PCB DESIGNING

# GET ALL THE SYMBOLS:

1. Make all the symbols you need into library by using part library editor
2. Use pins and labels for making symbol and shape it accordingly
3. Save the symbol in library

# MAKE SCHEMATIC OF CIRCUIT IN KICAD:

1. Make schematic in schematic area
2. Use your added component and other components which comes preloaded with kicad(Shortcut: “A”)

# ANNOTATE THE CIRCUIT

# PERFORM ELECTRICAL RULE CHECKING (ERC):

1. Check for errors

CERATE NET LIST: (It Save The Data Of The Schematic In a File)

# RUN “CvPcb to associate components and footprints”:

1. Associate all the components with their footprints
2. Create all the footprints needed and them all

# NOW RECREATE THE NET LIST

# OPEN “Pcbnew” AND READ NET LIST

# CREATE THE OUTLINE OF CIRCUIT BOARD AND ADJUST ALL THE COMPONENTS:

# SET THE TRACK,VIA VALUES :

1. Go to Design rules>design rules set track and via parametres
2. E.g. Global Design Rules

* Min track width : 0.006 in
* Min via diameter : 0.027 in
* Min via drill 0.013 in
* Min uvia diameter : 0.007874015748
* Min uvia drull dia : 0.003937007874

1. E.g. Net Classes Editor Default

* Clearance : 0.010
* Track Width :0.010
* Via Dia :0.027
* Via Drill : 0.013

# ADD TRACKS AND VIA

# ADD FILLED ZONES

# REMOVE ALL UNWANTED LABELS

1. Hover the label and press “E”
2. Select invisible

# ADD TEXT TO PCB

1. Add text to silk screen

# PERFORM DESIGN RULE CHECKING

1. Remove if any

# GENERATE GERBERS

1. Click plot schematic
2. Check

* F.cu
* B.cu
* B.Silks
* F.silkS
* B.Mask
* F.Mask
* Edge.Cuts

# AAAAANNNDDD PLOT IT

# GENERATE DRILL FILE :

1. Drill units : Inches
2. Zeros Format : Suprres trailing zeros
3. Drill map file format : PostScript
4. Drill File options : Merge PTH and NPTH holes into one file
5. Drill origin : Absolute

# TAP DRILL FILE

# NOW “GerbView” THE FILE

# LOAD ALL FILES IN IT AND CHECK YOUR PCB DESIGN

# KUDOSSS you are DONE