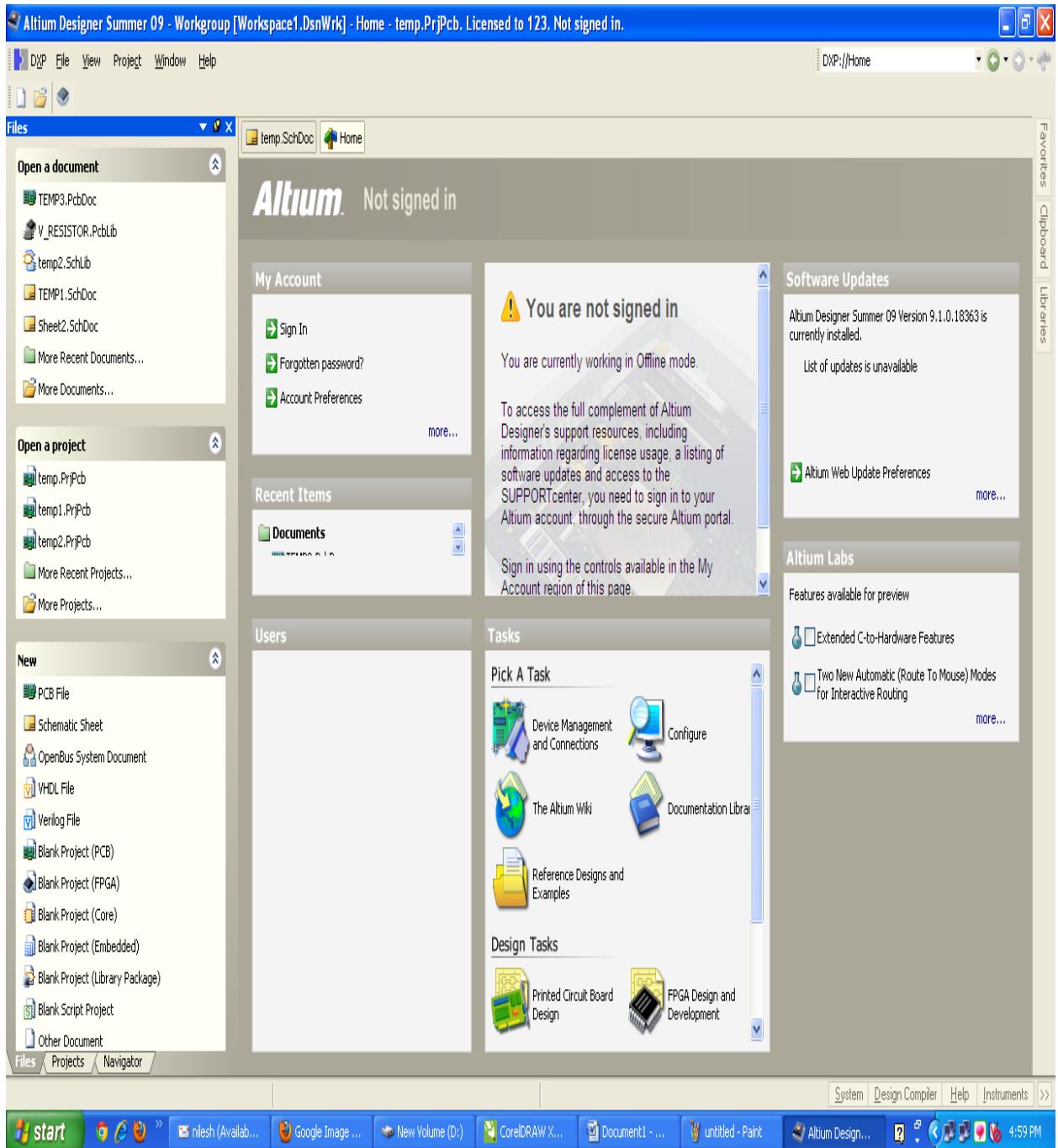


HOW TO DO PCB DESIGNING???

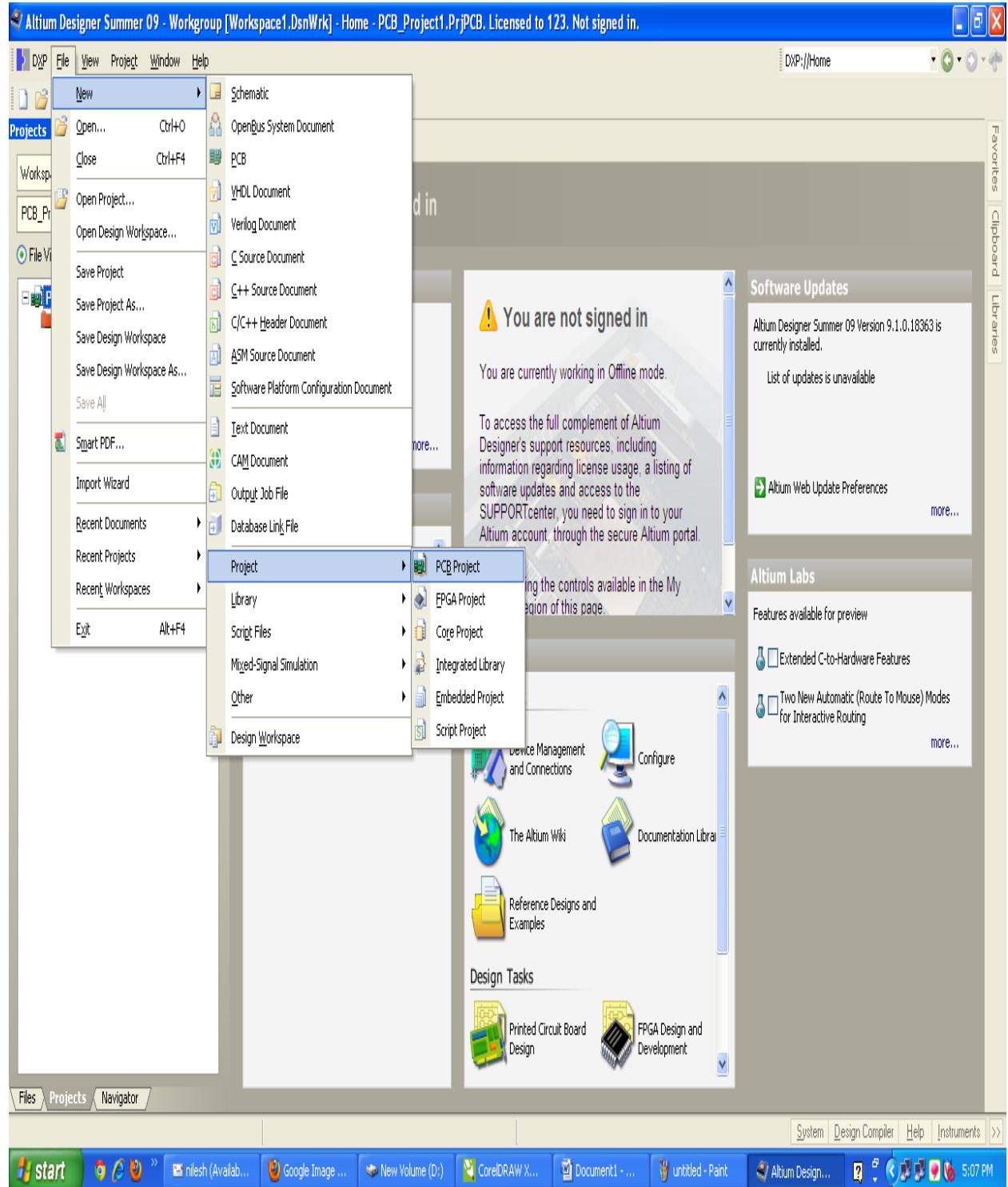
➤ START THE ALTIUM DESIGNER BY DOUBLE CLIKING ON ICON.



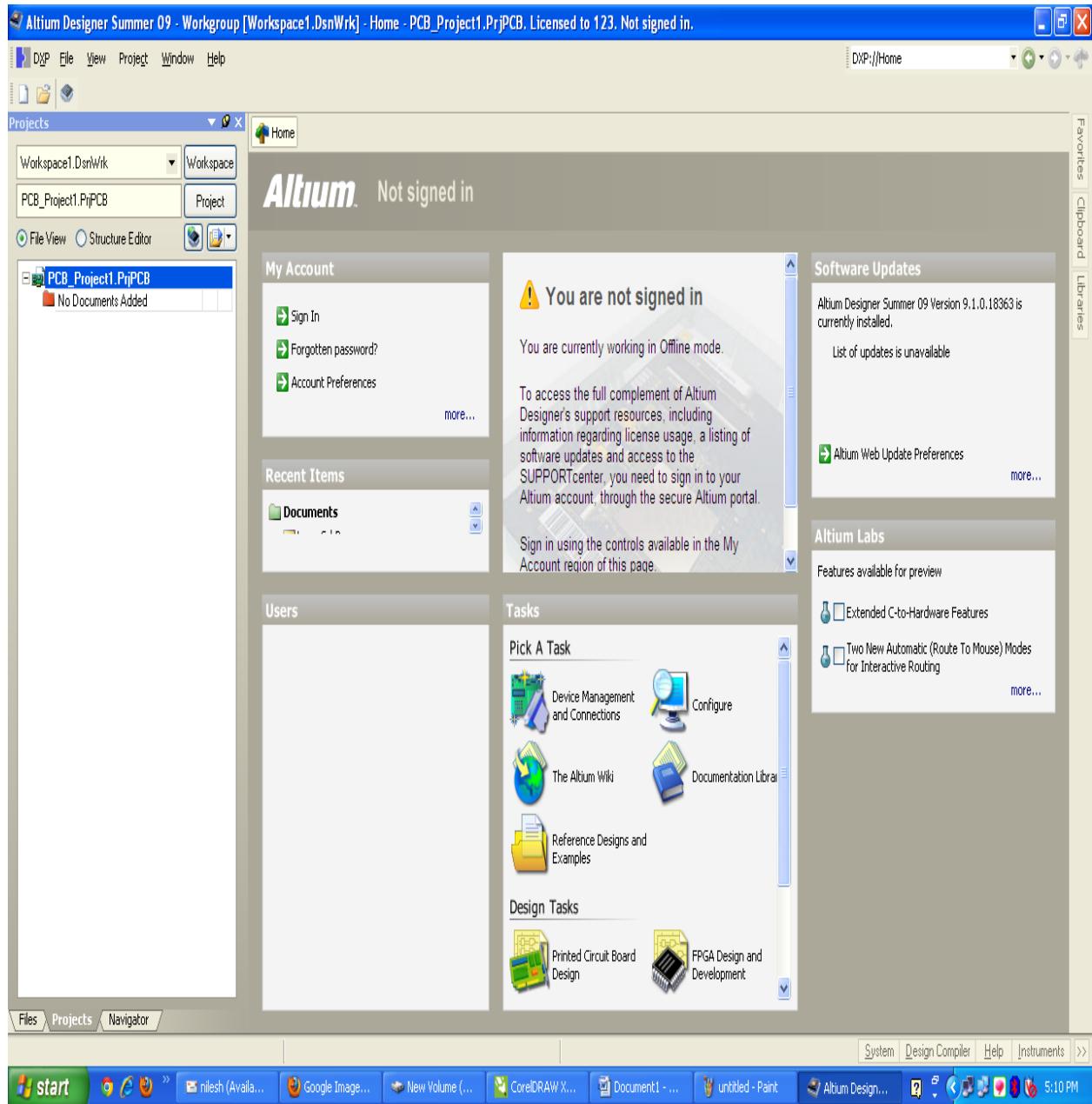
YOU CAN SEE THE PAGE LIKE THIS.



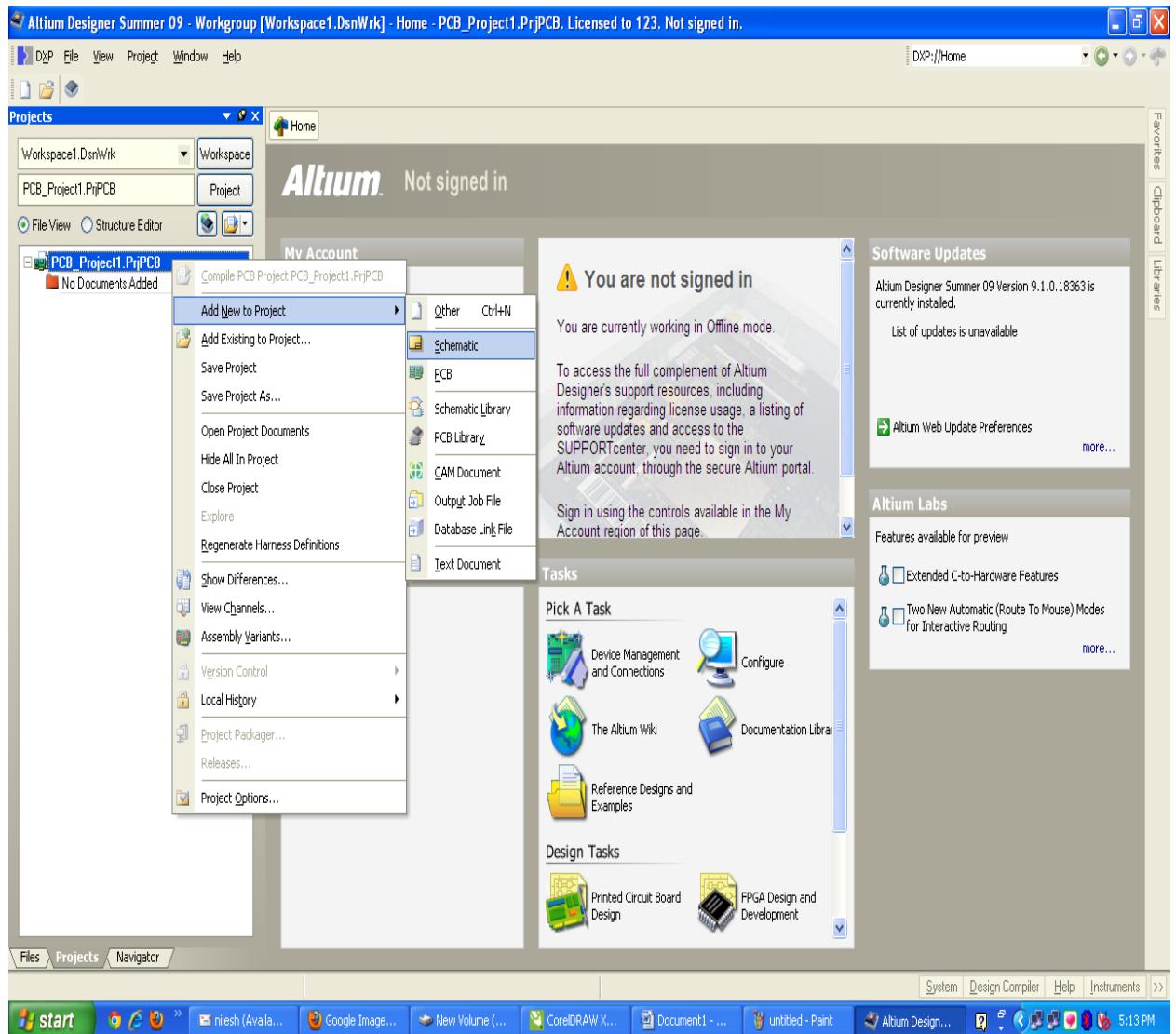
➤ NOW TO START PCB PROJECT GO TO FILE>NEW>PROJECT>PCB PROJECT AS SHOWN BELOW.



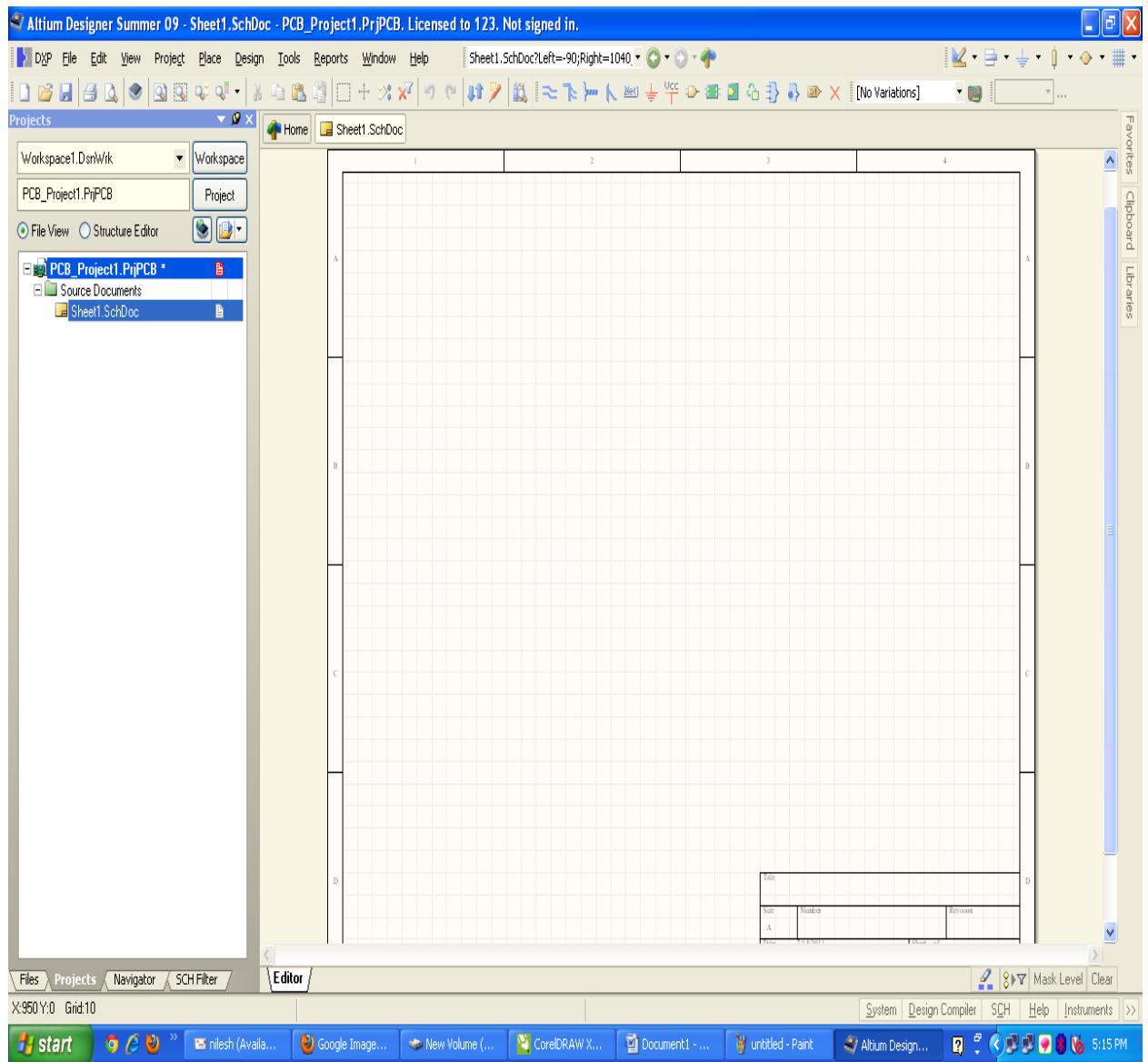
➤ THEN YOU CAN SEE THE BELOW PAGE.



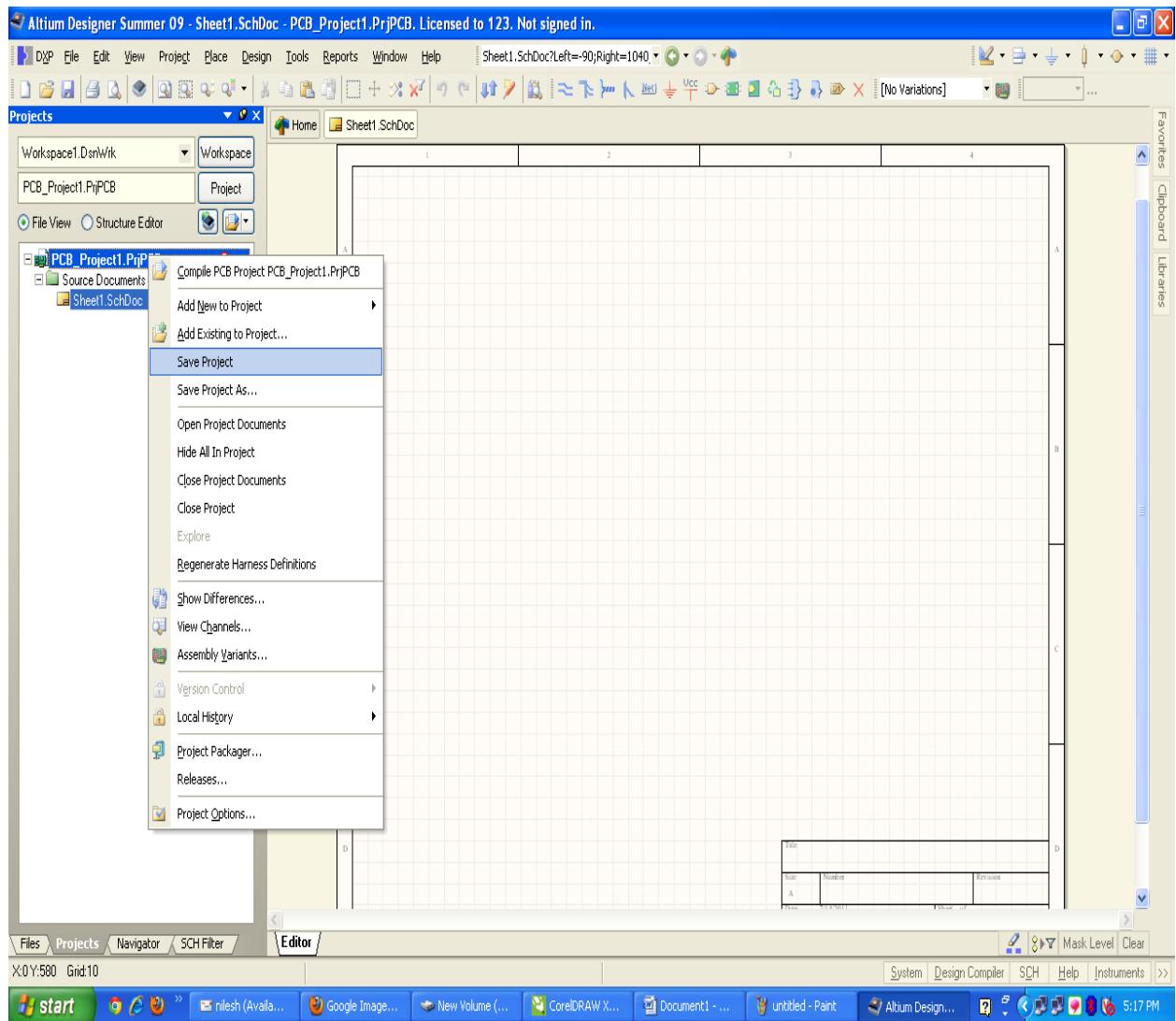
- THERE IS NO FILES IN THE PROJECT.
- SO WE HAVE TO ADD FIRST THE SCHEMATIC FILE TO DRAW CIRCUIT.



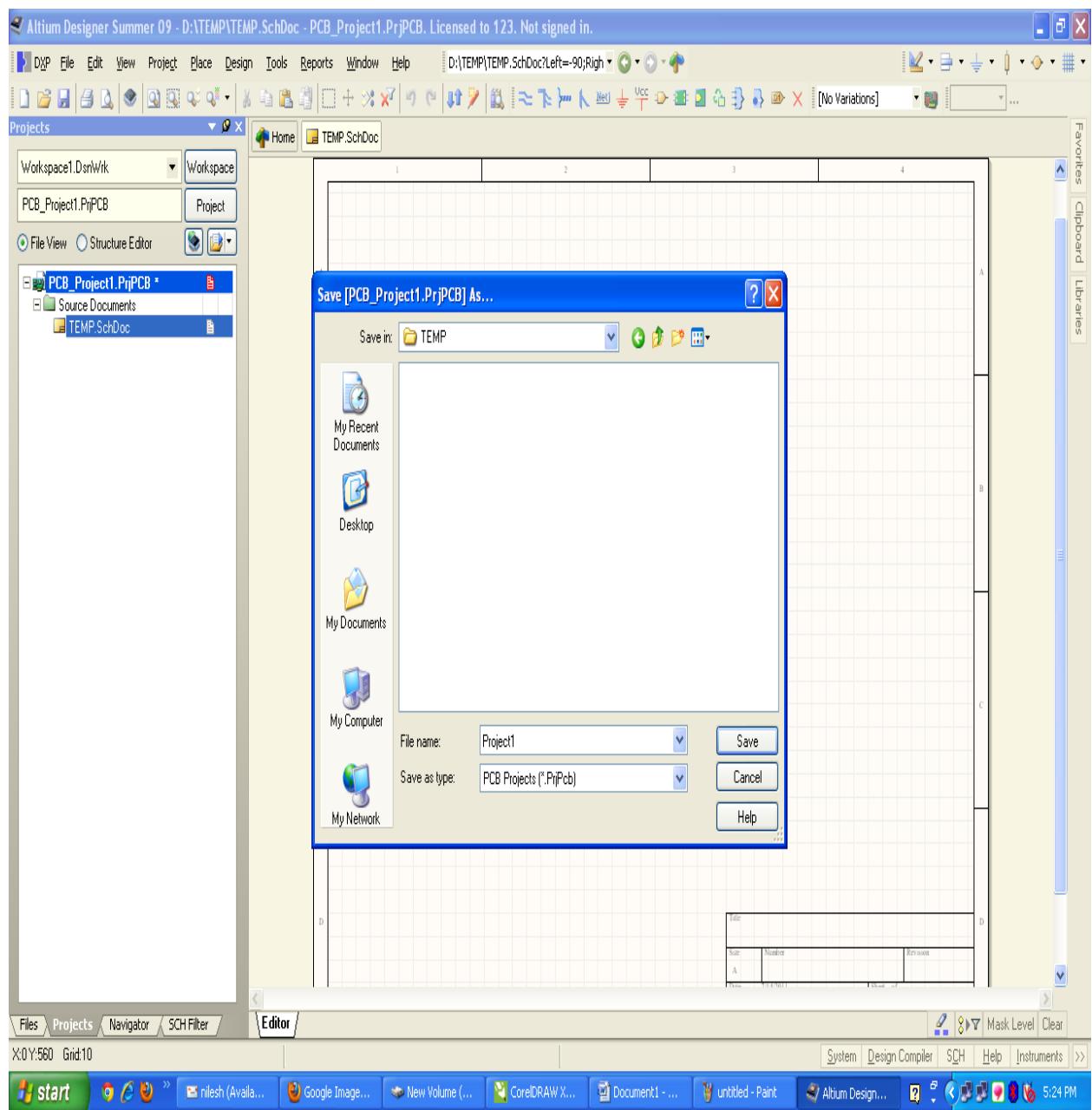
➤ THEN YOU CAN SEE THE PAGE LIKE GINVEN BELOW.



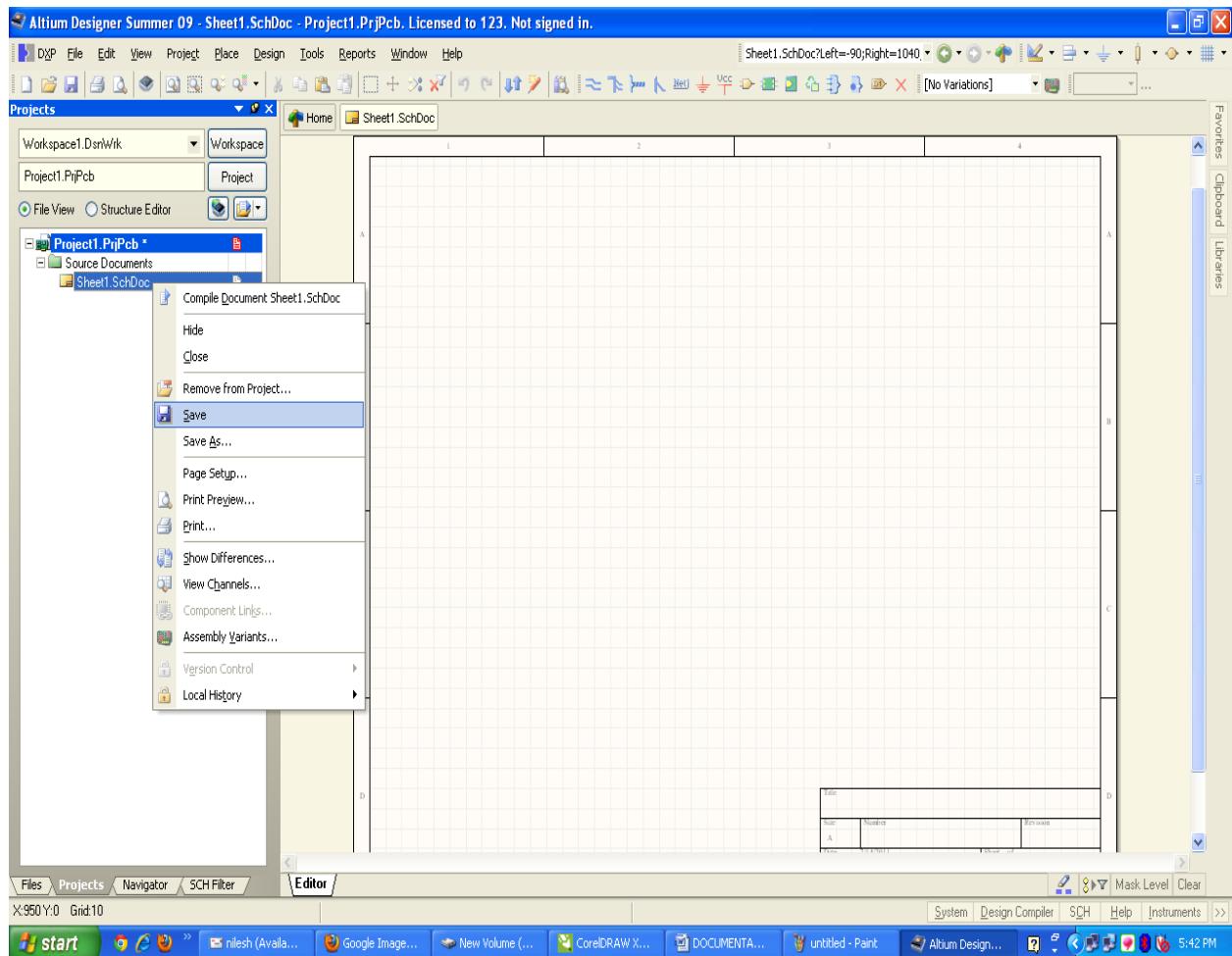
- NOW MAKE ONE NEW FOLDER AS PER UR REQUIRED NAME IN D OR E DRIVE TO SAVE THE PROJECT.
- NOW SAVE THE PROJECT AS GIVEN BELOW.



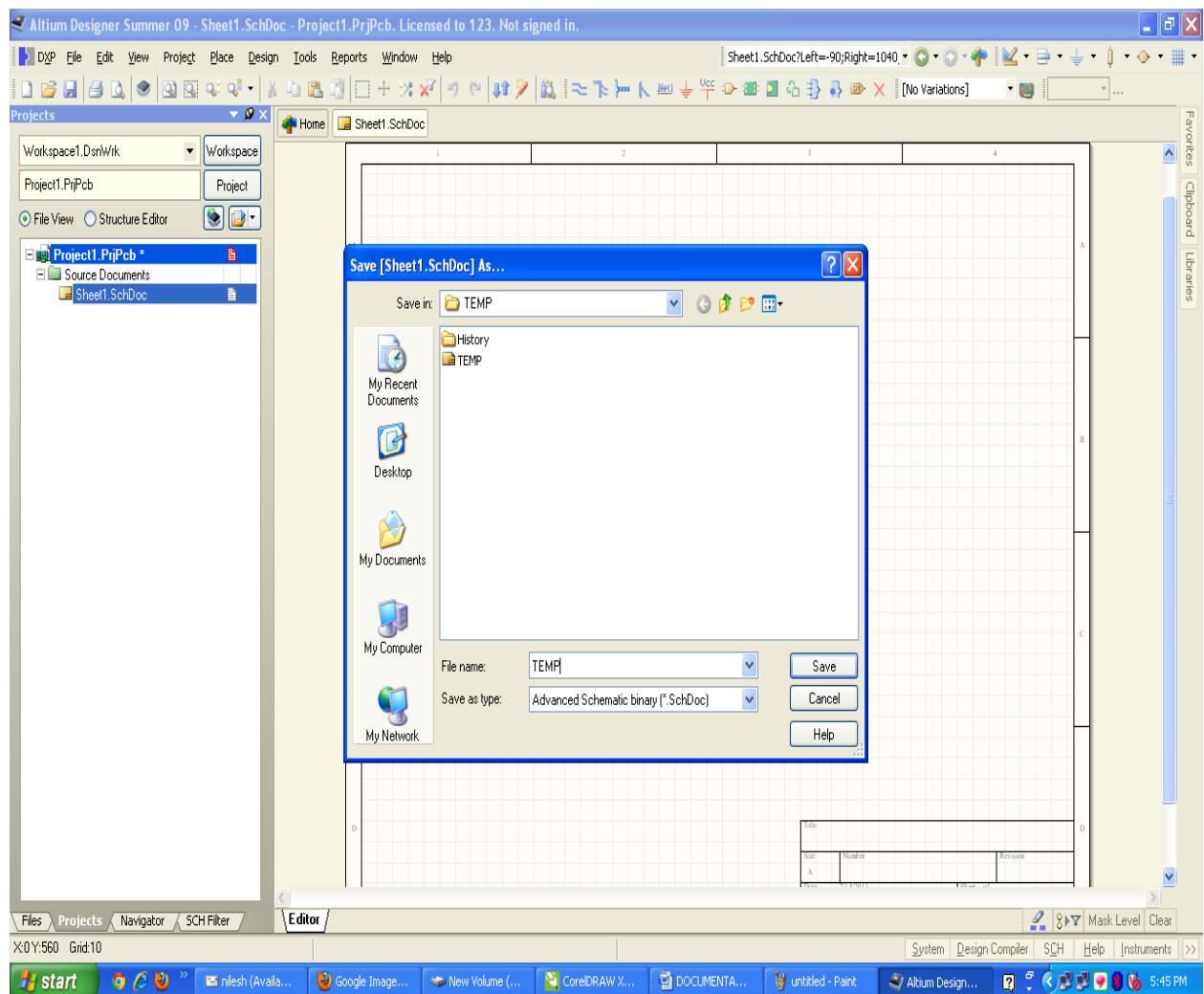
➤ GIVE THE NAME TO PROJECT AND SAVE THE PROJECT AS SHOWN IN BELOW FIG.



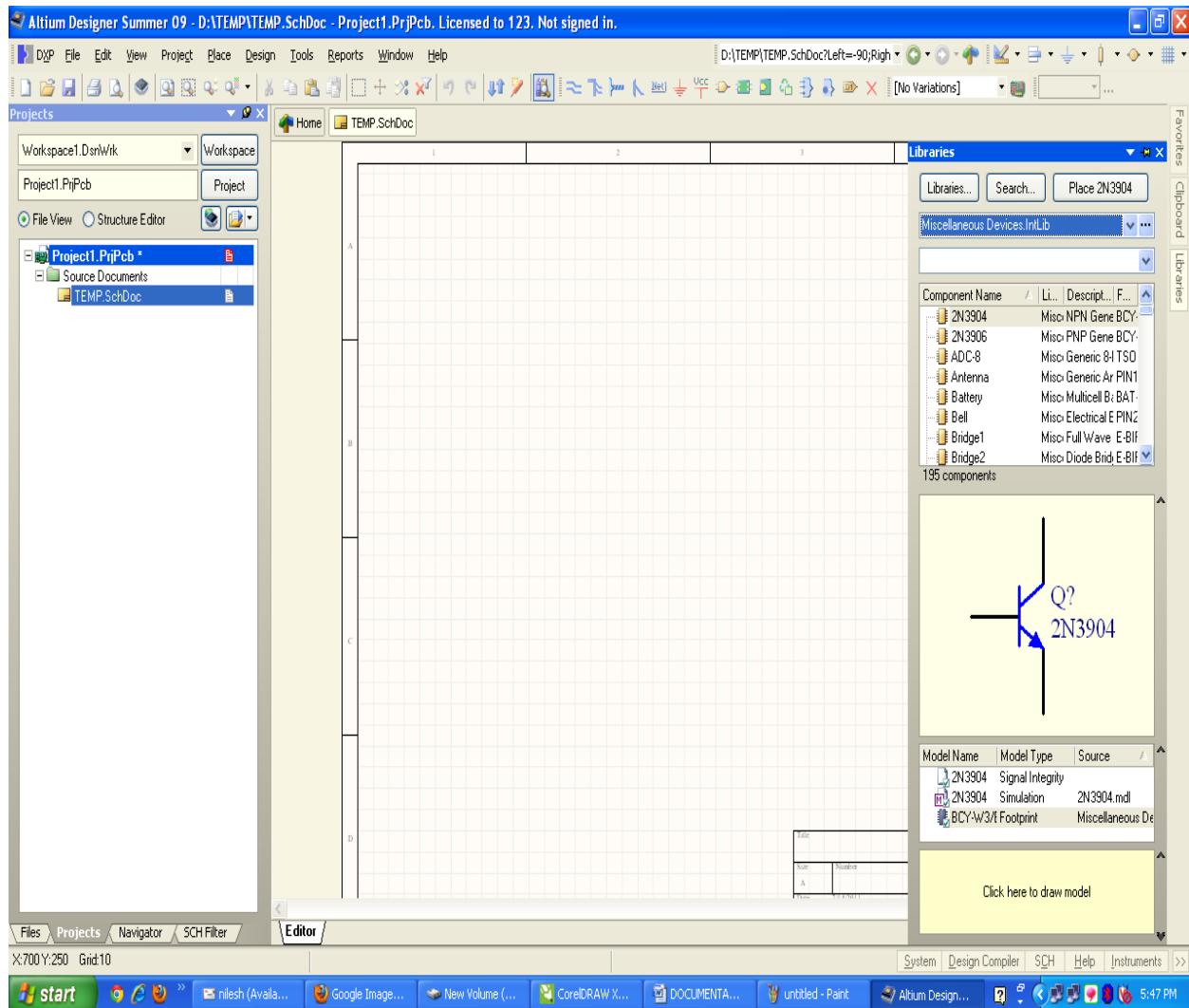
➤ NOW SAVE THE SCHEMATIC FILE WITH THE SAME NAME OF PROJECT SEE THE BELOW FIG.



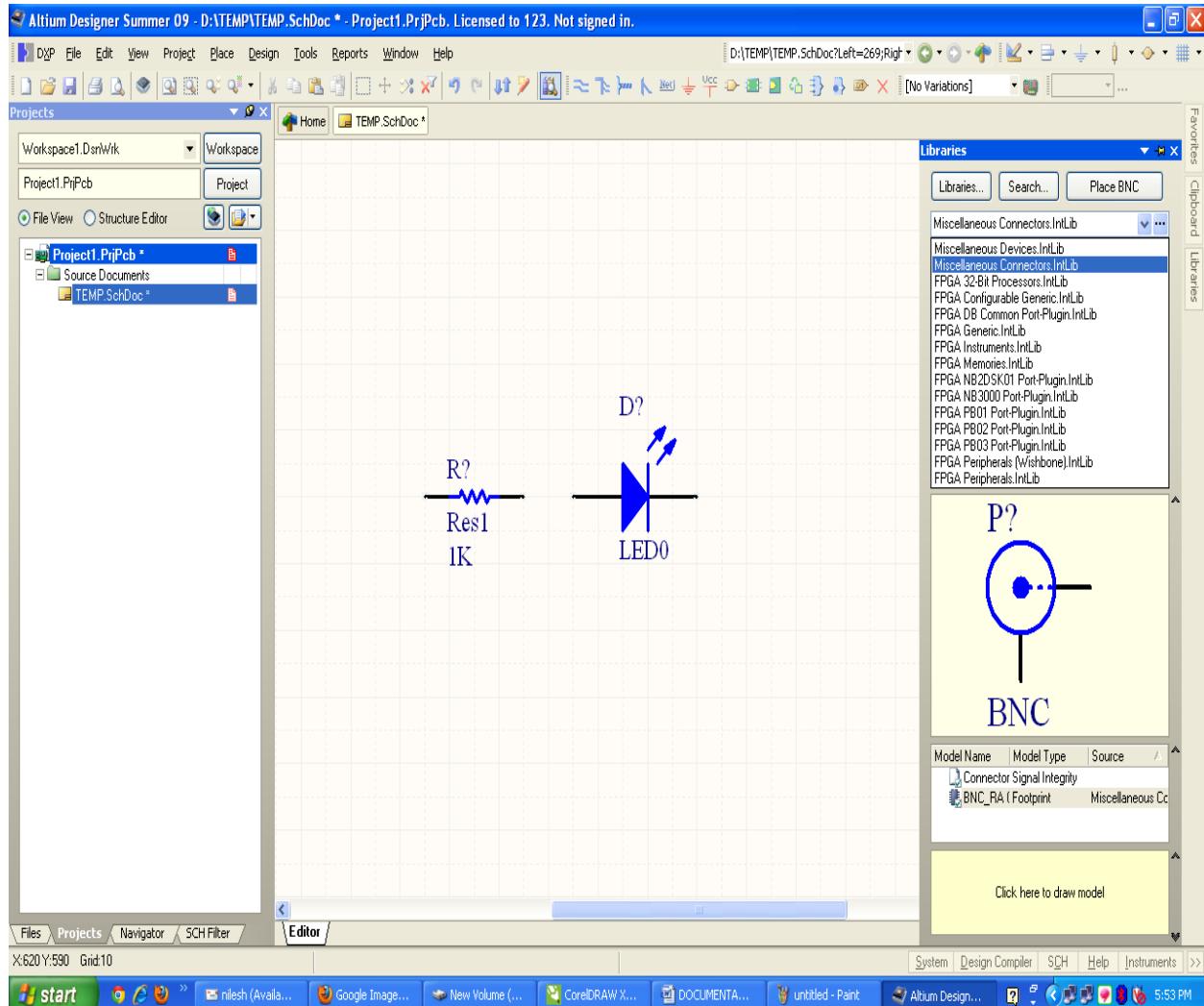
➤ SEE THE BELOW FIGURE.

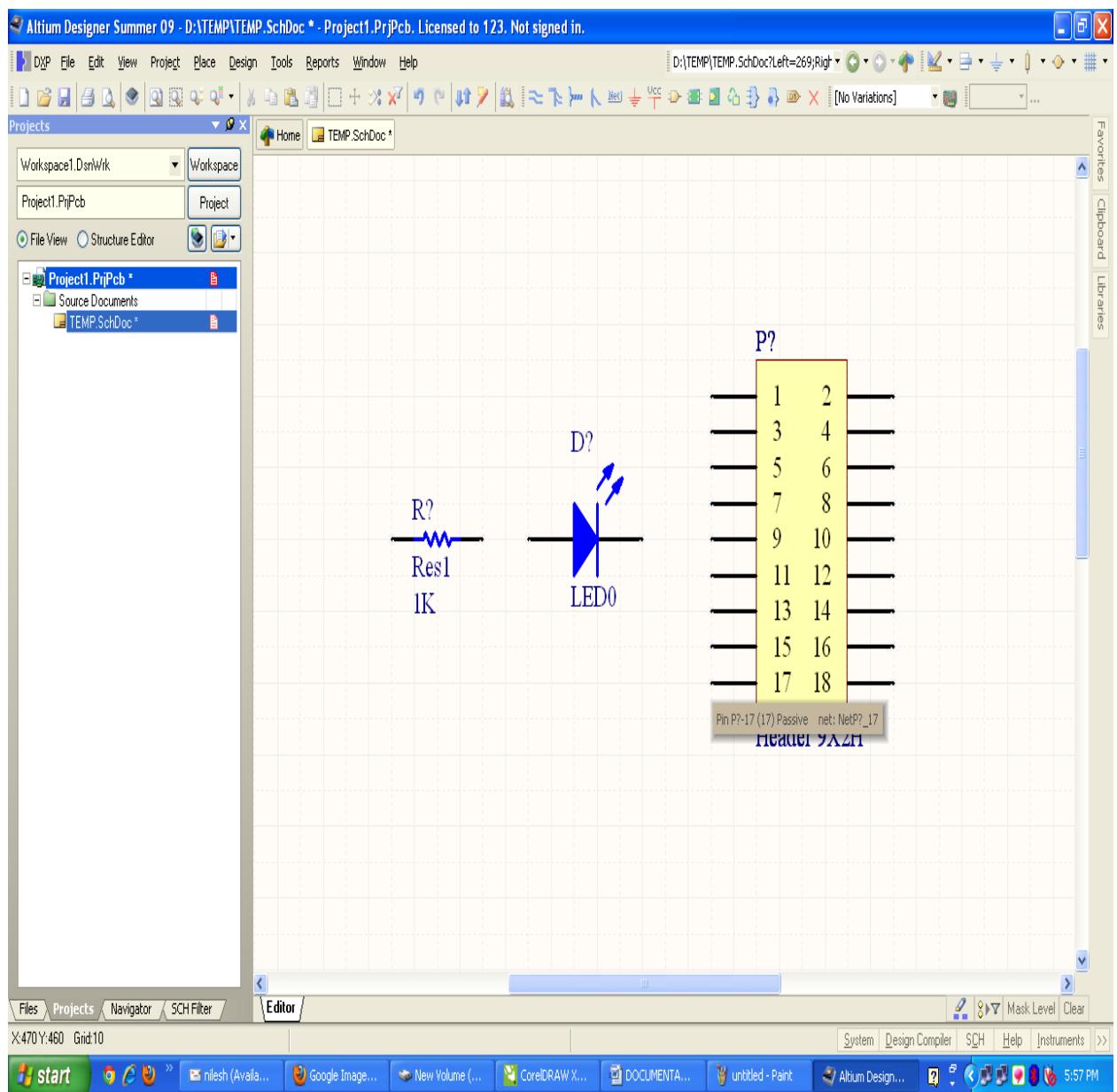


- WE CAN GET ALL THE REQUIRED COMPONENT AND CONNECTORS FROM THE LIBRARIES GIVEN IN THE RIGHT SIDE OF UR SCHEMATIC DIAGRAM.
- NOW OPEN THE LIBRARY TO ADD DIFF COMPONENTS AND CONNECTORS AS PER GIVEN BELOW.

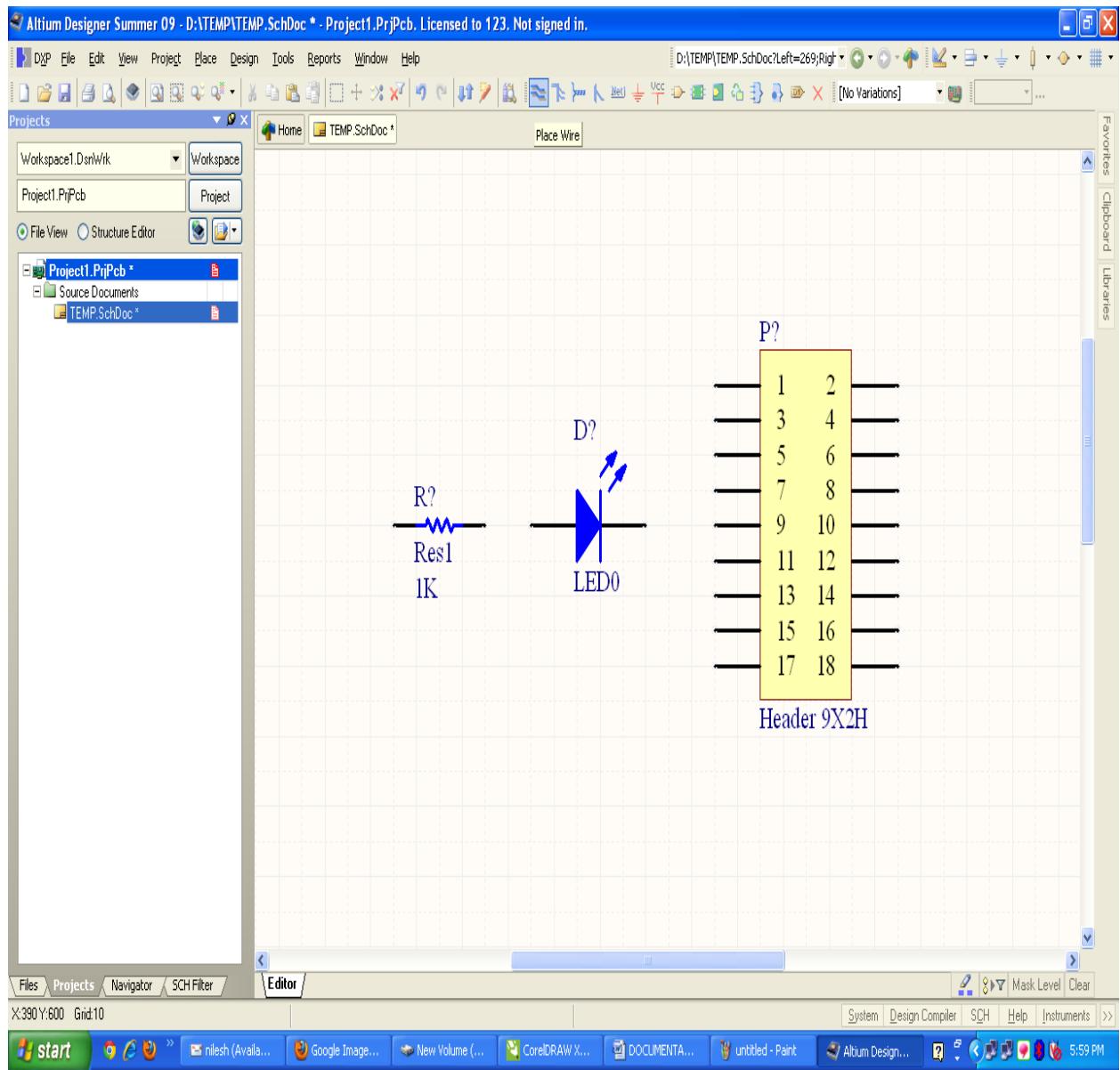


- TO ADD COMPONENT FROM THE LIST JUST DRAG IT FROM LIST AND DROP IT TO SCHEMATIC DIAGRAM.
- ADD REQUIRED CONNECTORS LIKE IC OR MALE FEMALE CONNECTORS FROM THE LIST.

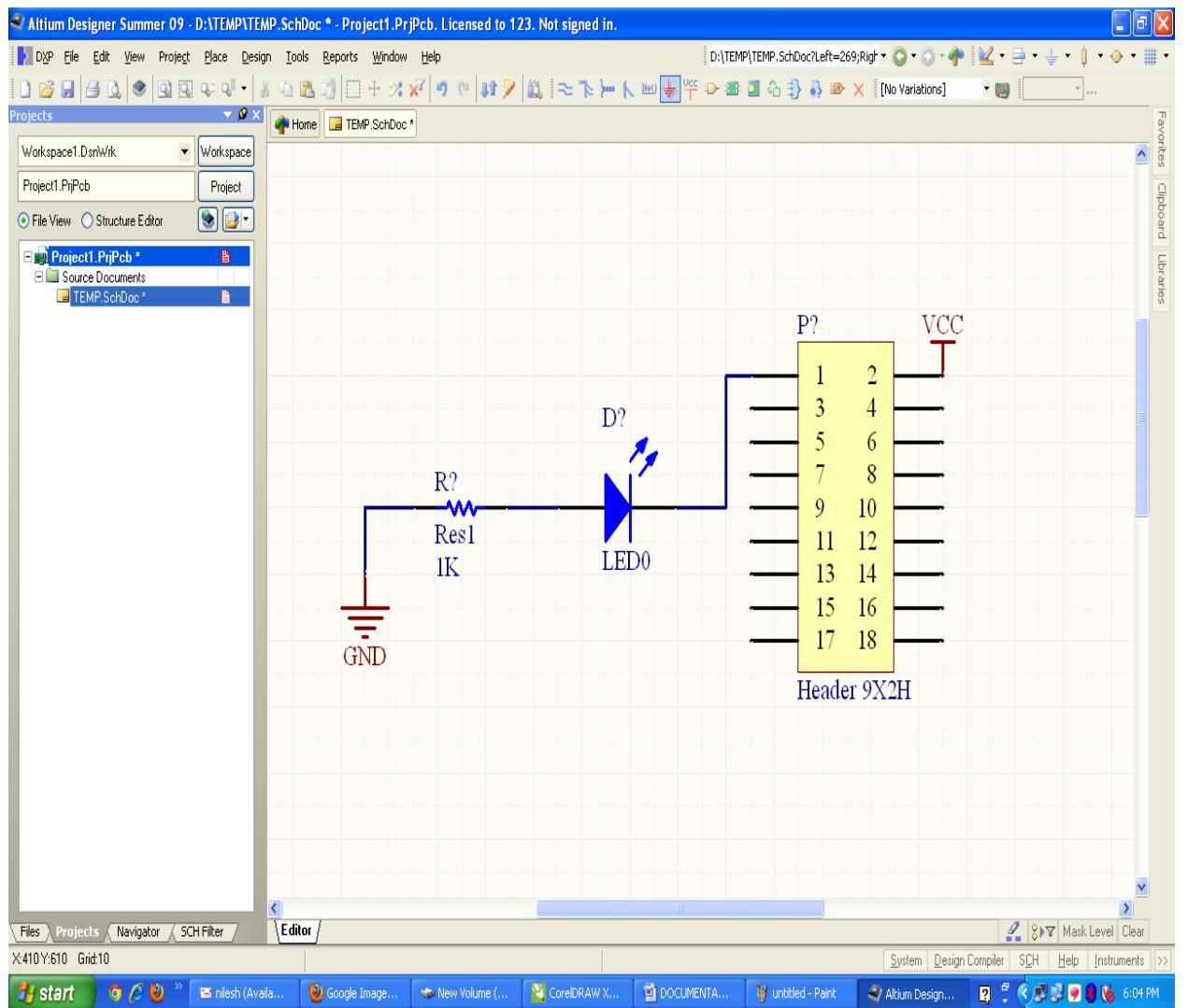




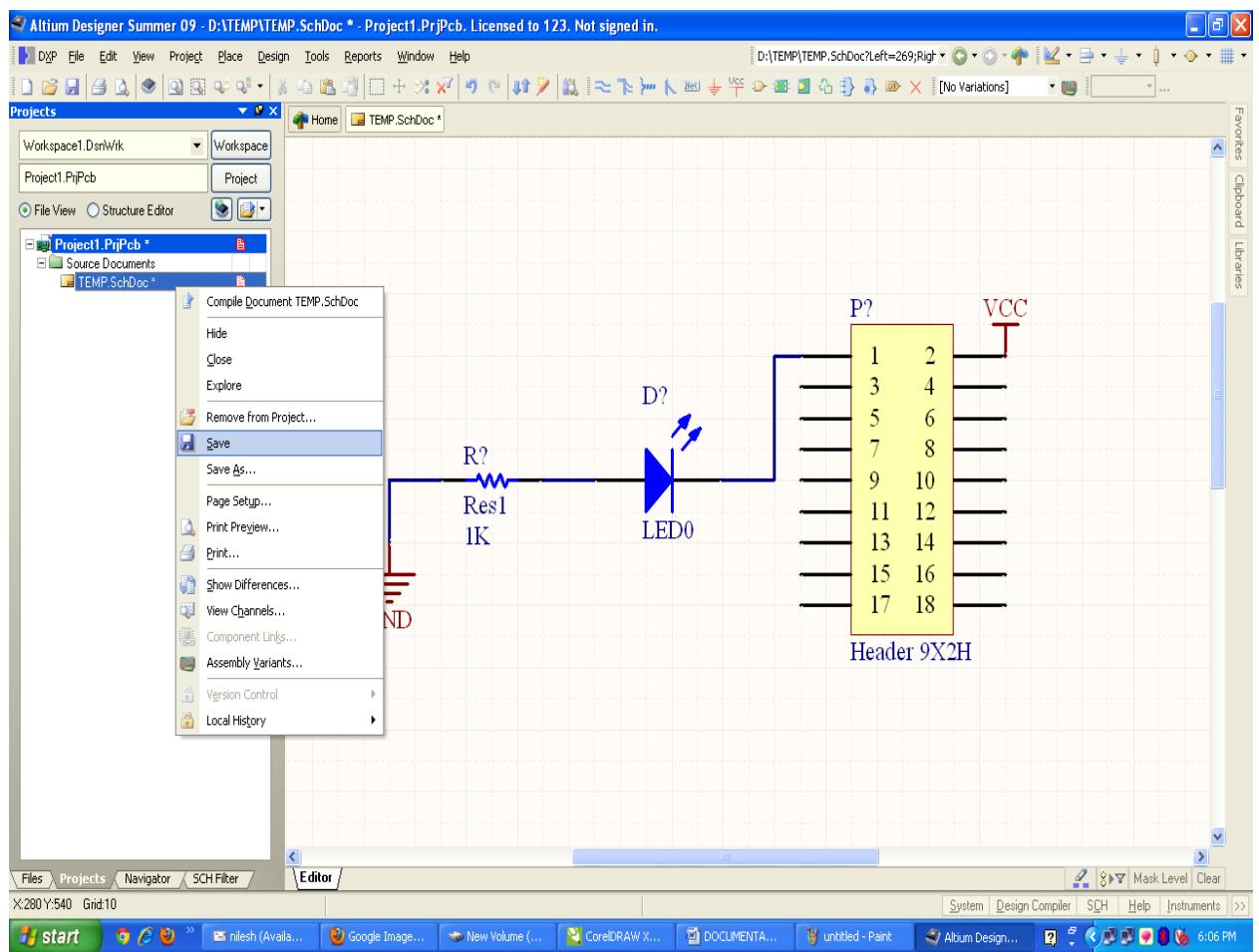
- NOW FIRST GIVE THE NAMES TO THE ALL THE COMPONENT AND CONNECTORS AT THE PLACE **P? D? R?**.
- YOU CAN GIVE NAME AS **R1 R2 R3... D1 D2 D3... IC1 IC2 IC3...**
- NOW TO CONNECT THE COMPONENT AND CONNECTOR TAKE **PLACE WIRE** AND CONNECT AS PER REQUIRMENT.



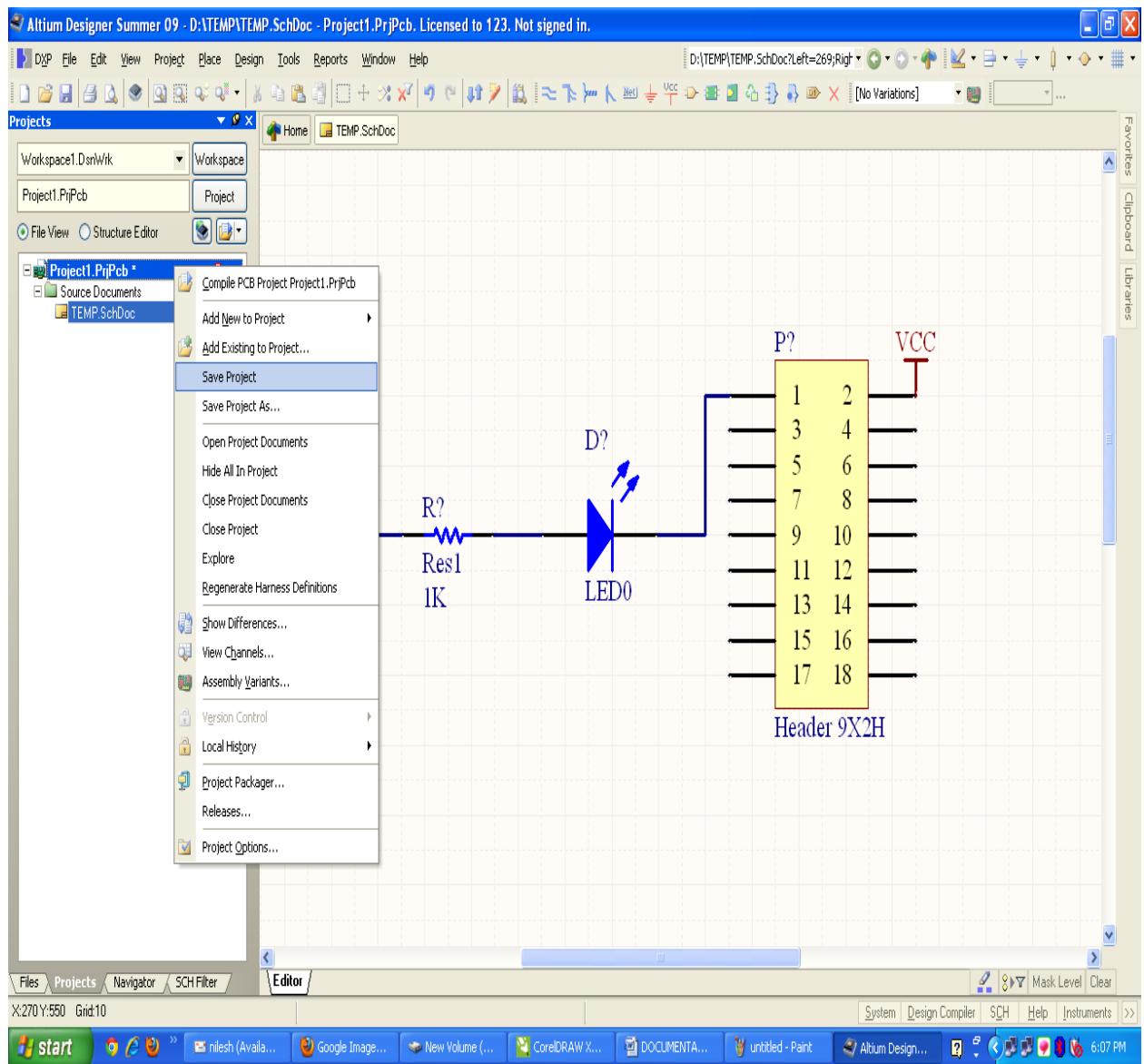
- CONNECT ALL THE COMPONECT AND CONNECTORS WITH THE BLUE LINE AS SHOWN IN THE BELOW FIG.
- THEN ADD REQUIRED GROUNDS AND POWER SUPPLYS VCC TO THE CIRCUIT FROM TOOLBAR GIVEN IN ABOVE ROW.



➤ NOW AGAIN SAVE THE SCHEMATIC FILE.



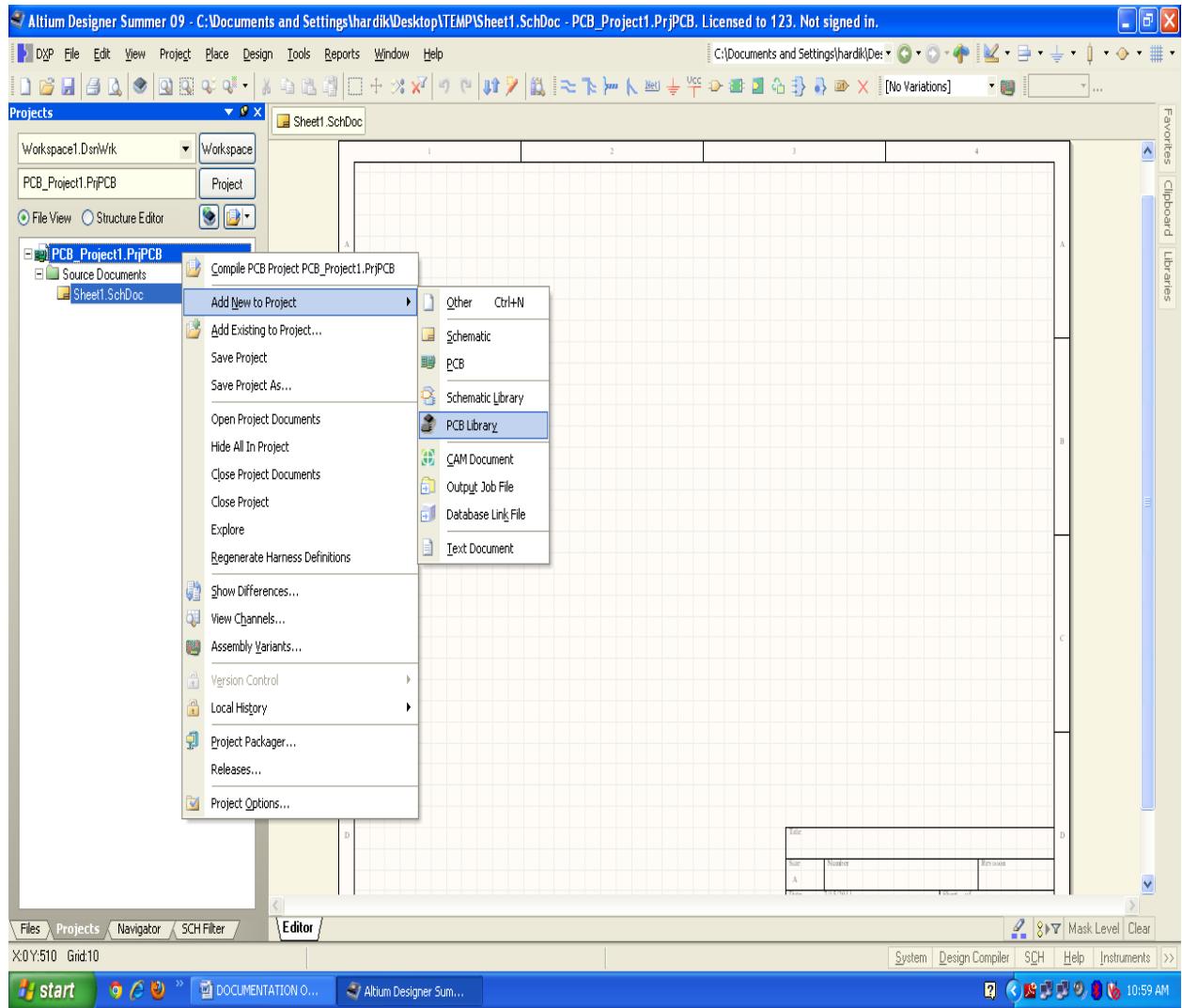
- ALSO SAVE THE PROJECT FILE.



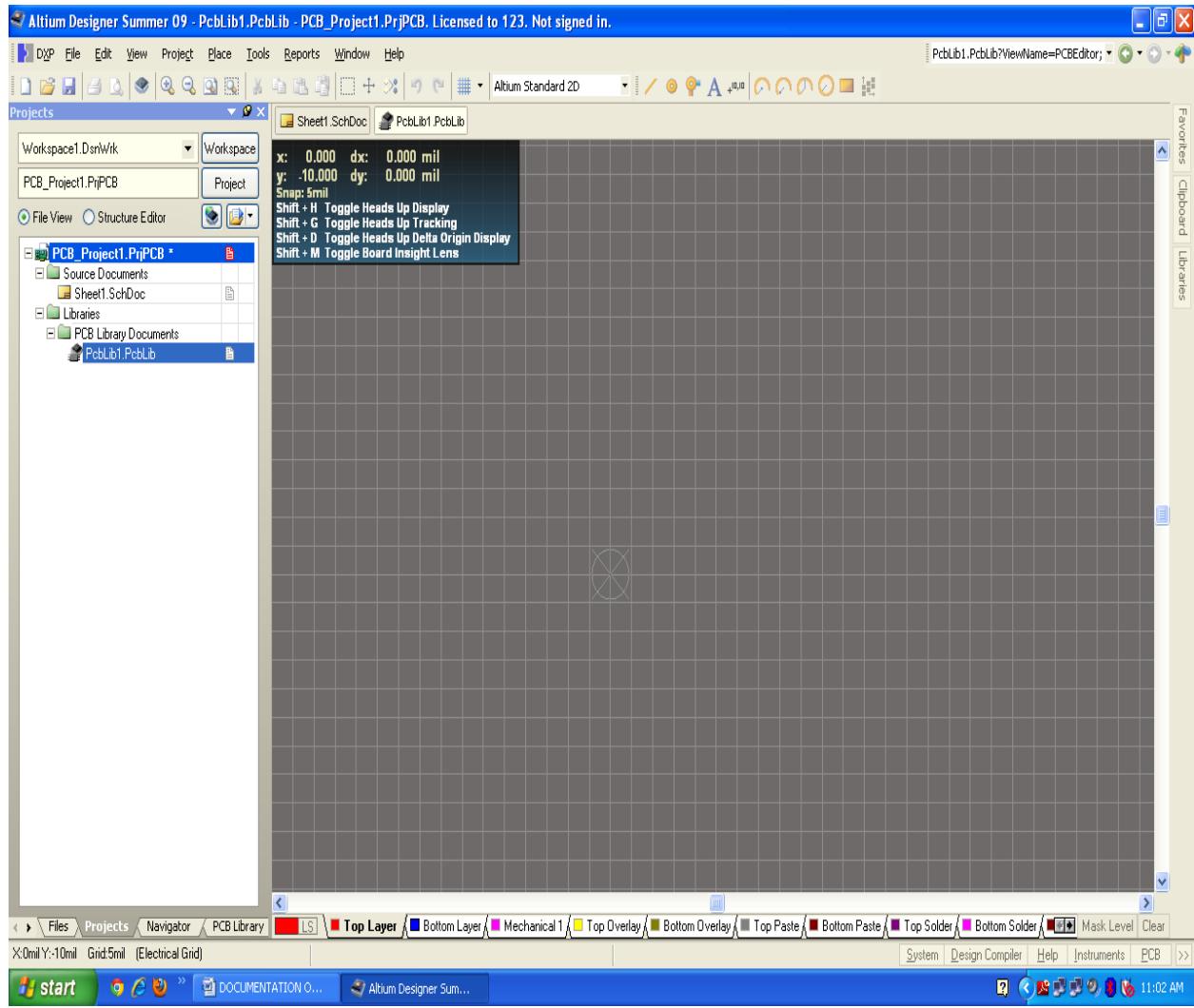
- TO PUT OUR ALL THE COMPONENT AND CONNECTOR ON PCB WE HAVE TO PUT FOOT PRINT OF ALL THE COMPONENT AND CONNECTOR.
- SO WE HAVE TO ADD ALL COMPONENT'S FOOT PRINT TO THE PROJECT FOLDER AND ALSO ADD ALL THIS FILES TO PROJECT.

HOW TO MAKE FOOTPRINT OF COMPONENT??

- FIRST WE HAVE TO ADD FOOT PRINT TO PROJECT AS GIVEN BELOW.

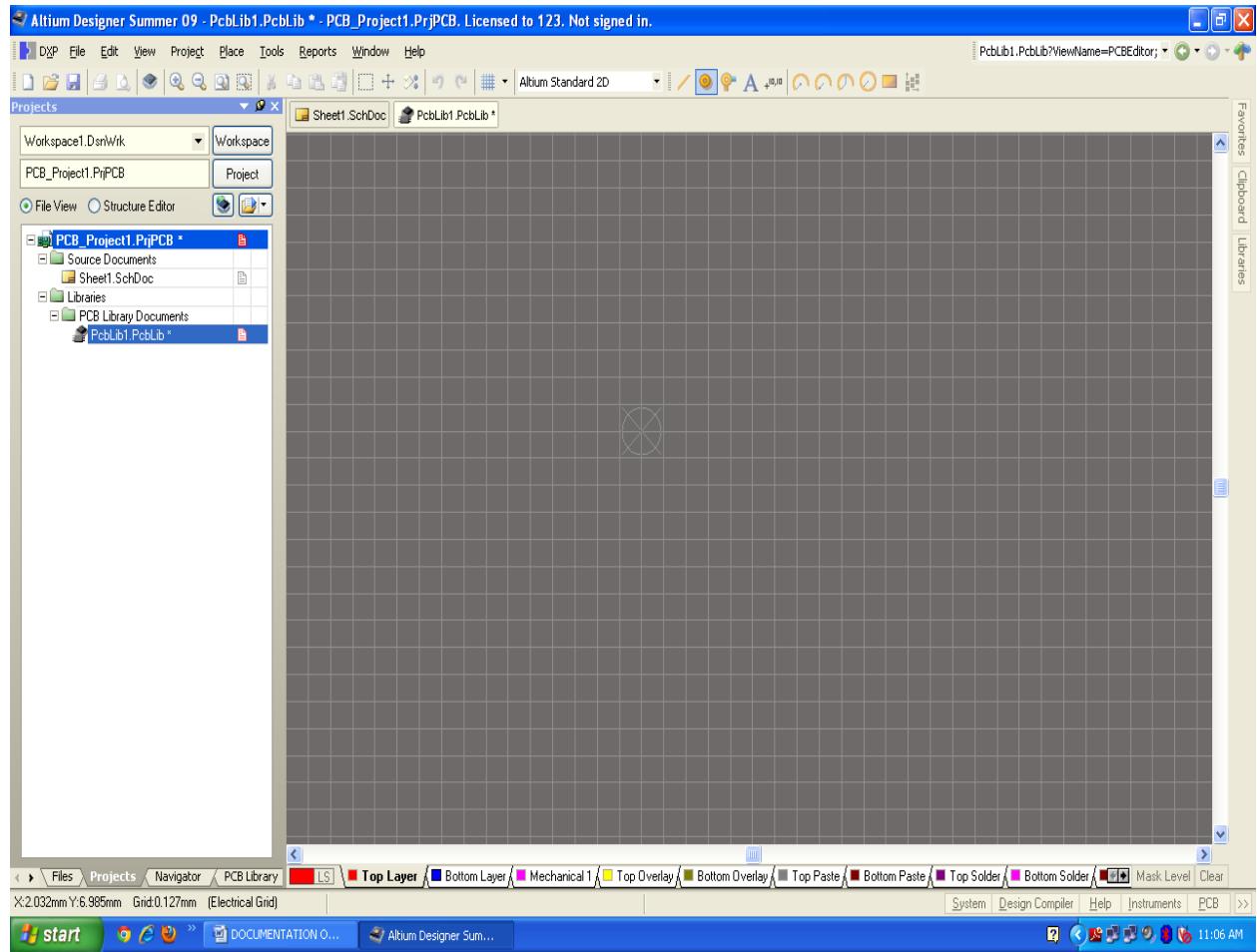


- NOW U CAN SEE THE PAGE LIKE THIS.

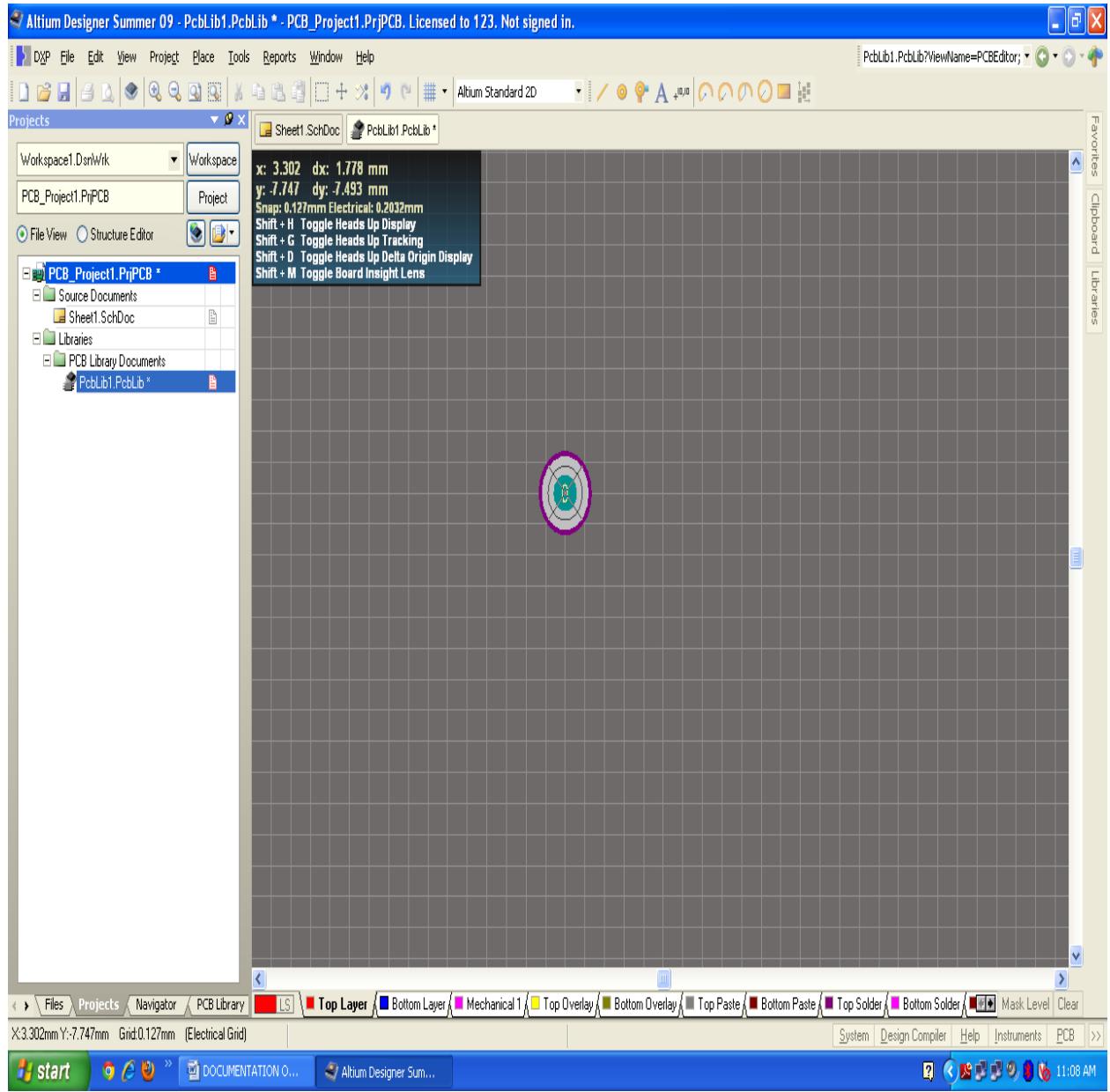


- THE CROSS SYMBOL IS SHOWS THE ORIGIN. WE HAVE TO START FROM ORIGIN.
- BY PRESSING "Q" WE CAN CONVERT OUR DIMENTIONS TO mm.

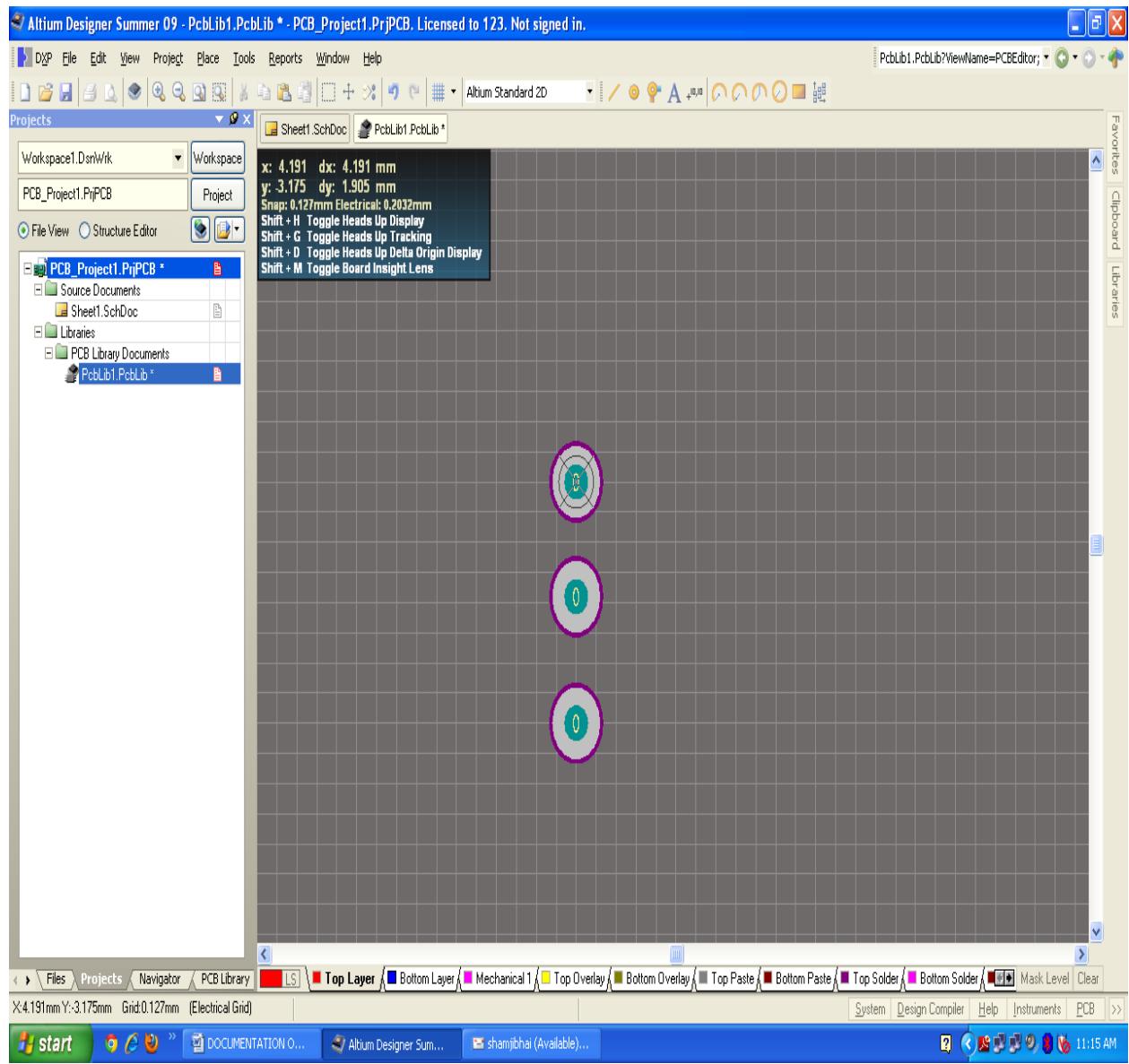
- AS PER OUR REQUIRNMENET WE HAVE TO PUT PEDS AND START FROM ORIGIN.
- YOU HAVE TO TAKE PEDS FROM THE TOOLBAR AS SHOWN IN BELOW FIG.



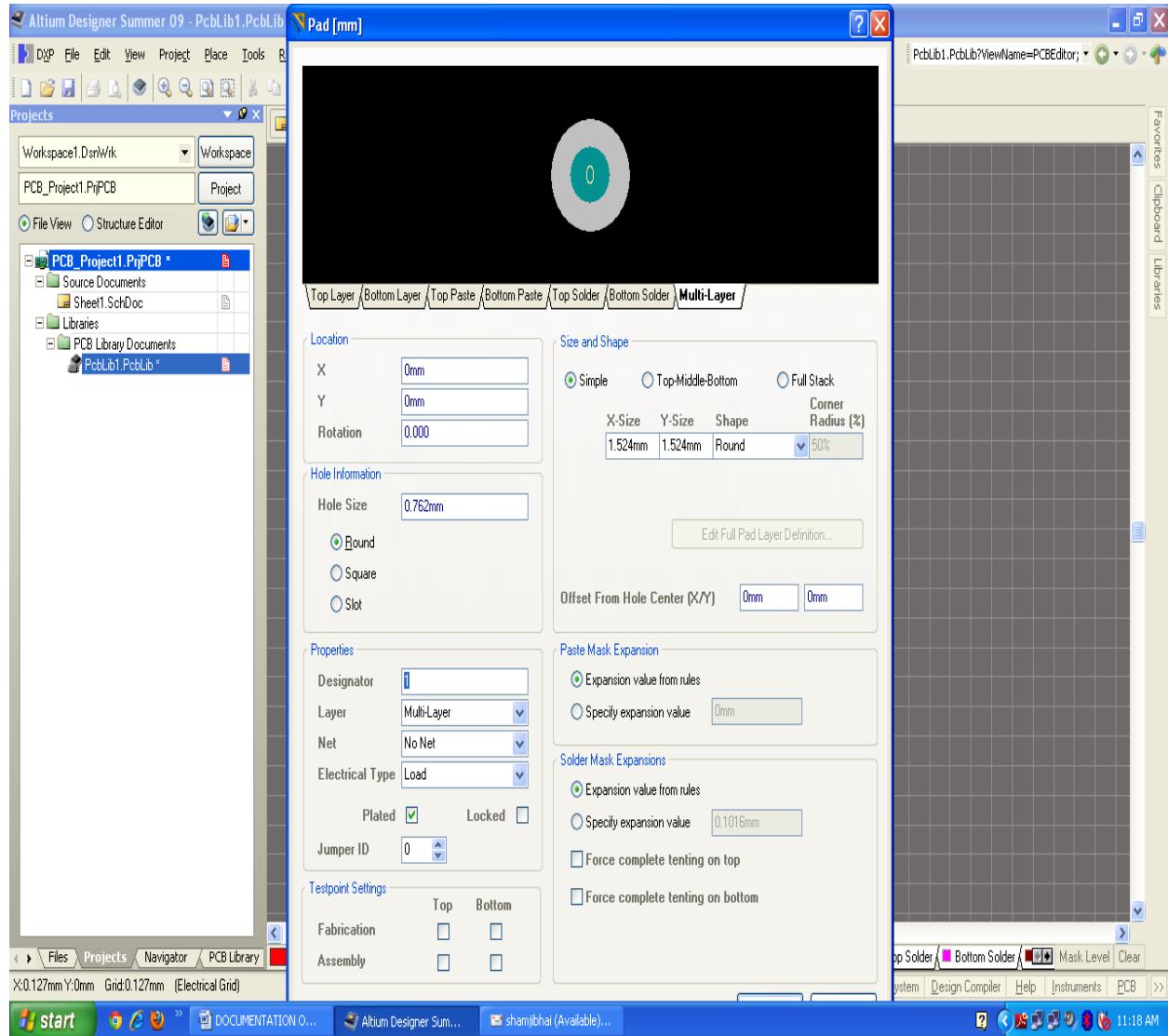
- NOW START WITH ORIGIN AND PUT FIRST PEG ON THE ORIGIN AS SHOWN IN BELOW FIG.



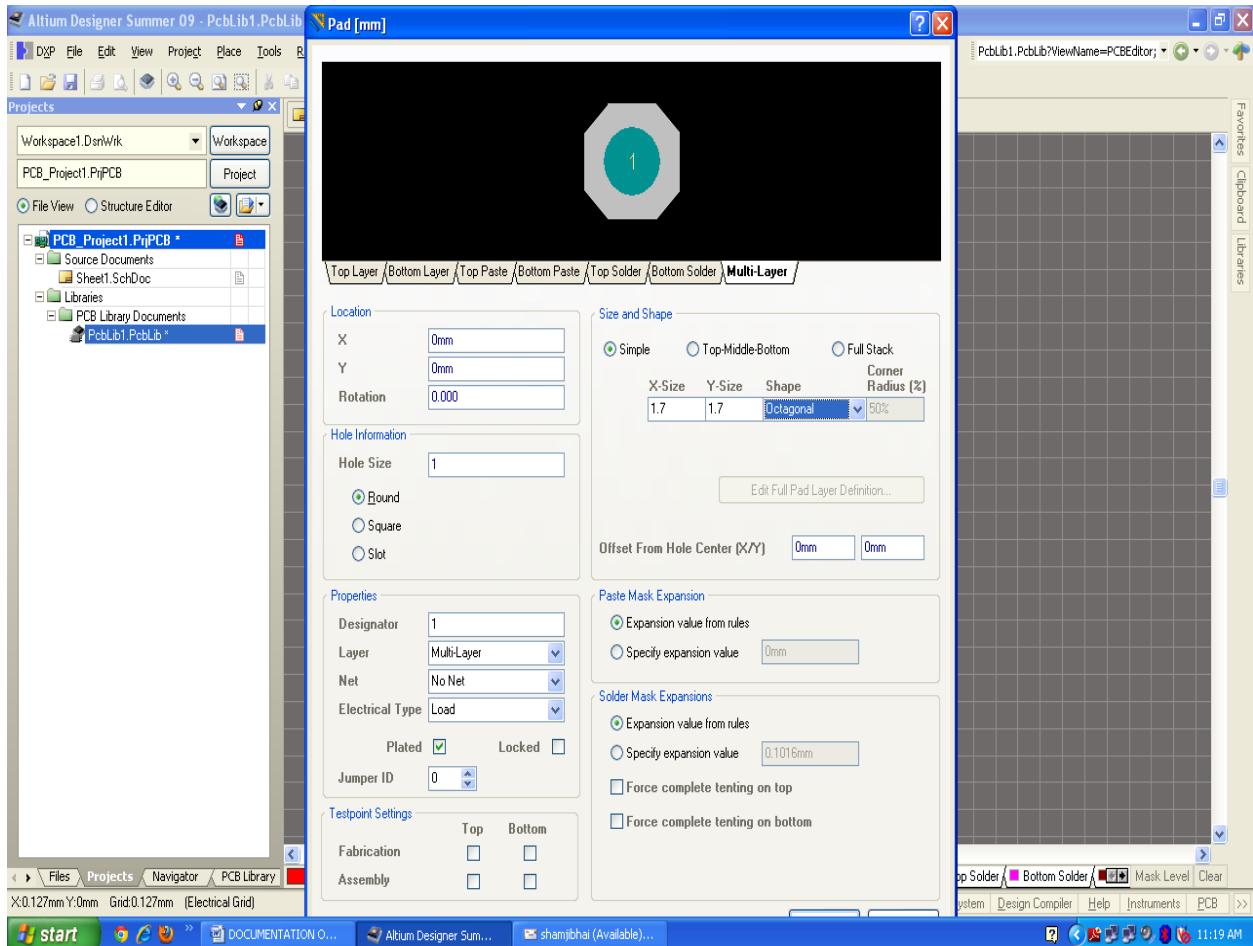
- SUPPOSE WE HAVE TO MAKE 3 PIN CONNECTOR'S FOOT PRINT.
- SO WE HAVE TO PUT 3 PEDS.
- AND ALWAYS REMEMBER THAT THERE IS 2.54mm DISTANCE BETWEEN TWO PEDS.(4 block center to center)
- PUT 3 PEDS IN THE ONE LINE WITH THE DISTANCE OF 2.54mm BETWEEN EACH OF PEDS.
- Put 7.62 mm between two column of peds.(12 blocks center to center)



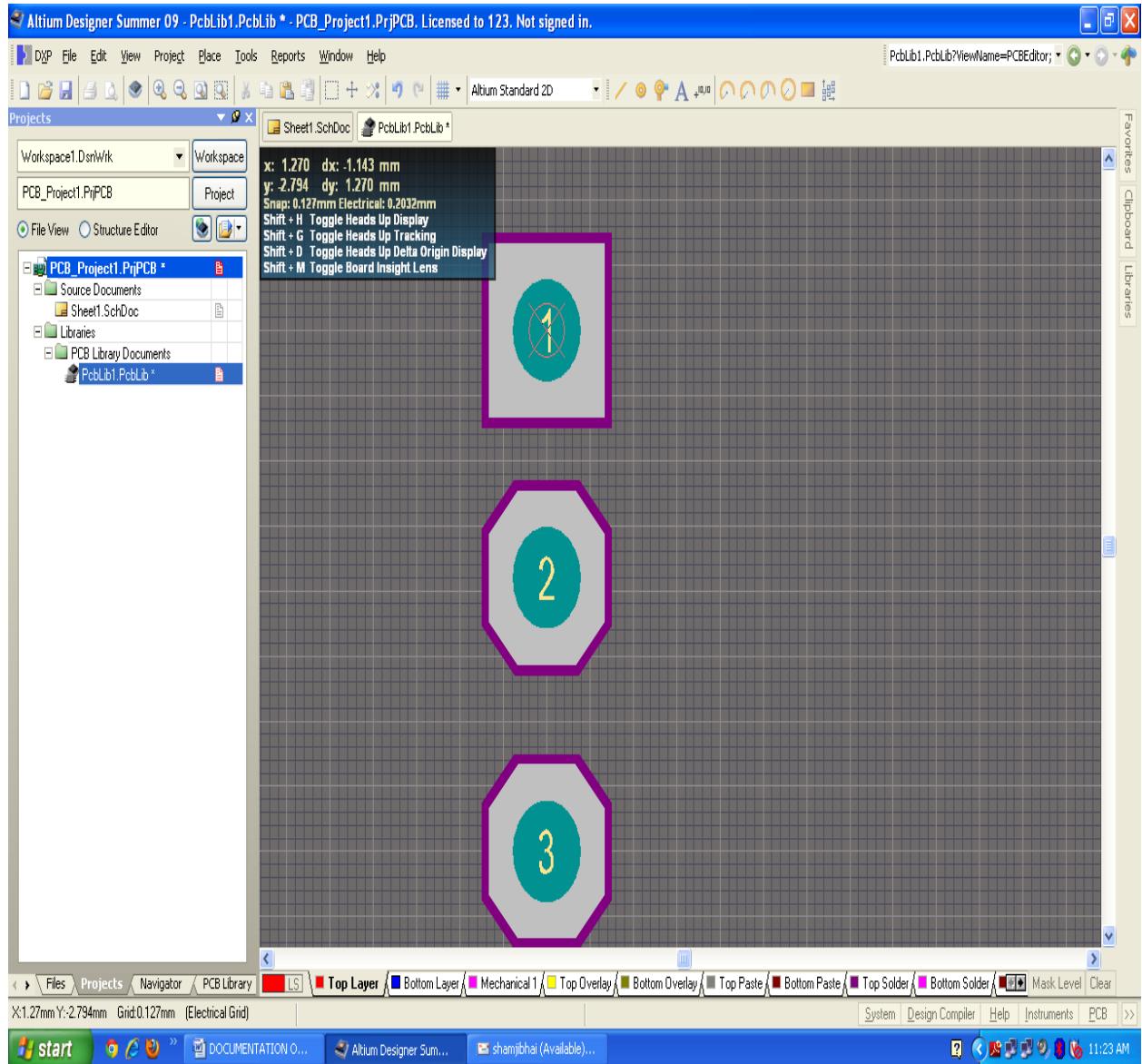
- NOW BY DOUBLE CLIKING ON THE PED WE CAN CHANGE THE DESIGNATOR OF IT AND ALSO CHANGE THE HOLE SIZE, X-SIZE AND Y-SIZE AS SHOWN IN BELOW FIG.



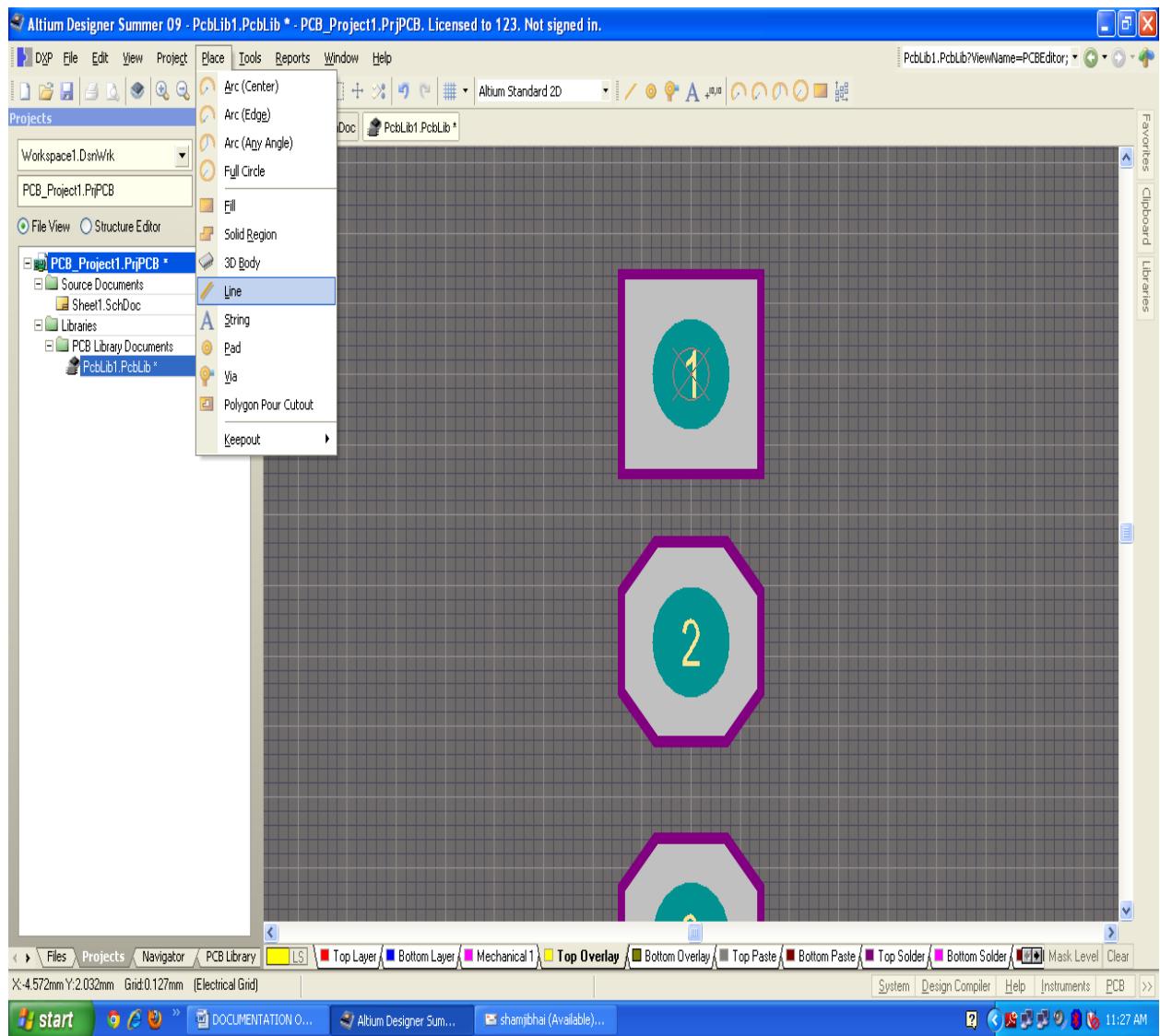
- MAKE THE HOLE SIZE ONE mm, X-SIZE AND Y-SIZE 1.7 mm AND ALWAYS MAKE THE SHAPE OF PED OCTAGONAL BECAUSE WE CANT TRANSFER ROUND SHAPE TO COREL DRAW TO PRINT THE PCB SO WE HAVE TO ALWAYS CARE FULL ABOUT THE SHAPE OF THE PED AND IT MUST BE OCTAGONAL OR RACTANGLE.



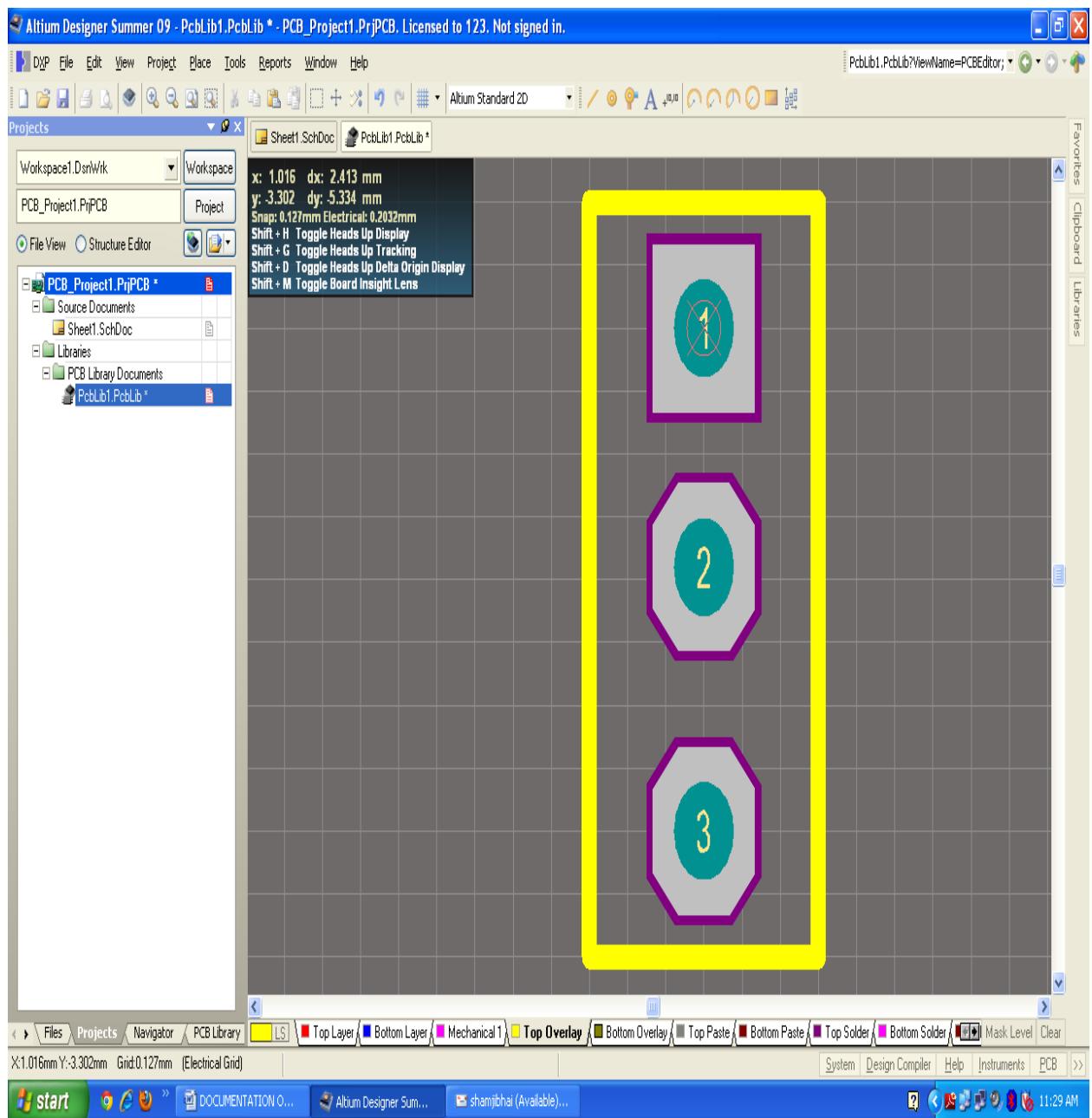
- FOLLOW THE ABOVE PROCEDURE WITH ALL THE PEDS AND GIVE APPROPRIAT NAME TO ALL.
- FINALY MAKE THE 1ST PED'S SHAPE TO RACTANGLE TO IDENTIFY THE 1ST NO PIN EASILY AS SHOWN IN BELOW FIG.



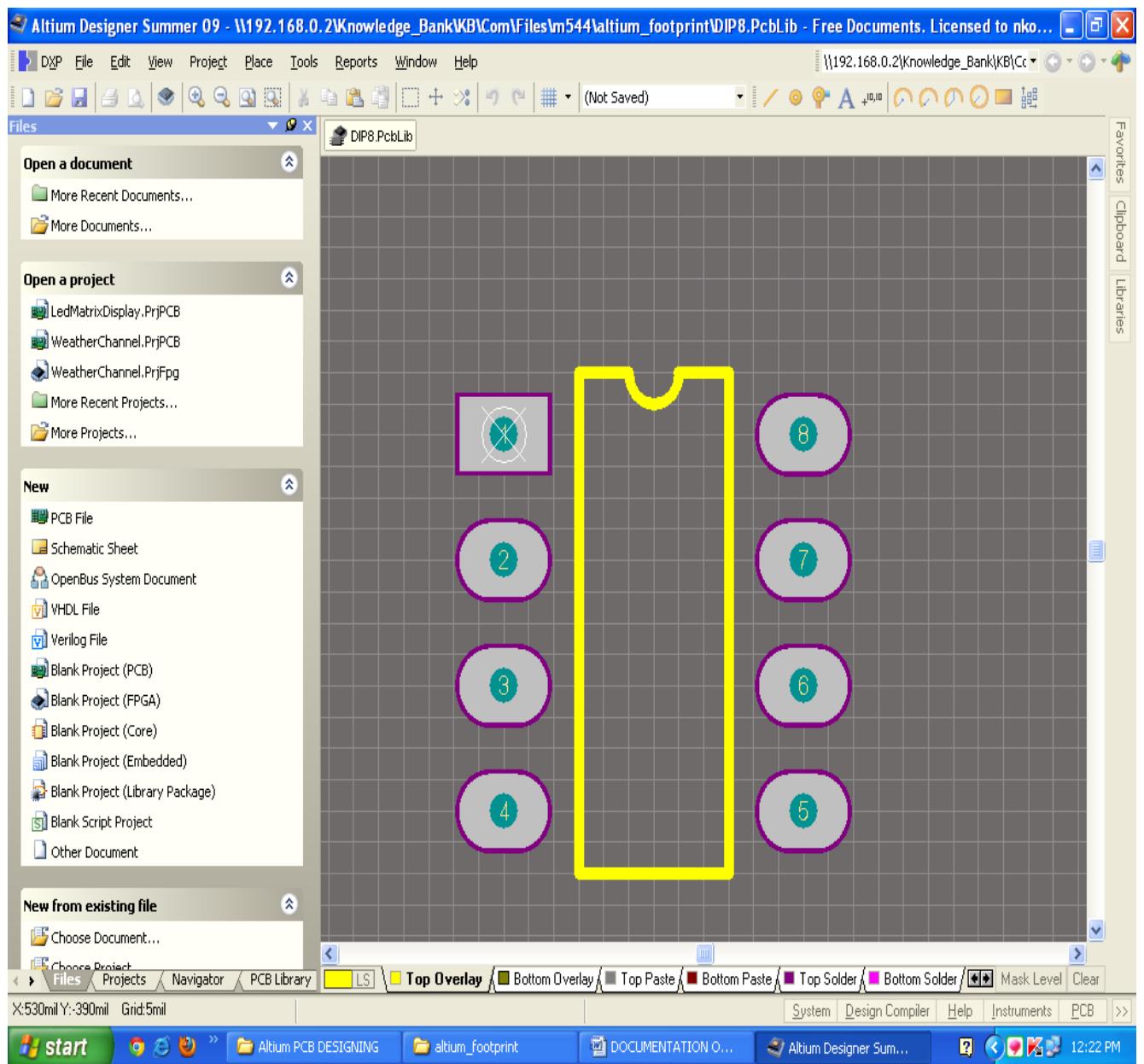
- NOW FROM THE PLACE LINE TOOL MAKE THE FANCING LINE TO ALL THE THREE PEDS AS SHOWN IN BELOW FIG.



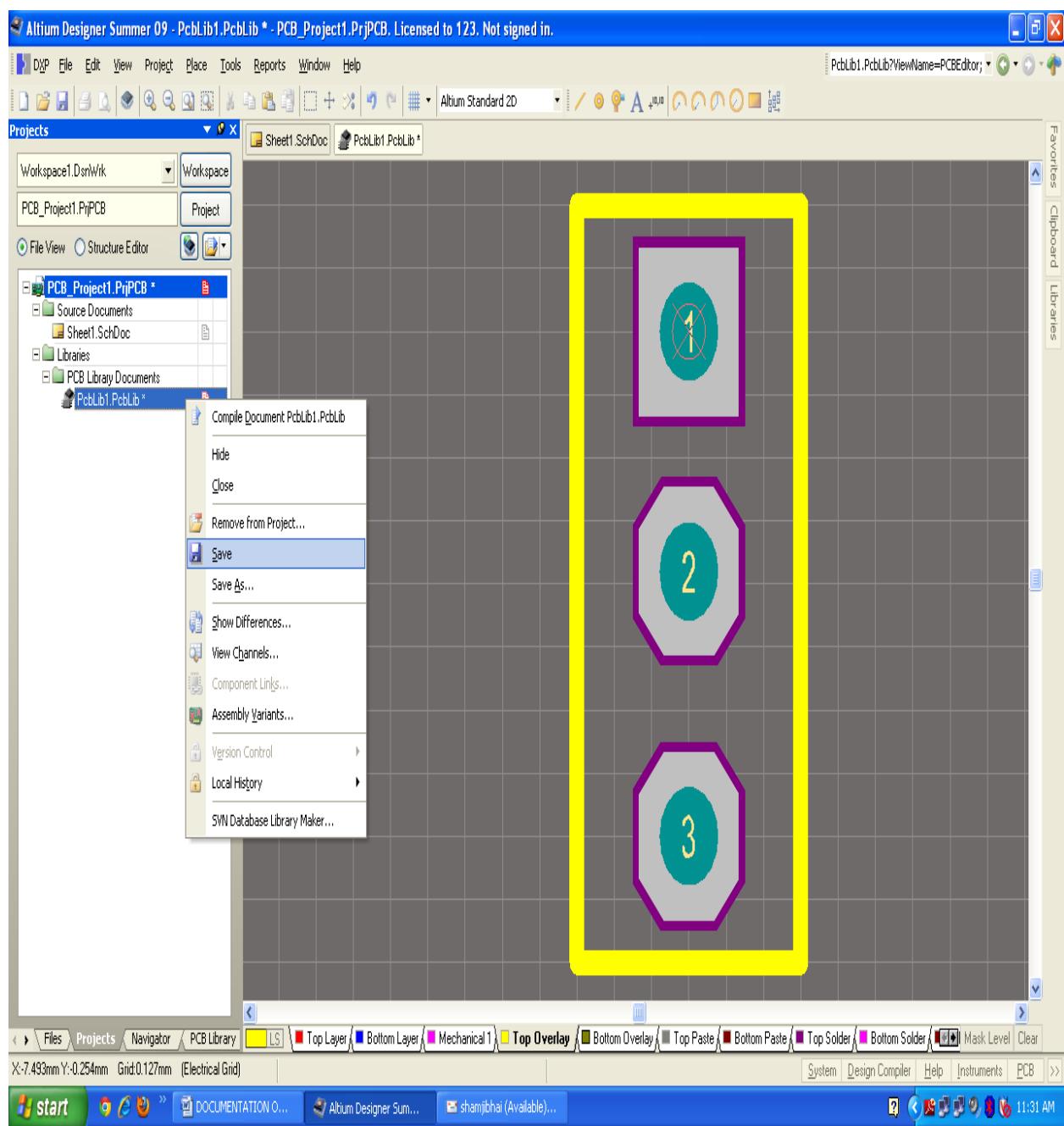
- SELECT THE TOP OVERLAY LAYER AND PUT THE YELLOW FANCING LINE LIKE THIS.



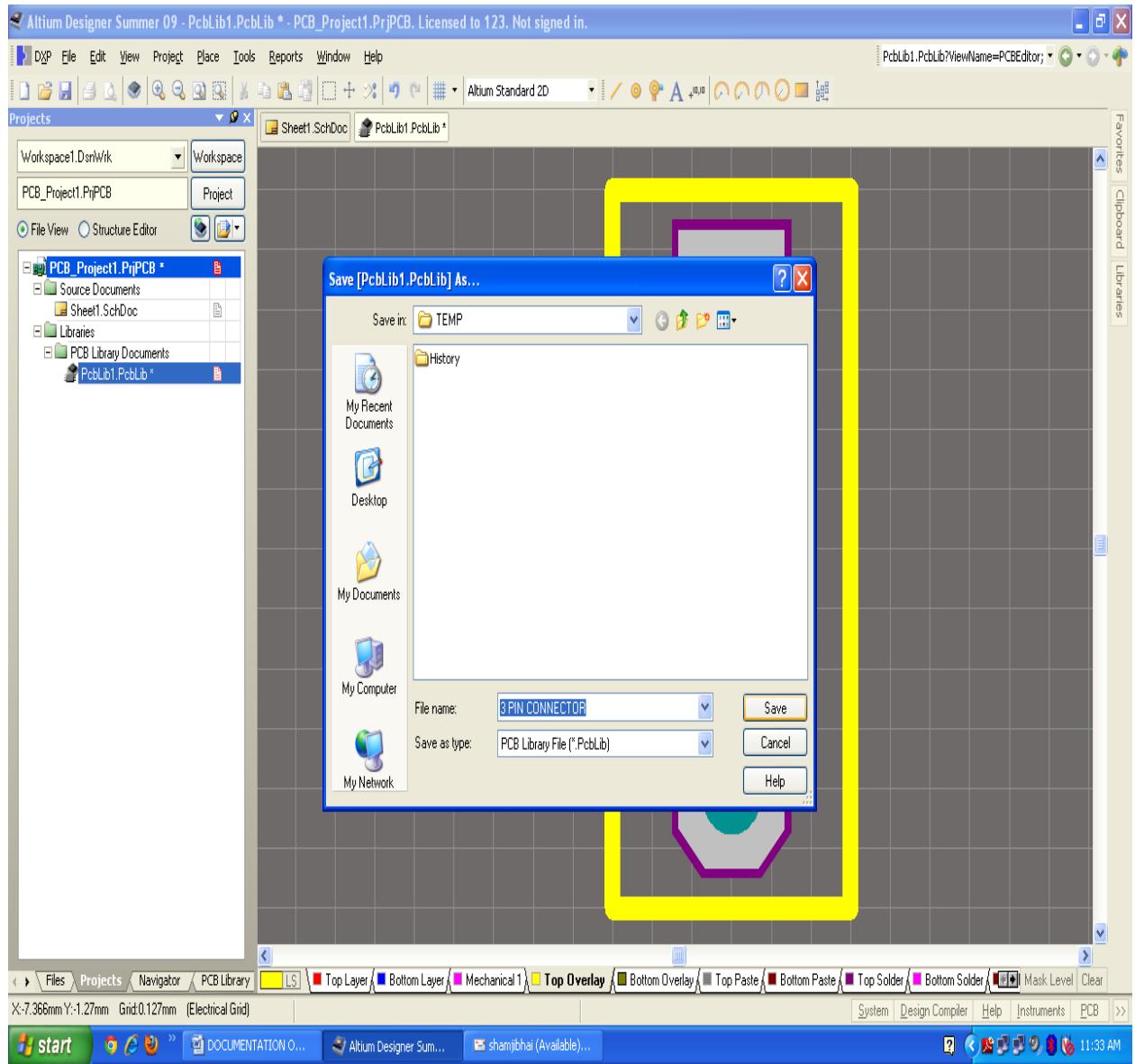
- IN THE FOOTPRINT OF IC YOU HAVE TO MAKE LINE INSIDE THE NODES SO WE CAN CONNECT ALL THE CONNECTION EASILY.
- AS PER SHOWN IN THE BELOW FIG.



➤ NOW SAVE IT WITH APPROPRIATE NAME AS PER BELOW FIG.

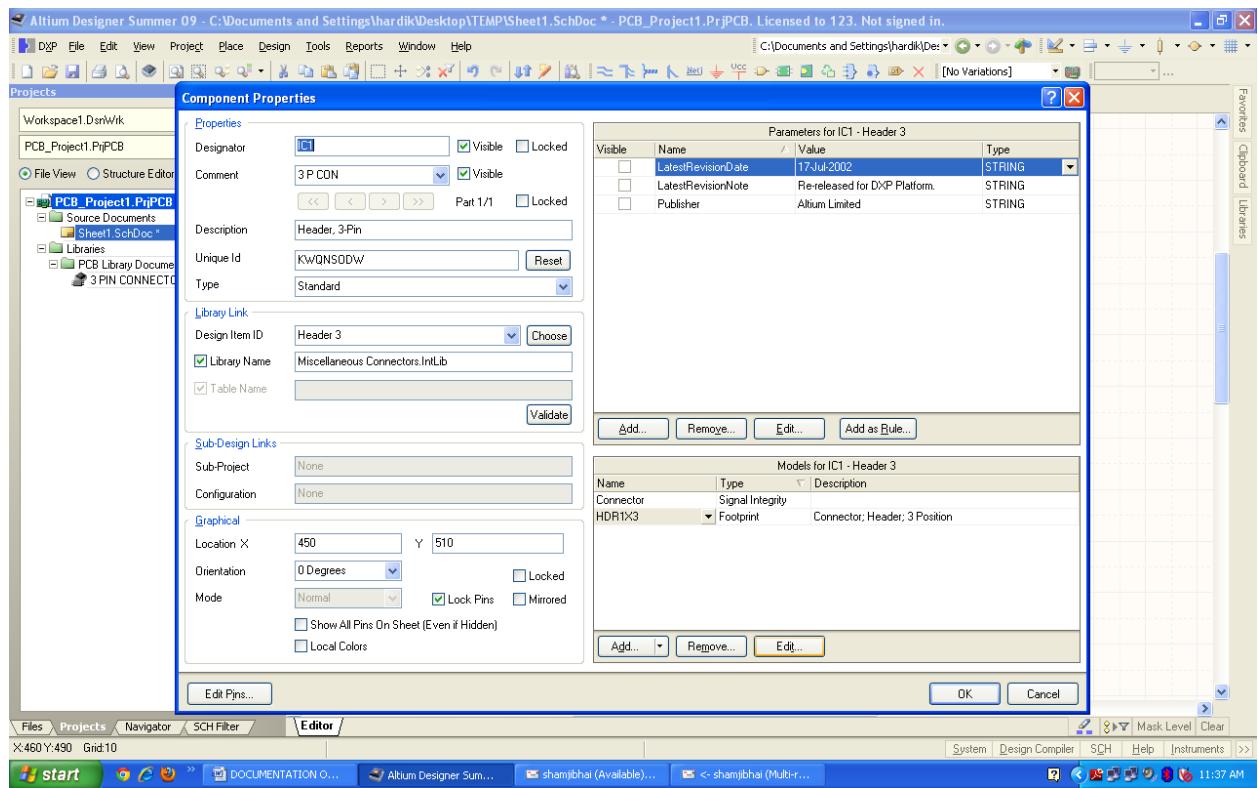


- GIVE THE NAME AS 3 PIN CONNECTOR AND SAVE IT AS PER GIVEN BELOW FIG.

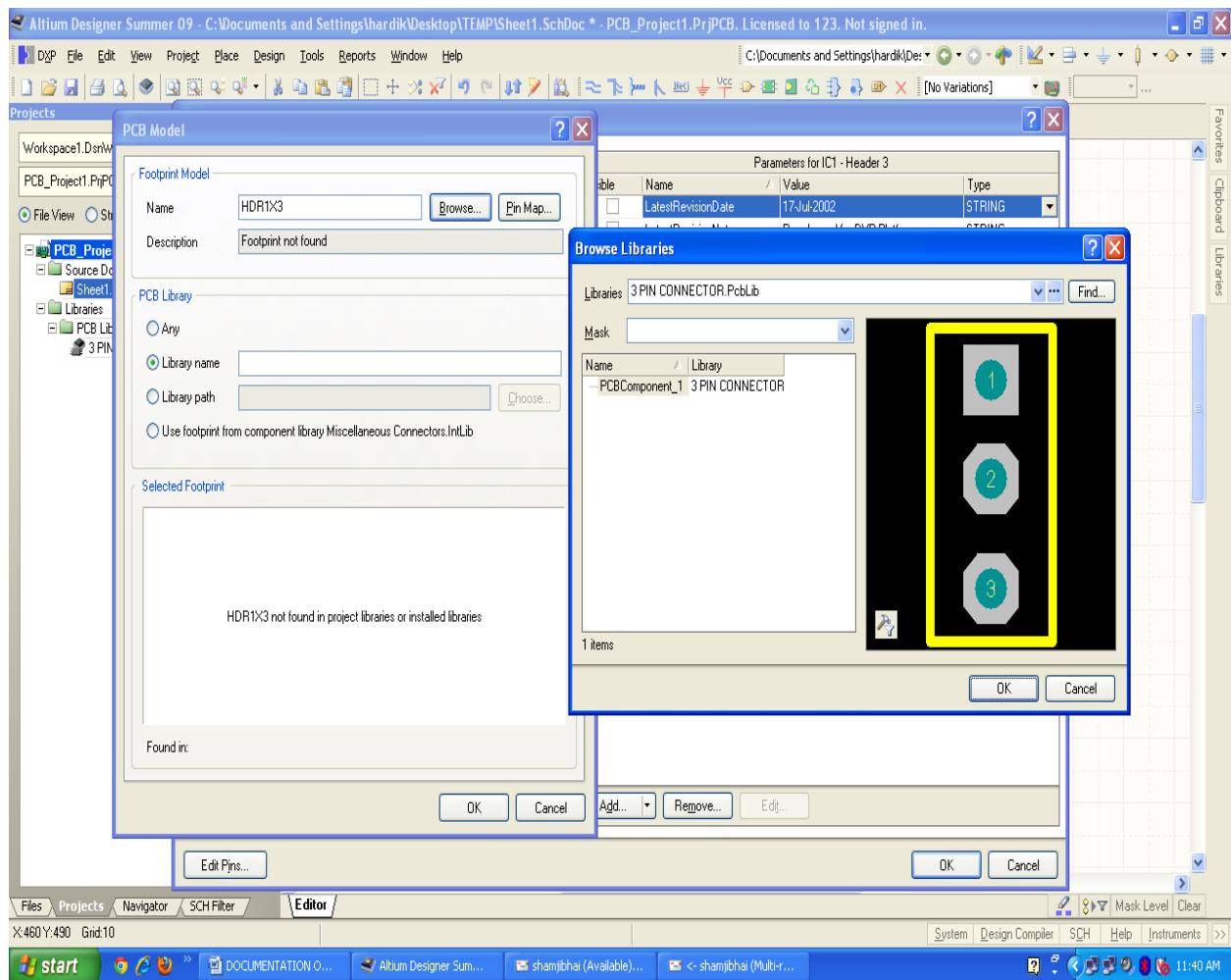


- SO LIKE THIS WE CAN MAKE ALL TYPE OF FOOT PRINTS AS PER THE NODES AND DIMENTIONS.

- BY DOUBLE CLIKING ON THE COMPONENT OR CONNECTOR IN SHCEMETIC DIAGRAM YOU CAN SEE THE WINDOW LIKE THIS.

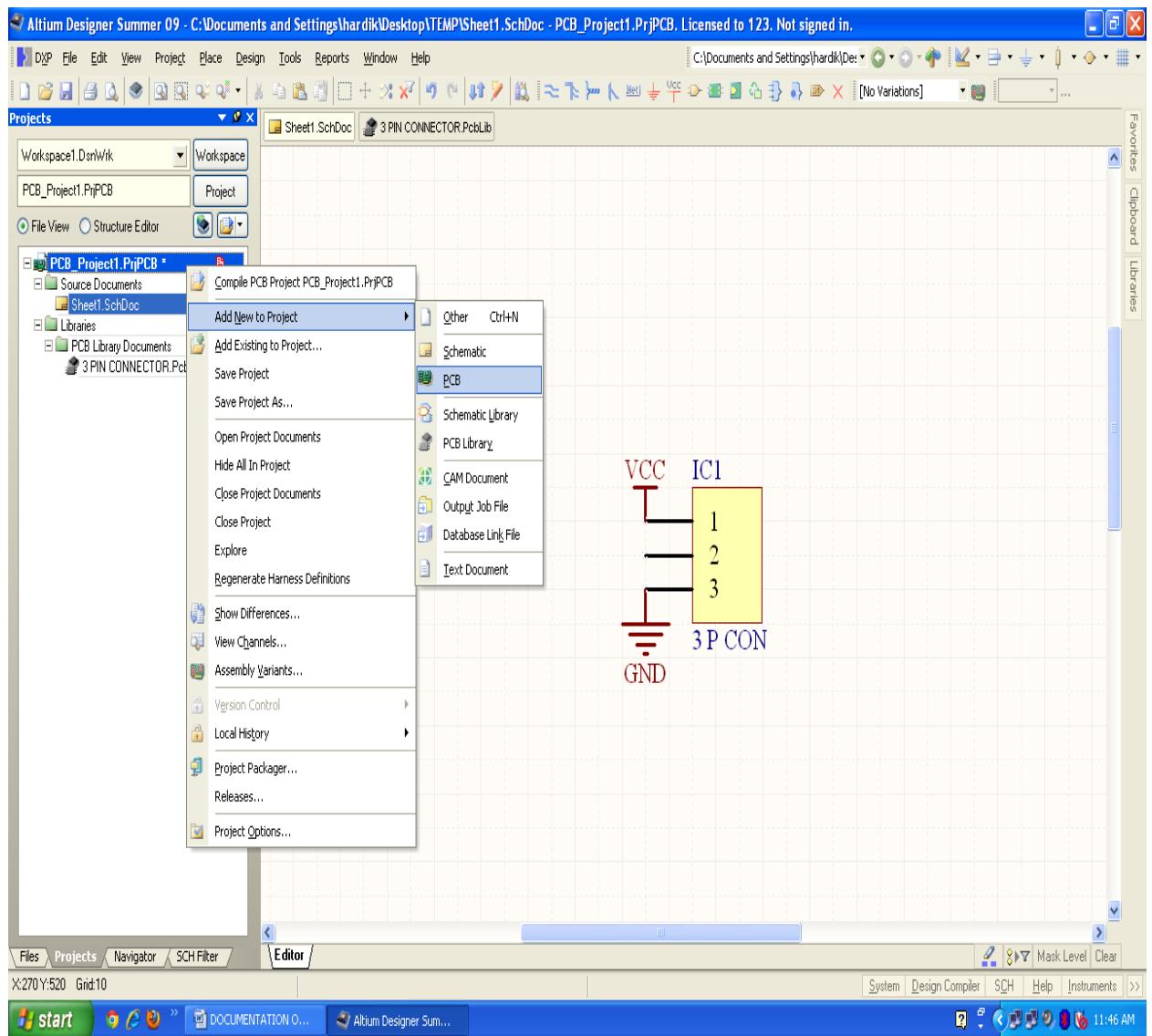


- NOW GO TO THE EDIT AND THEN CLICK ON THE LIBRARY PATH AND BROWSE THE FOOTPRINT OF 3 PIN CONNECTOR.

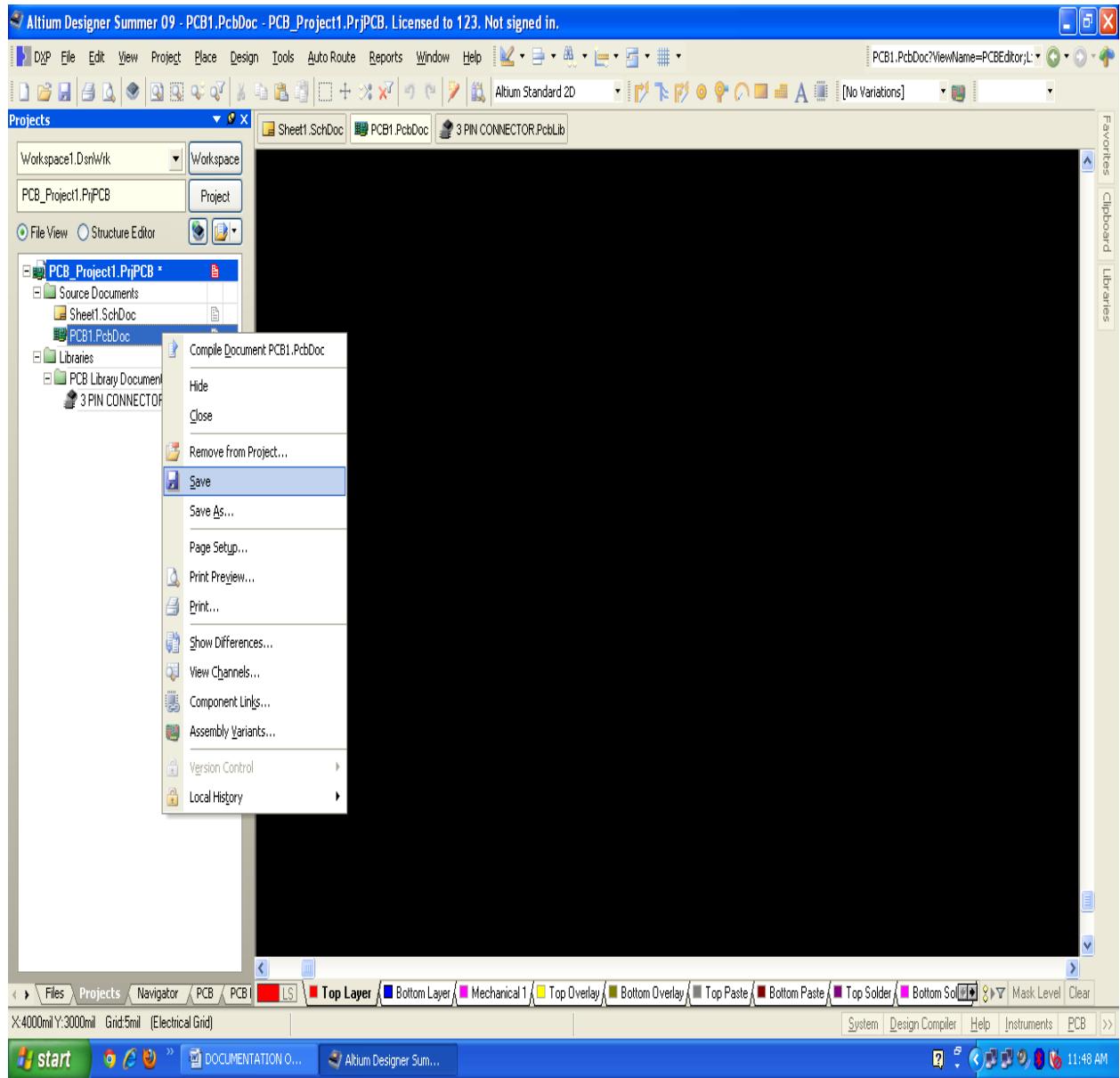


- AFTER SELECTING THE FOOTPRINT PRESS OK.
- LIKE WISE UPDATE ALL THE COMPONENTS AND CONNECTORS FOOT PRINT ONE BY ONE .

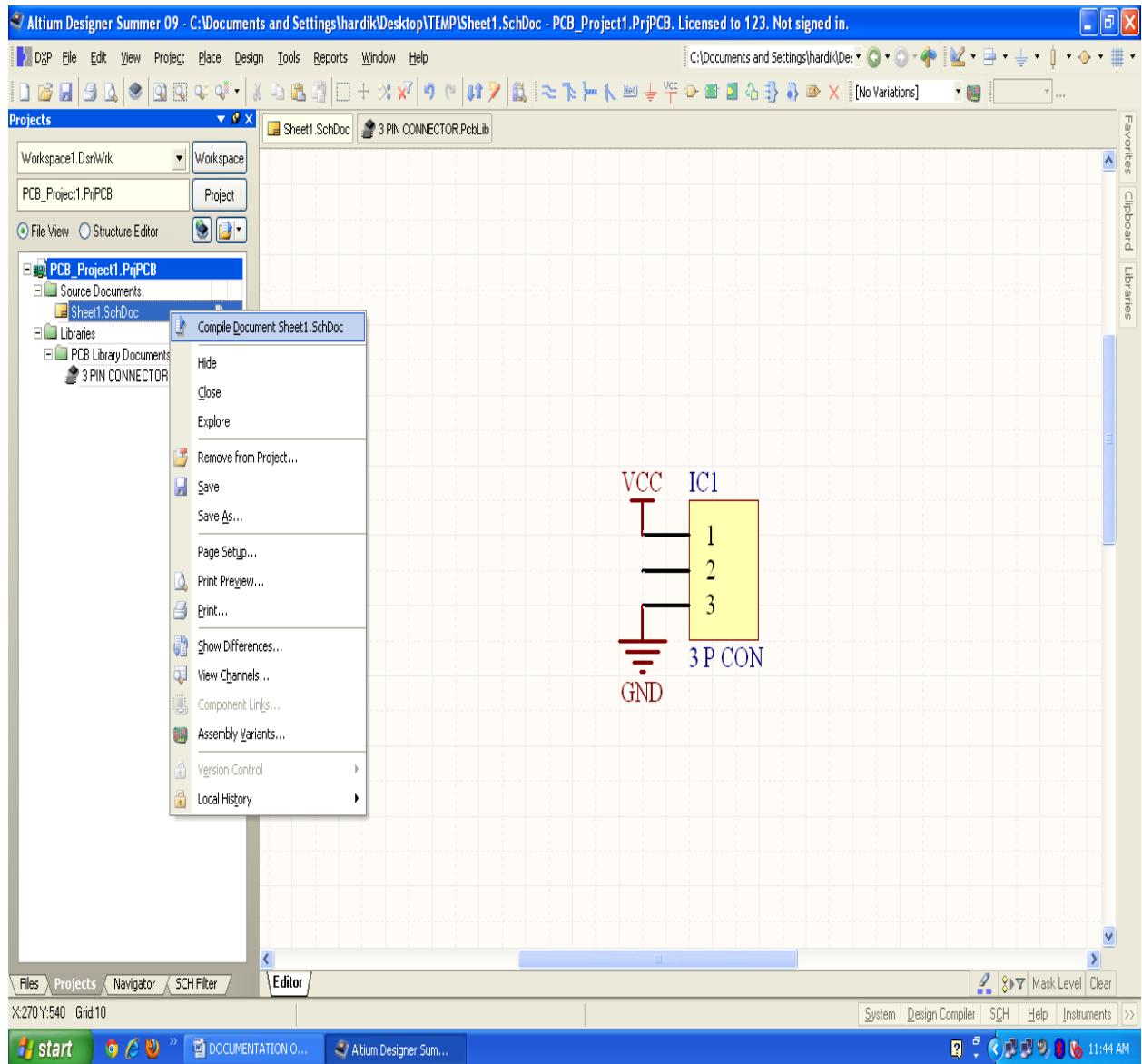
- NOW ADD PCB FILE TO THE PROJECT AS PER THE GIVEN FIG.



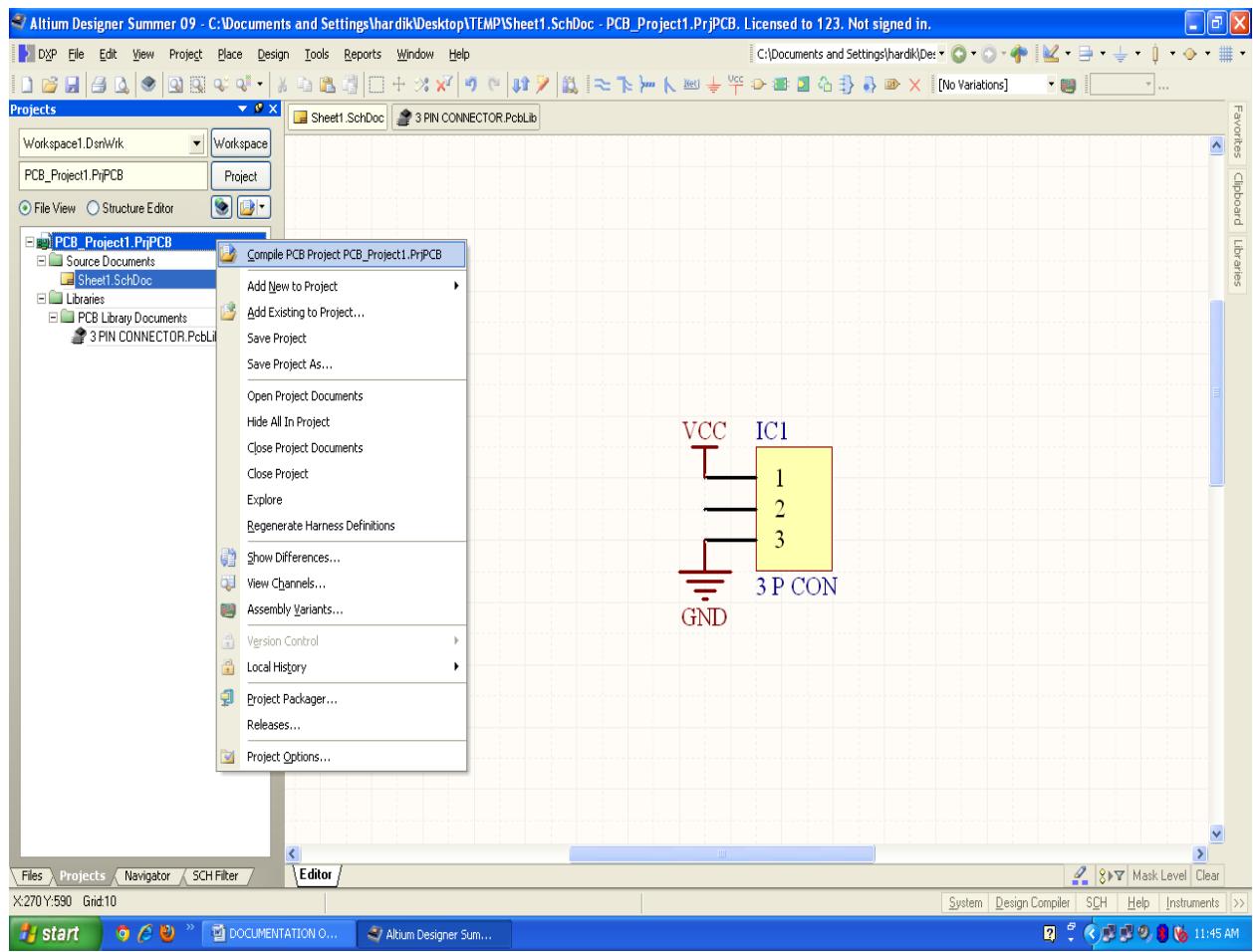
- YOU CAN SEE THE WINDOW LIKE THIS AND SAVE IT.



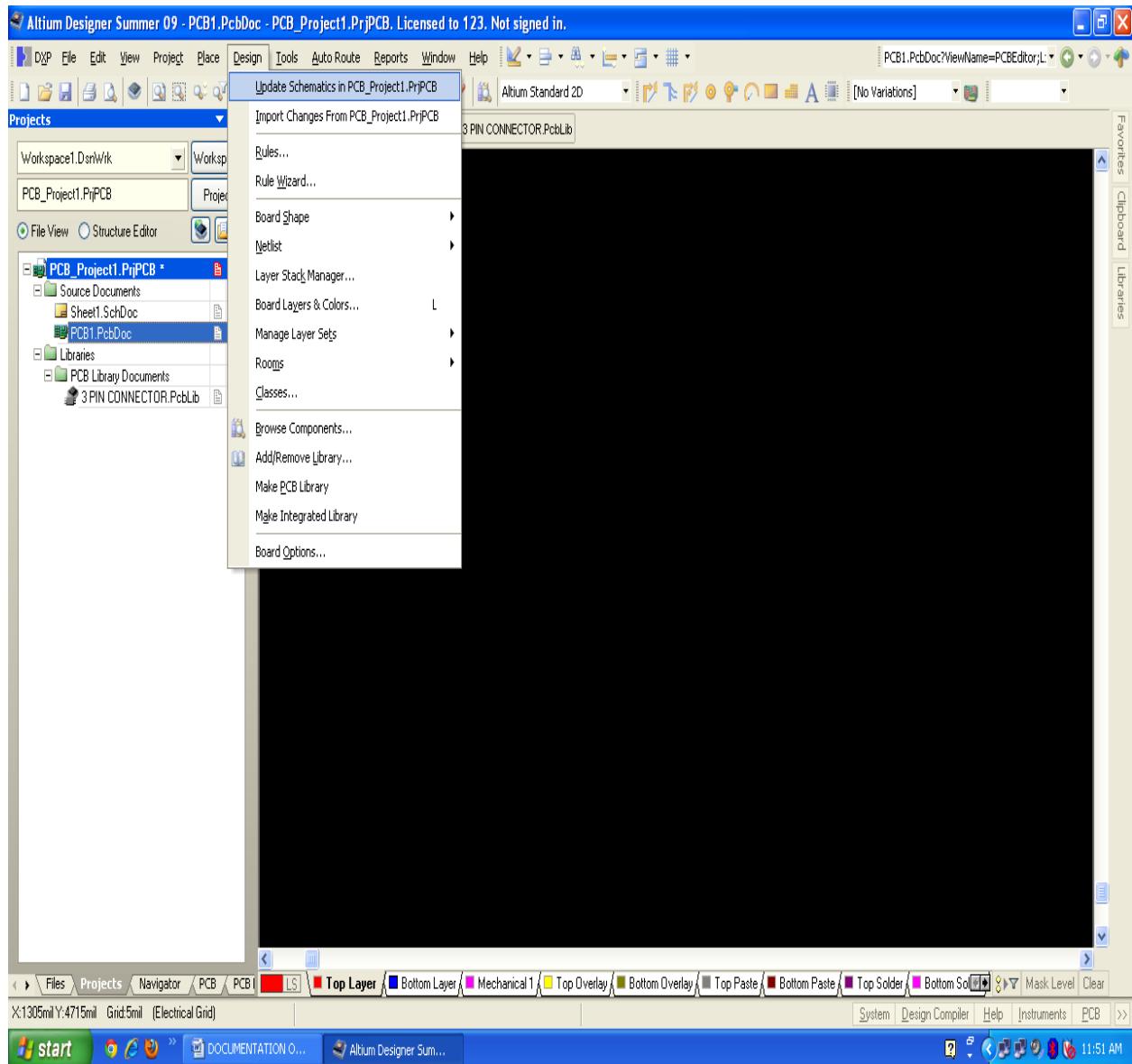
- NOW COMPILE THE SCHEMATIC DIAGRAM AS SHOWN IN BELOW FIG.



- NOW COMPILE THE PROJECT AS SHOWN IN BELOW FIG.

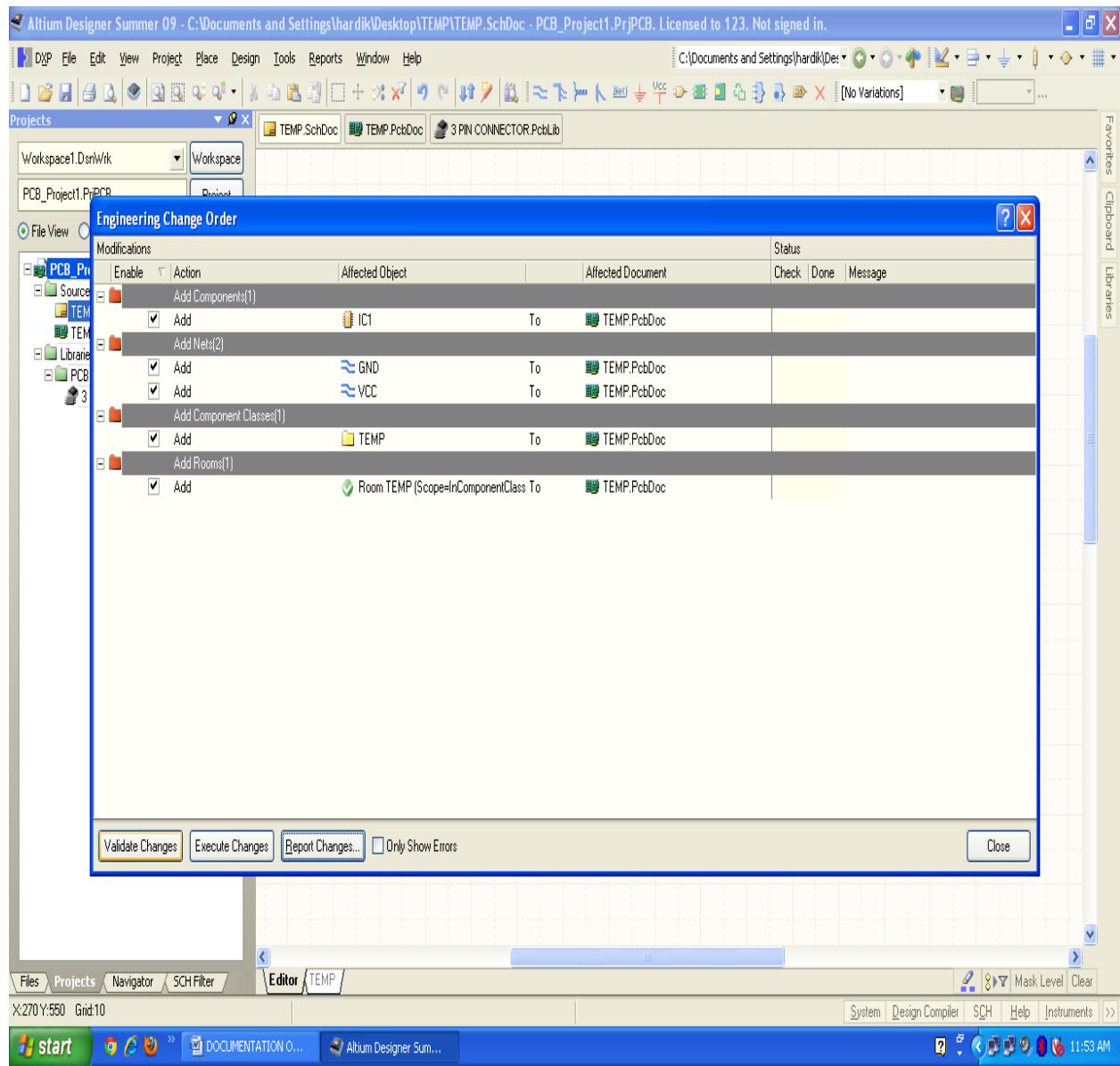


- NOW UPDATE THE SCHEMATIC ON PCB AS GIVEN BELOW FIG.

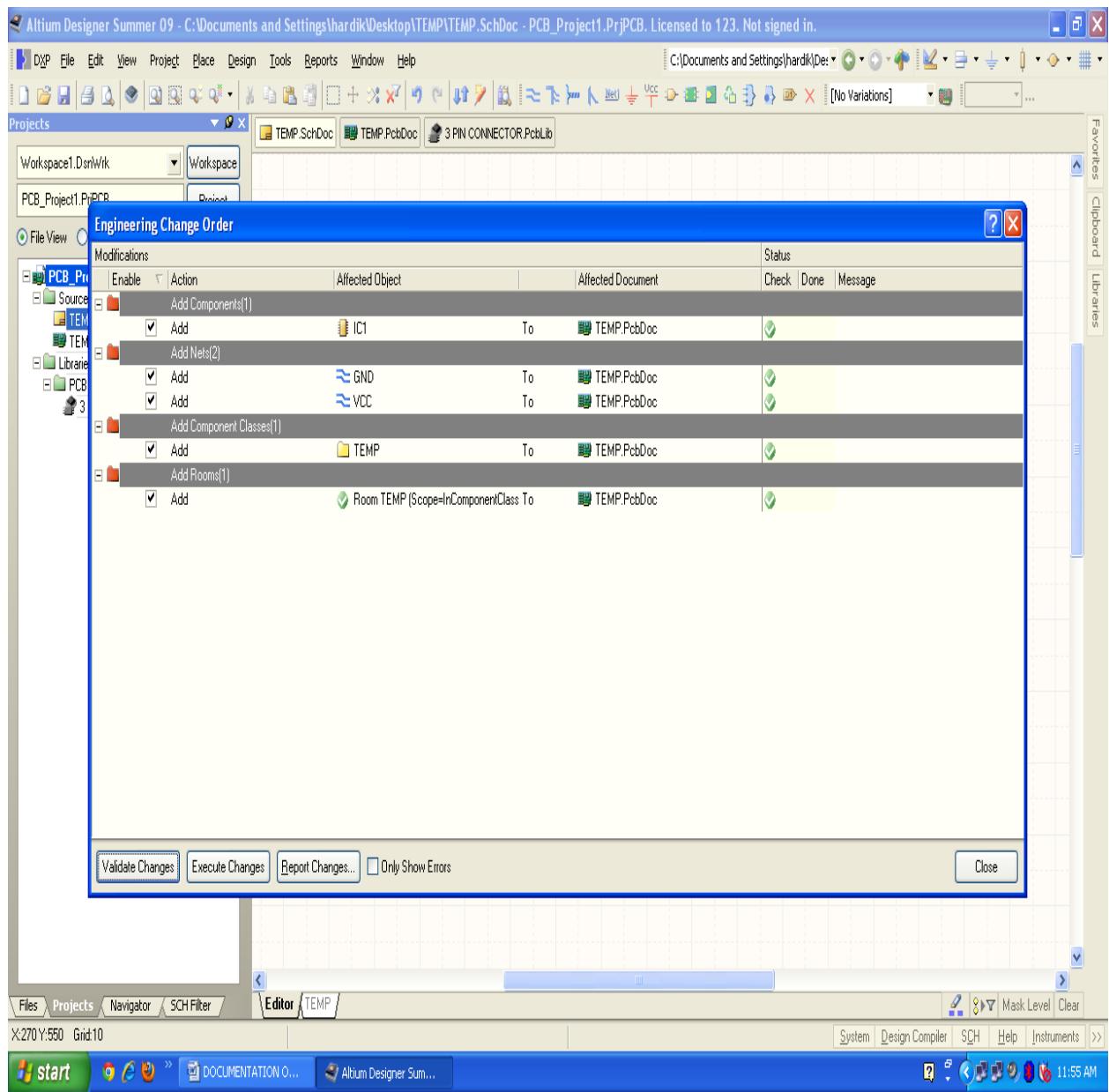


- ALWAYS REMEMBER THAT THE NAME OF SCHEMATIC AND PCB MUST BE SAME OTHERWISE SCHEMATIC CANT BE UPDATED ON THE PCB.

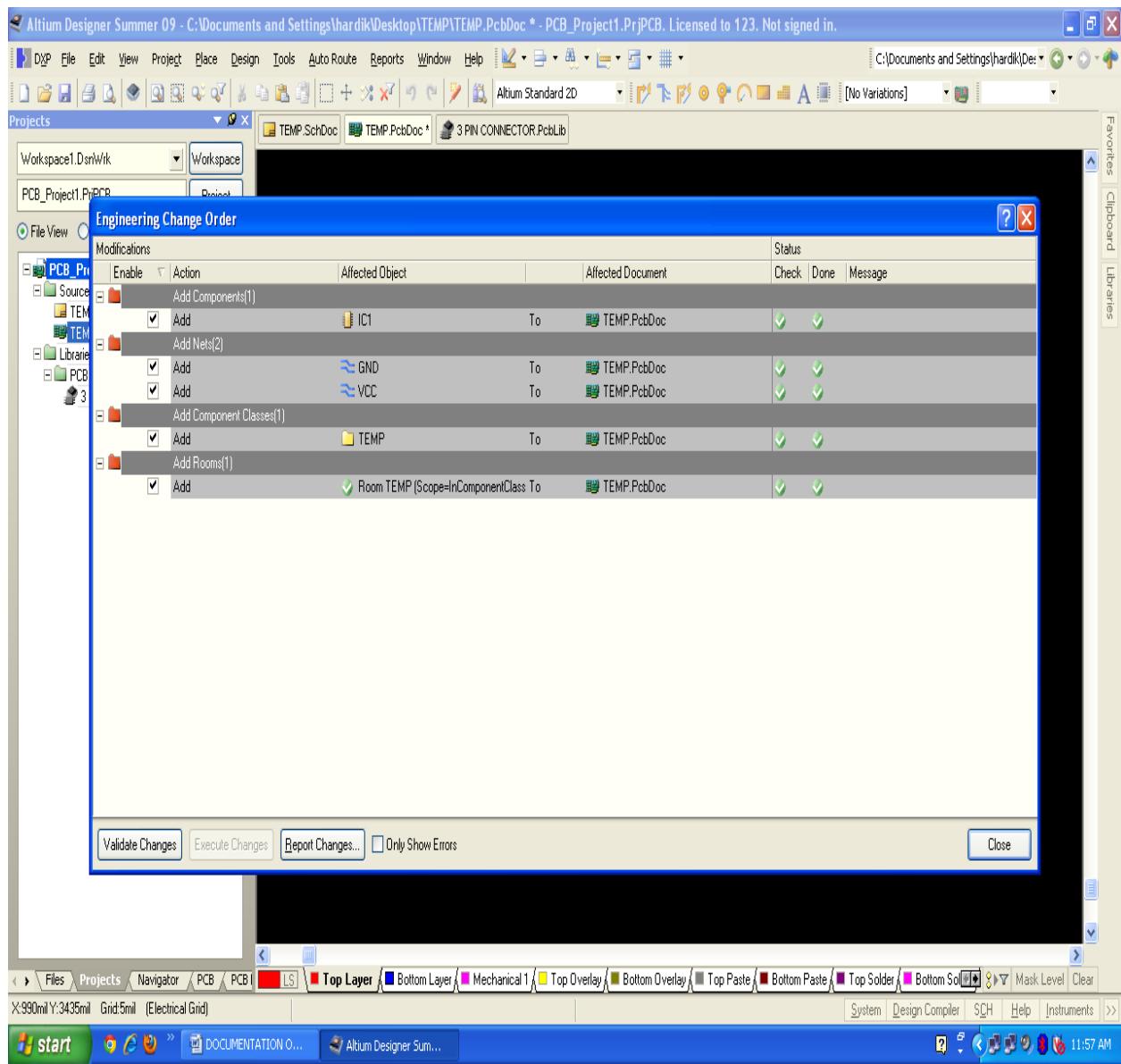
- AFTER UPDATING ON PCB YOU CAN SEE THE WINDOW LIKE THIS.
- NOW PRESS ON THE VALIDATE CHANGES.AS PER BELOW FIG.



- AFTER PRESSING VALIDATE CHANGE YOU CAN SEE THE STATUS CHECK AND ALL ARE RIGHT.

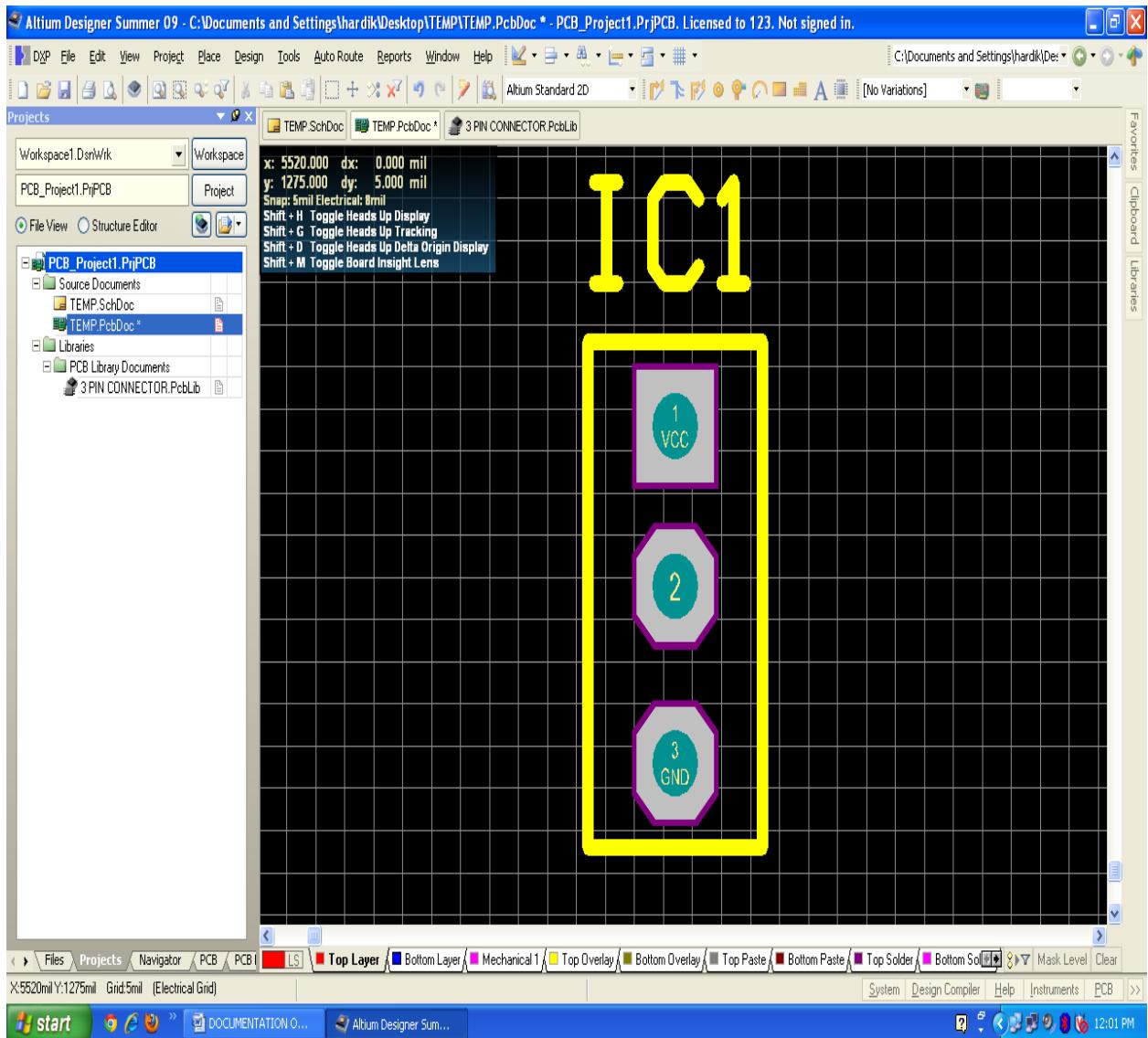


- NOW PRESS ON EXECUTE CHANGES AND SEE THE STATUS CHECK COLUM.



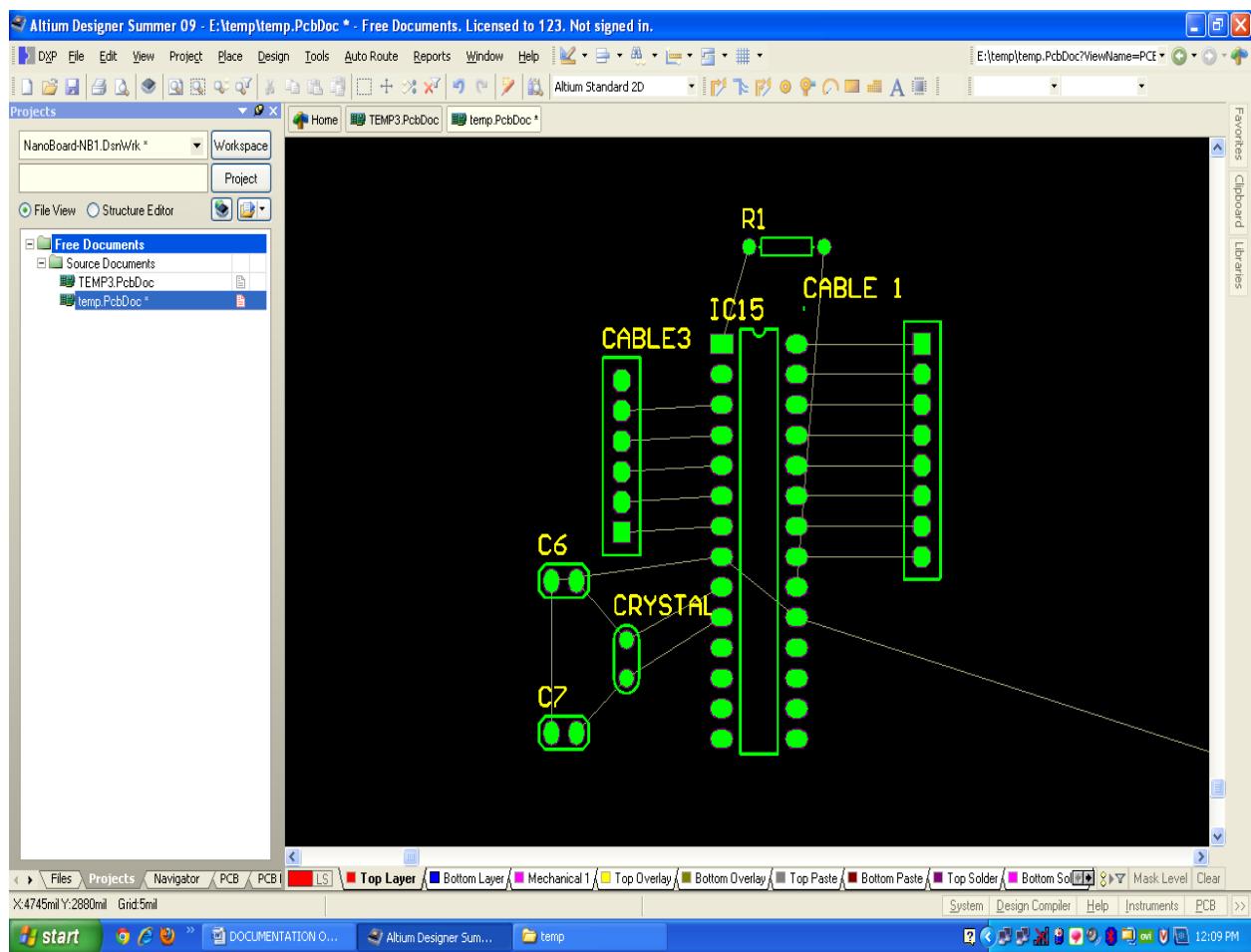
- ONE THING YOU HAVE TO REMEMBER IS IN EXECUTE CHANGES ALL THE STATUS CHECK ARE MUST BE RIGHT IF SOME THING WRONG IN SCHEMATIC IT WILL SHOWN AS WRONG MARK ON STATUS CHECK AT THAT TIME YOU HAVE TO SOLVE THE QUERY AND AGAIN COMPILE AND UPDATE TO PCB.

- AFTER UPDATING ON PCB YOU CAN SEE THE LIKE THIS.

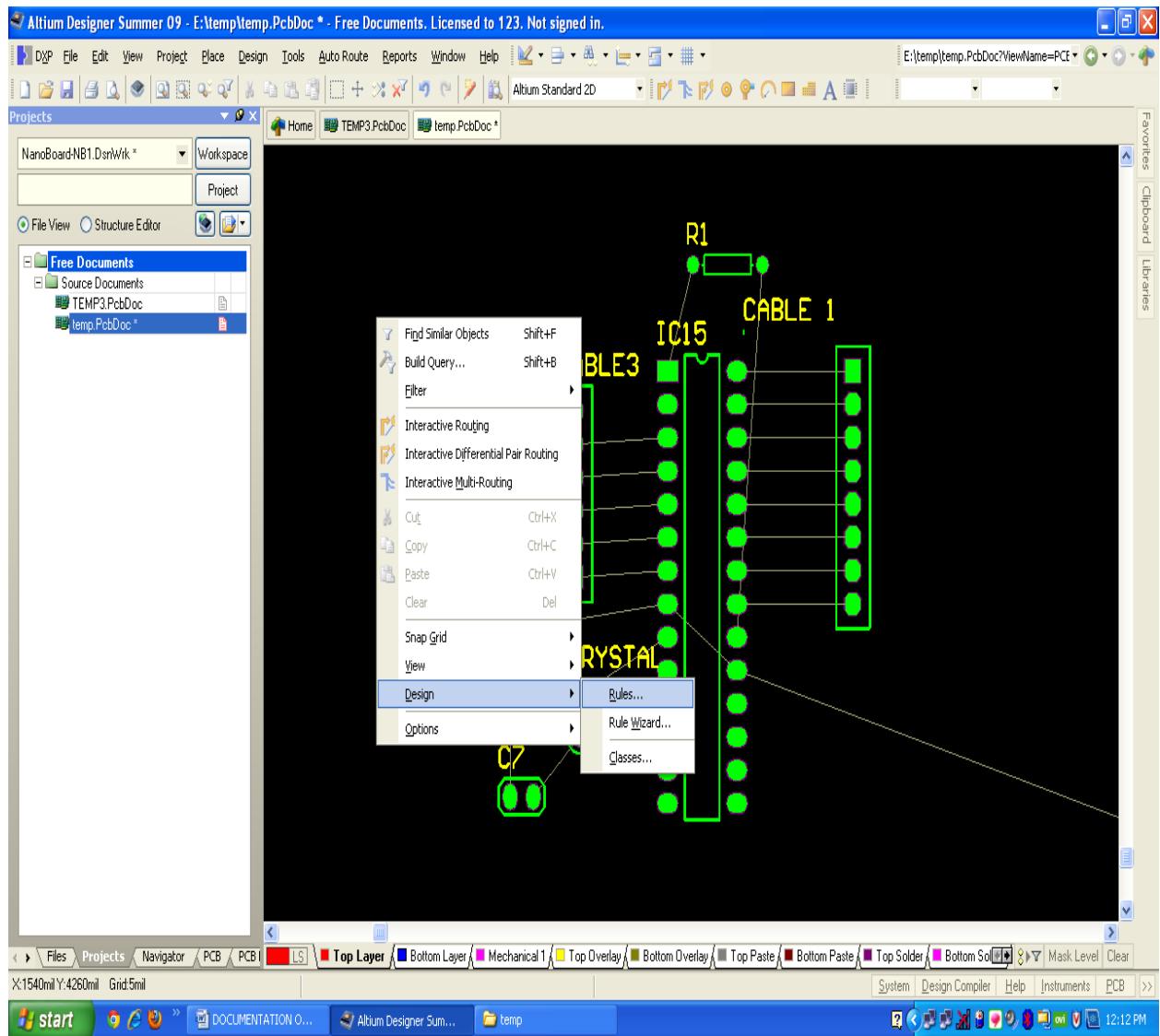


- AFTER UPDATING ALL THE COMPONENTS AND CONNECTORS TO THE PCB YOU HAVE TO JOIN ALL THE REQUIRED PEDS BY BLUE LINE (IN BOTTOM LAYER).
- BY THIS YOU CAN DESIGN YOUR PCB.

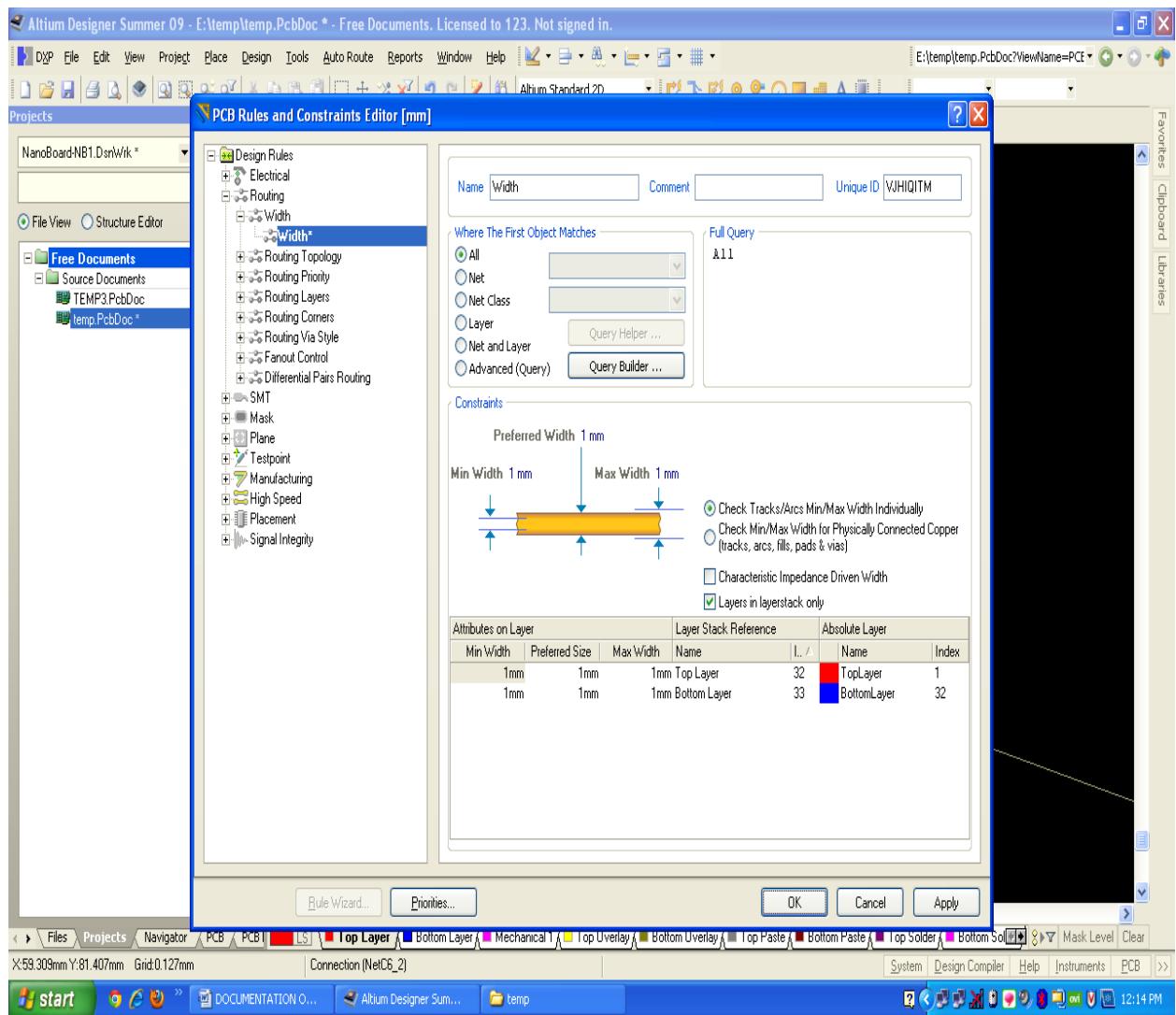
- NOW ALL THE COMPONENTS ARE ON THE PCB NOW WE HAVE SET THEM NEAR TO EACH OTHER TO REDUCE THE PCB SIZE.
- NOW AS SHOWN IN BELOW FIG PLACE ALL THE COMPONENTS AND CONNECTOR VERY NEAR TO EACH OTHER.



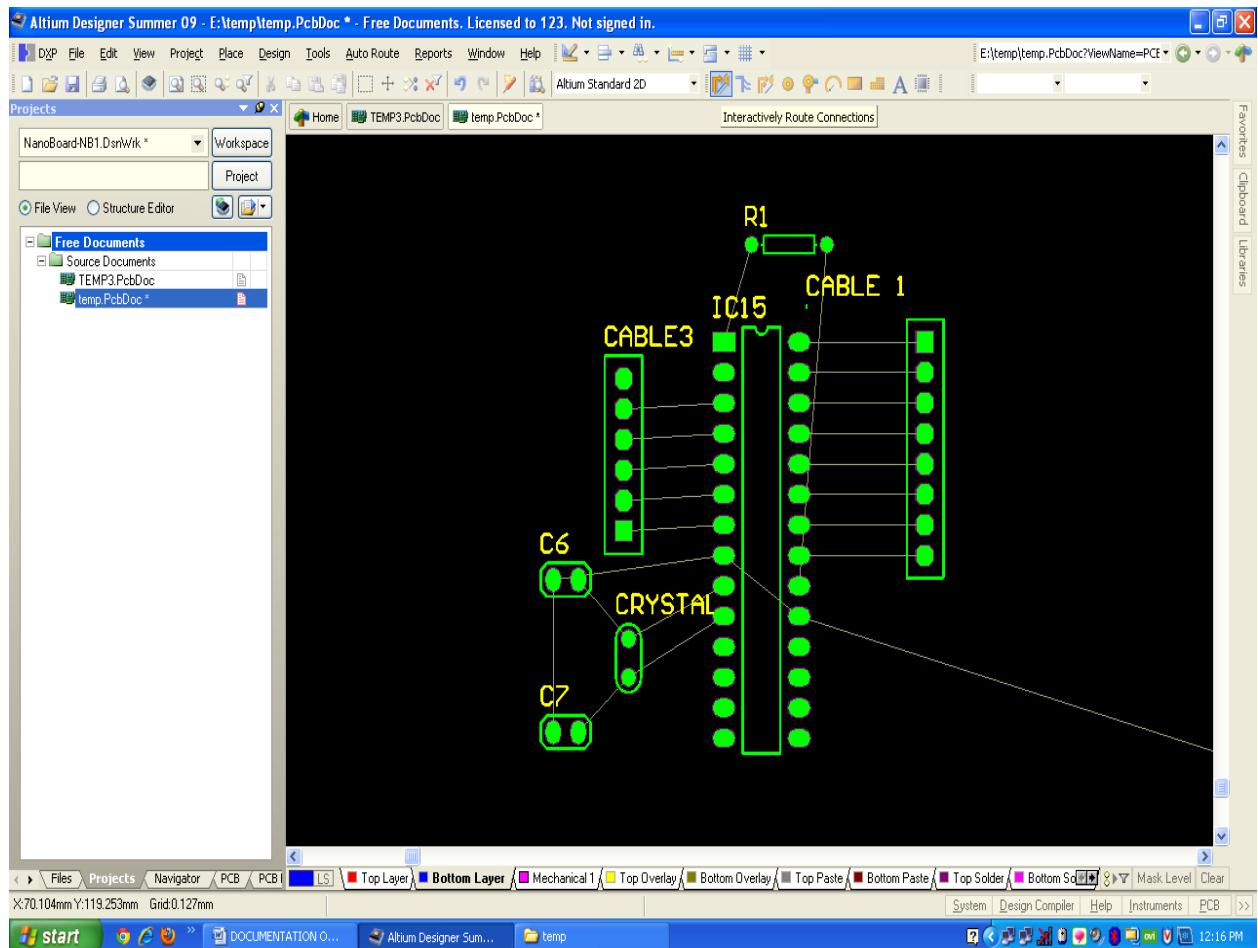
- NOW JOIN ALL THE CONNECTION IN BOTTOM LAYER WITH BLUE LINE.
- HERE THE LINE WIDTH IS VERY LOW SO WE HAVE TO INCREASE THE LINE WIDTH AS SHOWN IN BELOW FIG.
- RIGHT CLICK ON THE PCB AND GO TO DESIGN AND THEN RULES AS SHOWN IN FIG.



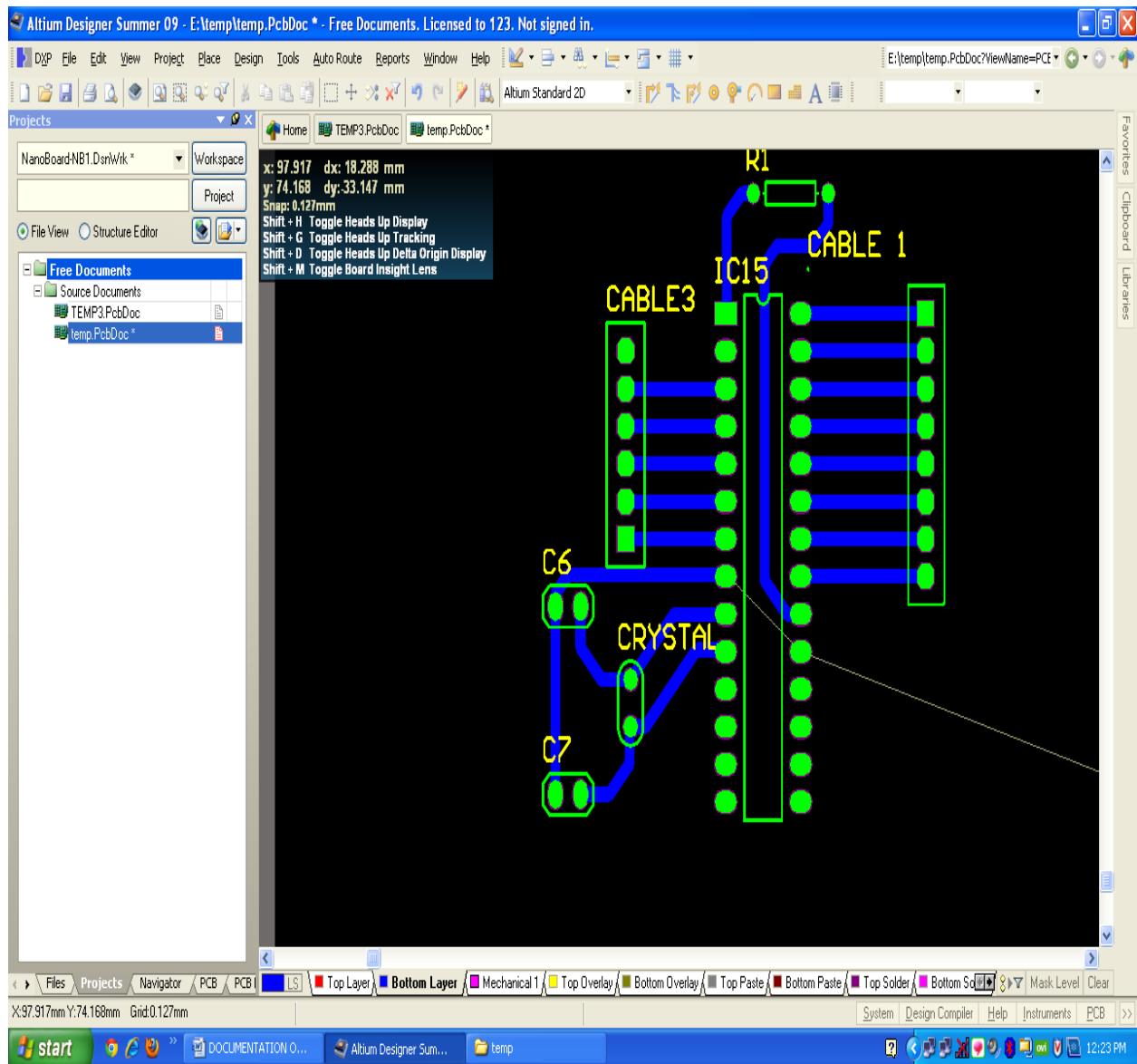
- THEN GO TO WIDTH AND MAKE PREFERRED WIDTH, MIN WIDTH AND MAX WIDTH TO 1 MM SIZE AND THEN CLICK ON APPLY AND OK.
- ALWAYS CHECK THAT ALL THE SIZE MUST BE IN MM ONLY.



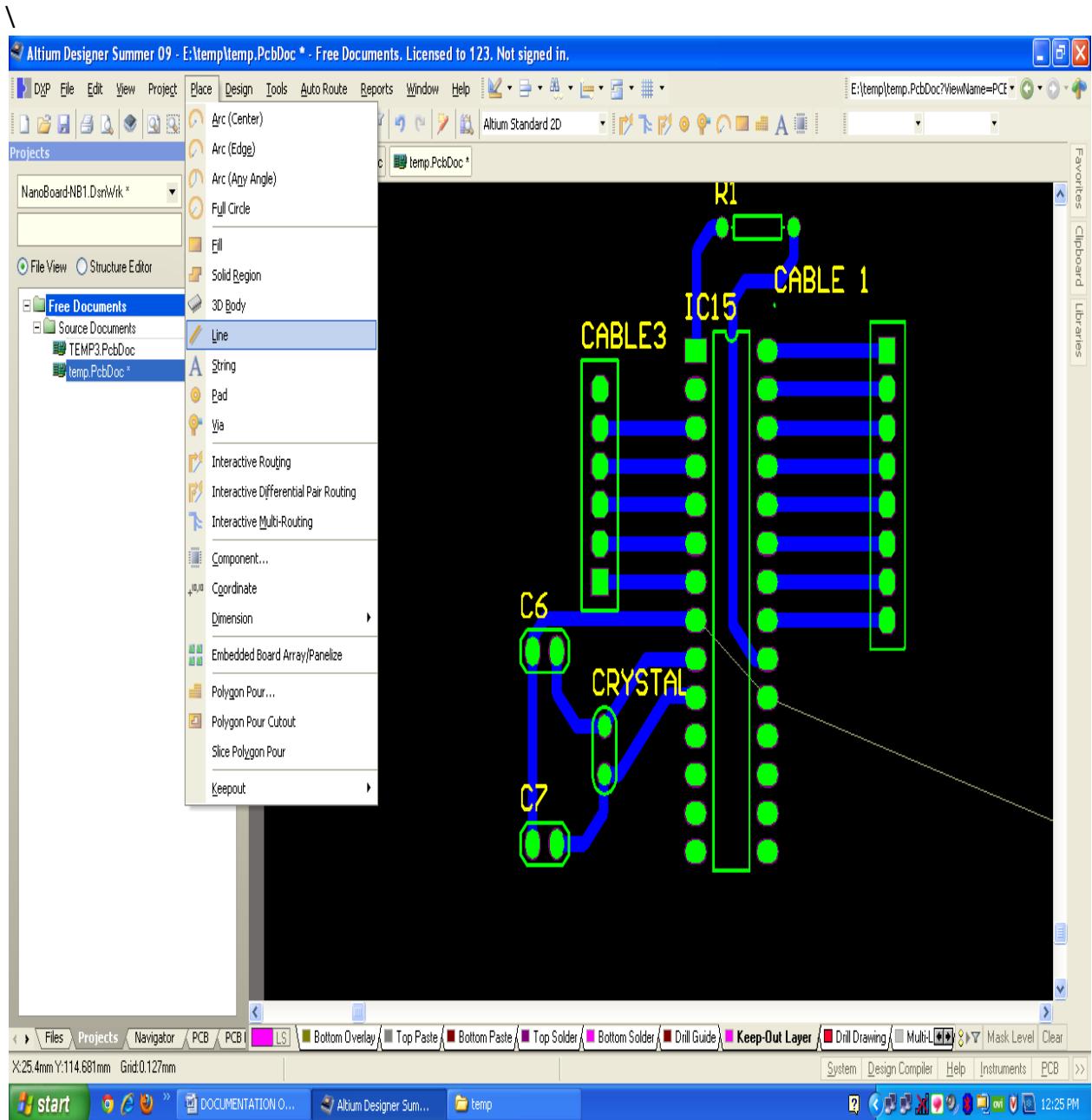
- NOW SELECT BOTTOM LAYER AND THEN SELECT INTERECTIVLY ROUTE CONNECTION AS SHOWN IN BELOW FIG.



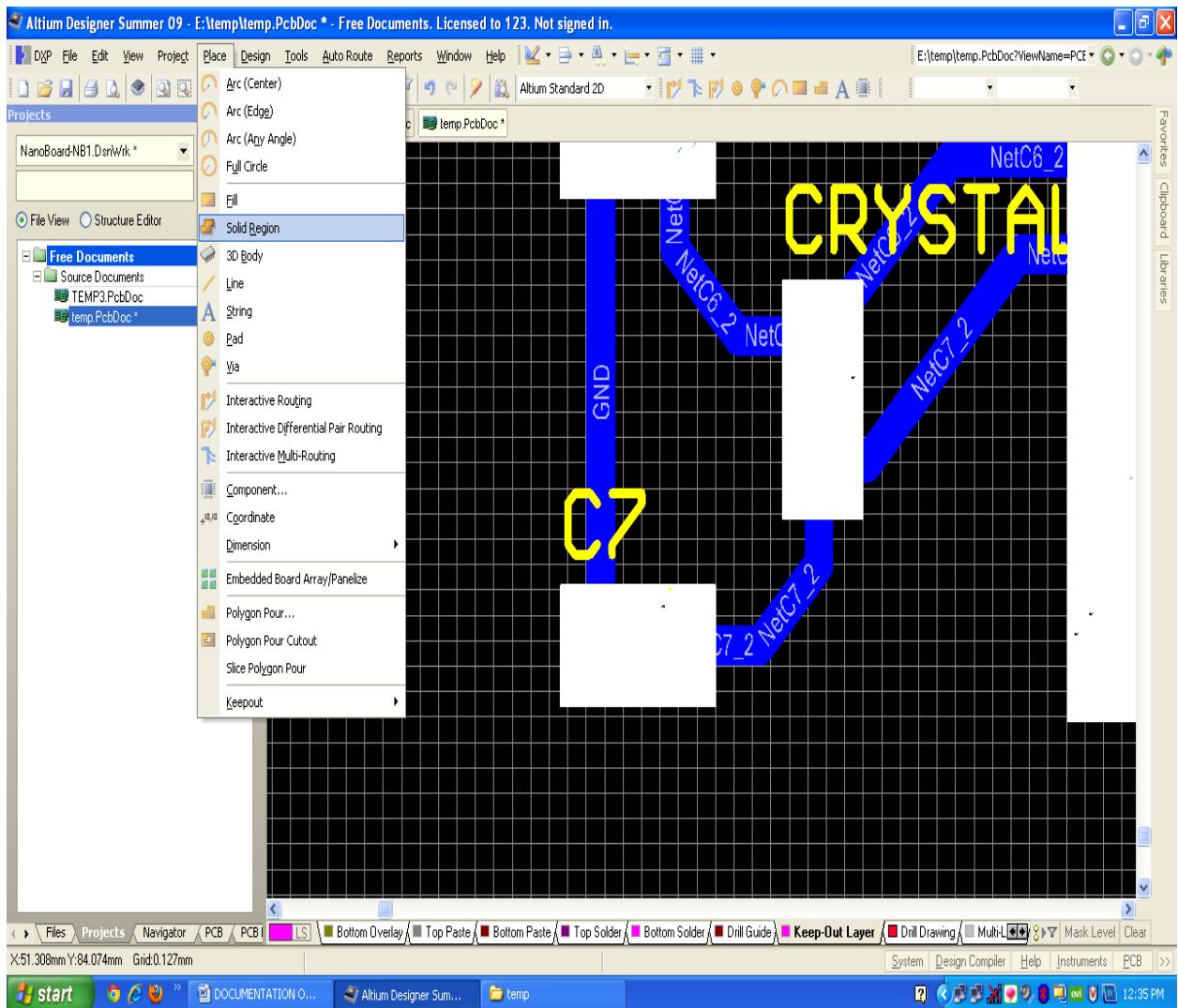
- NOW MAKE THE ALL CONNECTION AS SHOWN IN BELOW FIG.



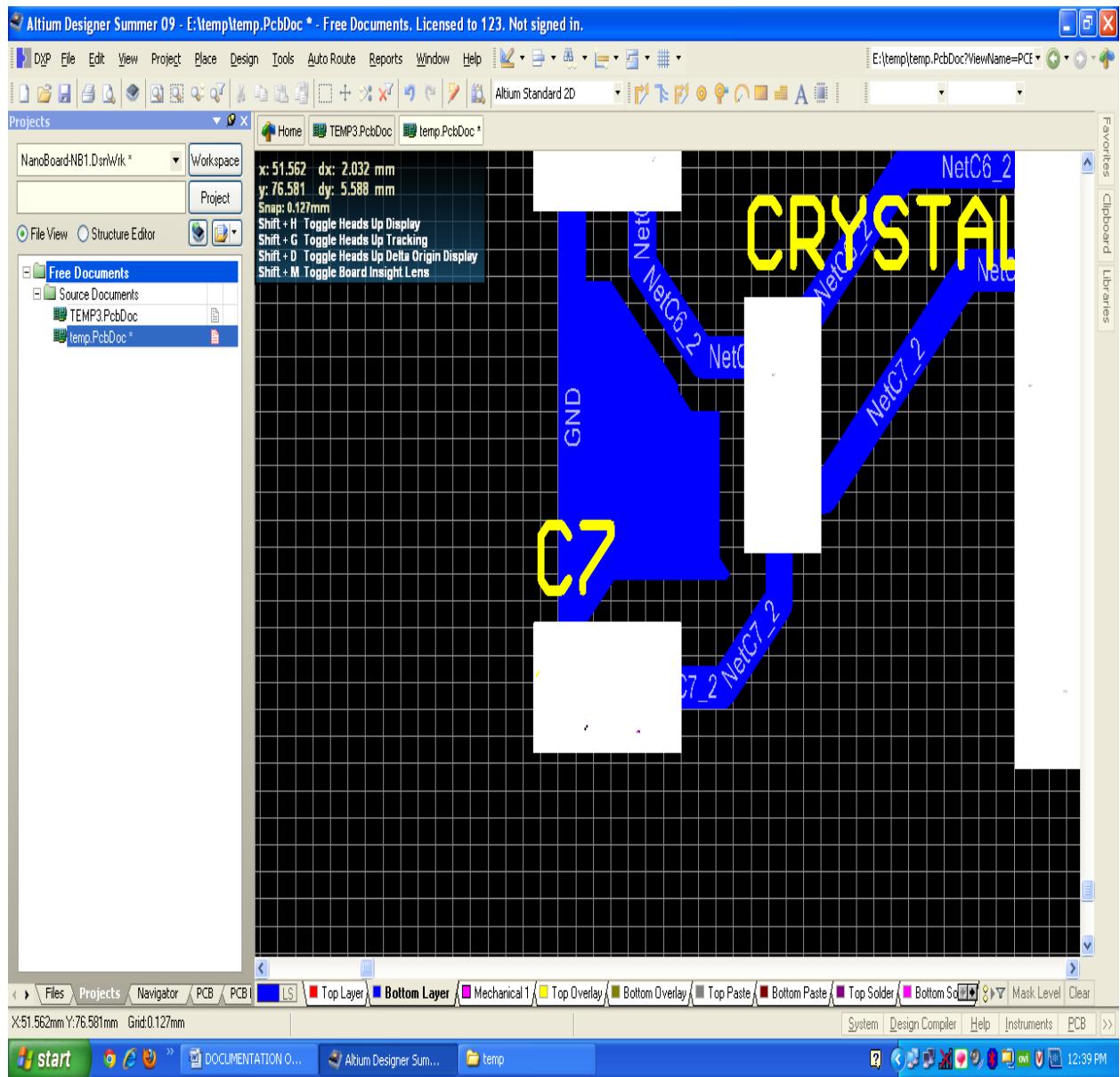
- AFTER FINISHING ALL THE CONNECTION PLACE THE KEEP OUT LAYER PINK COLOURED LINE.
- FIRST SELECT **KEEP OUT LAYER** FROM THE BOTTOM MENUBAR THEN SELECT THE PLACE AND THEN SELECT THE LINE AS SHOWN IN FIG.



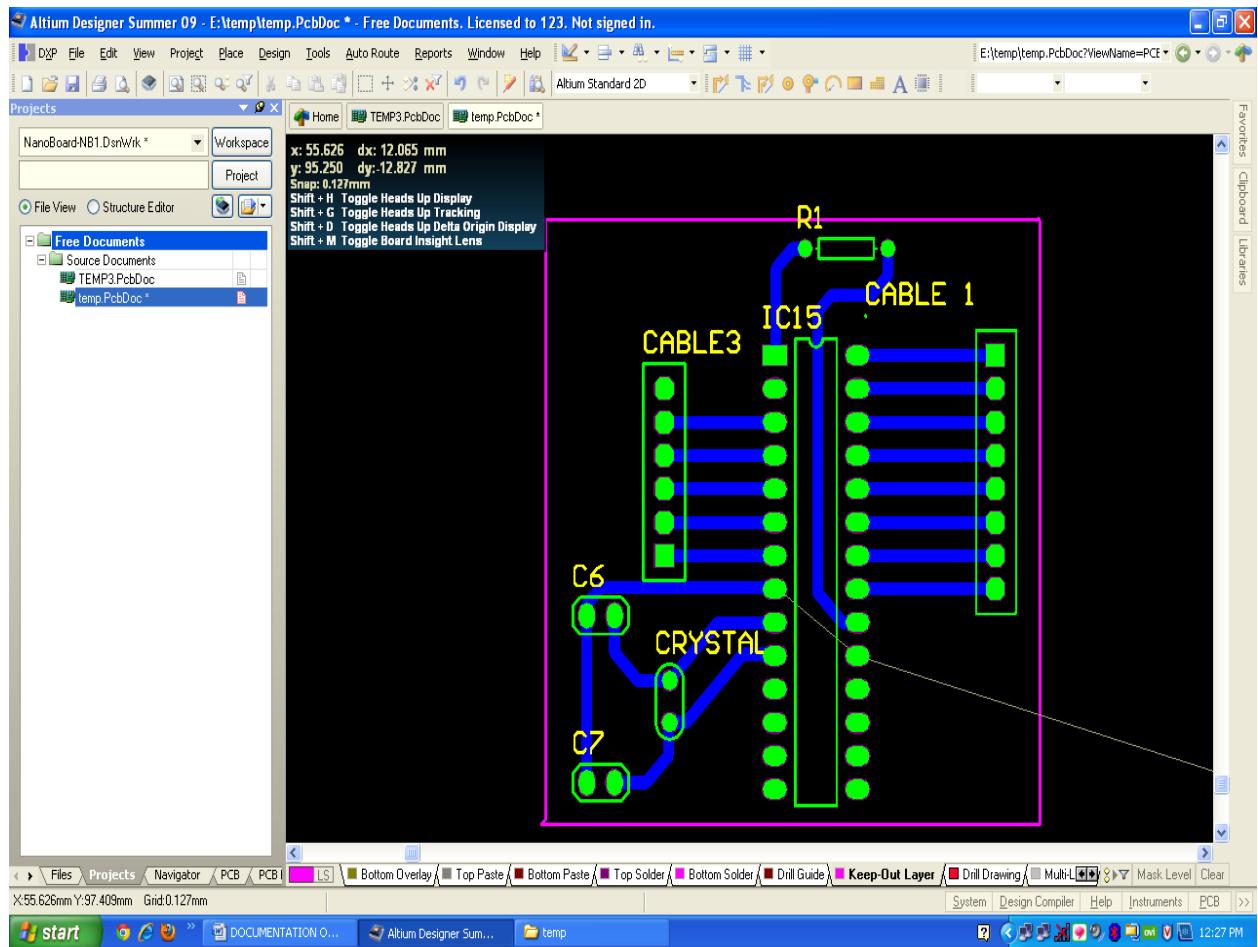
- NOW FILL THE GROUND PORTION WITH PLACE SOLID REGION.SHOWN IN FIG.



- FILL THE LIKE WISE PART IN BOTTOM LAYER AS SHOWN IN FIG.

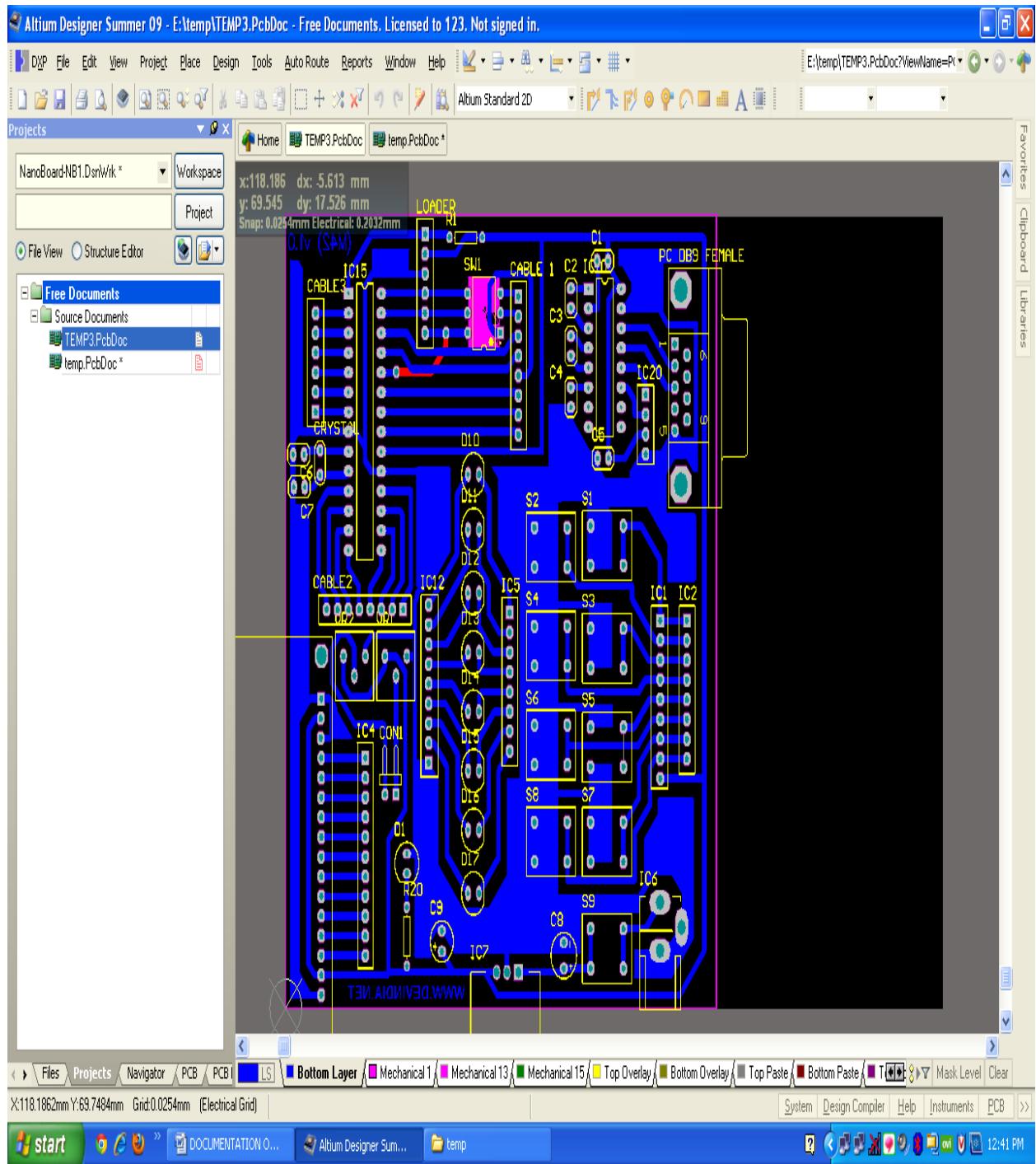


- NOW PUT THE KEEP OUT LAYER LINE LIKE THIS.



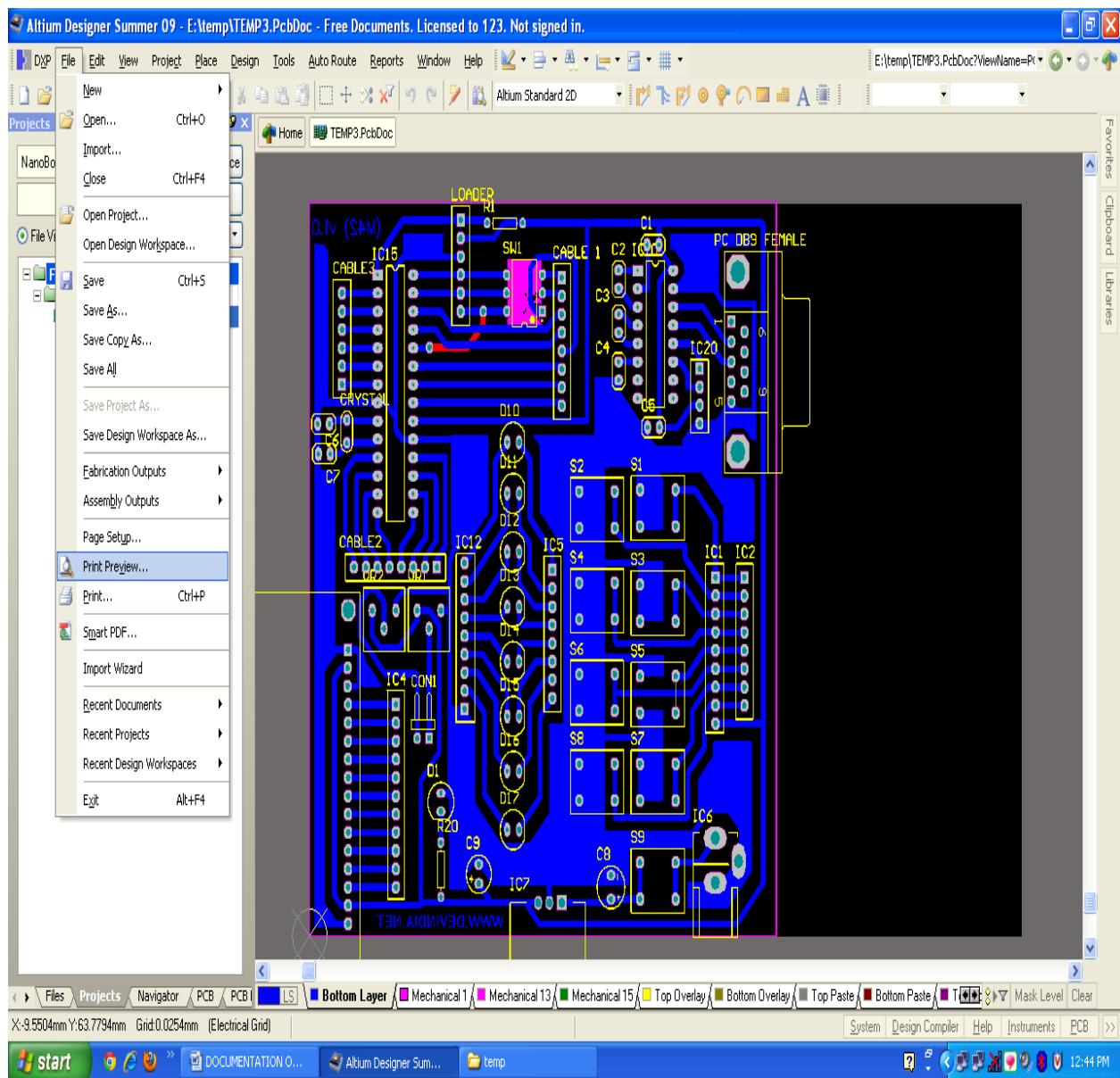
- YOU MUST CHECK THAT ALL THE NODE'S SHAPE MUST NOT BE ROUND.
- IT MUST BE RACTANGLE OR OCTAGONAL.
- AND ALSO SAVE ALL THE THINGS PCB, SCHEMATIC AND PROJECT.

- FINAL YOU GOT THE PCB LIKE THIS.

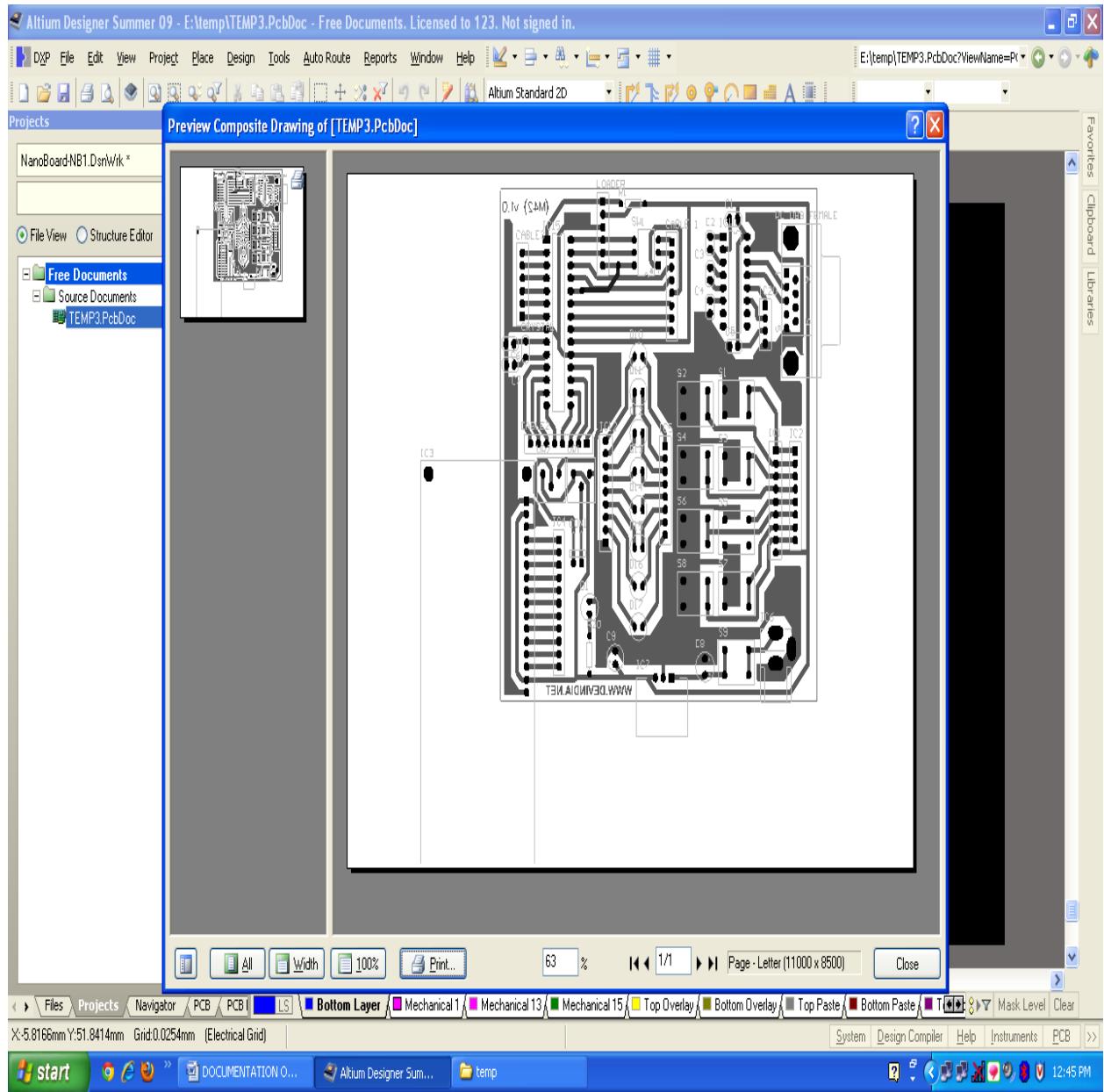


- THIS IS THE FINAL COMPLETION.
- ALSO PUT OUR COMPANY'S WEBSITE ADDRESS BY PLACE MENU AND GO TO STRING.

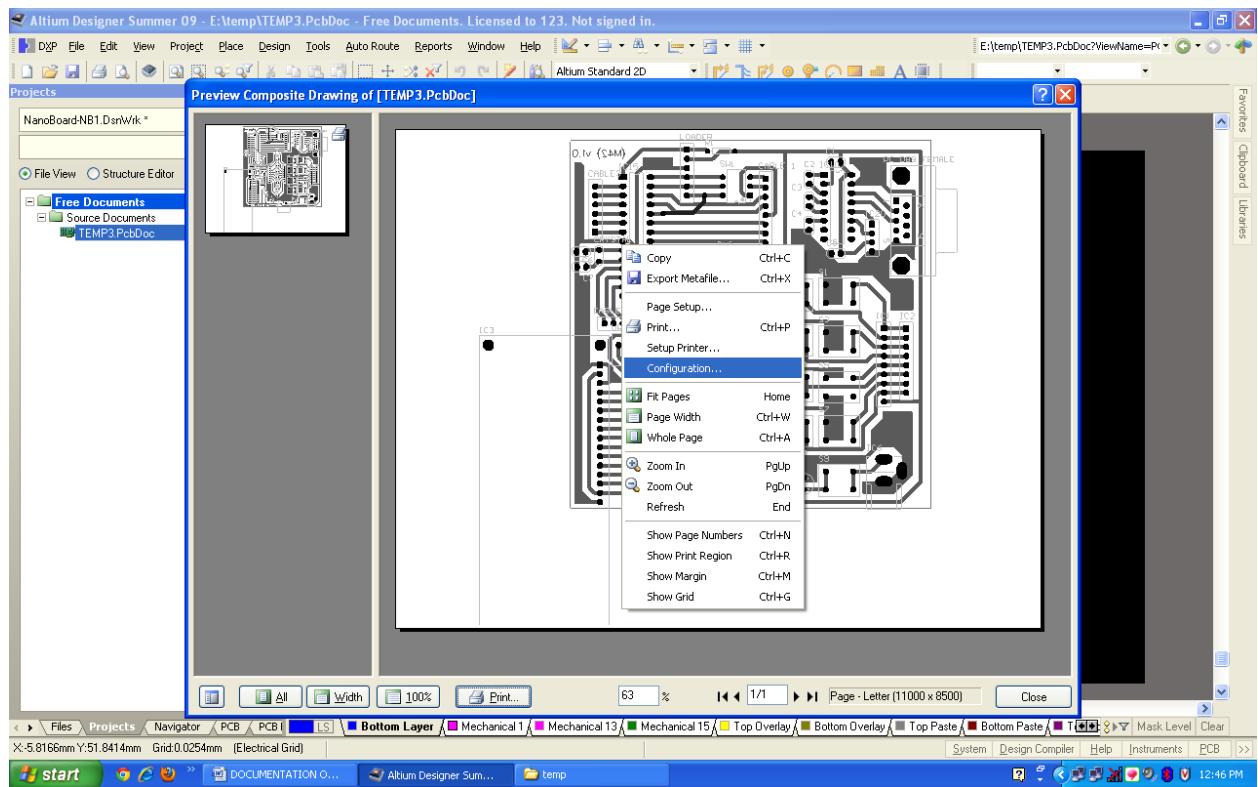
- NOW GO TO FILE AND PRINT PREVIEW.



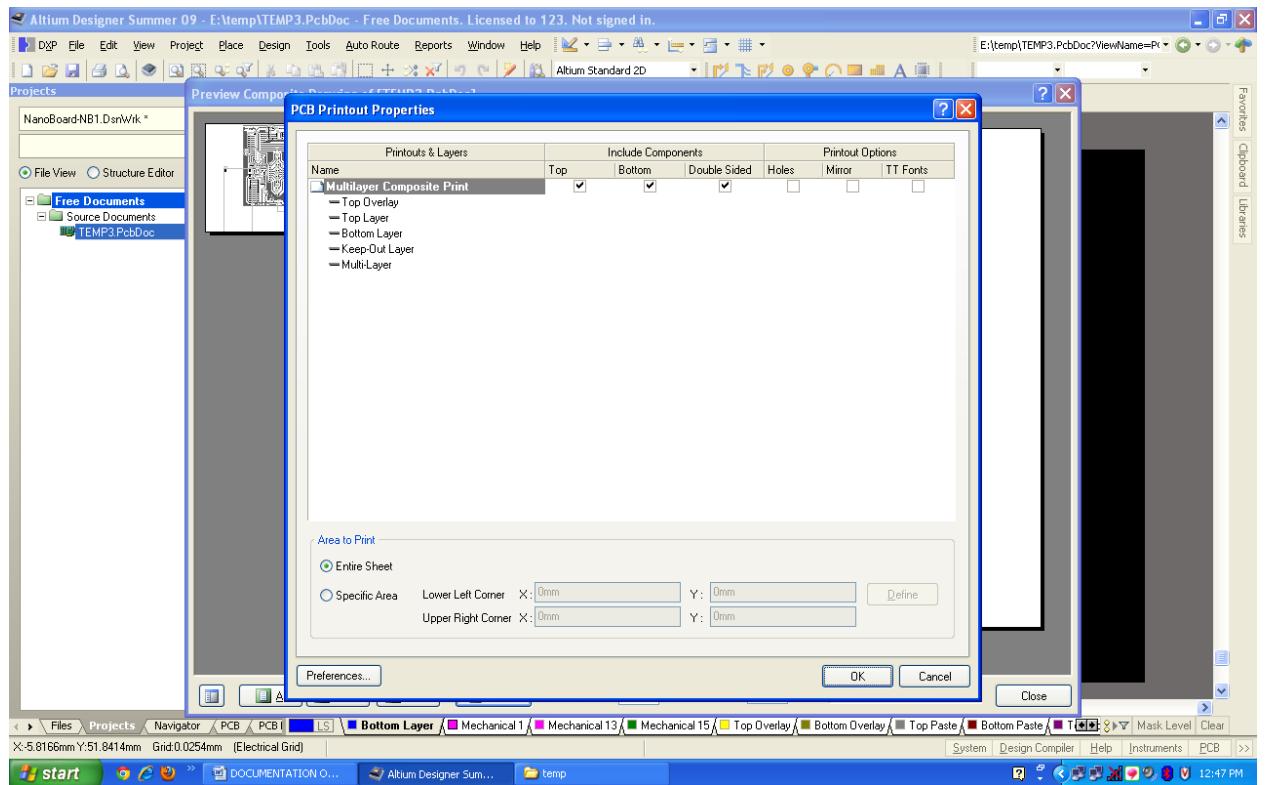
- THEN YOU CAN SEE THE BELOW WINDOW.



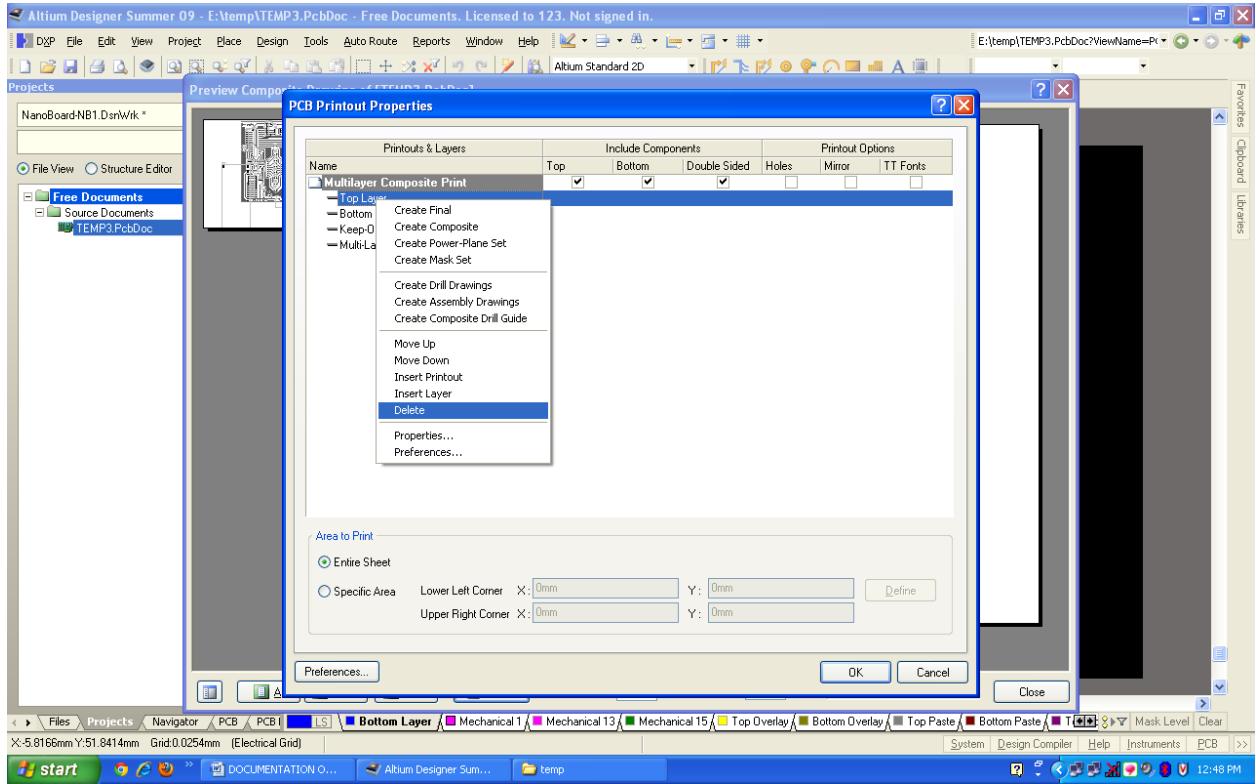
- NOW RIGHT CLICK ON IT AND GO TO CONFIGURATION.



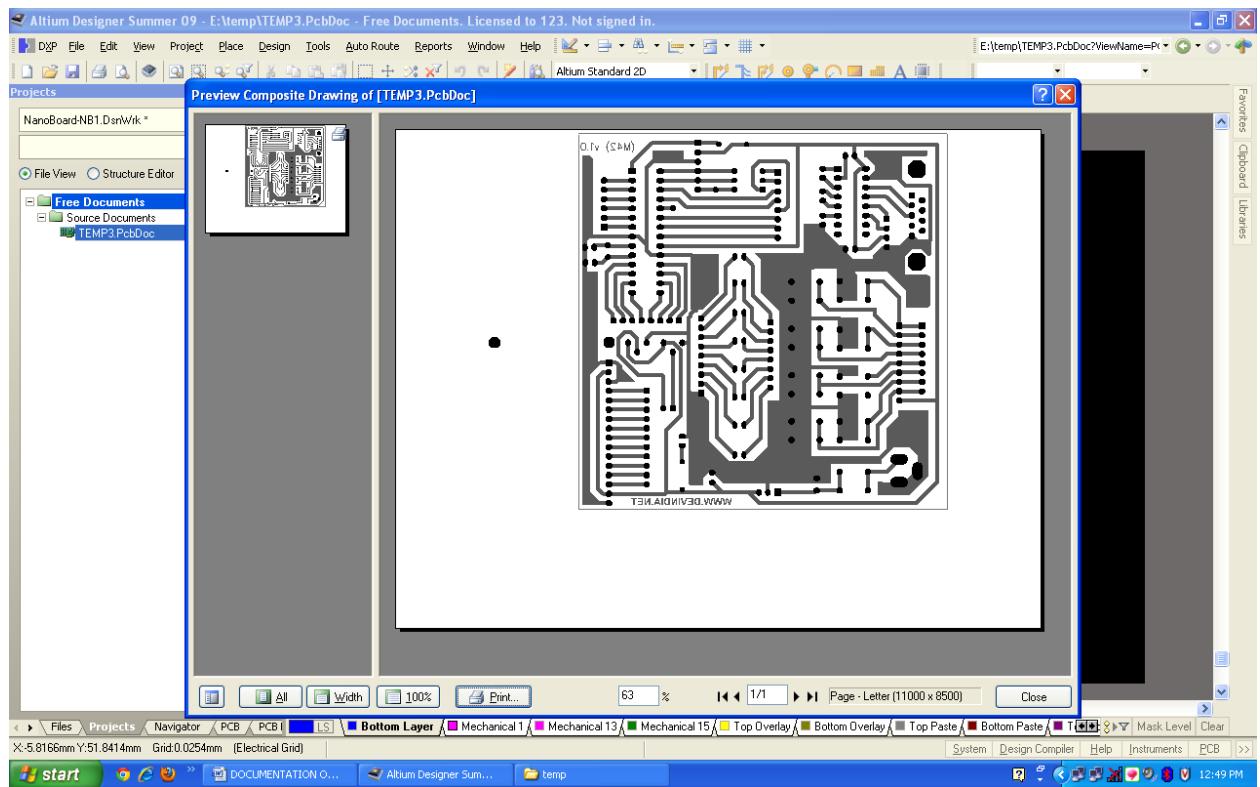
- THEN YOU CAN SEE THE BELOW FIG.
- FROM THIS REMOVE TOP LAYER AND TOP OVERLAY LAYER WE CANT REQUIRED IT IN OUR PCB DESIGN.



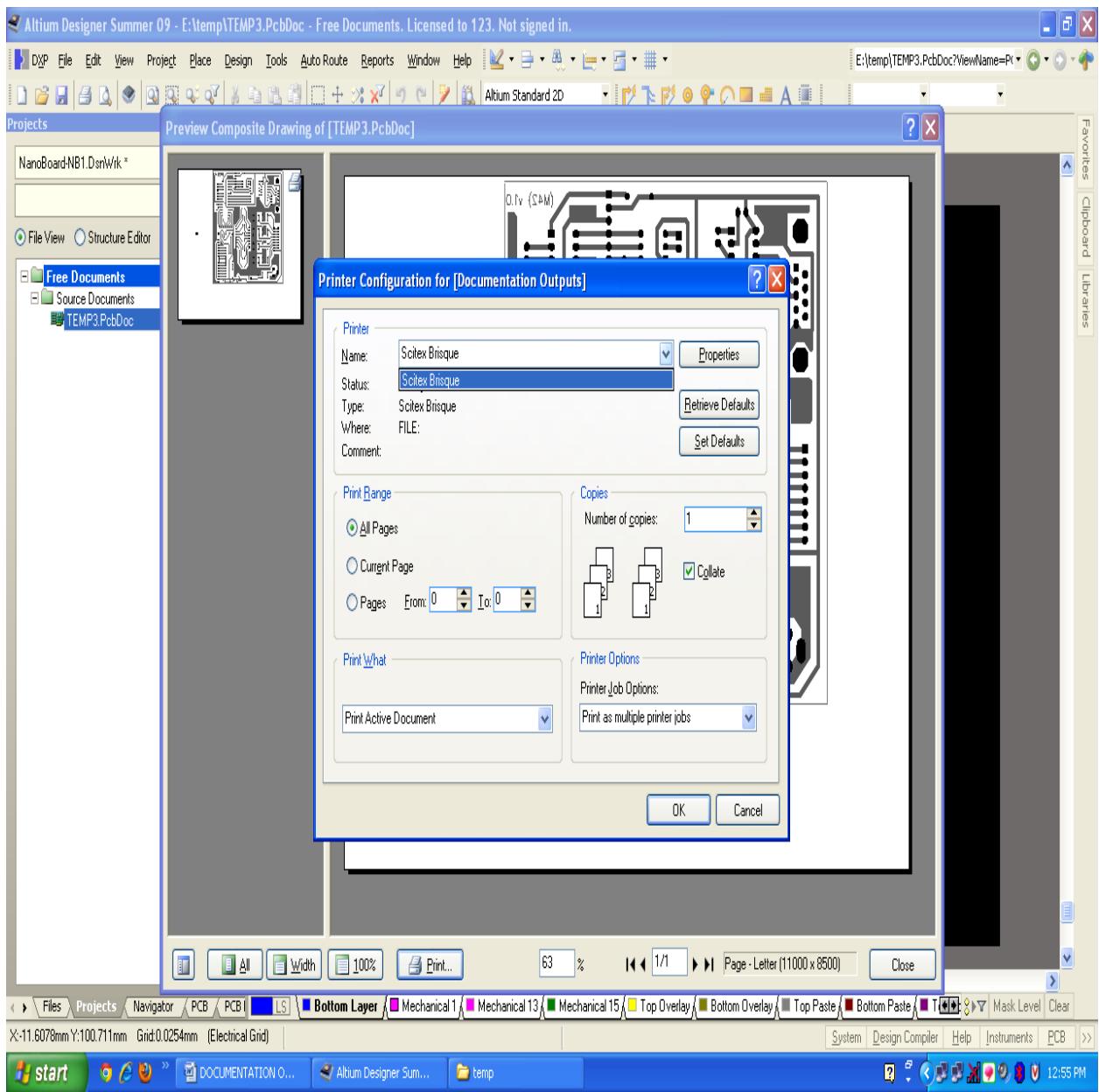
- YOU CAN REMOVE BY THIS PROCESS.



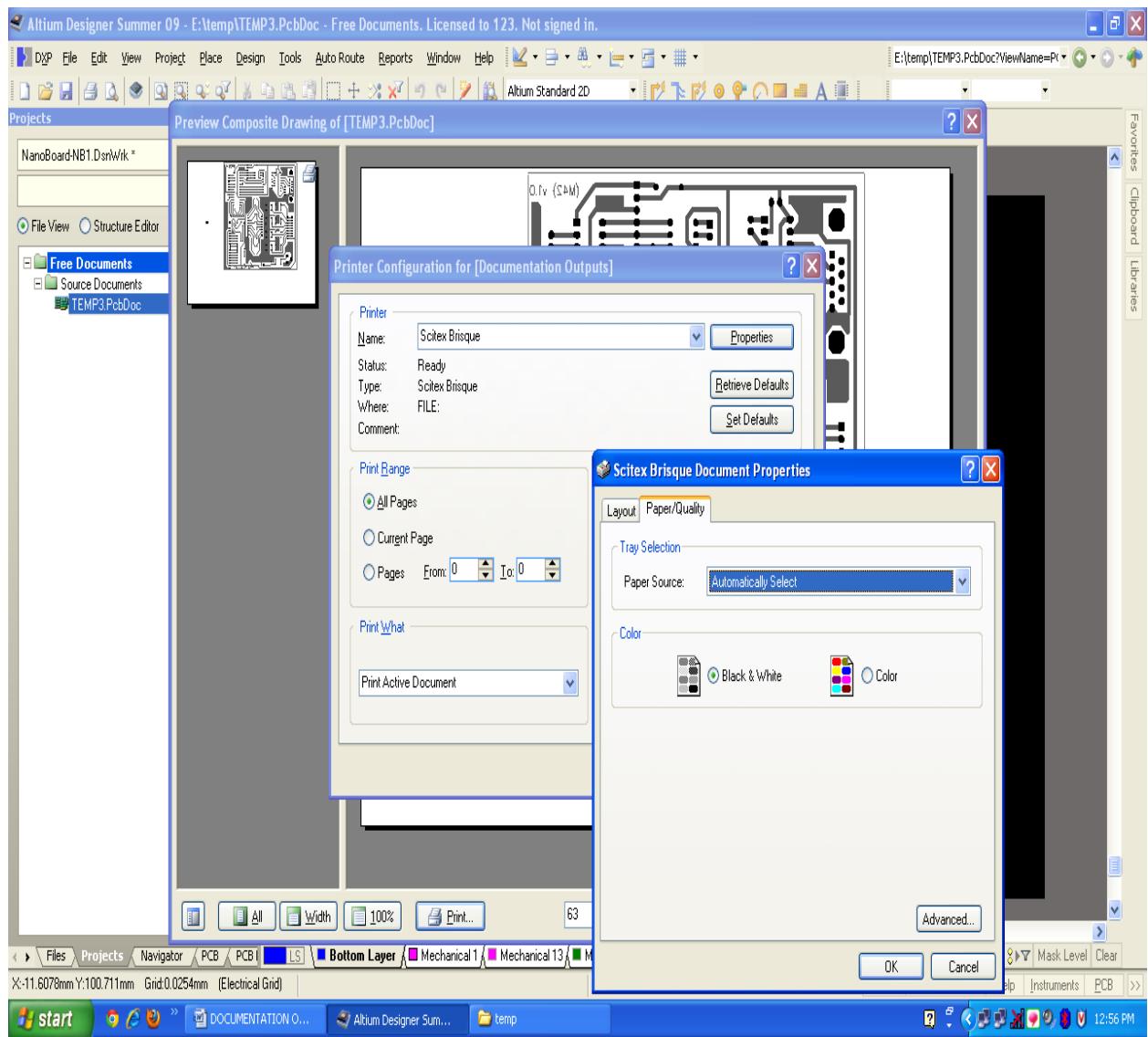
- THEN PRESS OK AND YOU CAN SEE THE FIG LIKE THIS.



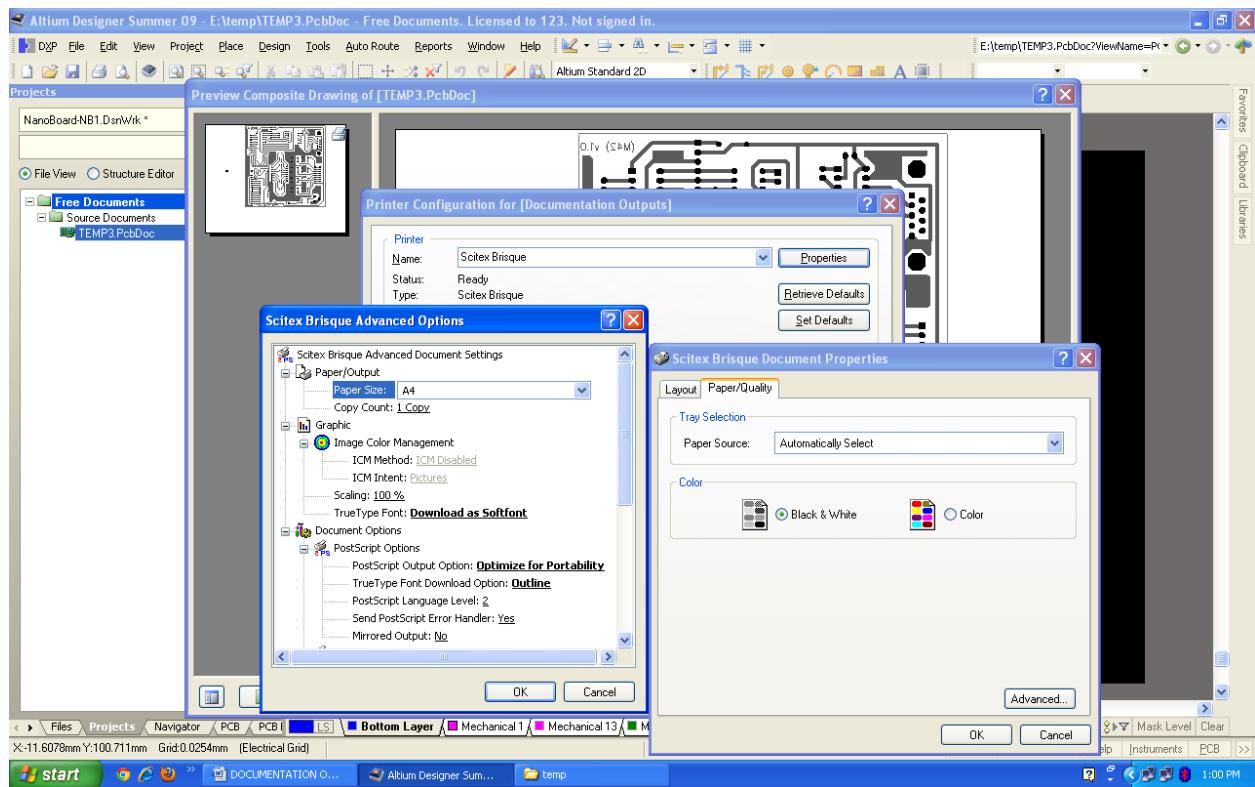
- NOW CLICK ON THE PRINT THEN SELECT SCITEX BRISQUE AS SHOWN IN BELOW FIG.



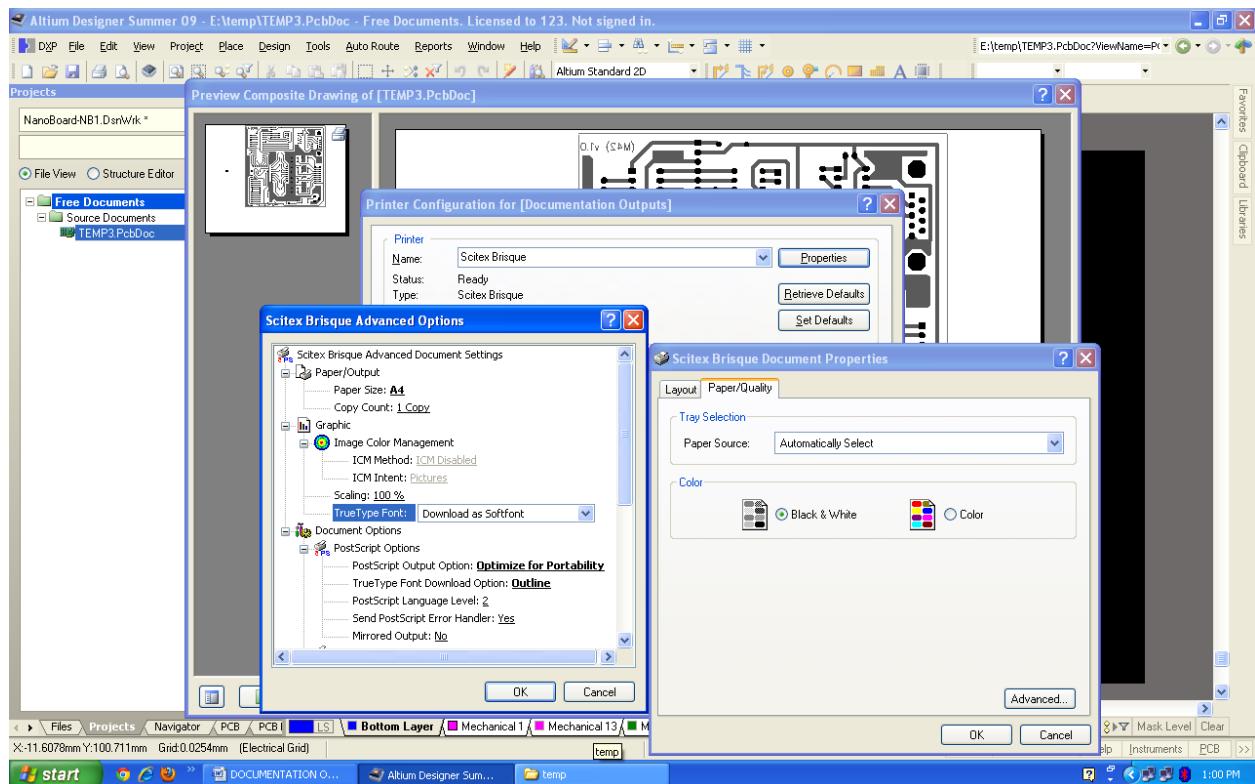
- NOW GO TO PROPERTIES AND SELECT BLACK AND WHITE ONLY.



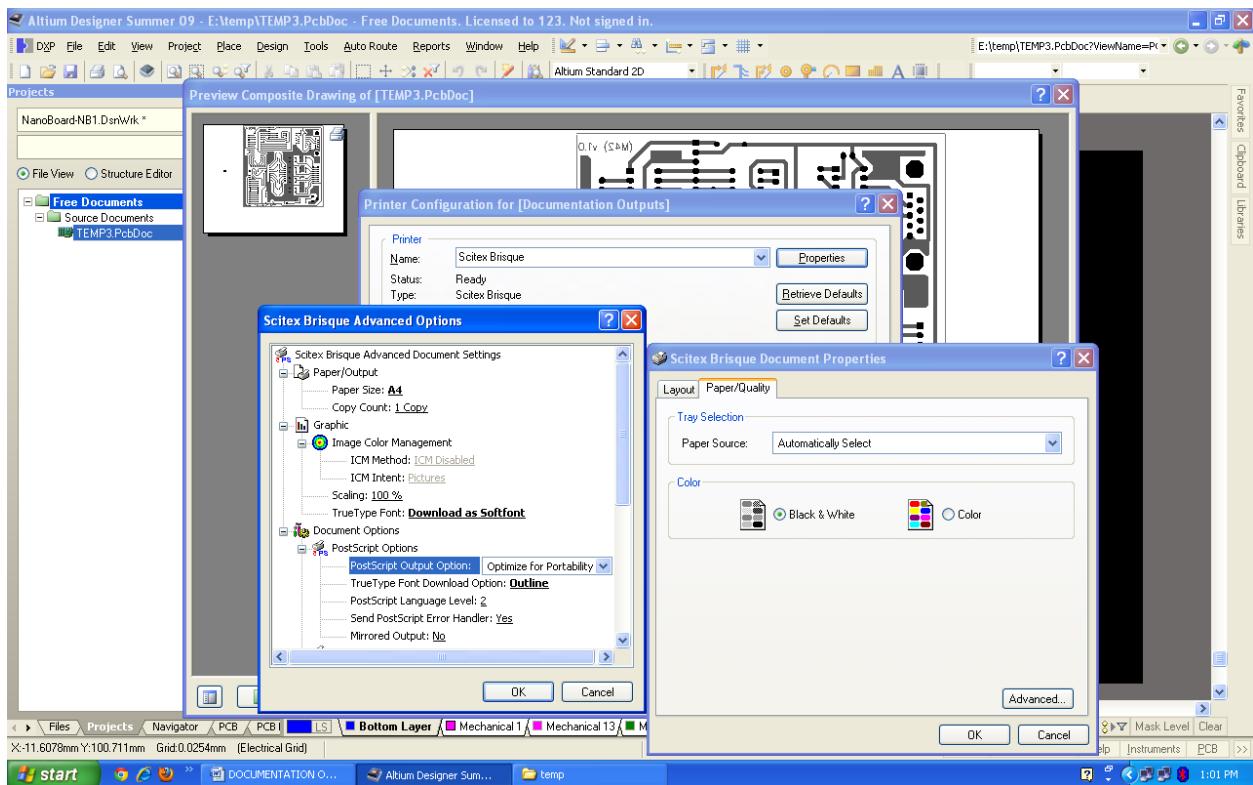
- NOW CLICK ON THE ADVANCE AND MAKE PAPER SIZE IN A4.



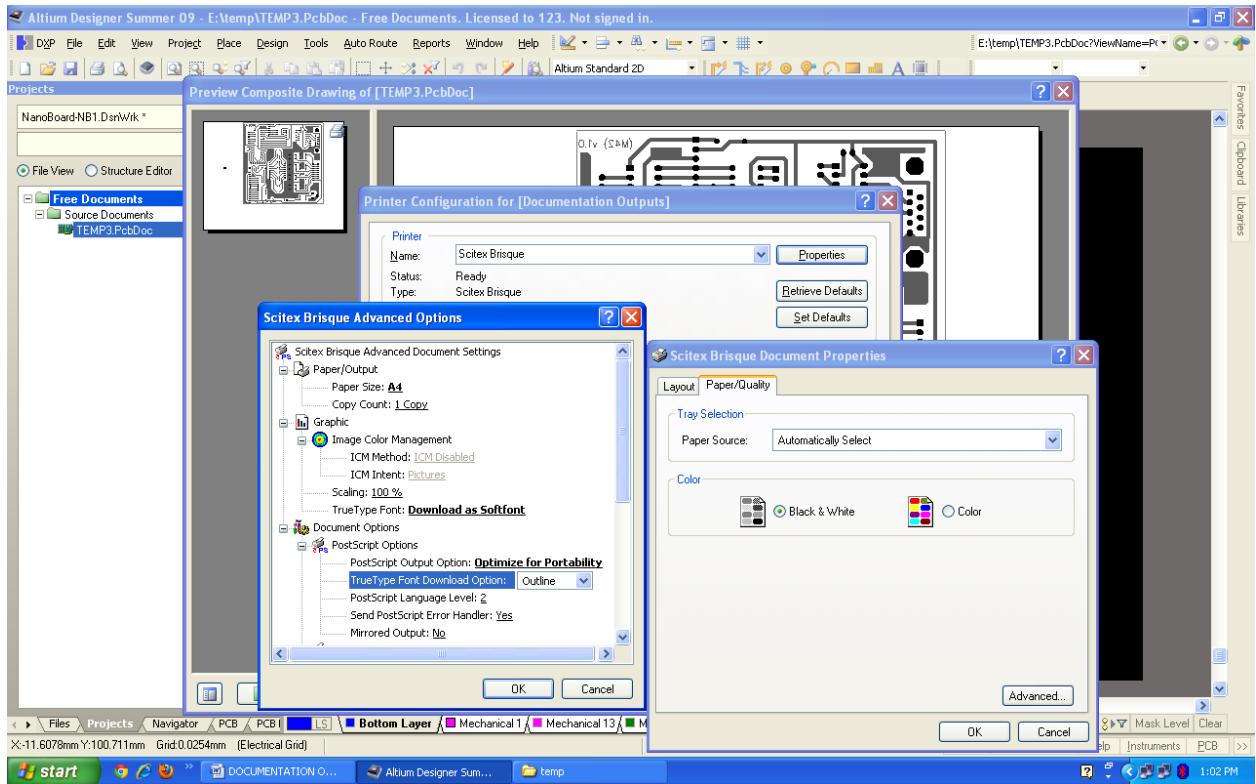
- THEN MAKE TRUE FONT TYPE TO DOWN LOAD AS SOFTFONT.



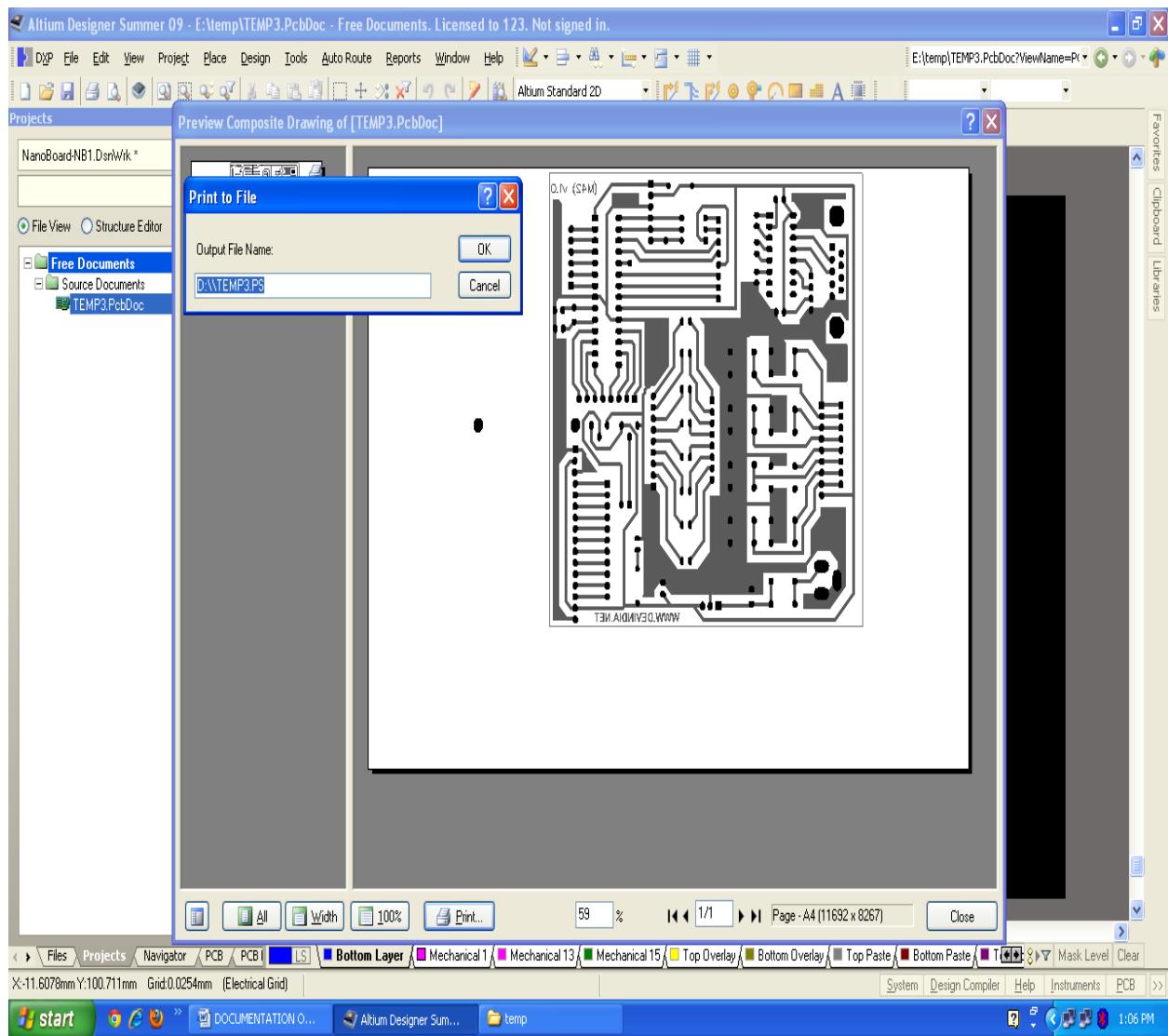
- THEN MAKE POSTSCRIPT OUTPUT OPTION : TO OPTIMIZE FOR PORTABILITY.



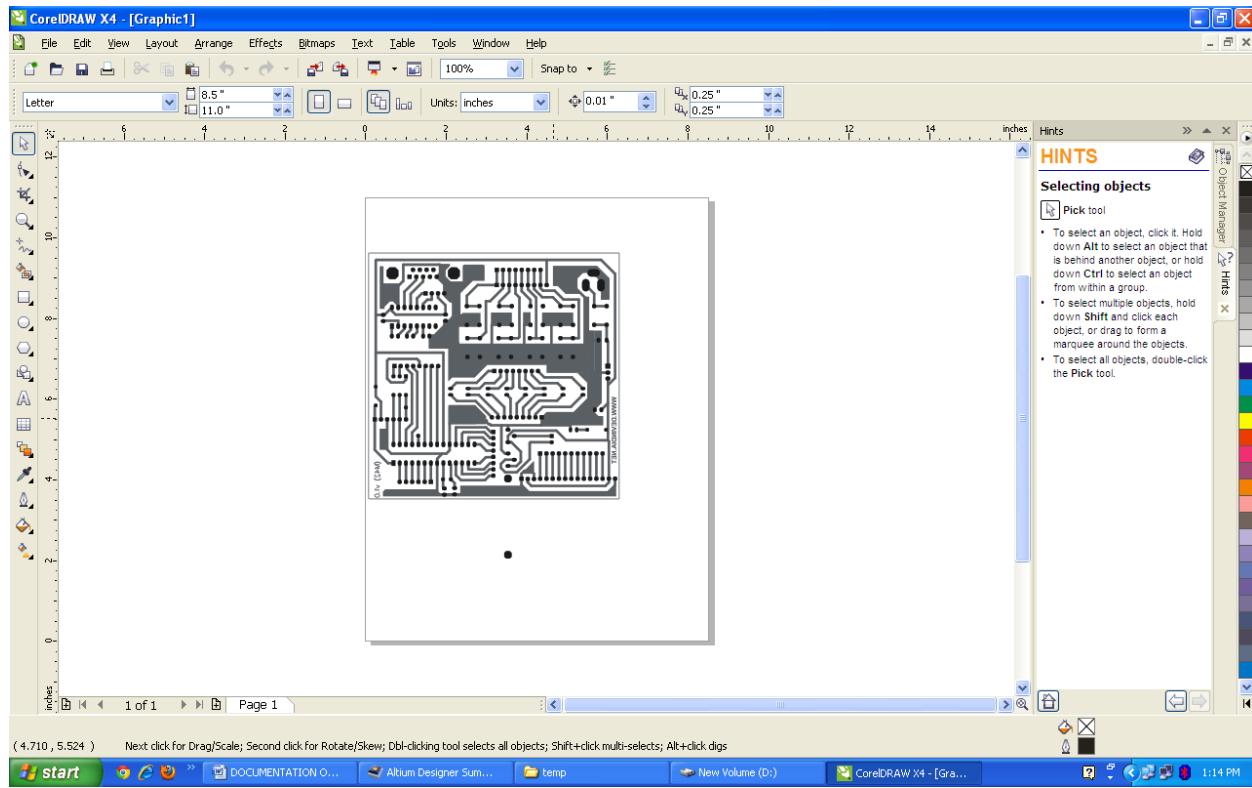
- THEN MAKE TRUE TYPE FONT DOWNLOAD OPTION TO OUTLINE.



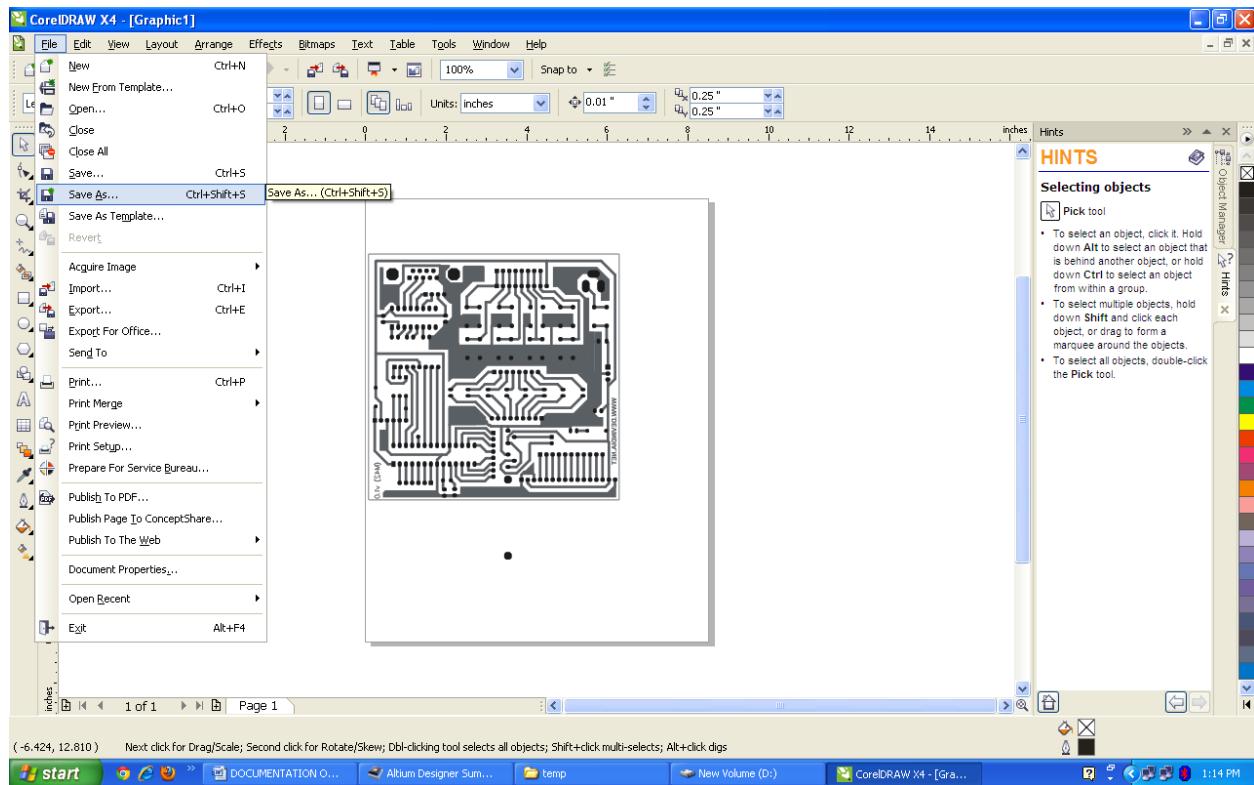
- THEN PRESS OK AND IN LAST IT WILL ASK FOR OUTPUT FILE NAME.
- GIVE THE APROPRIATE PATH AND THEN GIVE NAME TO FILE WITH EXTENTION .PS AS SHOWN IN BELOW FIG.



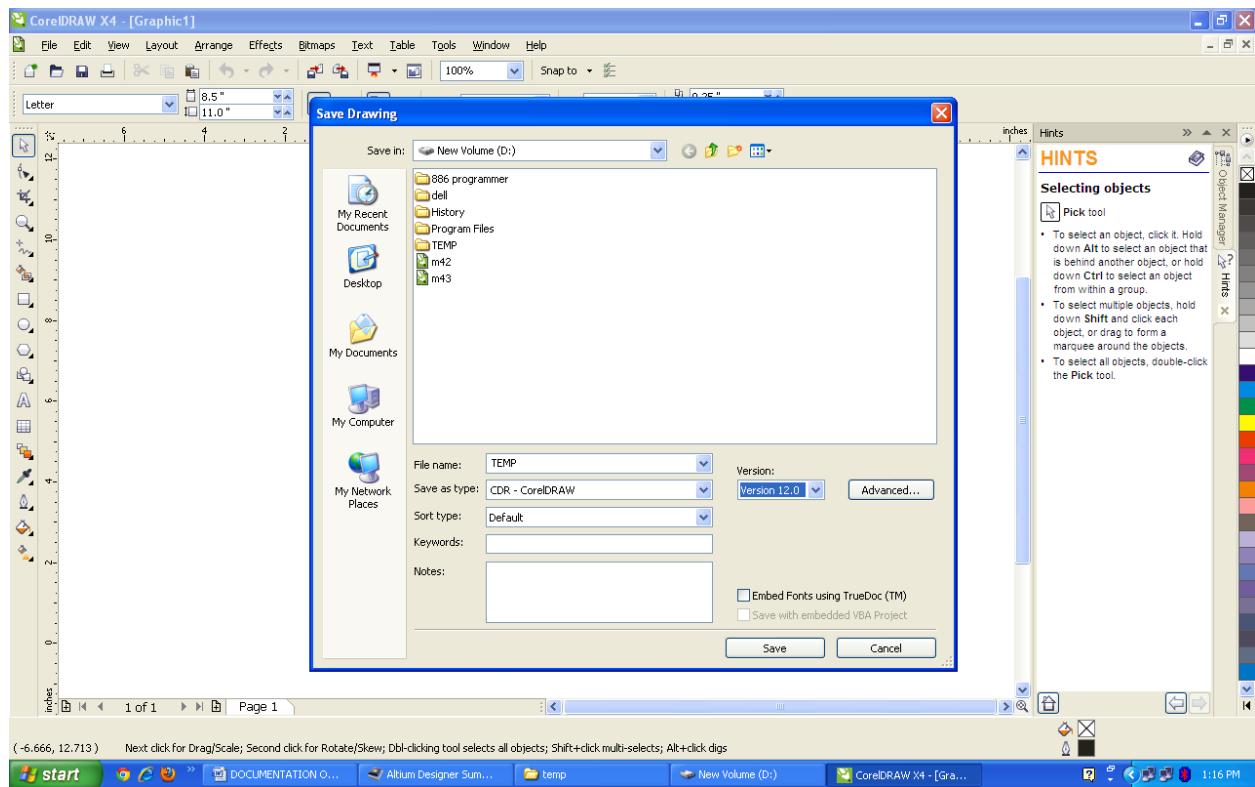
- THEN THE PS FILE WILL BE STORED AT YOUR GIVEN PATH.
- AND YOU CAN ALSO OPEN IT IN CORELDRAW FILE.
- THIS IS THE FINAL PROCESS FOR THE PCB DESIGNING.
- NOW OPEN THE PS FILE WITH IN CORELDRAW.



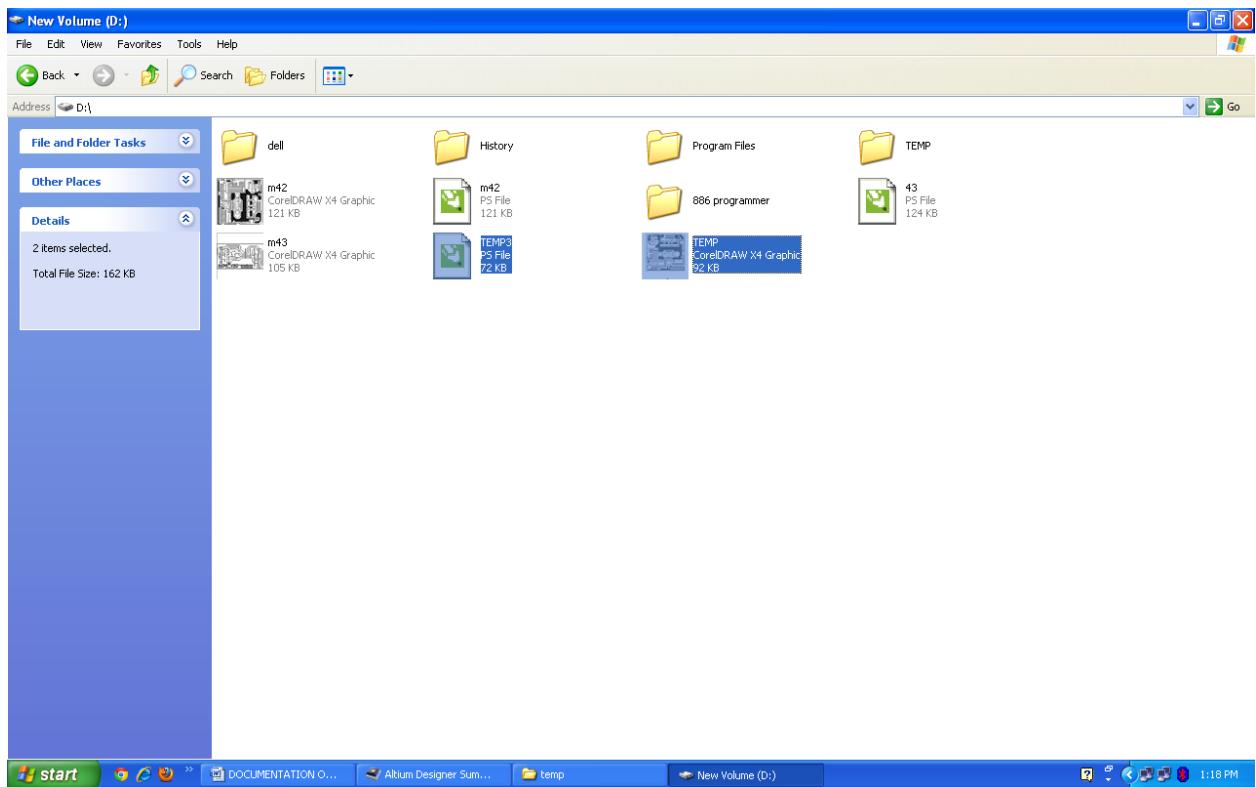
- NOW GO TO FILE AND GO TO SAVE AS AND SAVE THE FILE WITH NAME GIVEN BEFORE AND SAME PLACE WHERE THE PS FILE IS STORED.



- ALSO SAVE THE FILE IN VERSION 12 AND WITH CDR EXTENTION.



- NOW YOU CAN SEE THE PS FILE AND CDR FILE IN THE D DRIVE AS SHOWN IN BELOW FIG.



- THIS IS THE END OF PCB DISIGNING.