MAKE A PRINTED-CIRCUIT-BOARD (PCB) FOR YOUR ELECTRONIC DEVICE

11/10/2017 CLIFF CURRY

PART ONE: AN INTRODUCTION TO PRINTED CIRCUIT BOARDS. WHAT ARE THEY, AND HOW DOES ONE SPECIFY THEM TO GET WHAT ONE WANTS?

Start ->Multisim

File→open samples→ Mixed Signal → Pulse width modulator

Press the green arrow in the middle of the toolbar

Double Click on the XSC1 Tektronix thingy

Be amazed!?

Press the red button to stop the simulation

Transfer -> Transfer to Ultiboard -> Transfer to Ultiboard 14.1

Save the file with default name

Accept the changes to the netlist with OK

Use control-tab to switch between the schematic and the layout

1. In the upper left hand corner, look at the selection buttons. Disable all the selection buttons except “parts”
2. The mouse wheel zooms in and out. Click and drag to move parts. Click in open space to draw a boxes around several parts to select and move them as a group
3. Move the parts onto the circuit board.
4. The yellow lines are the interconnects (called “ratsnest”)
5. Looking at the schematic, move the parts to positions that are similar to the schematic positions.
6. When moving the parts, rotate them using cntr-r on the keyboard. Try to avoid having the yellow lines cross
7. Don’t fuss too much.
8. From menu, in the middle, choose Autorouter→ start Autorouter
9. Be amazed!?

IT IS GREAT WHEN IT ALL WORKS OUT

- If you have a few parts,
  - they are pretty big,
  - your traces are very thin, and you don't care
  - Your board is big enough to spread them out
  - all the parts are in the program's database
- Then life is good

NEXT STEPS

- File->Export
- Choose the layers Copper,Silkscreen,Soldermask,Board outline, Drill.
- Export
- upload the files to https://oshpark.com/
- Choose "super swift service" .
- Use your credit card.

PCB DESIGN AND LAYOUT IS A MECHANICAL ENGINEERING PROBLEM

- Its all about materials, dimensions, and topology

PRINTED CIRCUIT BOARDS: HISTORICAL CONTEXT

HOW THEY ARE MADE

PCB'S ARE DESCRIBED IN TERMS OF *INCHES*

- Modern parts are mostly described in mm, but the PCB's are still described in terms of "mils" or thousands of an inch.
- Small parts have lead spacing of 0.5mm=20mils.
- Smallest traces and spacing between traces for cheap boards, 10mils typically
- More sophisticated PCB manufacturing→ 6mils - 3mils. 3 mils is the width of a human hair

PCB'S PHYSICAL LAYERS.. PLASTIC AND COPPER

- Most boards are one layer of plastic with a TOP and BOTTOM copper layer.
  - My be cheaper to have only one copper layer, but is now non-standard
  - Quite a bit more expensive to have internal copper layers… usually an EVEN number are used: 2 layer, 4 layer, 6 layer boards.
- Drilled Holes with plating on their inside are used to connect the layers: called "Vias"
- The thickness of the plastic between the layers is not usually a big concern to the PCB manufacturer… but can be a concern to you as a designer.
- Thickness of the copper TOP and BOTTOM is usually 1.4mils, but can be more or less
- The TOP and BOTTOM are covered with plastic solder mask, a coating, except where solder is supposed to go.
- A help when putting the parts on the board is a layer of white text and shapes, called the "silkscreen" . This is standard on pcb's.
- A standard overall PCB thickness is 63 mils, but can be many other thicknesses

THE BOARDS ARE MANUFACTURED IN LAYERS, THE PCB'S DESCRIPTION IS ALSO IN TERMS OF ***LAYERS***

- There is a layer for copper shapes, a layer for pads, a layer for the drilled holes , a layer for the board outline, a layer or the solder mask…. And so on..
- The PCB manufacturing programs often have dozens of drawing layers, and they all use them differently.
- An example: a program will have four layers, called "refdes", "annotations", "text", "graphics". You can draw on each of the layers. Then the "silkscreen" layer (this is the white paint that is placed on the circuit board) is made from the combination of all four layers

