

# Lesson 1a: EAGLE Part Creation

Sam DeBruin

January 16, 2012

## **1 Introduction**

This addition to lesson 1 covers the EAGLE part creating process and can be used with or without concurrent completion of Lesson 1: EAGLE Part Creation and Schematic Capture.

This lesson covers making the EAGLE part for the ATmega328 microcontroller. This part can be found at

<http://search.digikey.com/us/en/products/ATMEGA328-AU/ATMEGA328-AU-ND/2271029>

Access the datasheet. It will be needed to complete the part.

## 2 Symbol

From the EAGLE Control Panel select File → New → Library

From within the library select Library → Symbol...

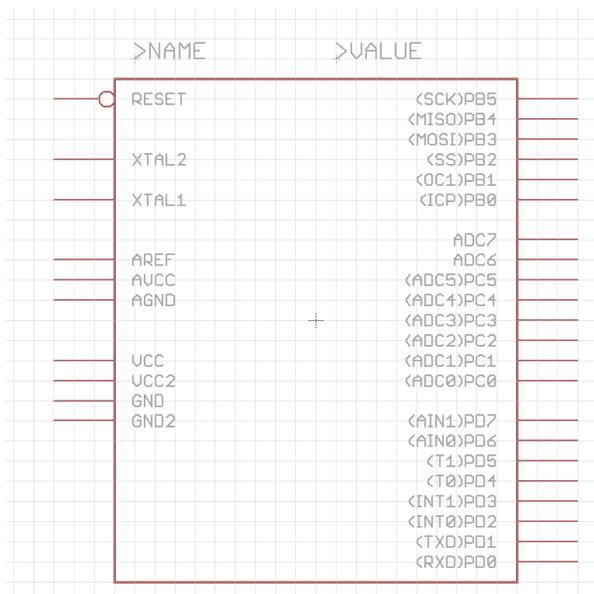
In the Edit dialog under Symbol type 'ATMEGA328-AU'. It will prompt you to make a new symbol, press 'Yes'.

In the command line, type 'wire' or press the  button.

Using the wire tool, make a 20x25 rectangle centered at the origin.

When making a complex part like a processor, care should be taken to group like pins together. If not, the schematic will be unnecessarily complicated. The ATmega that we are using has 32 pins as seen on the datasheet page 2 (TQFP package). One possible arrangement is presented below.

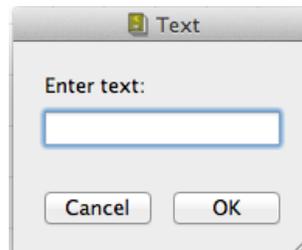
Type 'pin' or press the  button to place pins.



Once pins are placed, type 'name' in the command line or use press the  button.

Name the pins according to the picture on the previous page.

The last step in the symbol editor is to add labels. Labels are not mandatory but they will make the schematic much easier to read. In the command line type 'text' or press in the  button.



We are going to place two labels, one for the part's name and one for its value.

In the 'Text' dialog box, type '>NAME' and press enter. Using either a press on the middle mouse button or the pull down menu in the upper left, change the layer to '95 Names'.



Place the '>NAME' label, followed by '>VALUE' on layer '96 Values'. Possible placement for these labels can be seen in the figure on the previous page.

### 3 Package

We are now done with the symbol for this processor. Next we need to make the actual footprint that will appear on the finished board. Although the symbol can look like whatever we want, the package must be exactly the sizes and shapes that the part will fit on.

From the top menu, select Library → Package... and make a new package called '32-TQFP'.

In the command line type 'Grid mm' to change the coordinate system to mm. Alternatively, press the  button.

Although we can make our part based on the information on page 548 of the datasheet, it is easier to use a recommended land pattern:

[http://www.micrel.com/\\_PDF/other/unofficial-pcb-footprints/7x7%2032ePadTQFP%20Land%20Pattern.pdf](http://www.micrel.com/_PDF/other/unofficial-pcb-footprints/7x7%2032ePadTQFP%20Land%20Pattern.pdf)

This is the same as our part minus the large central pad. As we can see, the pad dimensions are 0.5x1.6mm.

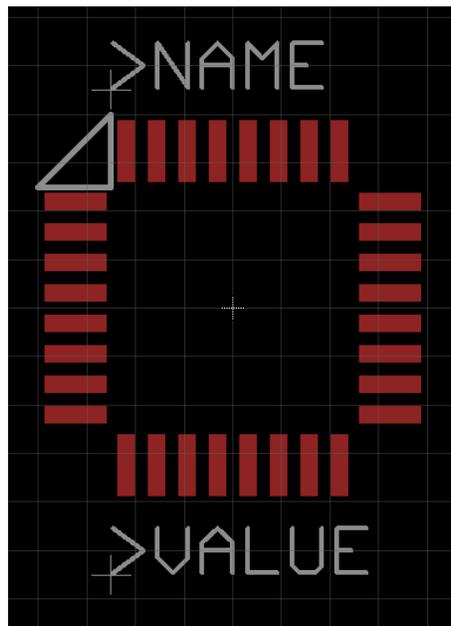
In the command line, type 'change smd 0.5x1.6' to change the pad size.

