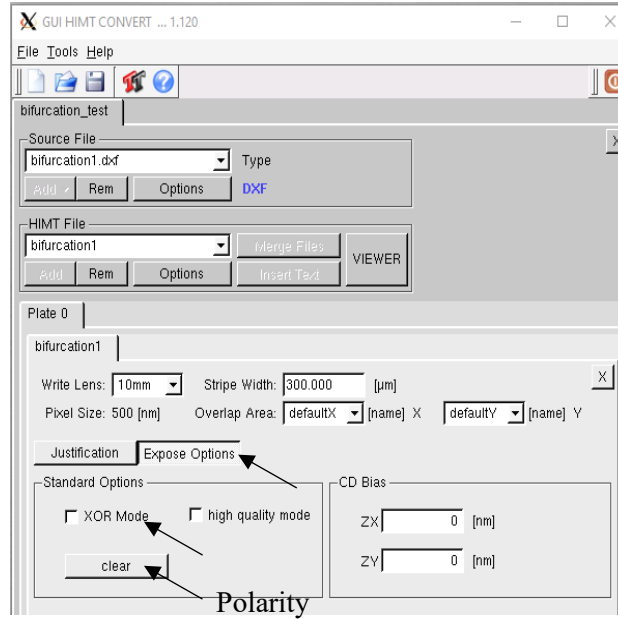
Polarity

p. In the event that there are shapes within a shape, click XOR Mode to perform a subtraction process of all overlapping shapes. In order to change the polarity click on the clear button (will switch to dark). Remember that the filled (black) region will be clear (no chrome) after mask development if the button reads Clear. If the button reads dark, then the filled in black region in the preview will be opaque (with chrome).

q. Click complete tasks at the bottom and complete expose jobs. If your design is very large or complex, this step may take a few minutes
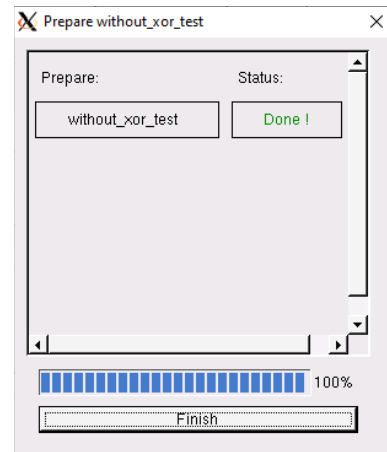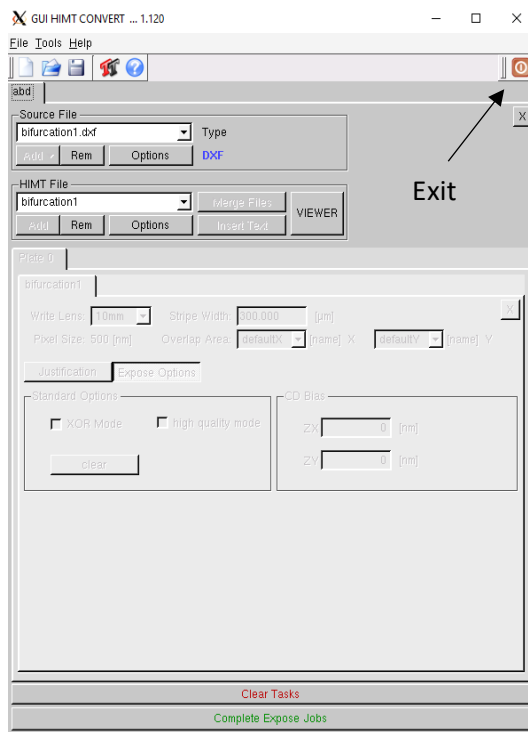

Exit

Conversion process tracking

r.   After the conversion is complete, click the finish and then the power button to exit the conversion interface.

