

AP-S STUDENT DESIGN COMPETITION RADAR BUILD & OPERATION GUIDE

MICHIGAN STATE UNIVERSITY

Jason Merlo, Stavros Vakalis, Cory Hilton, and Jacob Randall

1 Requirements

To build the Michigan State University AP-S Student Design Competition frequency-modulated continuous-wave (FMCW) radar board the parts listed in the bill of materials (BOM) found at the following URL will be required:

https://docs.google.com/spreadsheets/d/14kPDrza_Lf1kQ5H1xRQW3g6EeMugPi_6xZNXxwFOA4/

In addition recommended common lab tools and equipment are specified on the "Recommended Tools" page.

A laptop running Windows 10 with at least a 3rd generation i5 and 8 GB of RAM is recommended for real-time processing. Offline post-processing is supported by macOS, Windows, and Linux.

2 Radar Build Guide

The radar consists primarily of four main components: the radar printed circuit board (PCB), the USB data acquisition (DAQ) system, the Texas Instruments (TI) USB2ANY programmer, and the 12V power source. This guide will focus primarily on the assembly of the radar PCB as this will account for the majority of the time required to build this radar system.

Resources which may be helpful during the build of the board are:

1. Device schematics and PCB layout (Kicad format): <https://gitlab.msu.edu/delta/aps2020-competition/ap-s-2020-hardware>

Kicad can be acquired for free from: <https://kicad-pcb.org/>

2. Radar PCB Revision 1.0 Errata:

https://docs.google.com/document/d/1ynecgIvvKLD0n6N_bCJ5Vmr3iKfqa4PLDqKBhAX1aMA/

2.1 Radar PCB Assembly

2.1.1 Preparation

Begin by preparing an electrostatic discharge (ESD) safe working area with a static dissipative mat and a grounded wrist strap with a high-resistance to ground ($> 1\text{ M}\Omega$) to ensure the sensitive integrated circuits (ICs) are not damaged during handling.

Once the workspace is prepared, the assembly can begin. It may be helpful to sort the parts by passive and active devices, and through-hole devices to make the process faster; once the solder-paste is applied to the board, the board should typically be reflowed within 4-8 hours for best results (the solder paste manufacturer will usually specify the stencil life in hours in the datasheet).

2.1.2 Solder Paste Application

1. Begin by cleaning a blank radar PCB with a low-lint cloth and rubbing alcohol to remove any chemicals or oils from manufacturing and handling that may cause the solder to not adhere to the pads during reflow or otherwise cause the board to heat unevenly.

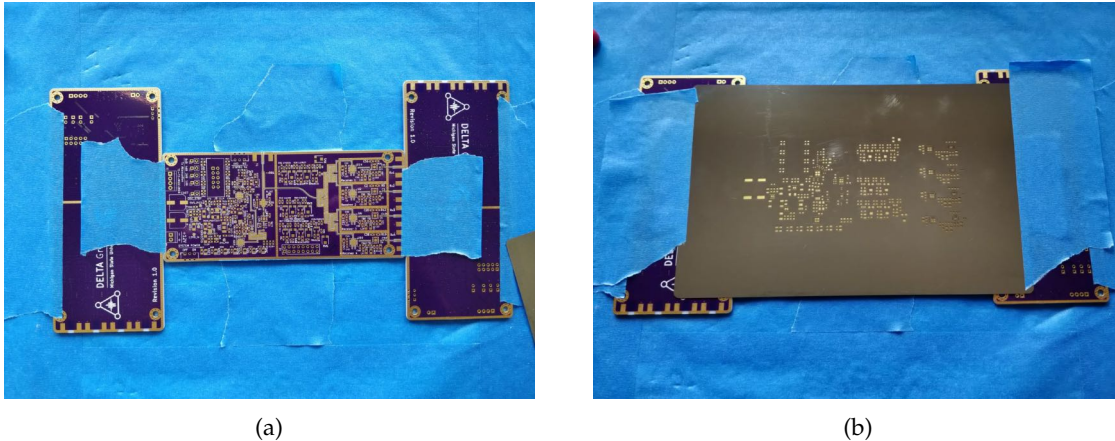


Figure 1: (a) PCB taped into place; (b) stencil taped over PCB for paste application.

2. Lay one radar PCB pad-side up on the work-surface and using spare PCBs of the same thickness (1.6 mm) place the two other PCBs on either side as shown in Fig. 1a. Taping the spare boards on either side of the board to be soldered will allow the solder stencil to lay flat while applying the paste.
3. Next, lay the stencil flat over the board and align all the slots such that only the gold pads can be seen through any of the slots. Take care to ensure all the fine-pitch IC pads are aligned properly. Once this is completed, tape the stencil down.

TIP: To avoid moving the stencil while taping it, apply masking tape to the edges of the stencil before aligning but leave the edges curled up so they don't stick initially, then once the stencil is aligned press the tape down. It may be helpful to fold the edge over on the tape on one side of the stencil to allow it to be removed easily and avoid smearing the solder paste on removal.