PHYSICAL NATURE OF THE CIRCUIT BOARD: MATERIALS

- "Glass-epoxy" is the plastic, fiberglass matts with epoxy resin impregnated in it.
  - This typically has a very high resistance…you can ignore its resistance mostly
  - The dielectric constant is about 4.7… use this to compute capacitance between traces.
  - At frequencies above 1GHz, losses and oddities in the plastic become important
    - You can get more sophisticated plastics, (i.e. Teflon), at a very high cost premium
- Copper is plated on the plastic
  - Usually 1.4mil (.035mm) thick, can be 2.8mil.
  - Resistance of copper IS important in high current situations. Inductance of the trace is important in high frequency situations. For low frequency, low current situations, you can pretend the interconnect is ideal.
  - Thickness is identified with the weight in oz. per square foot of copper "one ounce copper"
- "Solder-Mask", a plastic coating, is applied on the top and bottom
  - Solder mask really helps with soldering, both by hand or by machine
  - Really cheap board can come without solder mask (not recommended)

ALL THINGS ARE POSSIBLE. LET'S SAY THE "USUAL" PCB TO BE TWO LAYERS, 1 OZ. COPPER

- PAY ATTENTION to the specifications of the things that are important to you.. You can't assume that the PCB supplier will do the "usual things".
- Most important specifications ("traces and spaces") include the smallest traces the manufacturer allows and the smallest spacing between traces, pads, and holes, the smallest holes you can drill, and the tolerances on board shape.

HOW ARE THE PARTS CONNECTED TO THE BOARD? ---WITH "SOLDER" (A TIN-LEAD METAL)

Probably what you should do

- Hand Soldering
  - Use a good soldering iron, fine solder, maybe a microscope, sometimes solder flux
- Wave Soldering
  - A very messy process, useful for through-hole components and very high volume

Most often used method

- Using Solder paste and a "Reflow Oven"
  - A paste of tiny solder balls in flux is silk-screened onto the board, the parts are put on, and the whole thing is heated until the solder flows and the paste evaporates.
  - Can be done on an industrial scale, but also can be done on a small scale.

“TRACES”

- This is the name for the copper interconnect between the pads of the parts.
- PCB’s used to be made with a photographic process: A big layout was done with colored tape on plastic, then it was photographically made smaller.
- Later, a photo-plotter was used to be used to expose film to create the copper patterns. The photo-plotter had different apertures (round, rectangular, oval) for the light to come out. Then the light was moved around the film, exposing the film to create the traces.
- The file that controlled the photo-plotter was a text file that basically said “use this aperture”, “move from this coordinate to this coordinate”, “change to this aperture”, “move again” and so on.
- This is the still the standard way of describing the circuit board copper pattern.
- This still influences the layout programs… for example, right angles on traces are usually automatically made into two 45 degree angles. Pads are rectangular, oval, or round. Why? Tradition!

THE OUTPUT OF THE PCB LAYOUT PROGRAM IS A SET OF GERBER FILES, ONE FOR EACH LAYER NEEDED TO MAKE THE PCB. (THE GERBER COMPANY MADE PHOTO PLOTTERS.)

- Top Copper
- Bottom Copper
- Top Silkscreen
- Bottom Silkscreen
- Top solder mask
- Bottom solder mask
- Top paste mask
- Bottom paste mask
- Board outline

ULTIBOARD AND MULTISIM,
PRODUCTS OF NATIONAL INSTRUMENTS

"MULTISIM" IS THE SCHEMATIC DRAW-ER

MULTISIM WAS ORIGINALLY CALLED **ELECTRONICS WORKBENCH** AND CREATED BY A COMPANY CALLED INTERACTIVE IMAGE TECHNOLOGIES.