1. In order to load the substrate click on the 'load substrate' button on the main exposure set up window. Identify the proper alignment pin and vacuum region selection as per the reference instructions below. Move the alignment pins using the tool provided. (notify instrument manager if you lose the pin). Place the mask against the 3 pins and turn the vacuum on using the lever on the left of the stage.
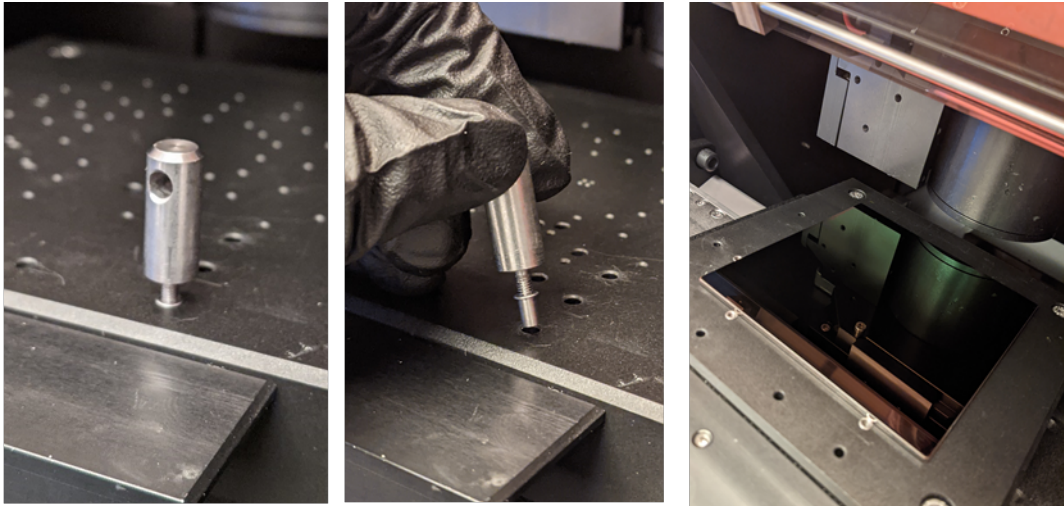
2. Vacuum settings should also be adjusted by turning them to meet the requirements based on the mask size
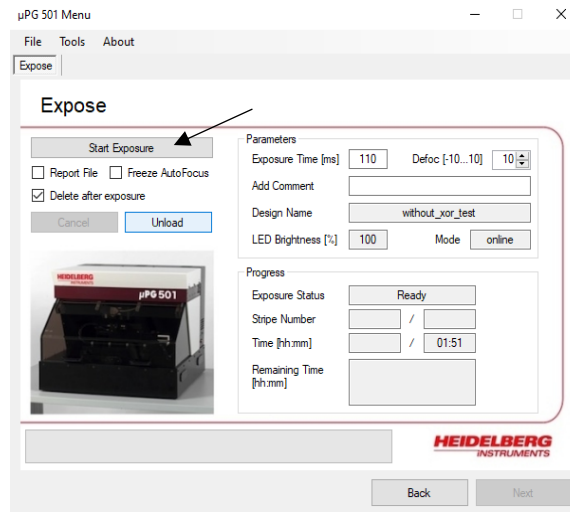