4. Next, apply the solder paste. Start by applying a small mound of solder paste at one end of the board (start on one of the short edges) and spread the mound into a line the full height of the short edge. Next, using the putty knife or squeegee, hold the blade at roughly a 35° angle and applying enough force to not allow the paste to slide under the blade, and push the solder paste across the board from one end to the other ensuring the paste evenly applies to all holes in the stencil. If some spots were missed, push the paste back over to the other side of the board until the solder paste is applied evenly to all locations on the stencil.
5. Once the paste is applied, remove the stencil.

TIP: To avoid smearing the paste, leave one edge of the solder stencil taped and carefully lift from the untaped edge.

TIP: If there is too little or no solder paste applied to a pad, use a fine-tipped solder paste syringe to apply a small amount of paste to the pad.

NOTE: If the solder paste is smeared or misaligned, it must be reapplied. This can be done by wiping the solder paste off with a low-lint cloth, then cleaning all the remaining paste off the board with isopropyl alcohol on a clean low-lint cloth. Once the board is cleaned, repeat section 2.1.2.

2.1.3 Surface Mount Component Placement

Once the solder mask is applied, the surface mount parts can be applied in preparation for reflow.

1. Start by placing all the ICs on the board. It is recommended to place the quad flat no-lead (QFN) packages first as these are the most prone to misalignment. Pay close attention to the orientation of the QFN parts to ensure their rotation is correct. It may help to have the PCB layout open while placing these parts.

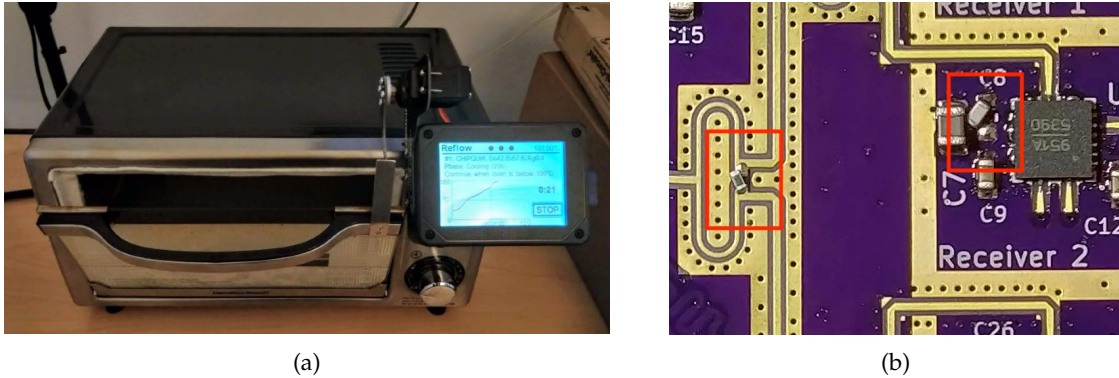


Figure 2: (a) Reflow oven with door open during cool-down period after reflow; (b) rotated surface-mount components after reflow.

2. Once all the ICs have been placed, place the remaining surface mount components (except the SMA connectors).

NOTE: View the errata listed above for any modifications to make while assembling the board (do not add jumper wires until after the reflow).

TIP: After placing each component it may be helpful to apply a gentle tap on the top of the component to make sure it is sitting evenly on the solder paste to avoid tombstoning.

TIP: Do not worry too much about aligning parts perfectly as they will slide into place as the solder paste melts during the reflow so long as they are not misaligned by more than half a pad-length.

2.1.4 Solder Reflow

Once all the surface mount parts have been placed, the board may be reflowed.

1. Begin by programming the reflow oven to match the manufacturer recommended solder paste heating profile. If using an oven with a Controleo as seen in Fig. 2a and ChipQuik-Sn42-Bi57.6-Ag0.4 solder paste, a profile created for this project can be found at:

https://drive.google.com/file/d/13rWLObJVVQnc_b_KsGbFKG-Ek4mt1dPP/

NOTE: Test the solder profile with an unused blank radar PCB as a dummy thermal mass to ensure the oven tracks the heating profile recommended by the solder paste manufacturer reasonably well. If changes are needed, modify the profile.

2. Once the profile is set, load the board with the parts added into the oven and run the profile.
3. Once the profile is finished and the board is cool enough to remove, inspect the PCB for any components that may have drifted or tombstoned during the reflow process; two components rotated during reflow of the board for this project shown in Fig. 2b – these can be easily fixed with a precision hot-air tool.
4. Carefully inspect the edges of all QFN chips and ICs for any shorts or excess solder. If shorts or excess solder is identified, use liquid flux with copper braid and a soldering iron to remove any excess. Once completed, remove flux residue with isopropyl alcohol and a clean low-lint cloth.

