The Design Process, Part 6
-
## Building the Library: Drawing things in the library - Parts

Sharon Hurtley-Durand
30 December 2021

The last step in building the part library is creating the Part which combines the Logic symbol or symbols with the Decal for the parts you will be entering into your design. This enables the logic symbol of the part you place on the schematic half of the software to identify with the footprint that appears in the layout half.

To the right is the Part section of the Library. This is displaying the MSP430F1471PM part selected from the library list below the display window.

When we select the Edit... button to the right of the list, we push into the part and the first window is displayed, which contains General information about the part.
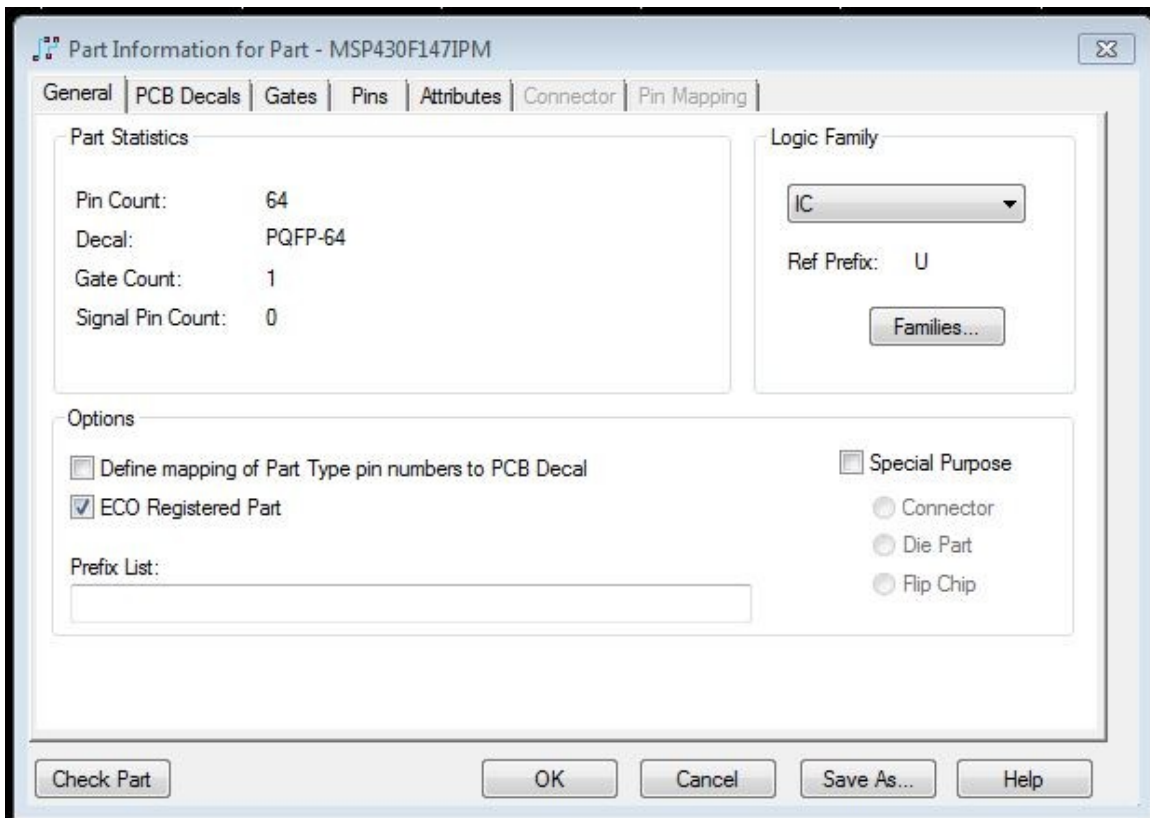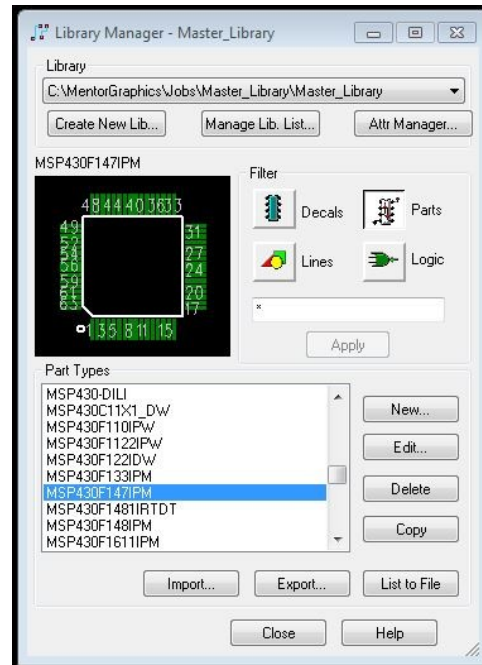
This is an overview of what has already been assigned to this part. Here it shows that the part has 64 Pins, uses the PQFP-64 Decal, has 1 Gate, and uses 0 Signal Pins.