Type 'smd' or press the  button to place pads.

Using the design on page 548 of the processor datasheet as reference, place pads in a counter-clockwise direction starting from pin 1. Make sure to place the pins in order or the numbering will be off.

Use the land pattern diagram as reference to place all 32 pins. Use the 'info' command or the 'move' command to move pads to their correct location. NOTE: If you use the 'move' command, hold the 'control' key while clicking the pad to snap to its origin.

The pads should look like the picture below. If you are unsure of your pad placement, print out the package using 1:1 scale and make sure the part fits nicely. Ask your SPLAB instructor for access to the part.



Using the 'wire' command place a pin 1 mark on layer '21 tPlace' like in the picture. This will be used when assembling the board.

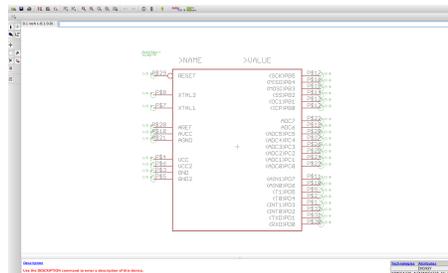
The final step in the package process is to add '>NAME' and '>VALUE' labels like in the symbol. These should be placed on layers '25 tNames' and '27 tValues' respectively.

## 4 Device

The final step is to link the symbol and package together in a device. The device is what gets placed into the schematic and includes both the schematic representation, symbol, and the board representation, package.

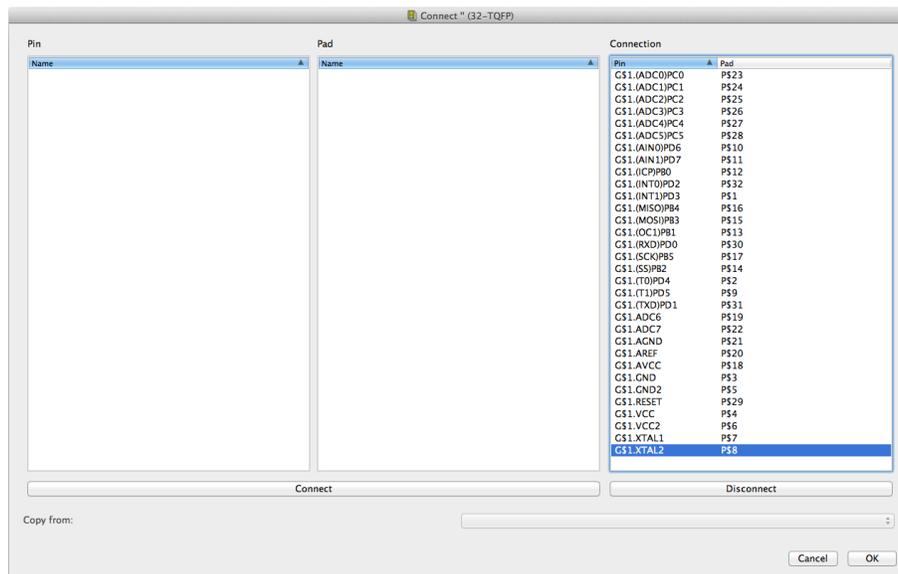
In the library go to Library → Device... and make a new device called 'ATMEGA328-AU'.

Using the 'add' command, find your symbol and add it to the device



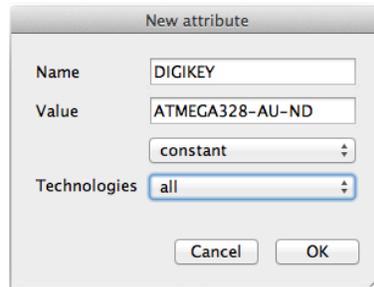
In the bottom right, press 'New' and select your new package

To match the symbol's pins with the package's pads, press 'Connect' in the bottom right. Connect the pins as specified by page 2 of the processor's datasheet. Use the image below to check your work.



Because we designed this device with a specific product in mind, it is good practice to include the part number to make part ordering easier. In the bottom right, press 'Attributes'.

The attribute that we are adding is the Digi Key part number, name it 'DIGIKEY'. Change the value to 'constant' and the technology to 'all'. The part number for the 'Value' field is 'ATMEGA328-AU-ND'.



The final step in the part creation process is to change the prefix in the bottom right to 'IC' to make naming smoother.

## 5 Part Complete

These are all the steps required to make a part to use in a design. There are some more complicated ideas - such as using multiple symbols with one package - but the above steps will make a working, well designed part. Attend later lessons or ask your SPLAB instructor for more information on more complex part creation or with any specific questions.

In order to add your part to a schematic, you need to import your new library. There are two ways to do this:

1. From the schematic, go to Library → Use... or type 'use' in the command line. Navigate to your library and press 'Open'.
2. From the Eagle Control Panel, press the dot next to your library so that it turns green:



If your library does not appear in the Control Panel, go to Options → Directories... and add the library's directory to the list of libraries.