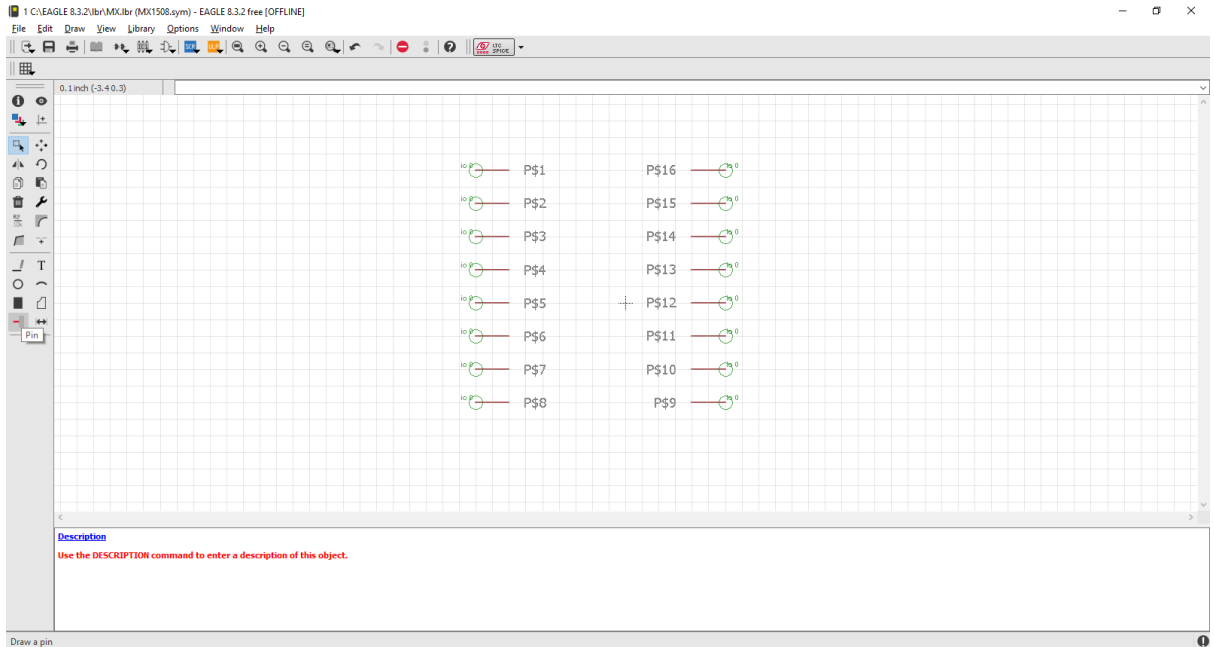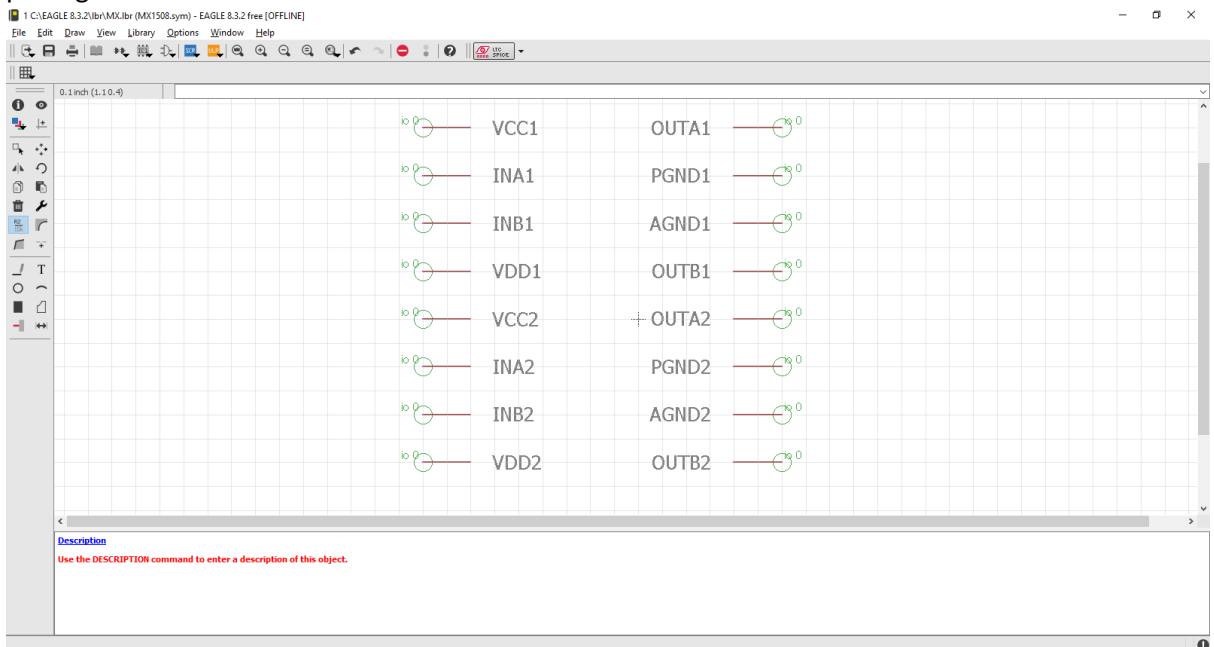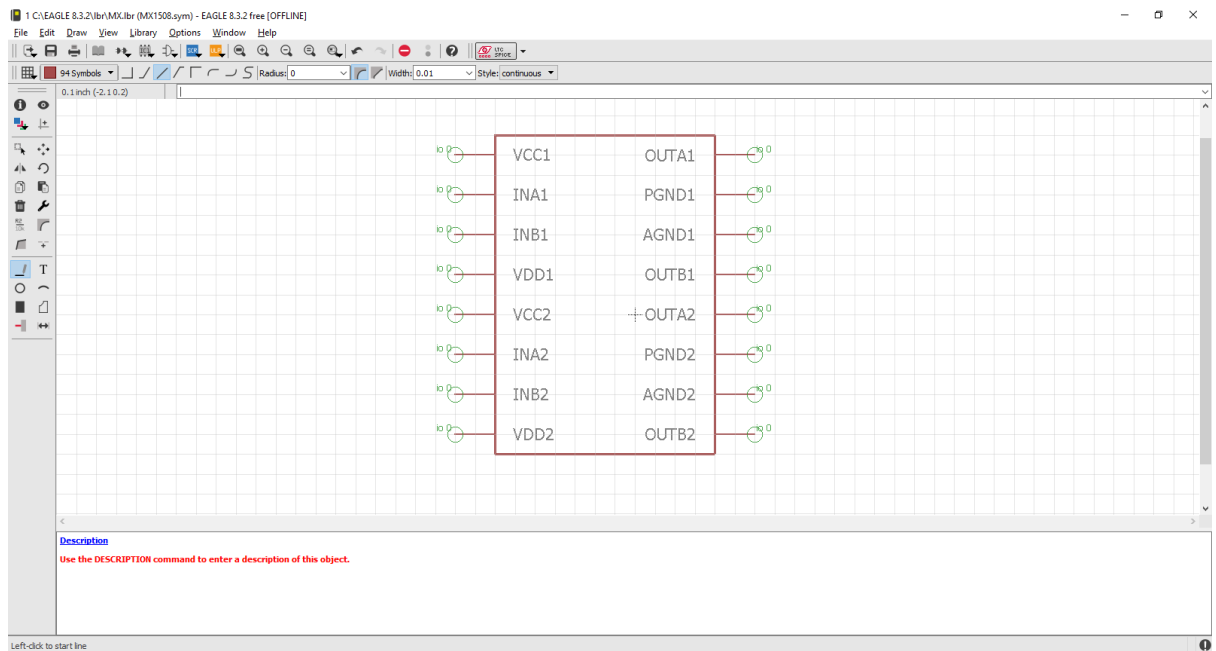# Step 2: Creating the MX1508 Symbol in EAGLE