"ULTIBOARD" IS THE PCB LAYOUT PROGRAM

ULTIBOARD WAS CREATED BY A COMPANY CALLED ULTIMATE TECHNOLOGY, FROM NAARDEN, NETHERLANDS

MANY OF THE ELECTRONICS CAD PROGRAMS ARE COMBINATIONS OF A PCB PROGRAM AND A LAYOUT PROGRAM, OFTEN FROM DIFFERENT COMPANIES

- This explains some of the inconsistencies in the CAD program's interface.
- You can see this while using Multisim and Ultiboard

FIRST, YOU DRAW THE SCHEMATIC OF YOUR CIRCUIT USING THE MULTISIM PROGRAM. THEN YOU EXPORT THE SCHEMATIC TO THE ULTIBOARD PROGRAM

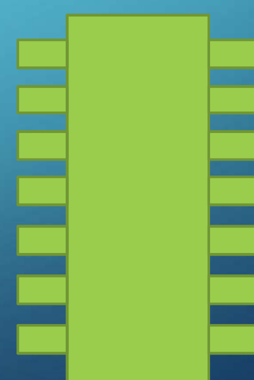
- The schematic drawing program uses the concepts of “COMPONET”, which are objects that have a given symbol and certain properties (the most important of which is “PINS”, or connection points), and “wires” or connections between the components.
  - Multisim also includes a simulator, a port of a Georgia Tech research project called XSPICE. This simulator, in turn is based on SPICE, which was a UC Berkeley research project. I do not recommend using the Multisim simulator, but it does work.

A SCHEMATIC DRAWING PROGRAM DOES NOT NEED ANY INFORMATION ABOUT THE PHYSICAL NATURE OF THE ELECTRONIC PART

- A PCB layout program does need all the information about the physical nature of the part, nothing about its electrical characteristics.

Schematic NAND gate

PCB program NAND gate



Fundamentally, the concept of "electronic part" is different for the two different kinds of program

ELECTRONIC PARTS COME IN STANDARD SIZES, RIGHT? SO, THE PHYSICAL DESCRIPTION (DIMENSIONS, BEST PATTERN FOR PCB LAYOUT, ETC.) SHOULD BE ALL BUILT INTO THE PROGRAMS

- YES and NO.
- Unfortunately,
  - 1. There has been a proliferation of part sizes as things have gotten smaller
  - 2. the drawings that come with the PCB programs are often just plain incorrect
  - 3. Soldering footprints that are appropriate for wave soldering or reflow soldering, may be inappropriate for soldering by hand in the lab.

STEPS FOR THE USE OF MULTISIM AND ULTIBOARD

1. Decide on the exact parts you are going to use in your design
2. Use Digi-Key to help you find parts, and to help you find the data sheet for the parts.
3. From the data sheet, find the physical drawing of the part and the recommended PCB land pattern.
4. Draw the physical part shape and PCB land pattern, using Ultiboard. Give this pattern a name.
5. Draw the part symbol for use in the schematic using Multisim. Use Multisim to specify the part shape that you have created in Ultiboard. Use Multisim to map the schematic symbol pins to the physical pins of the part shape.
6. Draw the schematic for your circuit in Multisim. Don't forget the connectors!
7. "Transfer" the design to Ultiboard
8. Connect the pins of the PCB land pattern with traces. You decide on the trace width and shape. You decide on the board shape
9. "Export" your Ultiboard drawing to create Gerber Photo plotter files
10. Send these files to OSH park for fabrication
11. Solder the parts onto the PCB's. For small parts, use a binocular microscope. tweezers, and fine solder.

