ECE453 Lab 5: FM Quadrature Demodulation / PCB Design Using Eagle

In this lab, you will work with your partner to design a printed circuit board for a quadrature demodulator IC and supporting components. The PCB will be designed using Eagle, a popular and low-cost program for circuit board design.

The PCB you design will be fabricated and returned to you in Lab 9, in which you will construct and evaluate the complete demodulator circuit. The quadrature demodulator circuit will then be used in a complete FM radio receiver system.

## Background Information

FM Quadrature Demodulation

For a more in-depth look at the theory behind quadrature demodulation of FM signals, review section 4.5 of the course notes or consult the supplemental lecture slides.

NJM14570 IC

The NJM14570 is an IC that, together with a number of supporting components, implements a quadrature demodulator. The IC contains an amplifier, a mixer, and an internal capacitance C'. A parallel RLC network must be separately provided in order to build the quadrature demodulator voltage divider.

Receiver performance is highly sensitive to the center frequency and quality factor of the parallel resonant circuit included in the quadrature demodulator voltage divider. Due to uncontrollable variations in component values, it is often necessary to include tunable components. Designs may include variable capacitive or inductive elements. In our circuit, a tunable circuit element called a trimming capacitor is used in parallel with a fixed capacitor to supply a total capacitance that can be varied about a central value.

Besides the supporting components described previously, a number of other peripheral components are required. A first order LPF is connected to the output of the IC to remove undesired frequencies generated by the mixer. Other components must be included for power supply conditioning. The complete circuit (with example component values) is shown below.
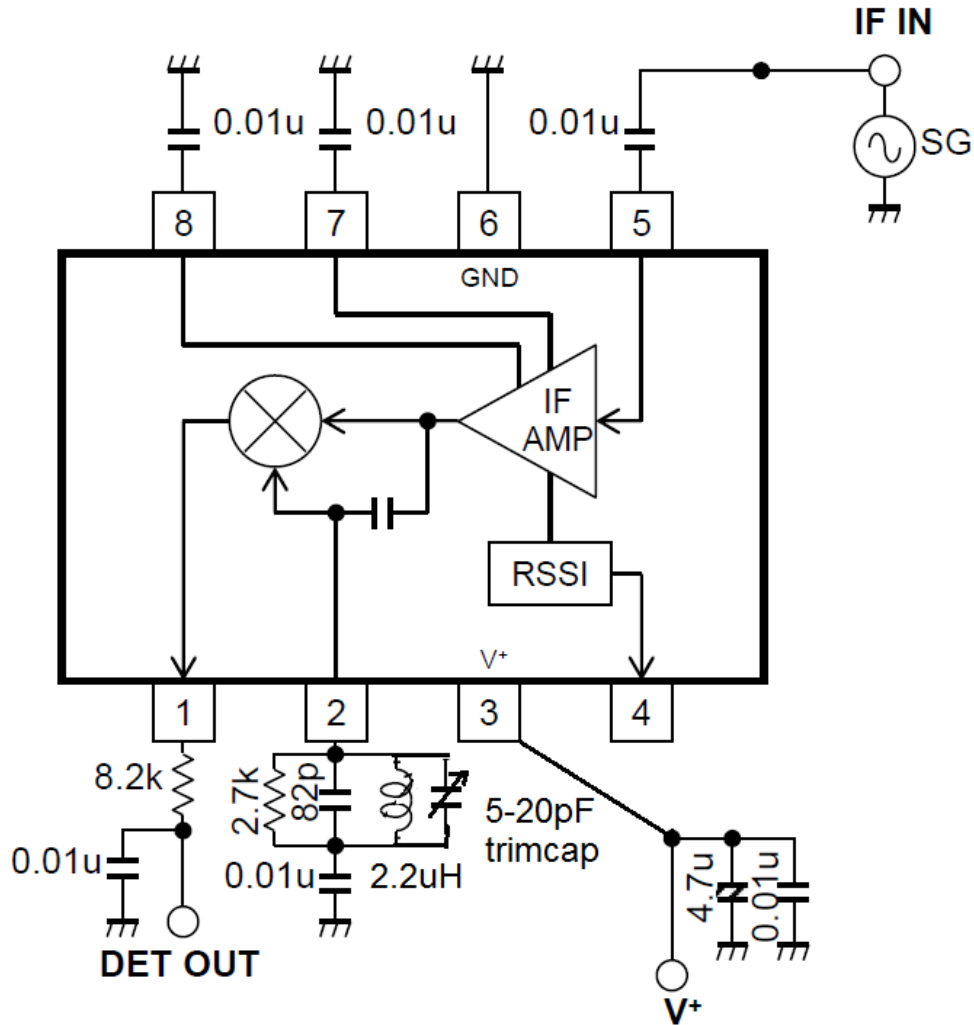
10.7 MHz FM IF demodulation circuit, based on NJM14570 demod IC

Surface Mount Technology

Like many integrated circuits, the FM demodulator IC is available only in a surface mount package. Surface mount technology provides improved RF performance and reliability, and minimizes cost in high-volume manufacturing applications. However, construction of prototypes becomes a more involved process. Instead of wiring through-hole component leads together directly, SMT components are soldered to a custom-designed printed circuit board that includes etched or milled traces. CAD programs have been developed to simplify the board design process. In this week's lab, you will use Eagle PCB design software to build a PCB for the FM demodulator circuit presented above.