NOTE: Once the reflow process is completed, jumper wires may be added per the errata. Solid core 30 AWG jumper wire was used for this project. For reference on adding jumper wires, follow: https://workmanship.nasa.gov/lib/insp/2%20books/links/sections/302_jumper%20wires.html

2.1.5 Through-hole Parts and Connectors

1. Once jumpers are installed, the through-hole parts can be soldered to the board by hand.



Figure 3: (a) Fully assembled radar PCB; (b) radar PCB integrated with power supply, DAQ, and TI USB2ANY programmer.

2. Once all through-hole components are added, solder the SMA connectors. For best results, solder the ground pins on the top and bottom side by hand using a soldering iron on high heat ($\sim 400\text{ }^{\circ}\text{C}$) with a medium-sized chisel-tip. Next, use a fine-tipped solder-paste syringe to apply solder paste down both sides of the center conductor pins and reflow using a precision hot-air tool.

Once all connectors have been added, the board will be complete and should appear as in Fig. 3a.

2.2 Antennas

The patch antenna arrays were fabricated on a Rogers 4350B two-layer substrate with 0.5 oz copper and a thickness of 1.524 mm. The antennas for this project were etched using a spin coating and etching process by our laboratory technician Brian Wright and an SMA connector was soldered onto each of the antennas once the etching process was completed.

2.3 System Configuration

To complete the radar system an external power supply, DAQ, and programmer must be used as is seen in the system in Fig. 3b.

1. The 12 V (10 V–13.8 V) power supply is connected through screw terminal J3 (take care to ensure proper polarity).
2. The analog inputs of the DAQ are connected to J2 near the middle of the board where pin 1 is marked by the 'L' shaped marker on the board silk screen and is typically marked on the insulation-displacement connector (IDC) cable in a separate color from the rest of the wires. The trigger input of the DAQ should be connected to the "MUX" pin on J1 to trigger the DAQ on the start of each new chirp from the transmitter.
3. The programmer is connected to the keyed IDC socket J9. This is only required to be connected while programming the transmitter, however the transmitter will need to be re-programmed each time the system is powered on as the transmitter has no non-volatile memory. The transmitter configuration used for this project can be found here:

<https://drive.google.com/file/d/1xWyCc04NwXAJl2SZQyp9YRtt0Kpb-MET/>

NOTE: To mount the board M3 mounting screws and standoffs may be used.

NOTE: Heat sinks may be desired as the radio frequency (RF) receiver front-ends will become very warm during operation.

3 Software Configuration

The software for this project was written using Python and Qt to build a platform independent real-time graphical user interface. The software was segmented into two primary components: the AP-S Dashboard and the Python Radar Toolkit (PyRaTk).

NOTE: PyRaTk is a standalone Python software library being developed by the distributed electromagnetic theory and application (DELTA) group at Michigan State University as a collection of tools for data-logging, playback, visualization, and processing of radar systems. At the time of writing it is still in a pre-beta state and is provided as-is.

TIP: It is recommended to use Anaconda for managing virtual Python environments when developing complex software. This tutorial will assume that an Anaconda prompt is installed on the desired system. The installer can be found at: <https://docs.conda.io/en/latest/miniconda.html>

NOTE: Ensure that conda environment contains Conda Forge as a source. If using an Intel CPU consider also adding the Intel source. To do so run `conda config --add channels conda-forge intel`

3.1 Radar Dashboard

1. Begin by creating a folder for the project and cloning both the PyRaTk and the AP-S Dashboard projects into the folder:

```
$ git clone https://gitlab.msu.edu/delta/aps2020-competition/ap-s-2020-software/  
ap-s-2020-pyratk
```

```
$ git clone https://gitlab.msu.edu/delta/aps2020-competition/ap-s-2020-software/  
aps_dashboard.git
```

2. Next create a conda environment from the `requirements.txt` file in the `aps_dashboard` file and activate the environment.

```
$ conda create --name radar --file aps_dashboard/requirements.txt
```

```
$ conda activate aps_radar
```

3. Once the environment is created, the software may be run. Simply install the PyRaTk and run the `aps_dashboard.py`:

```
$ pip install python_radar_toolkit && python aps_dashboard/aps_dashboard.py
```

This will open a dashboard window. On the left side there will be controls to load and save a database as well as save and play back new datasets inside the database. When cloning the repository, an `.hdf5` database will be included with the data collected during this project which may be used to playback without a radar.

TIP: The DAQ and the playback can be paused and resumed by pressing the space-bar key. The current recording or playback can be reset by pressing the 'R' key. Playback can be stepped forward by pausing, then tapping the '→' key. The program may be exited by tapping the 'esc' key.

3.2 Transmitter Programming

Due to the lack of on-board nonvolatile memory, the transmitter must be reprogrammed each time the power is reapplied to the system. To do so, it is easiest to use the Texas Instruments Clocks and Synthesizers (TICS) PRO software which provides the user a user interface to set the transmitter parameters and writes the registers accordingly. This software can be found at: <https://www.ti.com/tool/TICS-PRO-SW>. The transmitter configuration files used for this project can be found at: <https://drive.google.com/drive/folders/131e7e0wjbTGUE411TbyUZfJWKJl0so01>.