PHYSICAL ELECTRONIC PARTS THAT YOU DON'T HAVE TO DRAW (1)

- THROUGH HOLE
  - Resistors… 1/8w, 1/4w, 1/2w are all usually standard
  - Some capacitors have leads spacing of .1 or .2 inches, fairly standard
  - Integrated circuits in the DIP (dual inline package) are standard size
  - Transistors come in fairly standard packages.
- SURFACE MOUNT
  - A bunch of two terminal parts are rectangular surface mount packages described by there length and width in mils:
    - 0805= .08 in by .05 in
    - 1206=0.125 in × 0.06 in

PHYSICAL ELECTRONIC PARTS THAT YOU DON'T HAVE TO DRAW (2)

- Integrated circuits, transistors have some standard packages and package names
    - SOIC
    - SOT-23-5
- Proprietary integrated circuits that are listed in the Multisim component list:
    - Example: LT1179MJ from Linear technology.
        - You can trust that the schematic symbol and PCB footprint for these parts are correct.

WARNING

- BE VERY CAREFUL…
- YOU MAY BE DISSAPOINTED WITH ANY PACKAGE SHAPE THAT YOU DON'T DRAW YOURSELF
- Parts with similar footprint names from different manufacturers often are subtlety different. This is more true for the smallest, most dense of the surface mount packages.

EASIEST PRINTED CIRCUITS TO MAKE AT THE UNIVERSITY OF IOWA

- TWO alternatives:
  - THROUGH HOLE PARTS: using dip (dual inline package) IC's, leaded resistors and capacitors
    - VERY LARGE PARTS. Takes a long time to thread the leads through the holes.
    - Easiest to solder
    - Most of the new IC's are not available in through hole packages. You can't use modern parts.
    - Through hole parts are expensive!
  - SURFACE MOUNT PARTS using the largest packages: SOIC, 1206 or 0805 passives
    - Easy to solder
    - Very inexpensive
    - For small circuits, easy to design a board that uses only top copper… can be fabricated by the electronics shop

CIRCUIT DESIGN STEPS

These are electronic parts distributors that deliver through the mail. The digi-key service is best. Their web site is the best way to find available, affordable electronic parts that meet your specifications

- Draw a schematic to design the electronic performance
- Analyze the circuit to make sure it is ok
  - Use some SPICE (LTSpice is the best, can try using Multisim) to verify circuits
- Find parts that are available and work using Digi-Key or Mouser.
- For each part, make the PCB model in ULTIBOARD.
- For each part, make the schematic model in MULTISIM
- Draw the schematic in MULTISIM, using actual packages, pin numbers of the parts that you intend to use. Include connectors.
- Transfer the schematic to ULTIBOARD for layout.

PART 2:
DETAILS ABOUT USING THE SCHEMATIC DRAWING PROGRAM AND THE PRINTED CIRCUIT LAYOUT PROGRAM: HOW TO MAKE CUSTOM PARTS.

ULTIBOARD PACKAGE DEFINITION

ULTIBOARD WAS WRITTEN SO THERE ARE THREE DATABASES:

MASTER ---- YOU CAN'T EDIT OR CHANGE ANYTHING IN THIS DATABASE

CORPORATE --- CONTROLLED BY JOHN IN THE ELECTRONICS SHOP

USER --- WHERE YOU CAN DRAW YOUR OWN PARTS.

TO MODIFY THE MASTER DATABASE PARTS, COPY THEM TO THE USER DATABASE, FOOL WITH THEM THERE.

