Electrical and Computer Engineering

Electrical and Computer Engineering

Providing quality educational opportunities for students through teaching, research and professional service.

Welcome

The Electrical and Computer Engineering Department offers high quality

Degrees Offered

We offer B.S., M.S. and Ph. D. degrees in Electrical Engineering and
Electrical and Computer Engineering

ECE Equipment Inventory

List of ECE equipment as of 3/1/2019.

ECE Parts List

List of parts available in the ECE department.
Component Request Form
North Dakota State University
Electrical & Computer Engineering Department

Name: ____________________________ ECE Course #: ____________________________
E-mail: ____________________________ Or Group #: ____________________________
Date: ____________________________ Or Funding Info (if known): ____________________________

Authorizing Signature: ____________________________

CHECK ONE
(Use ONE order form for each distributor.)

☐ ECE STOCK
☐ Allied
☐ Amazon.com
☐ DigiKey
☐ Electronix Express
☐ JameCo Electronics
☐ Marlin P. Jones
☐ Mouser
☐ Newark

Other Vendor Name: ____________________________
Web Address (if applicable): ____________________________

**Note: the ECE Stock Location can be found at [http://www.ndsu.edu/ce/ece/parts_inventory.pdf](http://www.ndsu.edu/ce/ece/parts_inventory.pdf)

<table>
<thead>
<tr>
<th>QTY</th>
<th>**ECE Stock Location</th>
<th>ECE Stock or Part #</th>
<th>Description</th>
<th>Price/Unit</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

ECE Office Use Only
Electronics Design for Anyone and Everyone

Meet Upverter, the Free, easy-to-use electronic design tool that helps you bring your unique hardware ideas to life, no matter your engineering knowledge.

Start Designing Your Electronics Online with Speed and Ease
Start from scratch, upload an existing design, or choose from thousands of community designs to customize any way you want.
By using Upverter you will create gerbers, gerber files are used to create pcb’s. This is a typical pcb.
Why use Upverter?

In the near future you may or may not be using Upverter. There are many PCB generating programs available. It all depends on the application.

For Schematic Design and PCB Design, you may choose Multi-Sim (very good program and is heavily used by Engineers, It is also expensive and has a large learning curve. Digi-key has pcbartist, there is ki-cad, design-spark and many more.

Upverter.com was chosen as it is web-based, its free, and has an easier learning curve than others, plus it has a collaboration option that works quite well when you team-up on a project.
UPVERTER

*Power Point to aid in the creation of a PCB*

- Slides 1-12: Upverter Information
- Slides 13-24: Create the Schematic
- Slides 25-41: Create the PCB
- Slides 42-50: Create the Gerber’s
- Slides 51-53: Online Gerber-Viewer
- Slides 57-60: Order Procedures from Osh Park
- Slides 61-67: For a more advanced PCB using Arduino Atmega328P

ECE401
Upverter is web based

- Create your own account, password protected

*Collaboration is possible.* This makes it easy to work on a project together

Some problems- it is web based- Upverter does not like Internet Explorer, works ok with Mozilla, but seems to work best with **Google Chrome**

Also you can collaborate with different log-ins but you cannot be logged in to two different computers at the same time with the same log-in ID and password. (It doesn’t like collaborating with itself).

- go to upverter.com, and Sign up. Username is email, password and then maybe collaborate after you pick a partner
https://www.youtube.com/watch?v=QZbE-gGRtD4

This was very a useful website for getting started. There is a Part 1 for making the schematic and Part 2 for making the PCB. If you take the twenty minutes and work through it, you will find it rewarding.
After an account has been created, this is what you will see.

**Getting Started with Upverter**

**Your Profile**

![Upverter Dashboard](https://example.com/dashboard)

Figure 1: Upverter Dashboard.

Your Upverter experience begins with your profile page. On it you will find your projects, be able to search Upverter’s parts and projects, and connect with the Upverter community. Since Upverter is built for both professionals and individuals, we use what are called contexts to separate the workflow between...
After going to Upverter.com and creating an account

Create a New Project

- Create a New project

>Project

> New
User Manual is very handy and easy to get at, open from Lower Left corner of page
Adding Parts to the schematic. Verified vs. Generic
When creating schematic, do you have any issues like this? (Highlighted in Gold?)
Nets have to be named. Double click on the gold, this box shows up, and name your NET
Gold outline is now gone and circuit is finished- time to convert the Schematic into a CAM File
I started with a Microcontroller- using the Arduino ATmega328-PU IC, which is a 28 Pin DIP Package IC.
As you can tell, this schematic is different. This is a schematic of a 555 timer that blinks an LED, after schematic is finished, the next step is to create the PCB. Press “Q” to jump back and forth from Schematic to PCB.
In creating a schematic, you need to use verified parts. If on the schematic there is no checkmark, the parts will not transfer over to the Bill of Materials list of the PCB Editor. When installing a part, press “Q” and it will tab between schematic and PCB.
Remember only the Verified parts will work in creating a PCB
Reminder of all the tutorials available online and on YouTube.
Go to this tutorial to see how to change your schematic into a pcb.