Signal Pins are defined as unused pins on a part and are typically assigned to Power or Ground. I do not use these. They assume information in the background and I think it allows for confusion as it doesn't make obvious these connections. I prefer to use a Gate to show the connections. This will be shown when we get into building a real schematic.

To the right on this window is the Logic Family. This is where the function of the component is linked with the appropriate Reference Designator.
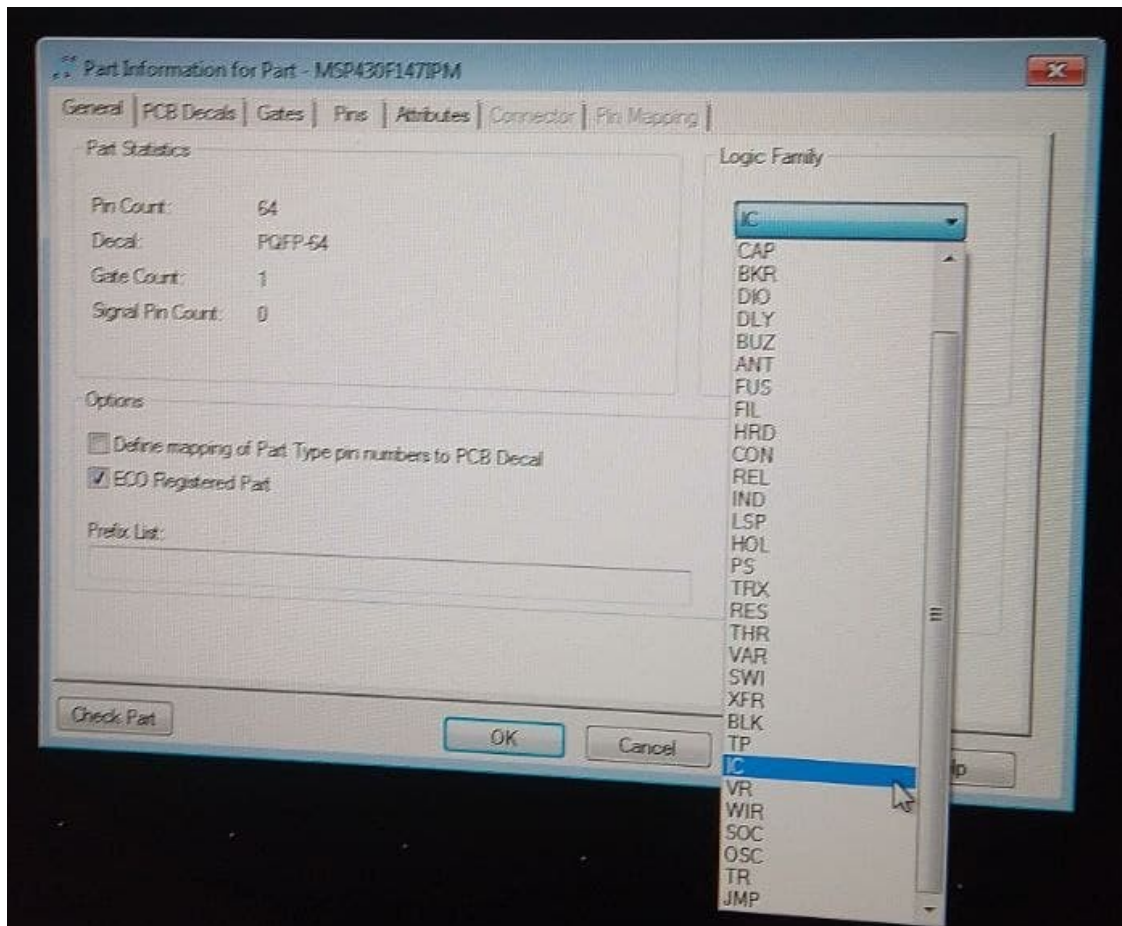
There is a pull down list of previously assigned types. In the above view the part is an IC and has the Ref Prefix of U. This is the letter that will precede a number on the schematic giving it its "local name and address".

