

# Pi Layout First Attempt

Max-Felix Müller

2020  
November

# Contents

<b>1</b>	<b>The Pi Layout</b>	<b>3</b>
<b>2</b>	<b>Schematic</b>	<b>4</b>
<b>3</b>	<b>Footprints</b>	<b>5</b>
3.1	SMA Connector . . . . .	5
3.2	Double Resistor . . . . .	5
<b>4</b>	<b>Layout</b>	<b>6</b>
<b>5</b>	<b>Thoughts And Possible Improvements</b>	<b>9</b>

## List of Figures

1	The basic pi schematic . . . . .	3
2	Pi network schematic . . . . .	4
3	SMA Connector footprint from the datasheet . . . . .	5
4	Two resistors sharing the center pad . . . . .	5
5	Upper copper layer . . . . .	6
6	Via placement . . . . .	7
7	Completed layout . . . . .	8
8	3D view of the layout . . . . .	8

# 1 The Pi Layout

The pi layout gets its name from the fact that it looks similar to the symbol  $\pi$  (pi) or even more similar to the upper case letter  $\Pi$  (Pi).

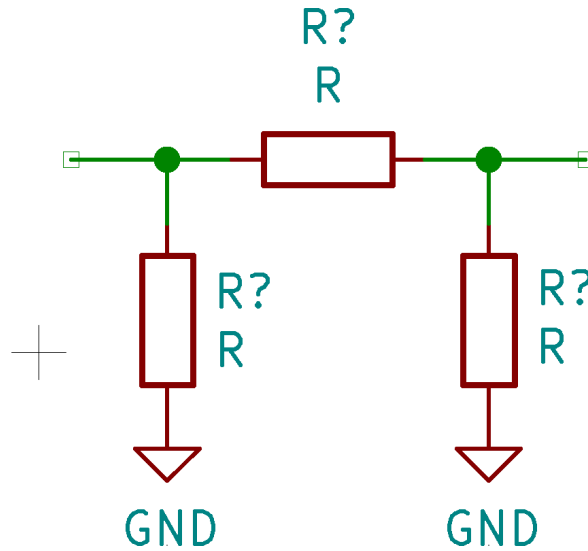


Figure 1: The basic pi schematic

In this figure a pi network is built using only resistors. The pi network can be built using all different components like resistors, capacitors and inductors. To design a layout only the component size matters, thus resistors are chosen as components in this example.

Alternatively components can be left out as well. For example when leaving the lower left resistor, a voltage divider is built. How much sense that makes obviously depends on the use case.

The pi network can be used to design for example RC filters, attenuators or even more complex filters requiring all three components to be populated. The complexity is limited only by the number of components, which in this case is three.

For this project an upgraded version will be designed. By adding an additional resistor (or other components) in parallel to the lower left and one in parallel to the lower right one. Therefore the maximum number of components is increased to five, giving the possibility to create more complex designs. One other use case would be to use two resistors in parallel instead of one for the pi network. When the layout is done the right way, as will be seen later, this will create a more even load on the RF signal and thus the performance will be increased slightly.

## 2 Schematic

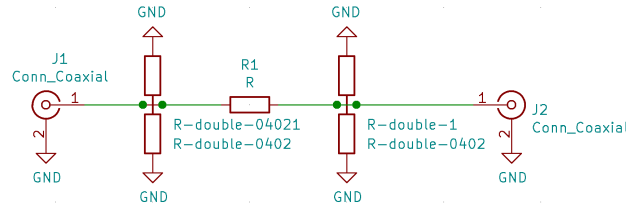


Figure 2: Pi network schematic

For the schematic the basic pi network was extended to make room for the parallel resistors. The layout in the schematic is done in a way similar to the final PCB layout. This will help to remember the intended design and to find the right component placement.

For the SMA connectors a standard symbol was used. KiCad provides them with the base install when selecting to add the libraries. They were copied into a project specific library anyways to keep all project files in one place.

The connection points on the wire segments between the parallel resistors is an artifact of the way those are handled. Instead of having two footprints on the PCB later a new footprint was created to combine the center pad into one single but slightly bigger one.

There is not much to the schematic. The design was already explained and the only major add-on are two simple connectors.

## 3 Footprints

### 3.1 SMA Connector

The footprint for the SMA connector was simply copied from the datasheet. It can be found in the link below.