REMEMBER THE MASTER DATABASE IS READ ONLY, SO EDITING PARTS DOES NOT WORK

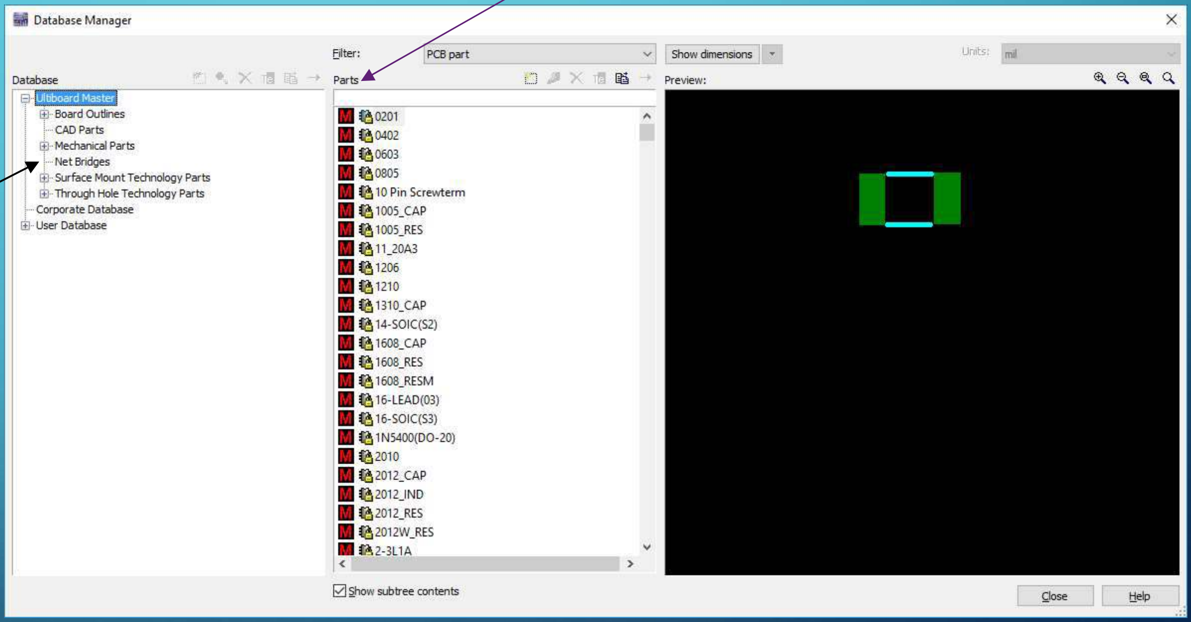
ULTIBOARD database manager

ULTIBOARD DATABASE– THE PARTS THAT ARE ALREADY THERE

This column, marked “parts” contains the “package”

To keep the database organized, Ultiboard allows you make up “groups”… arbitrarily named tree structure for the parts. The master database has a four level tree, with 6 branches at the first level.

You can copy this structure or make your own.



HOW DO YOU KNOW IF A SUITABLE PACKAGE IS ALREADY IN THE ULTIBOARD DATABASE?

- THERE IS NO EASY WAY ☺
  - Yes, it IS hard to believe.
  - Searching through the database is a pain… also, it is difficult to see the dimensions of the footprint so that you can tell if it is what you want. (often dimensions shown on the database picture are not the same as the manufacture's drawing, so you have to do some math to verify that the parts are the same)

THERE IS AN ORGANIZATION THAT TRIES TO MAKE PCB DESIGN AND MANUFACTURE STANDARD

- **Association Connecting Electronics Industries** or IPC
- IPC standards and publications are the starting point for ENGINEERING of circuit boards

IPC-7351 IS THE STANDARDS DOCUMENT FOR SURFACE MOUNT PACKAGES (WITH FOOTPRINTS)
IPC-7251 IS THE STANDARDS DOCUMENT FOR THROUGH HOLE PARTS

- There is an IPC naming convention….example for Surface mount component
  - Example: CAPC0402X20N This is a non-polarized chip capacitor that is 4mm by 2mm by 20mm high
- Example for through hole component
  - CAPAD800W52L600D150B This is a non-polarized axial diameter horizontal mounting lead spacing 8mm, lead width .52mm, body length 6mm, body diameter 1.50mm
- There are some packages (and some footprints) in ultiboard that use the IPC naming convention. This may be useful in finding parts (but, its not very)

HOW TO CREATE PARTS IN ULTIBOARD, OR "DRAWING THE PACKAGE"

- Defines the physical dimensions of the part that you are placing on the board
- Defines the "footprint", the copper, plastic, solder mask, silkscreen layers that are needed to manufacture the circuit board.
- ALMOST ALL PHYSICAL PARTS HAVE THE "PINS" (the electrical connection points) LABELED WITH NUMBERS, although sometimes you will have letters.