The button labeled Families is where you can add to, redefine or delete from the pull down list of types.
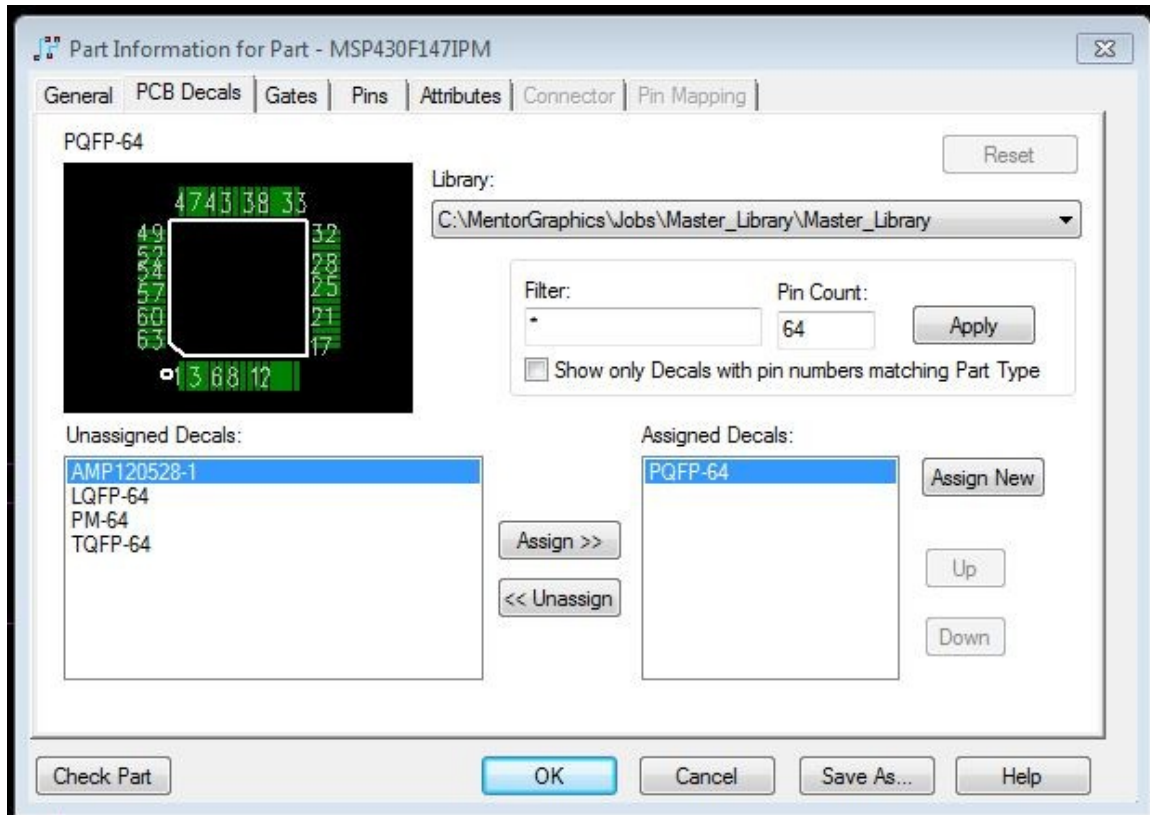
The next window in the Part Information group is PCB Decals. Here is where you assign which Decal is to be used for this part. The Decal for this part is PQFP-64. Decals can be named arbitrarily, but there is a desire to use names that are already in standard use. Here this name represents a Plastic Quad Flat Pack with 64 pins.

But, as the industry is always on going, there will be new types. However, the more popular types of packages will continue to be available. Often, depending on manufacturer or standardization authority, there may be several names for the identical footprint. Choose which ever seems appropriate. A good source for introduction to the various types of packages can be found at https://www.ti.com/support-packaging/find-packages.html

In the Decals Window under Library, there is a pull down list of all the libraries that are available. Select the library you wish to use and the apply whatever Filter you wish to it. The asterisk defaults to all possible Decals available with the given Pin Count. If you were to replace the asterisk with P, then it would only display those Decals that begin with the letter P.

The Filter then selects the Decals from the selected Library with a Pin Count of 64 and lists them in the Unassigned Decals. From this list, the PQFP-64 has been selected and Assigned.

The next window in the Part Information group is Gates.

Here there is only one Gate and it is a DIP-64 (Dual In-line Pack with 64 pins). The green drawing shows the appearance of this Gate.

Later we will show a part with multiple Gates. Gates can be added and deleted using the buttons above the list.