Design Requirements

Board should be rectangular, with max dimensions: 2.5" by 1.25"

The filename for your group's board will be indicated by your TA. (*example: ABA01.brd*). The same unique designator (*ABA01*) should be included on the silkscreen layer of your board design.

Two-layer design

No traces narrower than 6 mils (10 mils recommended)

No holes narrower than 13 mils; no vias narrower than 20 mils. (39 mil via drill size recommended)


Design Suggestions

The Eagle program should be preinstalled on EWS machines, including the ones in the RF systems lab.

It is not necessary to create a schematic before editing a PCB in Eagle, and for smaller boards like this one, a careful designer may save time by adding component footprints to a blank board outline directly.

Maintaining regular component spacing and alignment is easier when the grid is enabled. To turn the grid on: View -> Grid.

In the *add components* menu, select the following and add them to your design, in appropriate quantities and at appropriate locations. Press tab to rotate object. The following footprints have been found to work well:

BOM part	Part name (library)
resistors	R0805 (rcl)
fixed capacitors	C0805 (rcl)
MuRata trimming cap, 5-20pF	C1808 (eagle-ltspice)
2.2 uH inductor	C1210K (rcl)
NJM14570 FM demod IC	MSOP8 (atmel)
3.5mm audio jack	1503_02 (con-lumberg)
SMA 50ohm end launch jack	J502-ND-142-0701-801/806 (con-coax)

8 7 6 5

E$1

1 2 3 4

GND

Signal

GND

E$5
board edge

Ch1   Ch2

Ch1   GND

board   E$4
edge
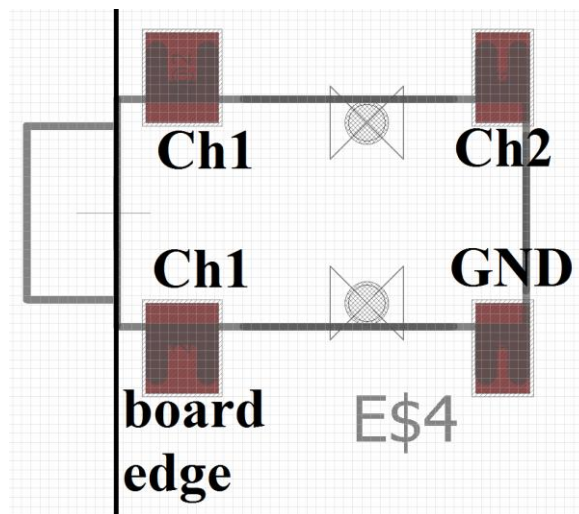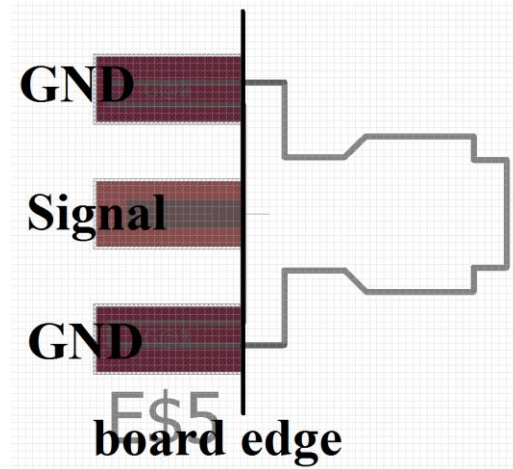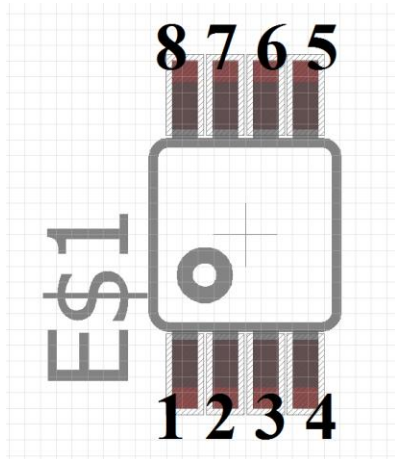
The outer dimensions of the board are defined by the board outline

    -set the board outline using the "wire" tool – a rectangular shape is easiest to work with

    -lines defining the board outline should be 6 mil or wider

    -ensure that your SMA and headphone jack components are aligned at the board edge

    -SMA connector pads should touch the board outline

Connect component pins with traces on the PCB using the Wire tool

    -select the appropriate layer, top or bottom

    -set trace width (at least 0.01 or 10 mils recommended)

    -keep traces short and direct for best performance, cleanest design

Define copper pours and ground planes as needed

      -select rectangle tool to create a copper pour

      -select appropriate layer (top or bottom), set rectangle boundaries

Vias are plated holes that connect top and bottom layers

      -round vias 39 mils in diameter are recommended

Add vias or large pads as connections to power and ground

Add silkscreen text to your board using the Text tool

      -content on the tNames (top) and bNames (bottom) layers will be printed as silkscreen

      -label your board with your group's assigned identifier

      -label V+, GND connections

Review/prepare board for fabrication

      -View the various layers (top copper, bottom copper, board outline, tNames, bNames) individually using the Layer settings tool. Does everything look the way you'd expect?

      -Save .brd file in Eagle. Name the file your group's assigned identifier.

Report Guidelines

There is no formal Lab 5 report writeup. The only requirement is that the .brd file is delivered to your TA. Your TA will provide instructions for this.

1. .brd file is functional, is correctly labeled, and meets all design specifications (20 pts)