ASIDE: (IN ADDITION TO PCB PARTS, THERE ARE THREE OTHER KINDS OF THINGS IN THE ULTIBOARD DATABASE)

- **Net Bridge:** A copper shape that has exactly two pins used to connect two nets, merging them electrically without creating an error.
- **Custom pad shape:** a named shape that can be used as a pad when defining a part
- **CAD part:** A 2d drawing, not specific to a PCB program, probably not really useful for anything in particular.

WHAT MAKES A "PART" IN ULTIBOARD: 5 THINGS

1. PADS…copper areas associated with a "pin" (an electrical connection to the outside world)
   - Pads include the **copper** (on all layers), the **solder mask**, the **paste mask**, the **drill hole info**
2. Attributes of the PADS.. Most important is the "NUMBER", identifies which pin is which on the package drawing
3. Silkscreen.. Paint that is printed on top of the solder mask to help with the assembly of the PCB
4. 3d data… used to visualize the package, a special feature of Ultiboard
5. Attributes of the PART (that is, the whole package). Most important is the "REFDES" (a letter, followed by a number, used in both the schematic and the layout to identify the part for assembly. Can be made automatically a part of the silkscreen

- You can also add other stuff to the part, including copper areas that are not "pads" (this means they aren't meant to electrically connect to things).

ALL THE THINGS ON THE PREVIOUS SLIDE CAN BE CHANGED USING THE "DATABASE MANAGER" IN ULTIBOARD

- Start Ultiboard
- Tools->database->database manager (don't forget.. Only user database can be edited, this tool only allows you to view the part in the master database.
- In the PARTS section, in the middle, there are Five choices Create, Edit, Delete, Rename, Copy
  - You can Create the 4 types of database things, (net bridge, pad shape, pcb part, cad part)
  - The create button takes you to a drawing screen, depending on the type of thing you are drawing. The 4 screens look the similar, but are used for different purposes. Some of the drawing screens have options that are not relevant to the task at hand. This is confusing: it is simply poor software design.

DRAWING A PCB PART IN THE PART EDITOR

1.Pads

1. Place->pins allows you to put down surface mount or through hole pads.
   - Notice the name "pins", mixed with the name "pads". Means the same thing here.
   - By changing the "properties", can edit the pads in many ways… replacing them, changing their size, changing their attributes (giving them a new number,) and so on. Notice the attribute "Number" changes the label "Name" on the pin. (inconsistent terms)

2.Attributes of the Pads

3. Silkscreen

2. Draw on particular layers
   - Silk screen– draw the outline of the part, other stuff you want
   - 3d info- draw several 2d shapes on this layer
     - Then EDIT->properties, (components and sheets dialog), 3D tab

4. 3d info

5. Attributes of the part

3. Change the "Attributes" of the part
   - Edit->properties with nothing selected
   - There are 3 "Attributes" of the part/sheet that are created automatically:
     - Footprint .. This is the name of what you are creating here. Looks like it is automatically populated when you save the file.
     - Refdes.. This is the kind of part (one letter), and the part number, concatenated
     - Value.. This is the value of passive parts, or the name of IC's

USE THE ULTIBOARD PART WIZARD

- Does some stuff automatically
- Drops you into the PCB Part drawing program at the end… so you can change anything you want after you go through the wizard
- Alternative is to edit an existing part, (with database manager)

NOW WE HAVE A ULTIBOARD DRAWING OF THE PART (FOOTPRINT + 3D INFO), WE CAN GO TO MULTISIM

MULTISIM PART DEFINITION IS STRAIGHTFORWARD

ASIDE… (MULTISIM CIRCUIT SIMULATION)

- Multisim is integrated with a simulator, based on XSPICE from Georgia Tech
- XSPICE is an extension of Spice3 from UC Berkley
- In the public domain. Used in very many open source software products
- Powerful and complicated. Has analog (time based differential equation solutions) combined with digital (event driven, discrete value (LOW,HIGH, TRISTATE), solutions)
- Models can be added without recompiling the source.
- National Instruments does not offer real support to its educational users, you are on your own.