Vacuum setting
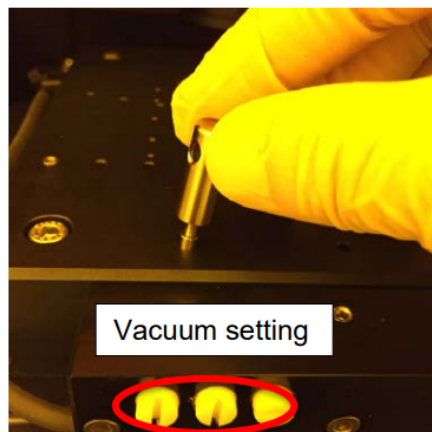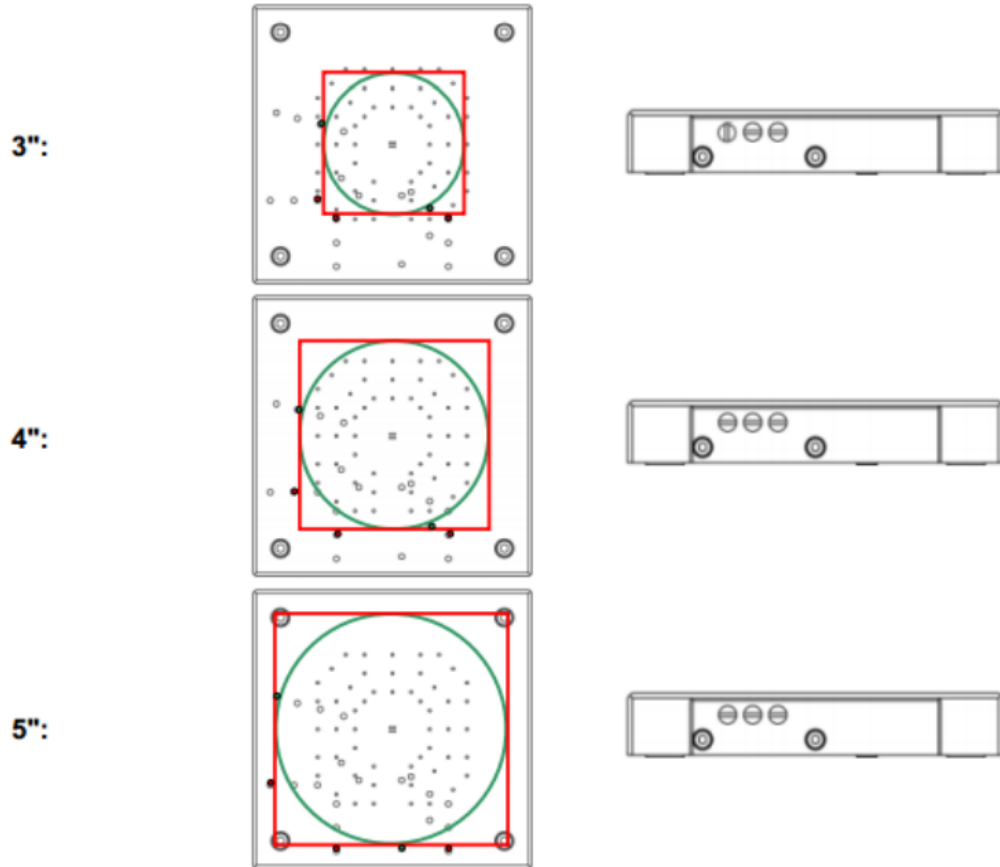
3. Follow the prompts for locating mask origin and size, Click yes or ok to the questions that pop up.

Exposure:
a. Click select design in the exposure setup window, and select the 'expose' file generated for the design in the jobs folder that pops up.
b. Ensure the exposure time is set to **110 ms and Defoc to 10.**
c. Click on Start exposure

d. The timer does not reflect true time to complete. The green bar at the bottom and the stripe number is a more accurate way to predict progress and estimated completion time.

e. After the timer runs out, click next and then unload to retrieve the mask

Timer not accurate
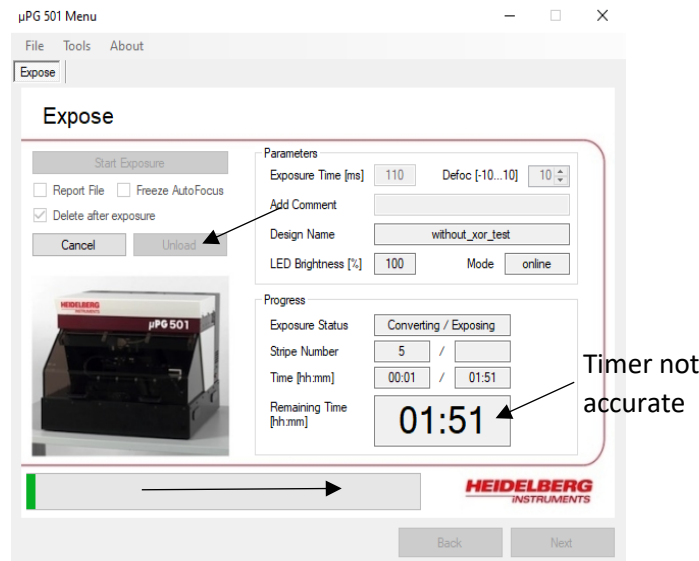
Developing masks

a) After retrieving the mask from the writer, prioritize the development of the mask. If you plan to develop the masks later, store the masks in a dark box or away from light.