1. Right click the library created previously and click open to open the library window. Select the Symbol ⬚ icon and name your new symbol MX1508

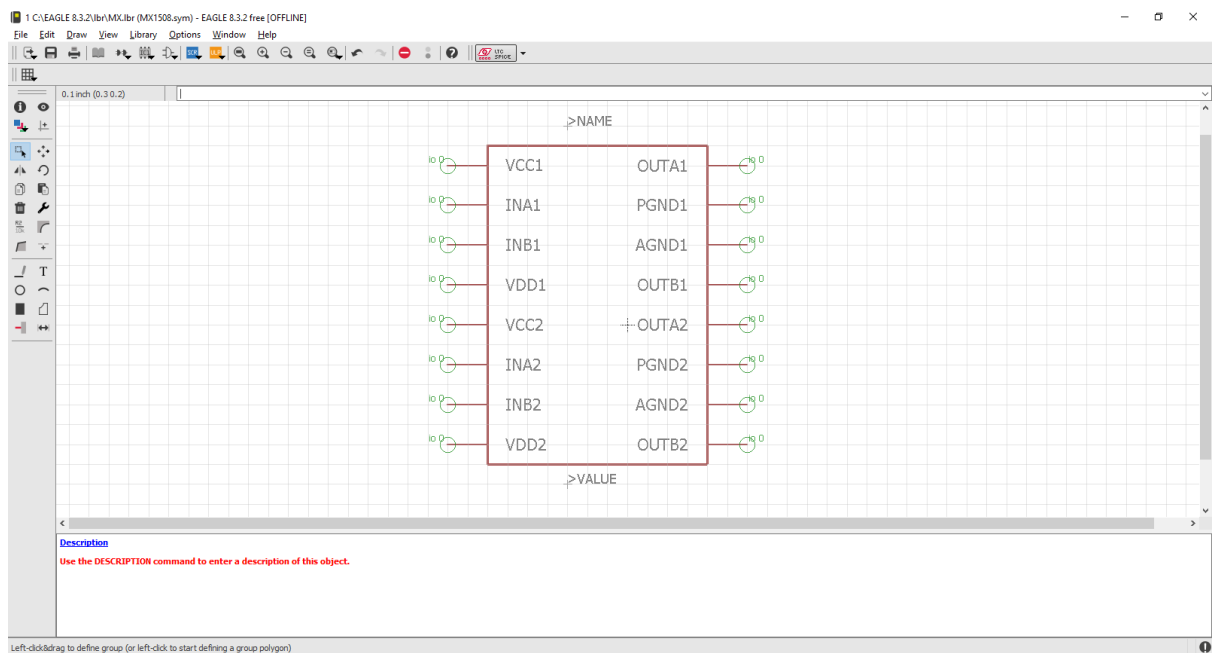2. Select ▭ pin and place 16 pins. Left click to rotate.