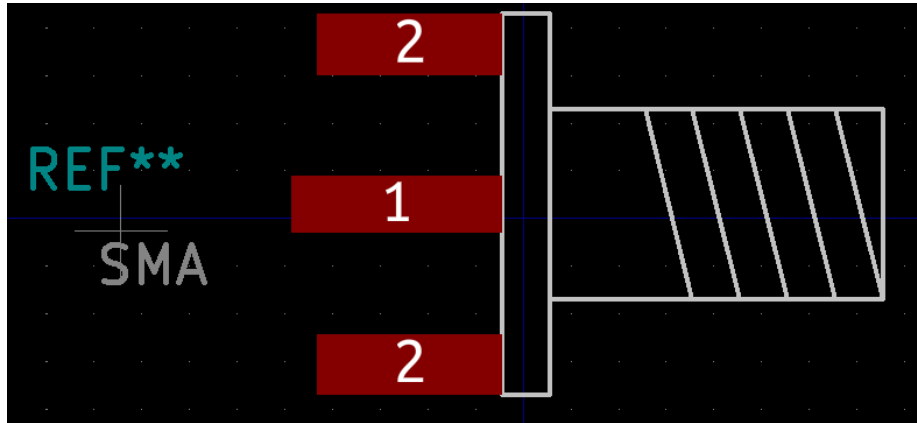


Figure 3: SMA Connector footprint from the datasheet

### 3.2 Double Resistor

For the parallel resistors the footprint had to be changed in a way that allows mounting of both resistors symmetrically. Also the pad in the middle has to be small enough to fit within the trace later on.

A first attempt was to duplicate a single resistor and overlap the center pads. That did not work too well with the connections in both the schematic and the layout as well.

Instead one of the center pads was removed and the other one was enlarged.

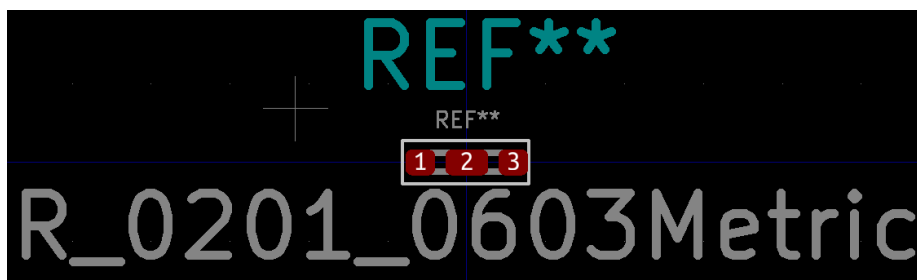


Figure 4: Two resistors sharing the center pad

## 4 Layout

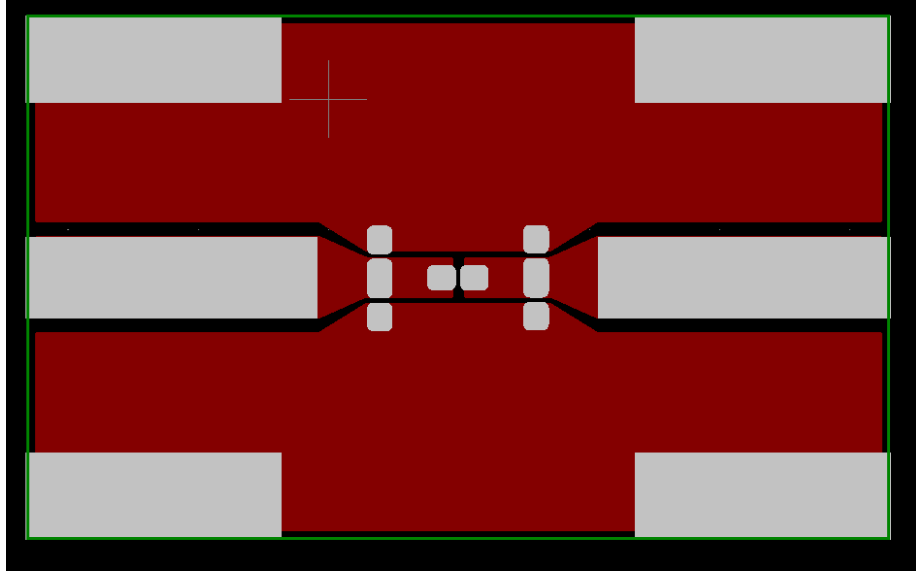


Figure 5: Upper copper layer

The copper on the top layer has the transmission line. A coplanar wave guide with ground plane was chosen for this design. That also suits the ground connections of the components.

The coplanar wave guide is dependent on the thickness and material of the PCB as well as the thickness of the copper layer. With a thickness of 1.6mm for the PCB and 35um of copper these are standard values.

The center conductor has a width that was chosen to be not too thick for the pads of the single resistor in the center. Final width is 0.8mm. With this value the distance to the ground plane on the sides can be calculated and for a 50 Ohm impedance the distance has to be about 0.1mm.

The width of the SMA connector pads was given by the datasheet. The distance to the ground plane was calculated from there.

For the part of the trace between the center and the SMA pad a linear approximation was used.