A "COMPONENT" IN MULTISIM

1. There is a tree based index in the database: called "Groups" and "Family".
   - Seems to be an arbitrary arrangement. You can add and delete these things, and use them in searches. You may want to adopt what the Multisim people have done, or use your own index for the parts you create
   - Note that a family has a default Reference Designator letter for parts you put in that family
   - There are many other fields in the database : Function, component, manufacturer, and so on. You can fill in these fields when you create your part, if you want. In some dialog boxes, you can use these extra fields to search or filter results.

A WHAT YOU SPECIFY FOR MULTISIM "COMPONENT" (4 THINGS)

1. You draw the schematic symbol
   - completely up to you: important thing is the number of pins.
   - schematic symbols can be drawn for multi-section packages (several symbols on the schematic refer to the same PCB footprint).
2. You identify the package from Ultiboard for the part
3. You map the schematic symbol pins to the footprint pins

You make the spice model and associate the spice model with the schematic pins

You can define pin properties for design rule checks (inputs, outputs, n/c, etc.)

4. You save it to a group and family

TOOLS->database->database manager (REMEMBER, YOU CANNOT EDIT THE MASTER DATABASE)
Copy parts from the master to the user database if you want to edit them.

DEFINING THE PART IN MULTISIM… HOW DO YOU FIND THE PART IN ULTIBOARD THAT IS THE RIGHT FOOTPRINT?

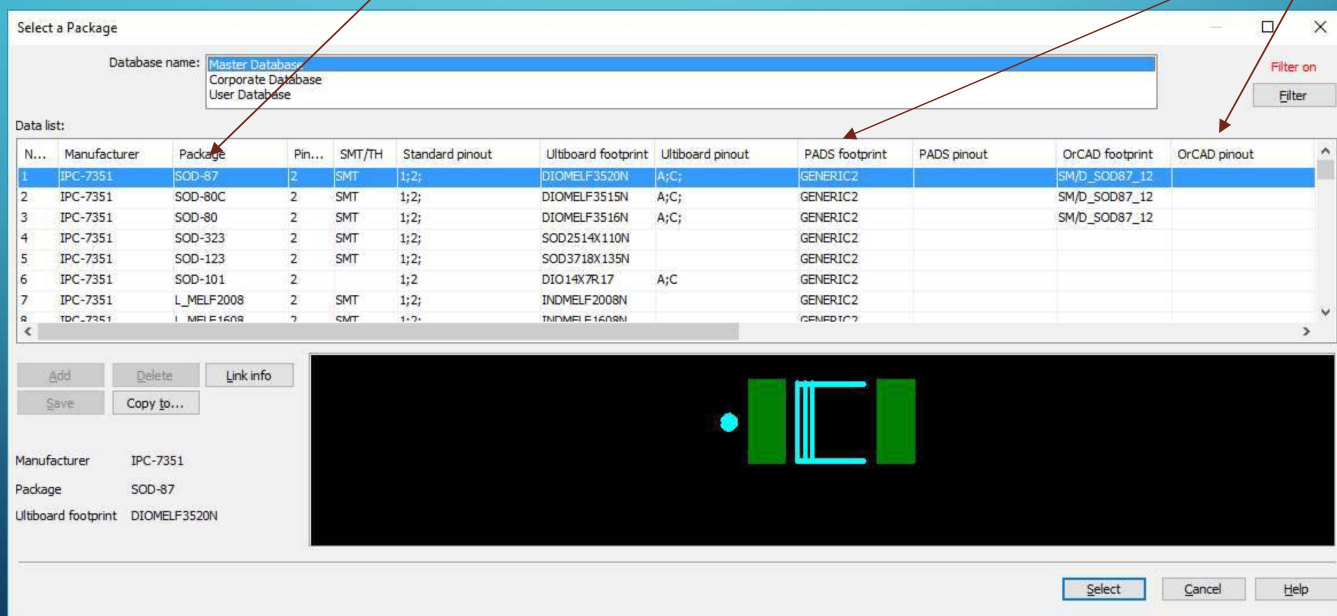
- Again, this is very difficult to do
  - Perhaps there is a footprint in the ultiboard Master database that is the correct one, or Perhaps there is one that is close, but not quite right
  - The only way to be sure is to DRAW YOUR OWN, then remember the name you have given it so you can enter it into the dialog in Multisim.
  - You are given a search dialog, to find the footprint in the Ultiboard database, (shown on the next slide).
  - Unfortunately, there are no dimensions shown in this search dialog, so it is easy to choose a footprint that looks OK, but is the wrong size.