3. Name the pins according to the datasheet. Use the Name tool as done when creating the package.

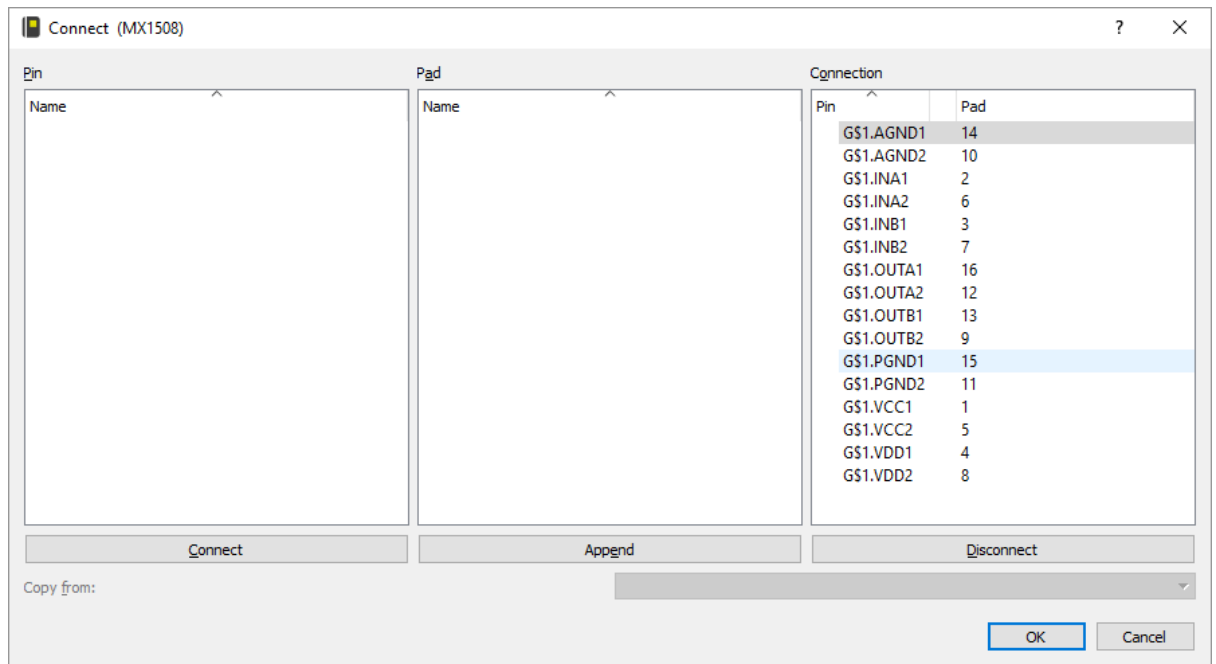4. Click the line icon and add an outline

5. Add name and value placeholders as done for the package using the layers 95 Names and 96 Values

Creating the MX1508 Device in EAGLE

1. Open the Library window again and choose Device ⬚. Name the device MX1508.
2. Add the symbol by clicking ⬚ Add. Select MX1508 and left click to place it at the origin. Hit escape and close the pop up window.
3. Click New in the bottom left and add the package.
4. Click connect in the bottom left and connect all the pins to the corresponding pads.

5. Click Description and add a short description for our device

6. Done!

References/Further reading

http://web.csulb.edu/~hill/ee400d/Technical%20Training%20Series/11%20PCB%20Layout%20with%20Eagle%20CAD.pdf

https://www.autodesk.com/products/eagle/blog/library-basics-part-2-creating-first-symbol-autodesk-eagle/

https://www.autodesk.com/products/eagle/blog/library-basics-part-3-creating-first-device-autodesk-eagle/