

## EBL CAD Procedure

### To create a pattern within a single field

1. Open CAD software (WecaS)
2. Click on Setting Icon
  - a. Choose field size (\*Note: the 2.4mm field size is not available for our system, even though it is displayed in the pull down menu)
  - b. Choose number of dots (# pixels in field)
  - c. Click OK
3. Click on Place Chip Icon or type command PC (this will place a square “chip” on the layout with the field size chosen previously)
  - a. Enter chip name (< 8 characters)
  - b. Enter chip origin coordinates
    - i. Typical value to use is 10mm, 10mm for now, and can move later
    - ii. Type in 10.0,10. (the last number must end with a . to indicate the units are mm) and hit enter
4. Click on Axis Icon or type command AX to change to chip coordinates.
  - a. This will change the origin of the chip to 0.0 dots
  - b. Spacing of dots can be read off screen (depends on field size and number of dots in field). E.g., for a field size of 600um and 60,000 dots, each dot is 10nm.
5. Draw patterns as desired using the various drawing icon types. *Note: all geometric shapes are automatically “filled” when drawn. To draw single lines, use the line icon, or circle generator icon.*

For example, to draw a square or rectangle:

  - a. Choose square icon or type command DS
  - b. Enter P1 coordinate, or click on position in layout for first point
    - i. Coordinates are entered in dot units. E.g., to start a point in the middle of a field with 60,000 dots, enter 30000, 300000
  - c. Enter P2 coordinate, or click on position in layout for second point
    - i. E.g, if each dot is 10nm, entering the P2 coordinate as 30010, 30010 will make a square that is 100nm on a side
6. The measurement icon can be used to measure and verify pattern sizes after they are drawn. Use the change unit icon to change the measurement unit to um.
7. Patterns can be copied, moved, rotated, etc. using the appropriate icons.

For example, to make multiple copies of a pattern in an array:

  - a. Choose copy matrix icon or type command CM
  - b. Identify element to be copied by clicking on it in the layout
  - c. Enter X, Y, and N where
    - i. X = shift in pattern in X direction (in dot units)
    - ii. Y = shift in pattern in Y direction (in dot units)
    - iii. N = number of total patterns
  - d. Ignore entries for L,H,R and D,P by pressing enter
8. Save file as a condition (CON) file by clicking the save icon. *Note: All files should be saved in your personal folder, and it is best to save each design into its own sub-folder. This will make it easier to transfer files to the EBL computer for*

writing. This will save a condition file (.CO6 extension) which contains the conditions information, as well as pattern files (.CCC and .CBC extensions) that contain the pattern information.

To create a stitched pattern (across multiple fields):

1. Open CAD software (WecaS)
2. Click on Setting icon
  - a. Choose desired field size (for one field) (\*Note: the 2.4mm field size is not available for our system, even though it is displayed in the pull down menu)
  - b. Choose number of dots
  - c. Click OK
3. Do NOT place the chip yet
4. Draw patterns as desired using the various drawing icon types (as described in previous section).
5. Click on Place Chip Icon or type command PC (this will place a square “chip” on the layout with the field size chosen previously)
6. If the layout does not fit within a single field size of the chip, use the matrix chip command to make multiple copies of the chip so that the layout is contained within the created chip matrix. Note: Choose NO for “All?” to only copy and matrix the chip itself, and not the contents within the chip.
7. When completed, save file as a condition (CON) file by clicking the save icon. Please save all files within your personal folder located in the “Users” folder on the desktop. It is highly recommended that you create a separate folder for each design. This will make it easier to transfer all of the files that are needed to the EBL computer.

To create a schedule file

1. Open the desired condition file into the CAD software (if not already present)
2. Click on the EXP icon to open an associated schedule file
3. Enter a value for the position offset to define where the pattern will be written on the substrate. (Note – this value would = 0 if the position on the substrate was defined in the layout file)
4. Enter the dose time offset (Note – this is the actual dose if a dose was not defined in the layout file)
  - a. The pattern can be stepped or matrixed by creating multiple entries in the exposure table.
  - b. An exposure/dose matrix can be performed by creating multiple entries with different doses and positions
5. Click on the Set Option button
  - a. In the 7500 EX window, typical settings are:
    - i. XY-Laser: Line (reads and adjust X-Y position with each scanned line)
    - ii. Z-Movement Mode: Preset

- iii. Z Preset: Type in known value for your substrate, or first guess. You will later verify this Z height when you set up the exposure for your sample
- iv. Z tolerance: 0.5um (fine). Use larger tolerance if want faster throughput.
- v. HS (height sensor) Error Process: choose “continue” button.
- b. In the Registration Window
  - i. Registration Execution mode: Choose N if not aligning the pattern to a previous layer. (If alignment is to be performed, see the “EBL Overlay Procedure”)
- 6. Save the schedule clicking the save icon. Please save all files within your personal folder located in the “Users” folder on the desktop. It is highly recommended that you create a separate folder for each design. This will make it easier to transfer all of the files that are needed to the EBL computer.

#### Importing Design Files and File Conversion

The EBL CAD software can convert either DXF (AutoCad) or GDSII files. To read in one of these file types and convert it into the EBL CAD software:

From the “Tool” pulldown menu select:

DXF converter for a DXF file

GDSII converter for a GDSII file

Follow the instructions given in the Elionex Instruction manual for file conversions. Note that there is a section on DXF conversion and a separate section on GDSII conversion.