MULTISIM "SELECT A PACKAGE" WHEN EDITING A PART

ULTIBOARD DATABASE : "PACKAGE" IS THE FIELD THAT YOU SHOULD LOOK AT

Package name is what the Ultiboard database is set up around

Other fields in database are not defined in any public document



USE THE MULTISIM PART WIZARD

- Does some stuff automatically
- Often, the way you will want to create parts
- Alternative is to edit an existing part, (with database manager), fairly easy to do also.

IN PLACE PART EDITING IN MULTISIM AND ULTIBOARD, AND EDITING THE DATABASE AFTER PLACING A PART OR COMPONENT

- When you edit the footprint or model of a component placed on the schematic, your changes will only affect that instance of the component, even if there are other instances of the same component on the schematic and even if they were copied from one another.
- If you want to make "permanent" changes to the component stored in your database, you can open the component properties and click "Edit component in DB". Any changes you make there will be saved in your database and will be reflected in any instances of the component you place in the future. To update instances of the component already placed on the schematic after making changes in the database, you can use the Update Components tool, founder under Tools>Update components...

SUMMARY: WHAT YOU NEED TO DO TO MAKE CIRCUIT BOARDS

CIRCUIT DESIGN STEPS

- Draw a schematic to design the electronic performance
- Analyze the circuit to make sure it is ok
  - Use some SPICE (LTSpice is the best, can try using Multisim) to verify circuits
- Find parts that are available and work using Digi-Key or Mouser.
- For each part, make the PCB model in ULTIBOARD.
- For each part, make the schematic model in MULTISIM
- Draw the schematic in MULTISIM, using actual packages, pin numbers of the parts that you intend to use. Include connectors.
- Transfer the schematic to ULTIBOARD for layout.

AN ATTRACTIVE WAY TO DO THINGS, IF YOU CAN MANAGE IT.

- In your electronic design, use only parts that are in the Multisim database.
  - verify that you can buy these parts, (in the package in the Multisim-Ultiboard database)
    - Practical options are Digi-key, Mouser, Electronics Shop.
  - Advice: Use surface mount 1206 or 0805 passives and SOIC packages for the IC's.
  - Advice: For connectors use "headers-test" , which are standard "0.1" headers.
- If you can achieve your design goals with these limitations, you are in great shape.
  - Don't need to create parts in Multisim or Ultiboard
  - Simulation will probably work
  - Place the parts on the circuit board, connect them up, and you are done.

HOWEVER, IT MAY TAKE YOU MORE TIME TO FOLLOW THE PREVIOUS INSTRUCTIONS THAN TO CREATE CUSTOM PARTS IN MULTISIM-ULTIBOARD

- And, often, you want to use a special part (perhaps a module from Ebay or Amazon) that has no footprint, schematic symbol, or model in the CAD program.
- Making the schematic symbol and the package footprint does not really take much time.