You will see that there are more than one possibility for the choice of decal: CAE Decal 1, CAE Decal 2, etc. Here you can put a choice for inserting one of the other options. You will be prompted to chose when adding the part to the schematic. These other options are identical in function but may present a better visual choice. For example, I use it to indicate a horizontal or vertical drawing of a part. Here are two options for a resistor.

You can rotate Logic symbols in the drawing, but in some instances, the result is not as desirable. There is a lot of text associated with the part: pin numbers, pin names, reference designators, values, etc. And these are not always easy to relocate.

And this is how these alternatives appear in the Gates window. When you add a Gate, the CAE Decal 1 button will prompt you to type in the name of the Gate. There is a further button that reads ... (3 dots) which will take you to a drop down list of possible Gates. Selecting more than one and assigning them will distribute them through the various CAE Decal boxes.
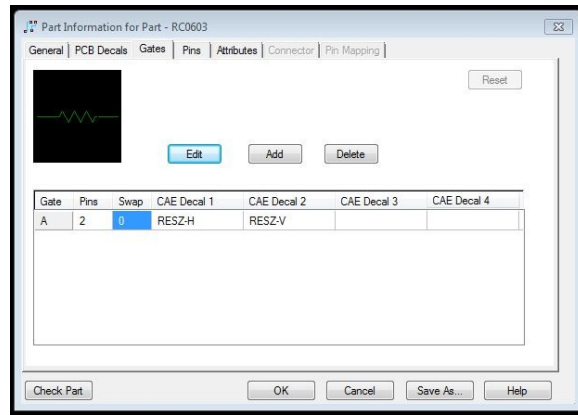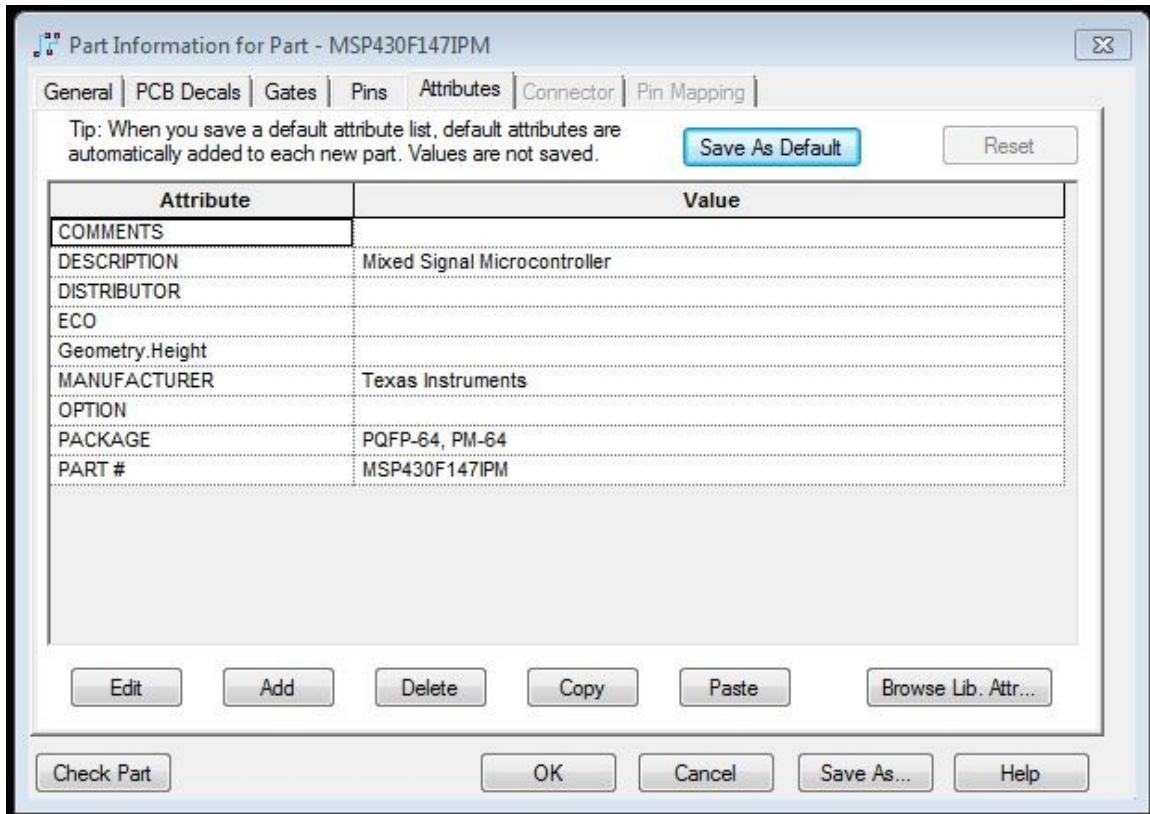