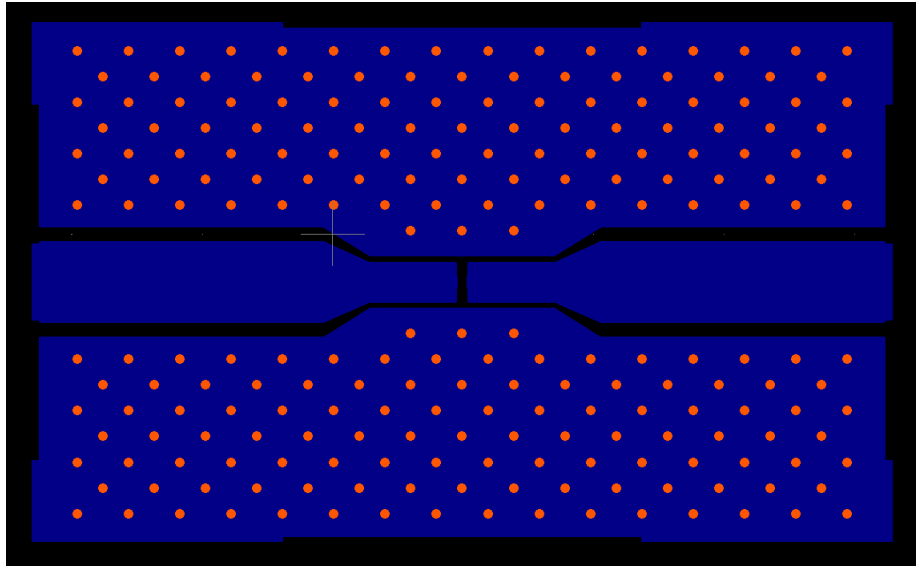


Figure 6: Via placement

The vias were placed in a 0.5mm grid. For the via size the limits of JLC PCB were copied from their website. A smaller via diameter makes for a maximum number of vias and thus a better connection between the two ground planes on the top and bottom layer.

It can not be seen here, but the vias in the center do not overlap the pads of the components.



The final complete layout can be seen here. Dimensions were added as measurements on the drawings layer.

Since KiCad offers this feature, also a 3D view can be generated. The 3D models of the components are not included since the resistors on the parallel resistors would have been placed on the wrong location.

The PCB has the standard thickness of 1.6mm since the SMA connectors have this distance between the edge and the center conductors.

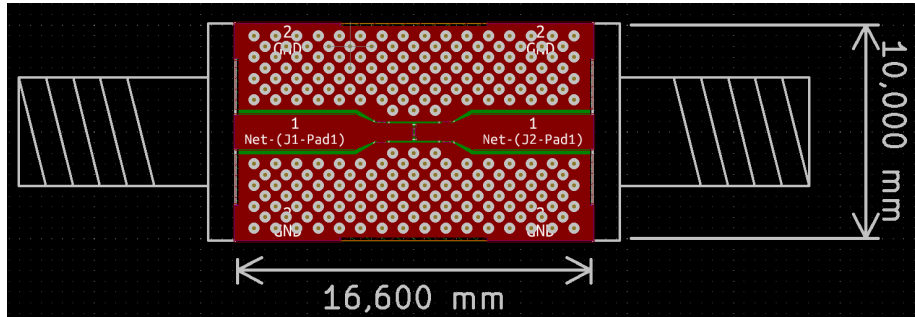


Figure 7: Completed layout

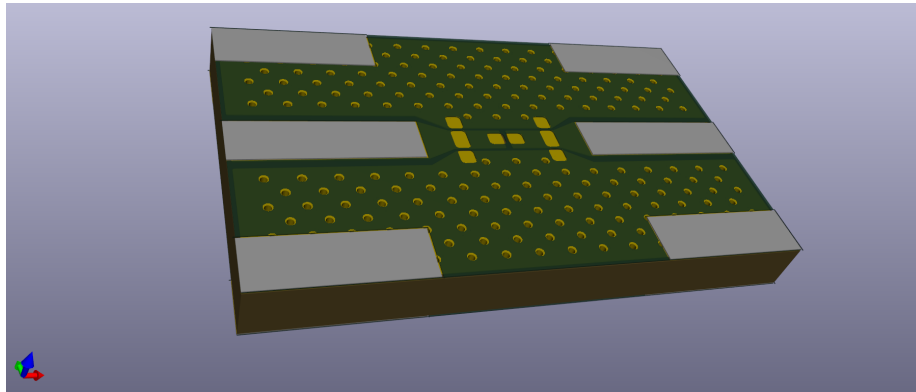


Figure 8: 3D view of the layout

## 5 Thoughts And Possible Improvements

I'll start with some thoughts about the schematic. The symbol of the parallel resistors could be improved to remove the "ugly" green dots where the wires meet.

Also the name of the parallel resistors symbol is a little long to look nice.

Other than that, there is not much to say about the schematic, it is quite straight forward.

A problem I already identified with the layout is the size of the components. Zooming on a PC is so easy, but soldering the 0201 resistors won't be.

Then there is the "problem" of the length of the PCB. With nearly 17mm the length will play a significant role at higher frequencies. This should work up to 2 GHz without much of a problem though.

The length could be shortened by remaking the footprint of the SMA connector. Its center conductor is not longer than the ones of the shield, it is just the footprint recommended in the datasheet.

Instead of the 0201 even 0402 components would fit on the PCB without making it larger. Maybe the distance between the components would have to be reduced, which is good for higher frequencies as well.

Additionally, when staying with 0201, the component distance can be further decreased.

There are enough reasons for another layout in the future.