https://www.youtube.com/watch?v=E44qD_H6Ftw
Click PCB Layout. This should show up. The VERIFIED Parts and the NETS (These are connections made in the schematic and altogether is called the Rats nest.)
First step in creating a PCB is the dimension of the PCB. Double click anywhere on the canvas and this shows up.
After filling in dimensions, in this case changing mm to mils, and typing in 2000mils x 2000mils, we create a pcb of 2” x 2”
Go to the Layers tab on the right hand side, scroll down to Mechanical Layer and shut OFF, the square goes away, highlight all the visual components and move down slightly so the ORIGIN cross sits slight above all the visualized components.
After moving all components out of the way, and turning the Mechanical Layer back on.........
If this is what is shown the first step in PCB Fab is complete. Now highlight your parts and insert them in the box.
Always start with your main components and add the rest in accordance with your schematic.
Move components around in straight lines, this also organizes the pcb, so it resembles the layout of the schematic.
Next step is to determine the width of the trace you want to use and start adding traces. Power and ground need wider traces than signal and control traces. Try 20mil.
After adding all the traces (remember 1 and 2) changes from top to bottom layers.
Add some text.
Add whatever is required for text
Using the Pour tool start at the ORIGIN and highlight the outside edge start bottom left/ go up/ to the right/ down/ then start to the left and stop- hit escape. No need to go all the way around 360 degrees. This should show up.
Press 2 - this will be the bottom layer and then using the pour tool start at ORIGIN, highlight the box and hit escape. This will show up.
Add VIAS to connect the top and bottom layers together if required
PC Board is now done
We’ve covered basic UI and schematic design in the first part (http://youtu.be/QZbEg6RtD4) and are now tackling the layout side of the project.
Once finished, you will need to create the Gerber files which are then exported to companies that make PCB’s, such as Advanced Circuits, or in our case we will use OSH Park.com. 
Upverter to Gerber files exported. 

Go to dashboard and press  --------- Project  --------- Export Files

Once seeing this image of your schematic, **scrolling down you will see**…….

Your schematic.....
Scrolling further......
Bill of materials - IF you used verified components - Generics won’t show up

Bill of materials....

Scrolling further.....
Scrolling down you will see this they are to export all files, such as Schematic in PDF format, or Bill of Materials

Export files...

Gerber files are also saved along with NC Drill files, which are then zipped and sent to.....

A Gerberviewer.com

For checking
You may download your schematic as a pdf
Or as a .png file....... Which is a Higher resolution Schematic image.
Download the projects bill of materials as an Excel spreadsheet... convert to pdf... etc.
To generate Gerber files – use the Gerber Format (RS-274X extended) *read directions if required*
It will automatically download these files, which contain all gerbers as well as xln drill files.
It also will compress the files automatically, which are needed to check them for accuracy. Send the zipped files to gerberviewer.com.
Open Demo Design or select Gerber274X, Excellon1/2 or Zip-File:
Choose File: No file chosen

Show format settings
mm
inch

"Online Gerber Viewer" integrated in your local company intranet?
Contact us: info@easylogix.de

PCB Investigator
Professional CAD/CAM Station - Try it now for free!
This is one of the layers that was downloaded from Upverter files.
Shows full PCB with Traces

If you like what you see, your PCB files are ready to order…. Go to.....
After looking at your files using the gerber-viewer.com website or similar, they need to be OK’d by Dr. Maassel or the TA or myself to make sure that all is ok for ordering.

These next slides show how to order with OshPark.com, It works well as you will see an instant picture of what your actually creating.

**ALL emails/orders must have the SD401-F19-XX on it. Nothing will be approved with it.**
JE1	After looking at your files in the
    Jeffrey Erickson, 1/30/2018

JE2	Jeffrey Erickson, 9/4/2019
ALL orders and emails must have this format shown.

SD401-F19-XX for Senior Design ECE401, Fall of 2019 Semester, Group number XX.

This to be included on Parts orders, PCB orders and emails.
To Order: Go to oshpark.com create an account, Name and email address, DO NOT Pay for anything!
You will see this, it gives information about your board. IF this shows up and are agreeable with the price, scroll down and press Continue.
Now you can VERIFY all details of your board. Do not purchase/ files are emailed to Purchase – Jeffrey.Erickson@ndsu.edu
This is the final- but DO NOT ORDER unless it is for yourself- send the files to Jeffrey.Erickson@ndsu.edu
The check mark says it is a Verified part. This is required to transfer to the PCB Design.
Add some components, such as LED’s and current limiting resistors
Insert a battery, in this case BT1, change the 3.3 Volt to 9V
Add your grounds
You will need Data sheets handy, indicating Pin Layouts, for VCC, GNDs, Input and output pins.
Insert a Voltage Regulator such as an LM7805 5 Volt Regulator
The Connections between components, grounds, power, etc. are called Nets. If highlighted in Yellow there is an issue and must be corrected. Usually a naming issue.
Any questions please stop by, but remember to use the Tutorials and Users Manual on Upverter