The next window in the Part Information group is Pins.
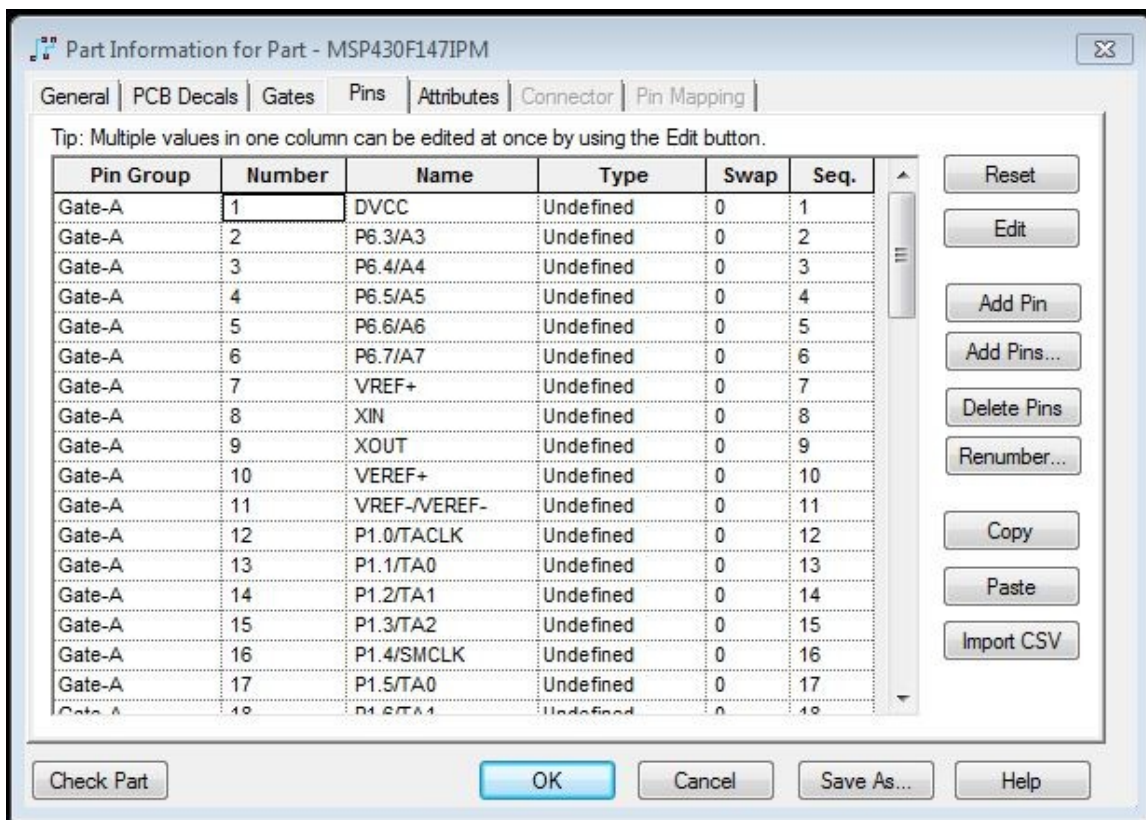
Here there is a list of all the pins in this part. The names are entered by the user and they are derived from the Data Sheet on this part. These names will show up internally on the drawing of the Logic symbol. The names are not required and the Name column can be left blank. However, these names convey essential information on the function of the part. Since this only displays a part of the list, the slide bar to the right will enable the used to access the rest of the list. Here, since there is only one Gate, the Pin Number and Pin Sequence are the same.

I do not use the Swap function. It enables you to swap electrical connections on pins that have an identical value. More complex parts have sub functions that make seemingly identical valued pins not identical in all instances. I prefer to manually swap connections when necessary.

I also do not use the Type function. This applies more in the use of simulator programs and we do not use them.

The final window in the Part Information group is Attributes.

Here you can enter information about the part that may be valuable to the Bill of Materials for purchasing parts, suggesting alternative parts that will fit the same footprint, etc. You can add and delete categories as needed.

The category Geometry.Height will set the height limit for the part. This can be essential in order to ensure that the part can be used if there is a limit to the height because of the shape of the enclosure, an overhanging part, or other such obstacles. The software will alert you if the assigned limits have been exceeded.

ECO was used more in the past to designate whether parts were lead-free or other environmental issues. Currently it's assumed that all parts conform to international environmental standards.