b) Three glass dishes labeled 1,2, and 3 are available on the wet bench. Fill each dish with the appropriate solutions as identified below,

c) Dish 1: 1 part AZ400K and 5 parts DI water - 45 sec immersion

d) Dish 2: CEP 200 45 sec – 1 min immersion

e) Dish 3: AZ 400K – unlimited immersion

f) AZ400K is a photoresist developer. While CEP 200 is a chrome etchant.

Last Edited: May 28, 2023

i. Diluted AZ400K solution in dish 1 can be used for developing the un-crosslinked (by laser) photoresist. Immerse the mask into the solution and rock the dish gently holding one corner. In about 45 sec – 1min, a faint pattern of the design will begin to appear. When you visually observe no more changes, rinse the mask and move to solution 2.

ii. After rinsing, transfer the mask to Dish 2 using a flat end tweezer but be careful not to create scratches on the mask surface. Immerse the mask completely into the solution and gently rock the dish holding one corner. The chrome etchant will etch the chrome in the exposed region in the photoresist layer. After this step, rinse the mask with DI water.

iii. Rinse the mask with water and dry it with nitrogen/air. Carefully and thoroughly examine the mask, under the microscope if required. If the design does not look fully developed, cycle between steps 1 and 2 again (you may do this as many times as you want). If the design still does not look good, increase the concentration of solution 1 a bit and repeat steps 1 and 2. After the design looks good, proceed to step 3.

iv. In order to remove the rest of the photoresist on the mask, immerse the mask in the AZ400K developer in Dish 3 and rock the dish gently for 45 secs. **DO NOT go back to step 2 after this step.**

3) Cleanup:

1. DO NOT close software when finished and leave computer ON
2. Remember to close the lid when finished and log out of the "laser pattern generator" on LMACS

Troubleshooting

1) 'Start Conversion Interface' not loading

1. Shutdown the computer a wait 2 minutes before restarting it.
2. After restarting, wait 2 minutes before logging in (the login password it written on the top of the monitor)
3. Once logging in, do not start the HIMT software until a Microsoft pop-up appears stating that the driver has been properly connected. This should happen within the first 5 minutes of logging in.
4. Once the pop-up has appeared, the laser pattern generator has made proper connection to the computer and you can start up the software.

2) Design does not appear in 'Viewer'

1. Typically you should also see ERROR NO BOARDERS FOUND or ERROR NO STRUCTURES FOUND when viewer is loading. In this case, go back to autoCAD and check for the following:
   a. All lines are polylines or circles only. Any other line type (spline, array, arc,...etc) will convert properly. This will be shown when clicking on any given line and seeing the line type in the properties (left panel). Please see 'notes before use' for details on converting line types. If you have used an array, you can select

the entire array, type **explode**. Then for the same selection, type **join**, and all
individual array elements will be converted to closed polylines.

   b. No lines are overlapping. This is most common when the original design file
   was imported into autoCAD and not created in autoCAD. Click on any given
   line and delete it to check if there are any random line segments hidden
   underneath. All hidden lines must be removed before converting design.

   c. Small line segments/polylines outside the write field area. Check that there are
   no lines/shapes/artifacts outside the bounds of the intended design. If you are
   unsure, highlight your entire design and copy it into a new file to remove any
   potential unwanted artifacts.

2. Check that the units are correct when loading your design. The software makes the
   initial assumption that you are using mm. Double-check your units in autoCAD and
   make the correct DXF unit conversion here: