BUNGARD

PCB Design Using CCD2 CNC Machine.

Physics Lab

School of Sciences and Engineering

Lahore University of Management Sciences

CHAPTERS

CHAPTER 1: INTRODUCTION

Bungard CCD2 is an automatic circuit designing machine. It simply prints your designed circuit on a board by:

1. Routing its path defined by you
2. Drilling the given points spotted in the circuit

To start a PCB design using CCD2 machine you first need to have **Gerber/CadCam/Excellon** files of your PCB design. Almost all PCB designing softwares are able to generate Gerber output. These files are the input files for this machine.

(Note: To get introduced to the idea of how isolation milling/engraving method is used to design PCB circuit please watch the Youtube video http://www.youtube.com/watch?v=TYAfNTvgTak)

Important Note:

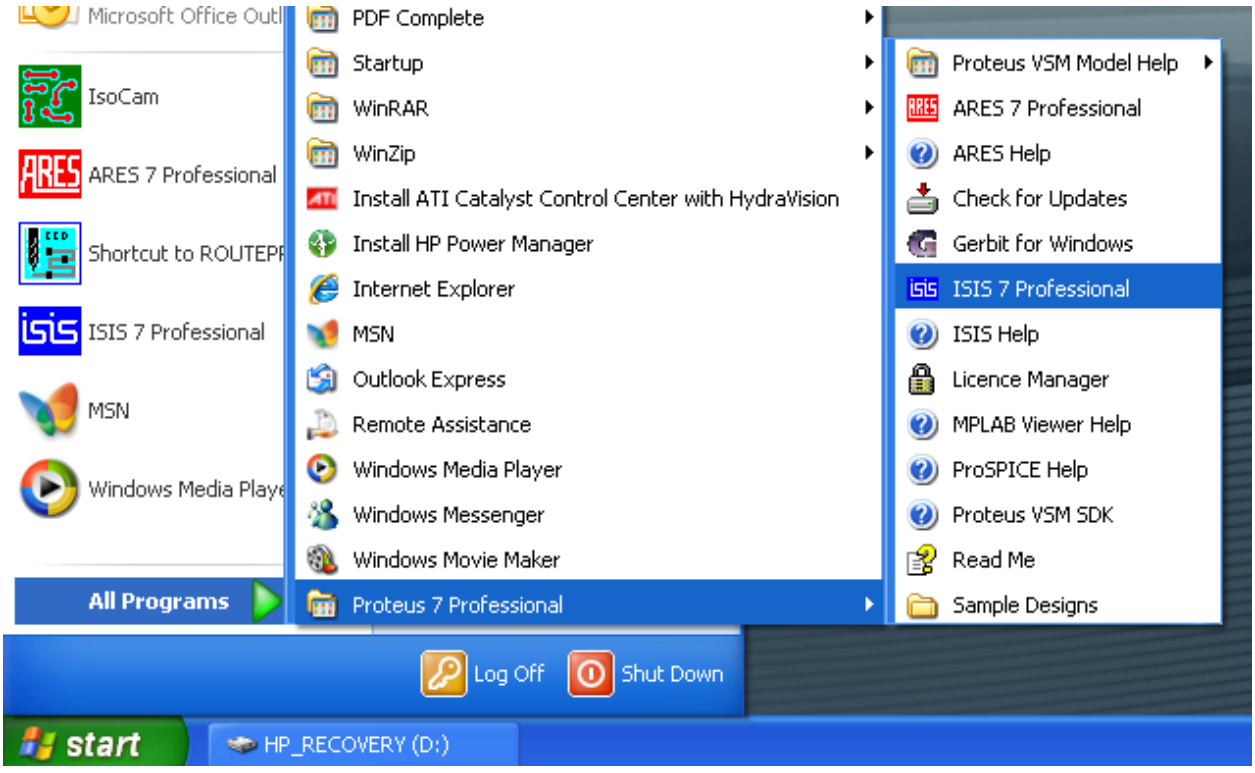
1. Please note that only Single Layer PCB designs are possible on CCD2. It can work on one layer at a time. Dual layer design is also possible but requires extra consideration and will also be complicated to handle.
2. Also make sure that in your circuit the minimum track clearance is no less than 0.6 mm , i.e., the distance between any two adjacent tracks on design is not less than 0.6 mm. This limit is imposed because minimum of 0.6 mm routing bit is available in physics lab.

CHAPTER 2: PCB Designing Using PROTEUS and ARES

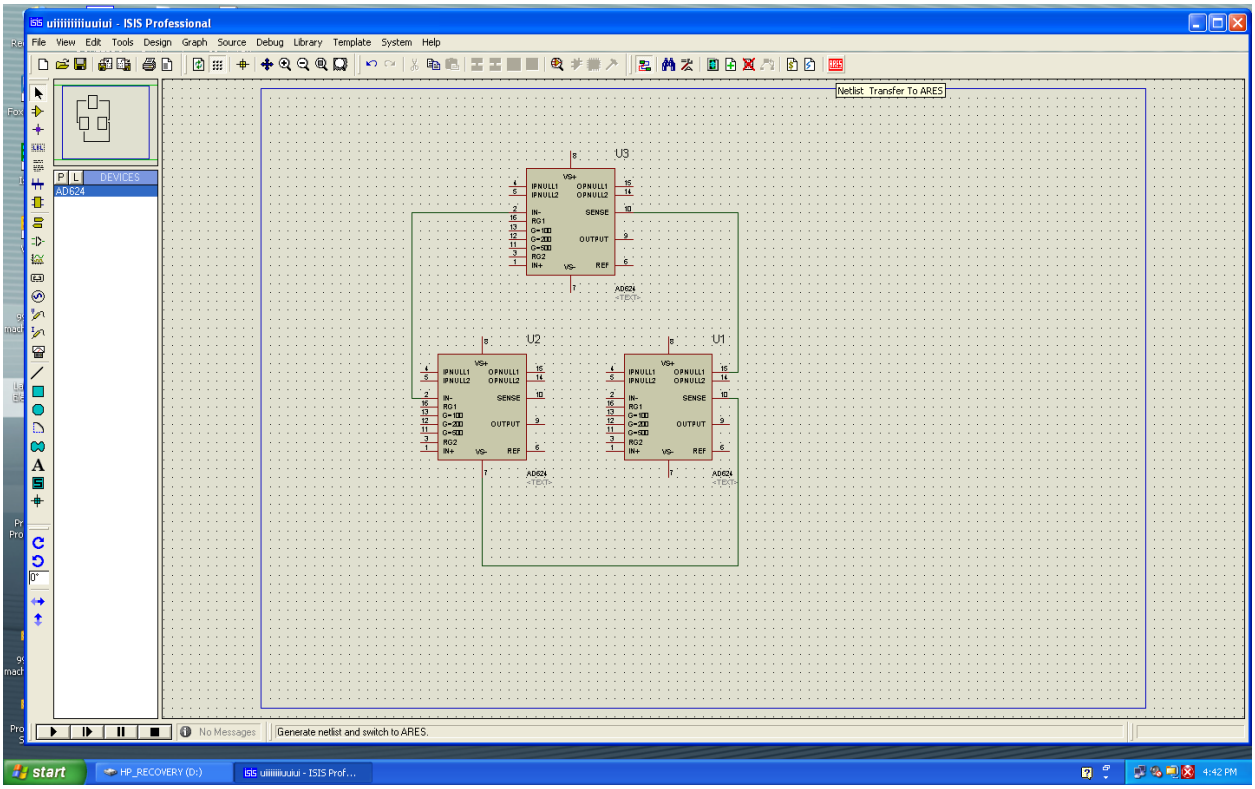
Proteus ISIS is a circuit designing and simulation software. ARES is used for designing PCB Layouts. First you make your required circuit on Proteus, test it, and then make the desired PCB layout in ARES. Here is small demo of how to make PCB using Proteus ISIS.

PCB design using Proteus ISIS: A basic overview

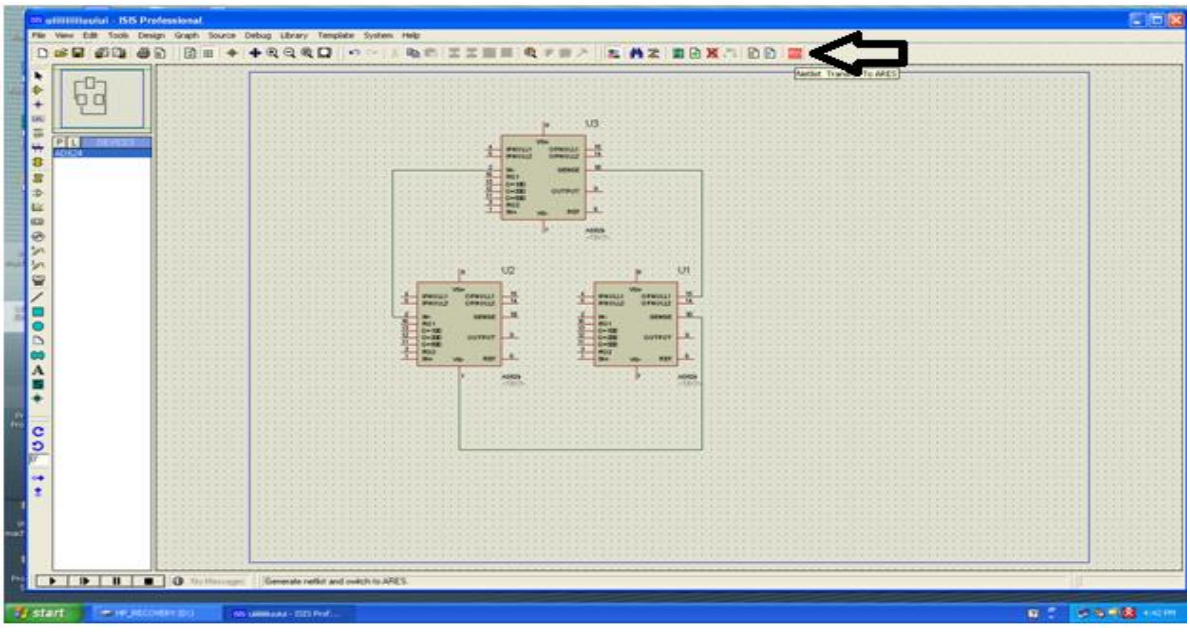
1. In physics lab computer located with the PCB machine, you can open Proteus ISIS from start menu



2. After opening Proteus you can make any circuit design using Proteus available components. Make sure that the components you use must have their PCB footprint available in the package (or in other words, they must have the component packages for ARES). For further detail use Proteus Help menu.

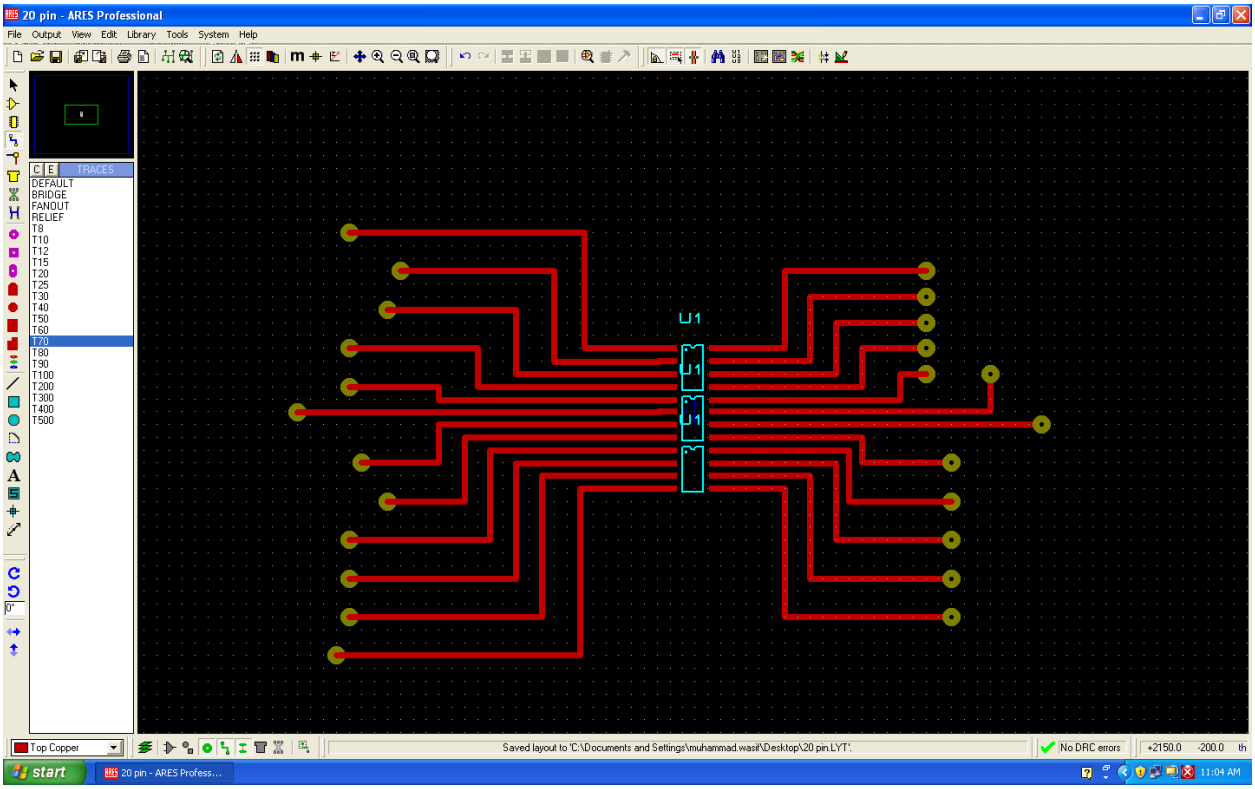


3. Once finished with the design click on the red button shown in the figure below. It will generate the Netlist of components and open the ARES interface.

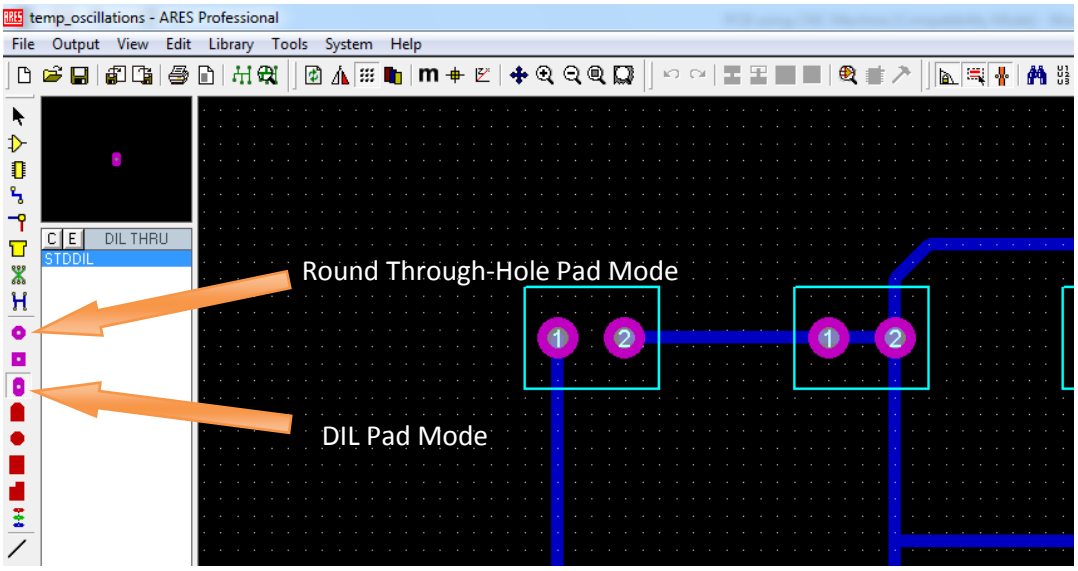


4. Once the ARES is opened as shown in the figure below, you can perform the manual or autorouting to interconnect the component and route the traces tracks. You can also make your PCB design directly in the ARES without going through the Proteus ISIS.

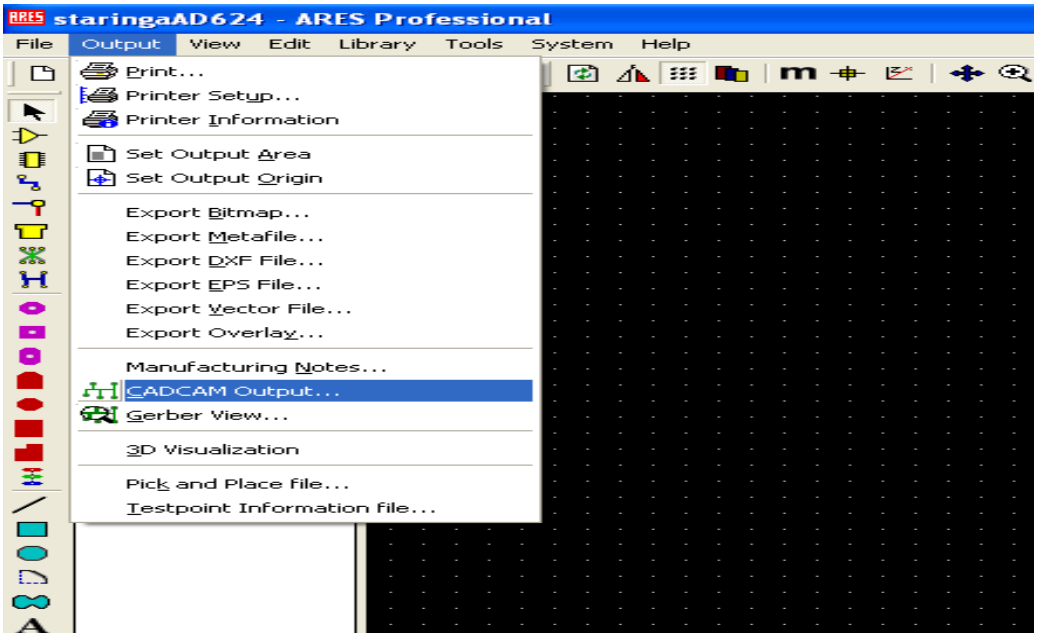
   (Note: If you don't require the simulation part of Proteus, you can directly open the ARES and design your PCB layout by selecting the desired component packages from the library).



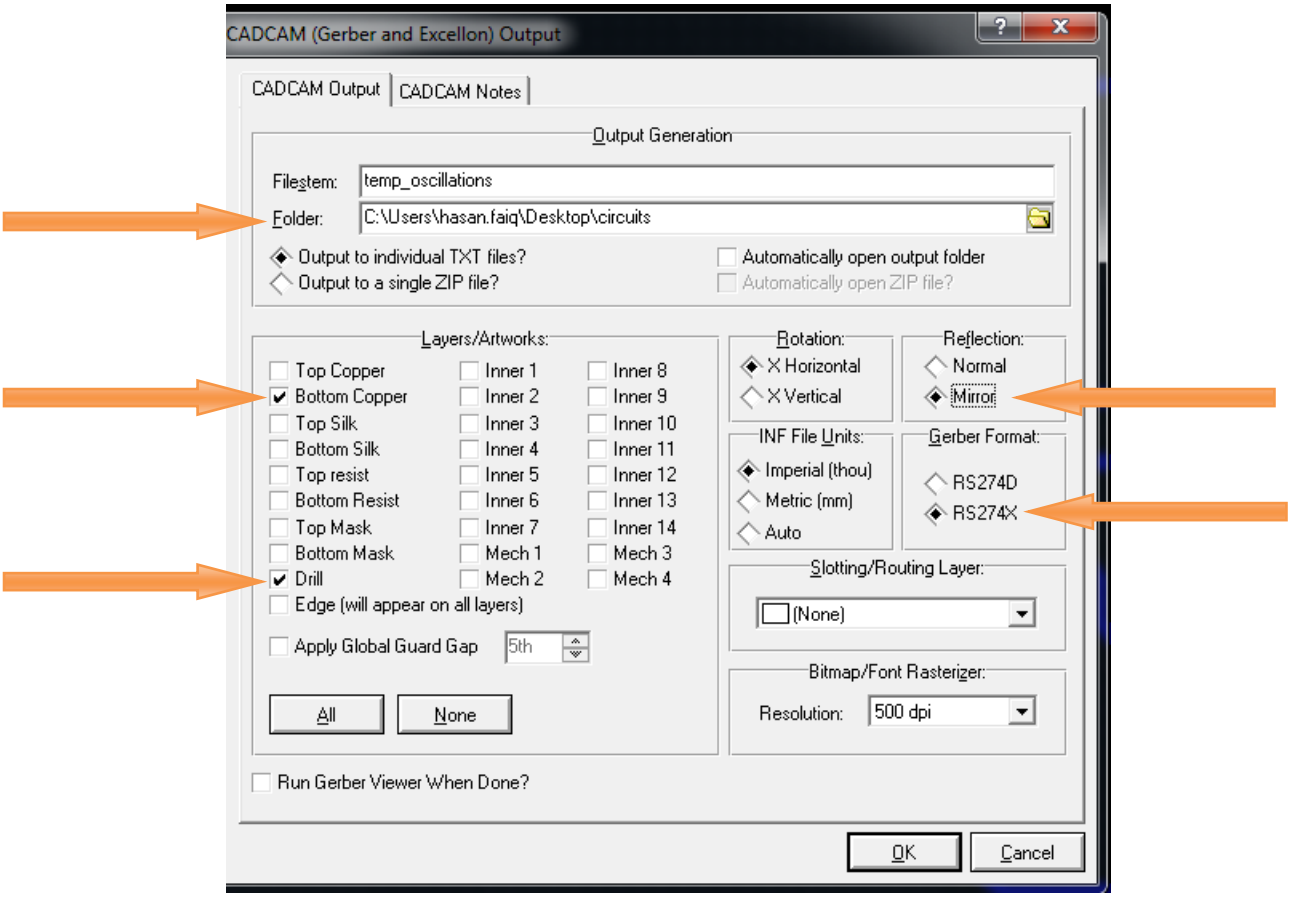
5. The "Track Mode" should be greater or equal to T40. This is because of the while the Machine is routing with 0.6mm bit, the layout get enough width for its routing paths. Also use "Round Through-Hole Pad Mode" and "DIL Pad Mode" in the circuit wherever required. This increases the copper space for a component pins and helps you get enough space while soldering.



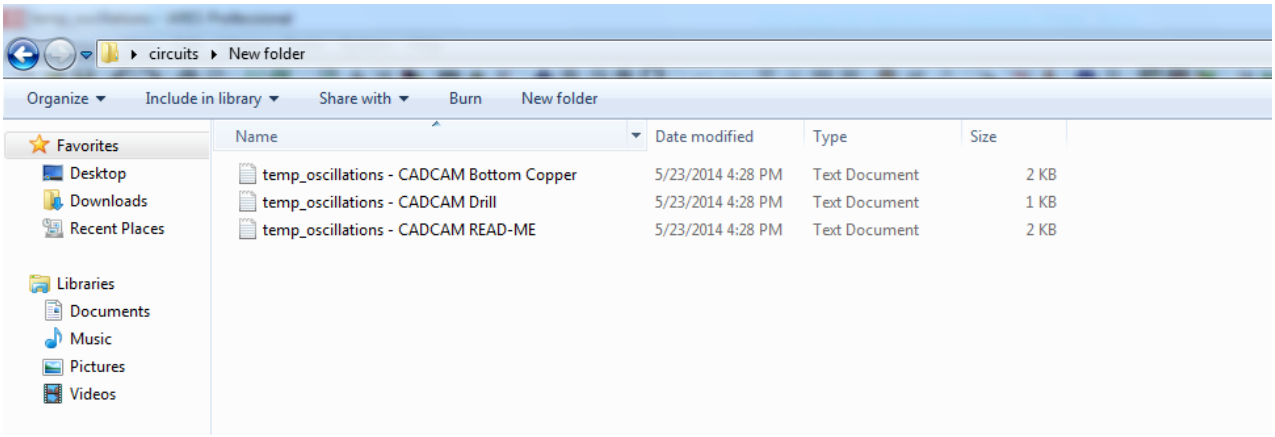
6. From ARES go to the output menu and select CADCAM output (which is the same as Gerber).



7. From CADCAM ouptput window make sure that **Mirror** and **RS274X** is selected. Now specify the output folder where you want to save the Output GERBER/CADCAM files. Please note that on the left it shows all the layer that it will export: Top Copper, Bottom Copper etc. files. You only need to check Bottom Layer and Drill boxes, rest are of no use here.



8. Now open the folder where you saved your CADCAM/GERBER/EXCELLON output files. It will contain at least three files: Bottom Layer, Drill and Read-Me files as shown in figure below. We just need Bottom Copper and Drill file.



CHAPTER 3: ISOCAM

ISOCAM is basically a PCB layout designing software just like ARES and is compatible with the Bungard CCD2 machine. It will generate the required milling (routing) and drilling files that can be fed into the software of machine to make it work. If you are familiar with this software, you can make circuit of your choice on it. But in our case, we are using ARES. So we will load the ARES CADCAM files in it and generate the requisite milling and drilling files for CCD2 machine through it.

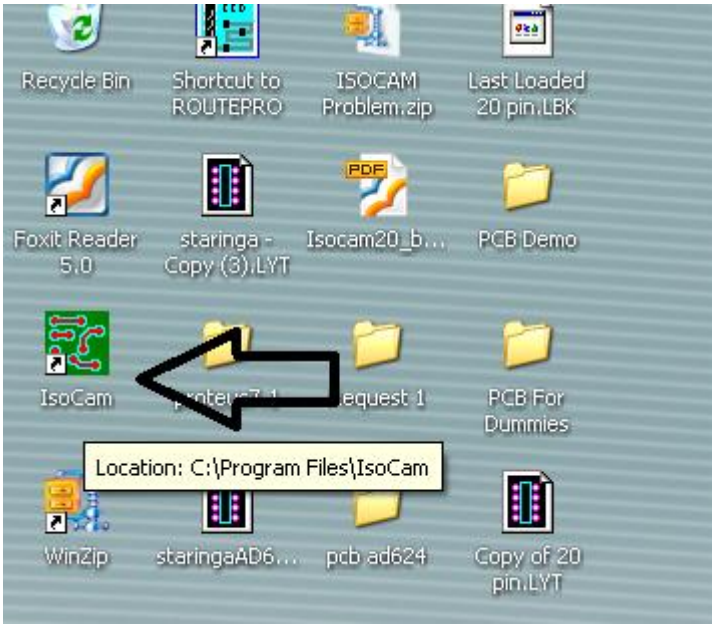
This software works with a DONGLE KEY. Dongle key is a USB that make this software run. It means that you have to plug this USB (Dongle Key) into the USB port of your computer and then run the ISOCAM software.

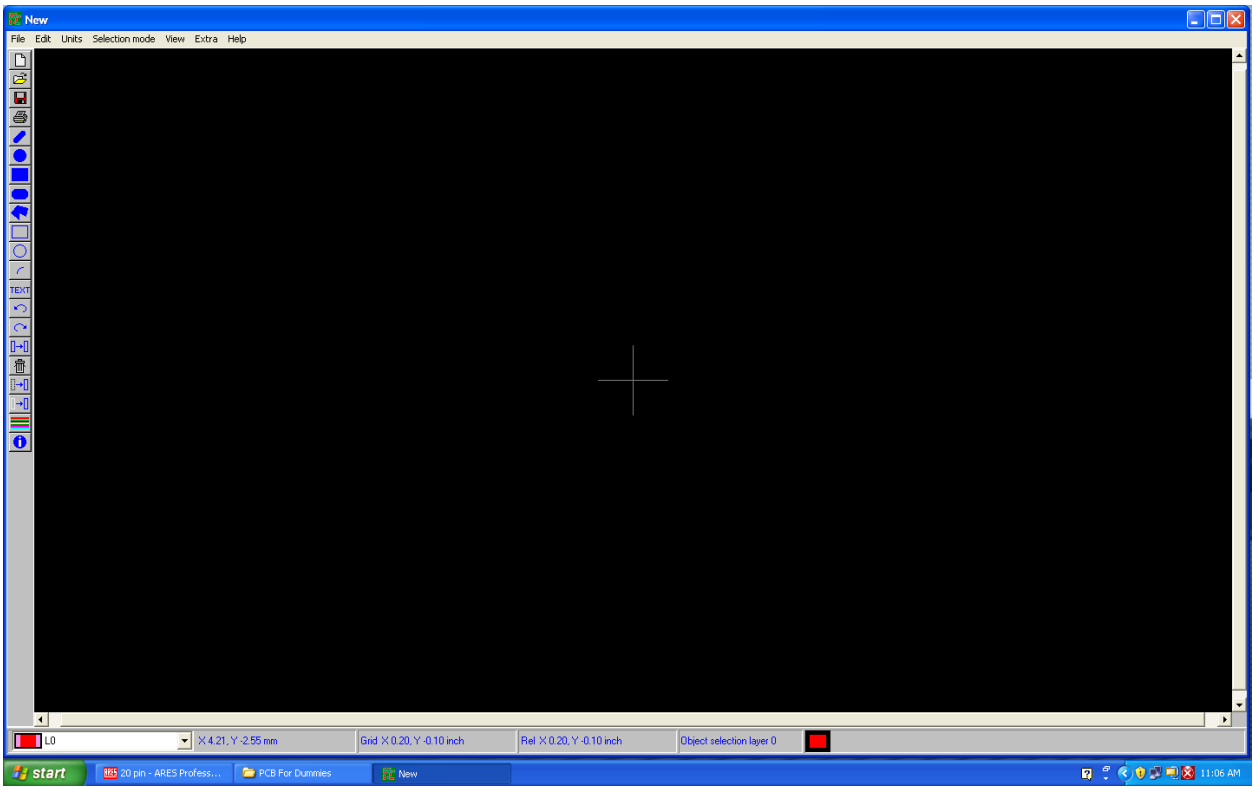
Using ISOCAM to generate Milling and Drill data

Before using ISOCAM make sure that the USB Dongle Key is connected with the computer, see the figure below. Please grab the Dongle Key from the Lab Instructor.

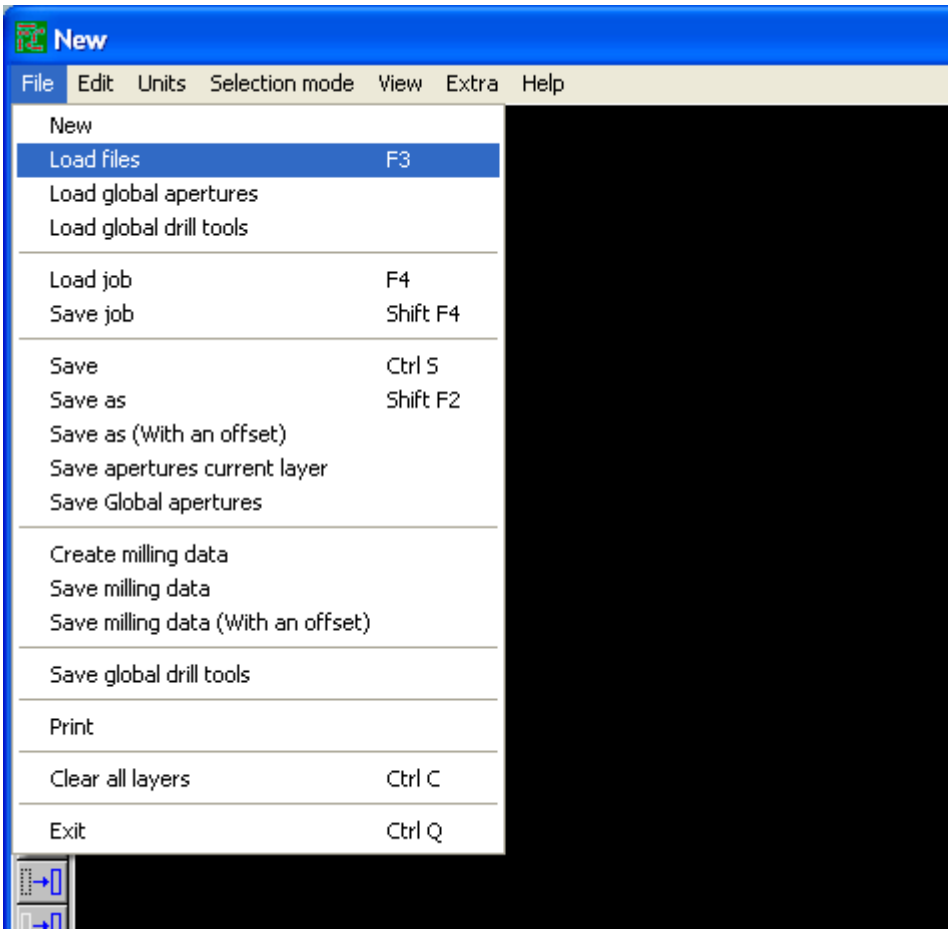
(Note that without dongle ISOCAM will not work hence the entire PCB machine will be useless, so make sure that the dongle is not lost in any case).

1. Now open the ISOCAM on located on Desktop. Or you can use the path C:\Program File\IsoCam if you find problem with it.

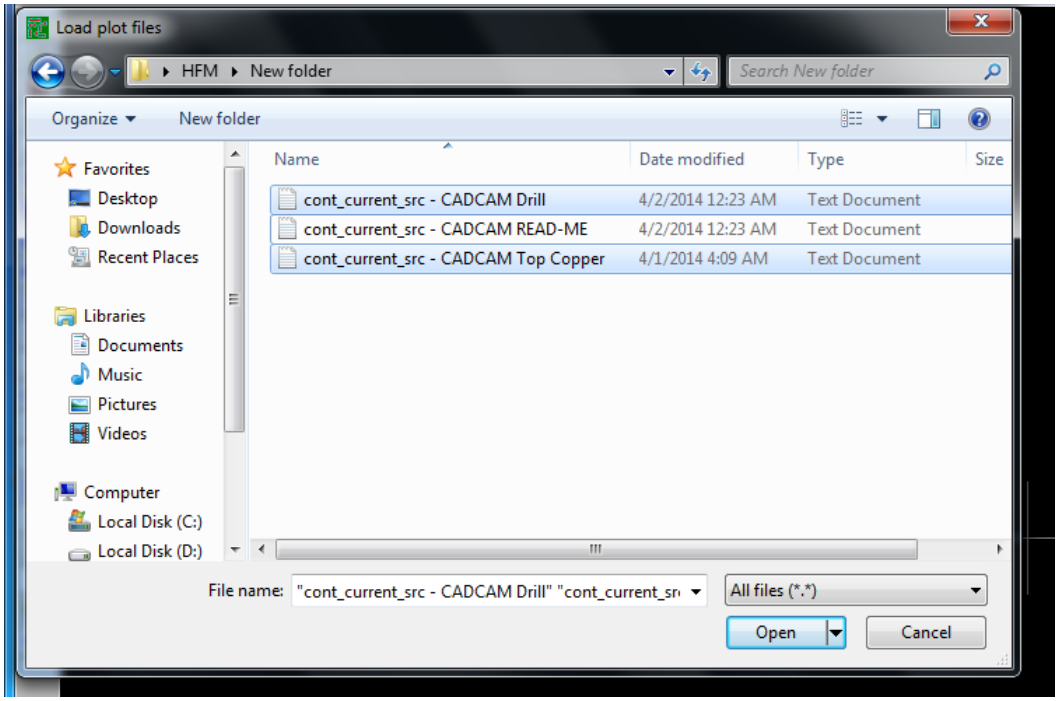




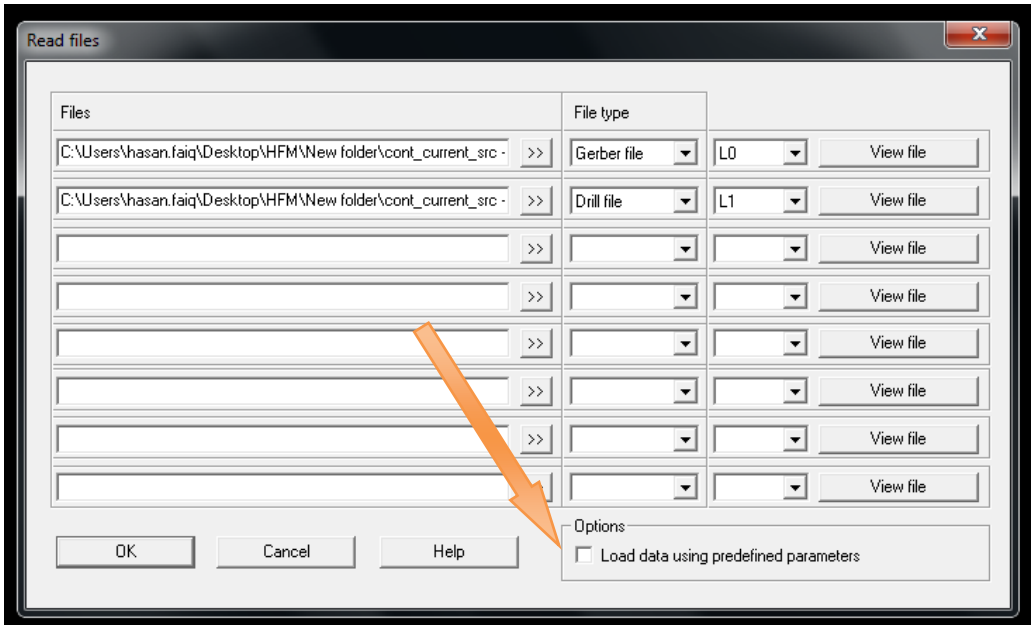
2. Go to the file menu and click on **Load Files** or press F3



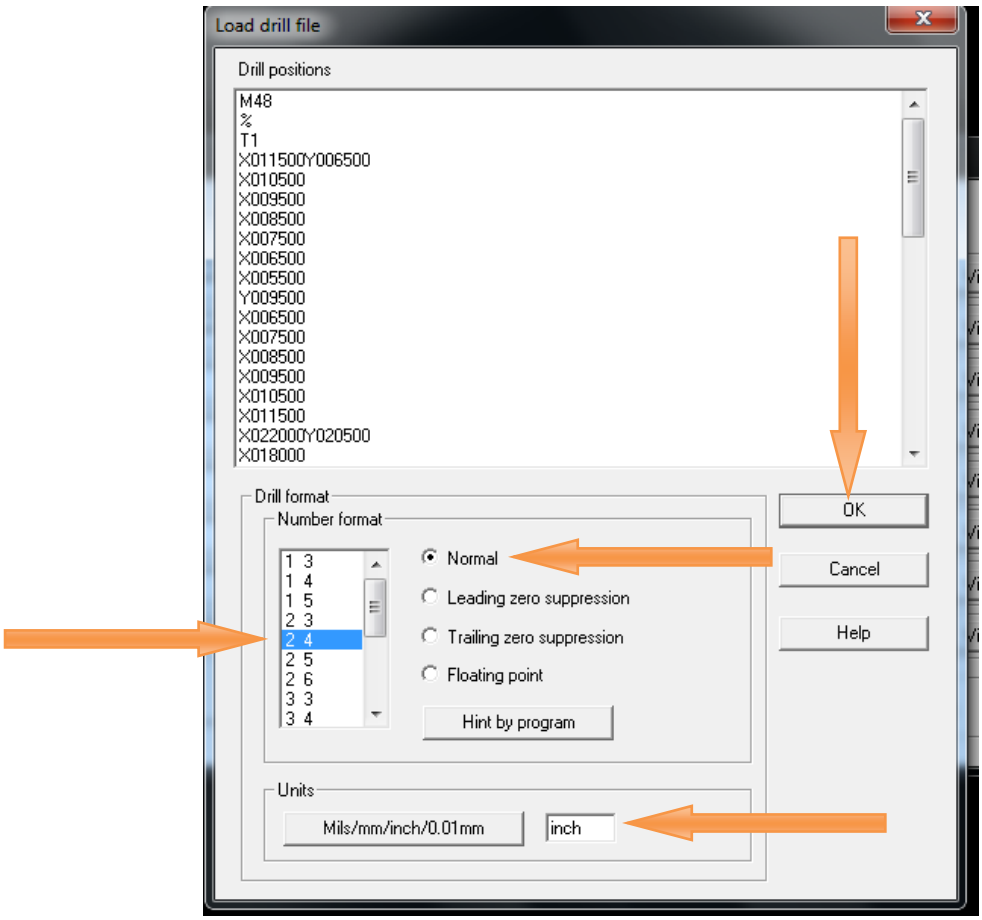
3. Now navigate to the folder where all the Gerber/CADCAM PCB design files are located. Only select Bottom Copper and Drill files.



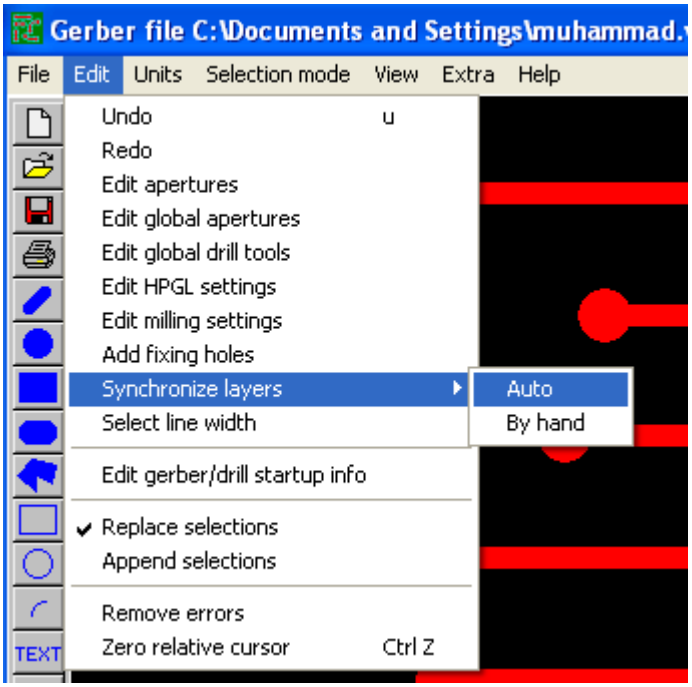
4. You will see that the files are uploaded with the File Type of “Gerber File” and “Drill File”. Make sure that “**Load data using predefined parameters**” option is unchecked, then click OK



5. As you have designed your Layout in ARES, Click on “**Mils/mm/inch.0.01mm**” to select the unit “inch” for your drill file. Number format should be “2 4”and “Normal. Click OK.

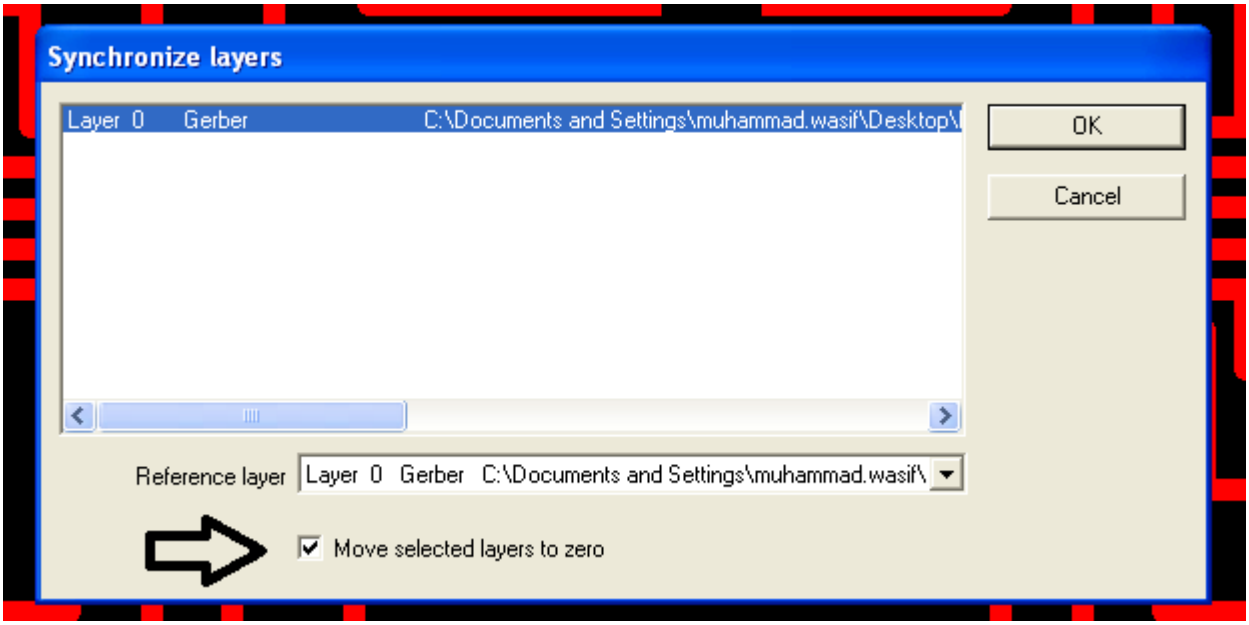


6. Now go to **Edit** menu select **Synchronize layers** and click on **Auto**



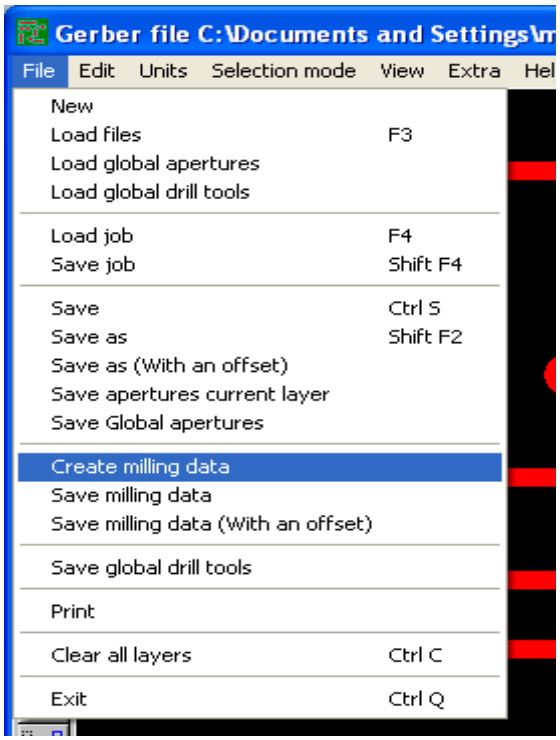
7. Make sure that "**Move selected layers to zero**" option is checked, then press **OK,** it will synchronize all layers to zero (starting) point of the software spatial plane. Synchronization is necessary to accumulate Bottom Copper and Drill layer exactly on each other.

   (Note: Here zero point means the origin of the XY plane of your PCB and Layers mean Bottom Layer and Drill Layer).

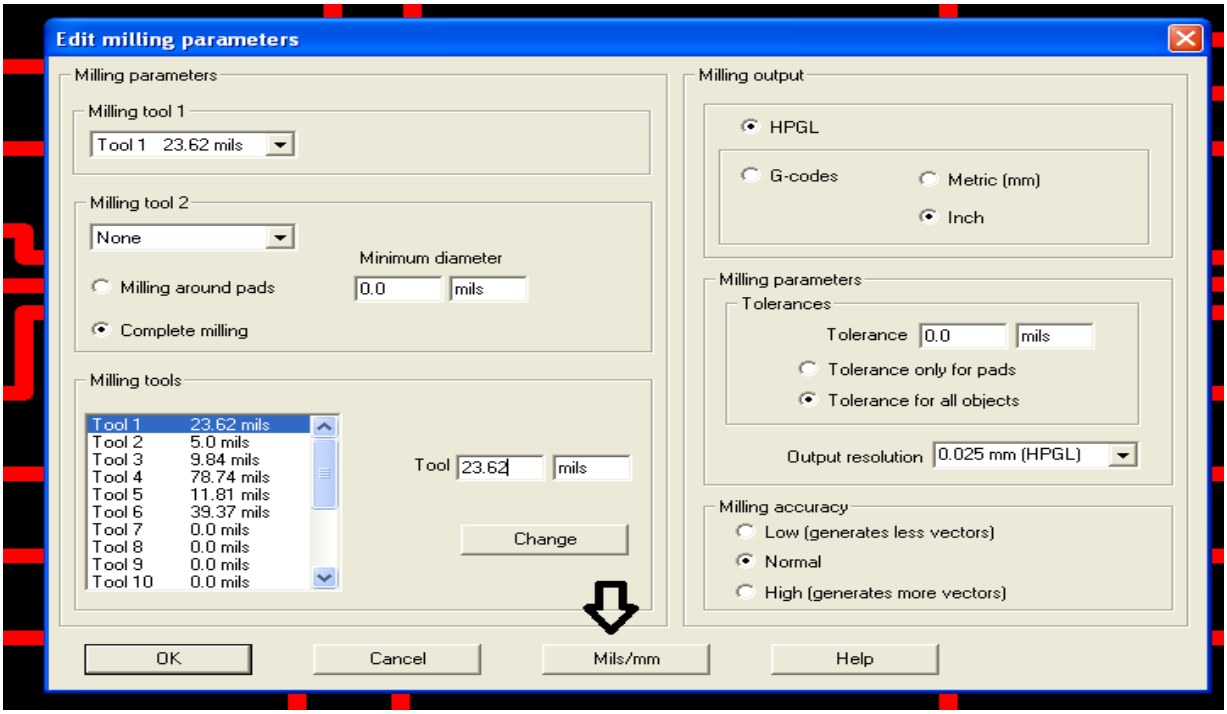


8. If you are unable to synchronize these layer by "Auto" mode, use "By Hand" mode and select any part of each layer (by clicking on it) to make them synchronize.

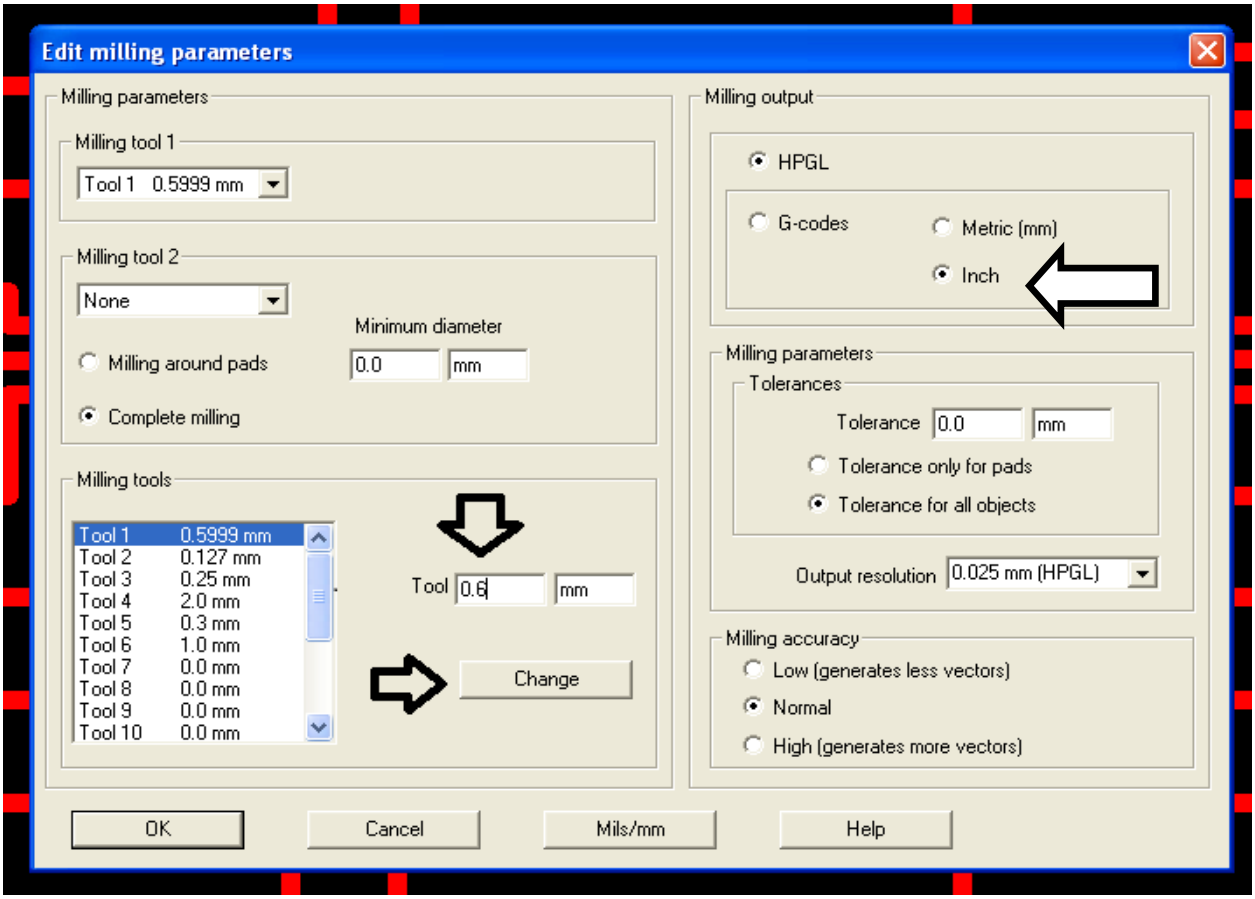
9. Now go to file menu and click on **Create milling data ,** the ISOCAM will create the milling/routing data in CCD2 machine format



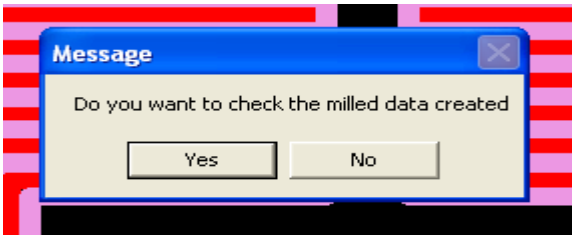
10. Make sure the **Tool** size is 0.6 mm, if the unit are in mils then click on **Mils/mm** button as shown



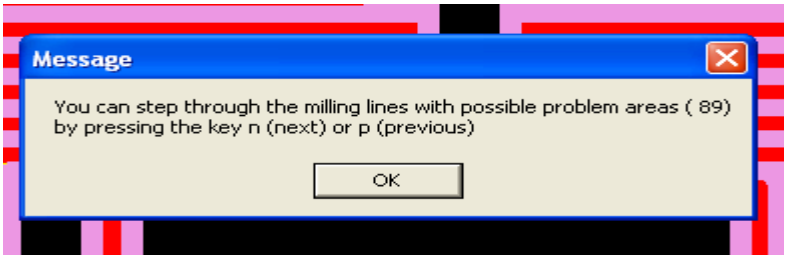
11. Now enter **0.6** in the **Tool** option and then click on **Change** and then finally click **OK** as shown below. Also make sure that the Milling output should be on "**Inch**" option.



12. Now it will ask you if you want to check whether the track clearance is satisfied with the milling tool (0.6 mm) you selected previously, click **yes** to proceed.

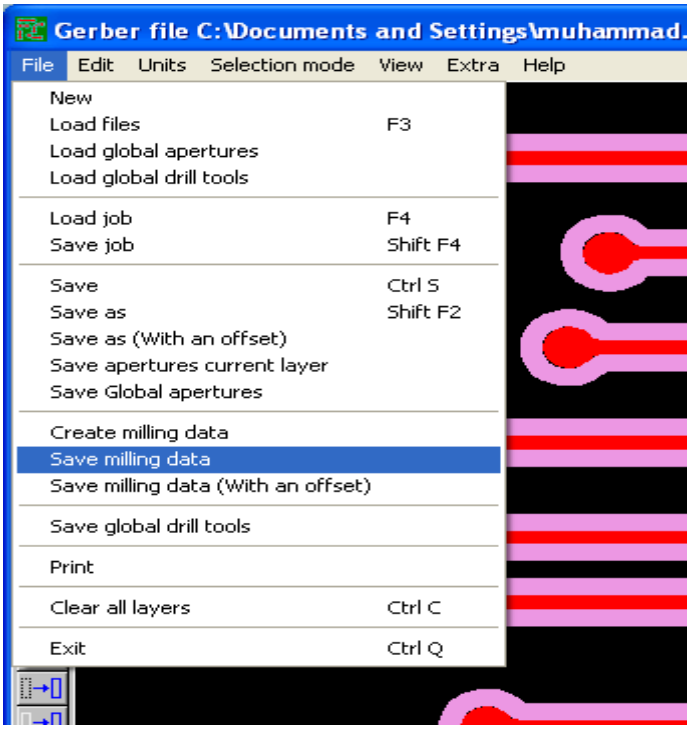


13. Press **OK**

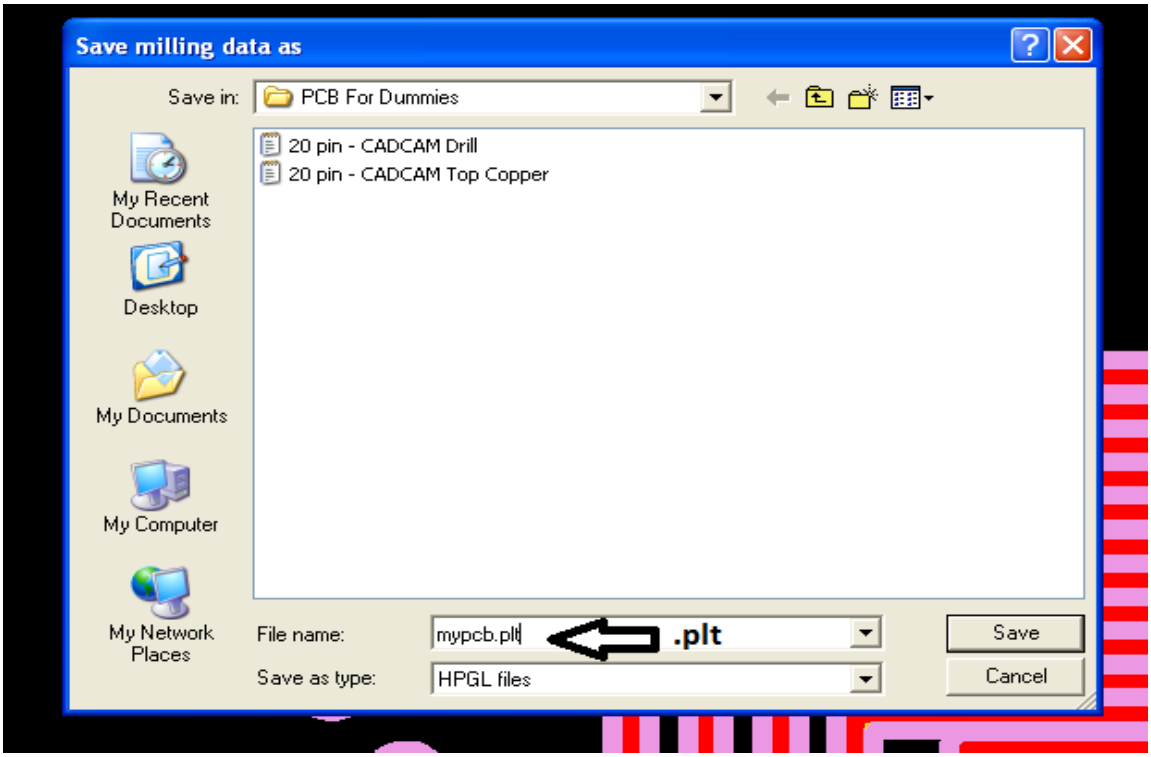


14. If your Layout paths are very thin and the milling data is unable to drive these path especially in between the adjacent circles, change the milling tool to 0.3mm or less so that you may get your desired path.

15. Now goto file menu and select "**Save milling data**"

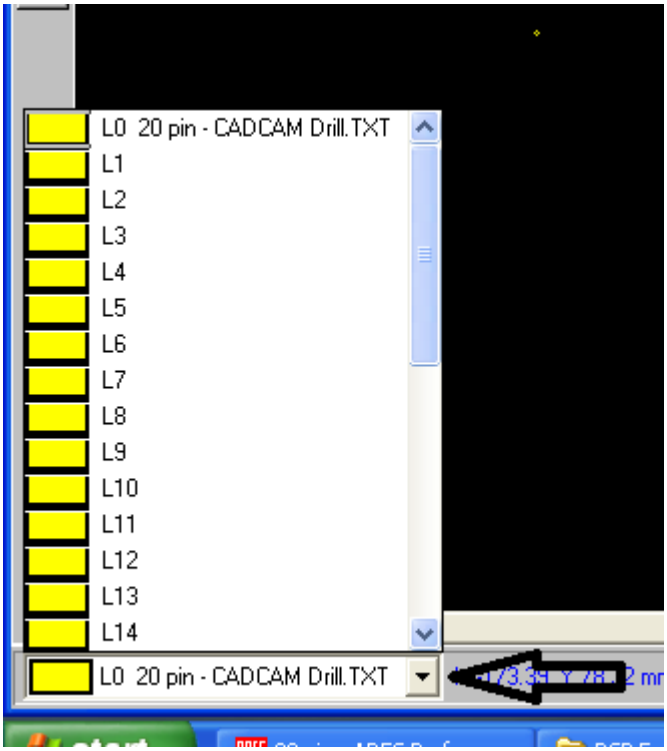


16. Enter any name, but make sure it ends with the extension **.plt** as shown in below



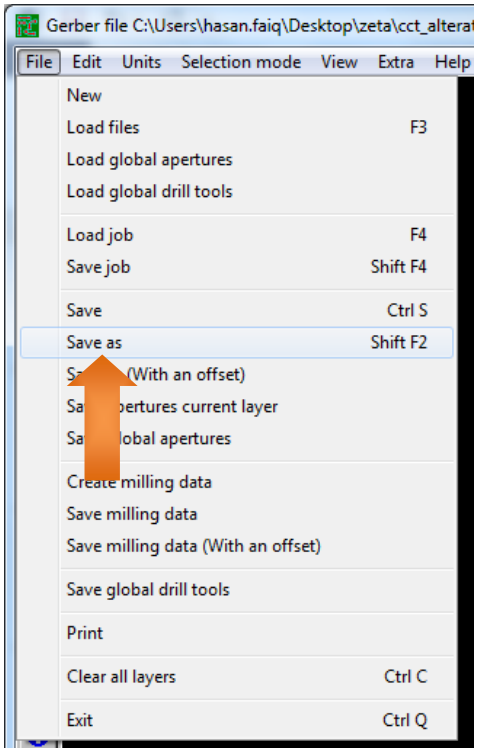
17. So you have successfully created the milling/routing file.

18. Now to create the Drill file: select the drill file from the layer menu at bottom left of the ISOCAM software

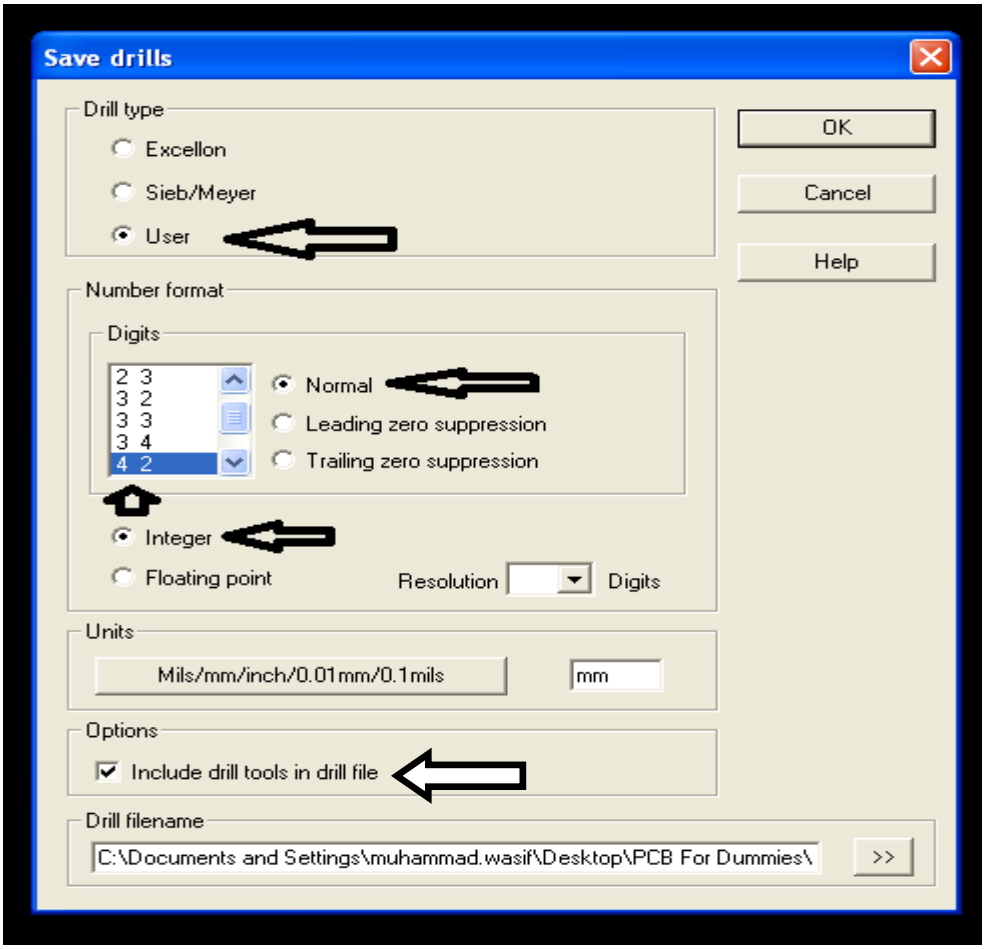


19. After selecting the Drill file you will observe the file name with full path on the top of ISOCAM window e.g C:\Documents and Setting\.....\20 pin – CADCAM Drill.TXT

20. Now goto the **File** menu and select **Save As.** Write the file name and save it with an extension **.ncp**



21. Make sure the following options shown in Figure are selected, then press **OK.**
    Do remember to check "**Include drill tools in drill file**"



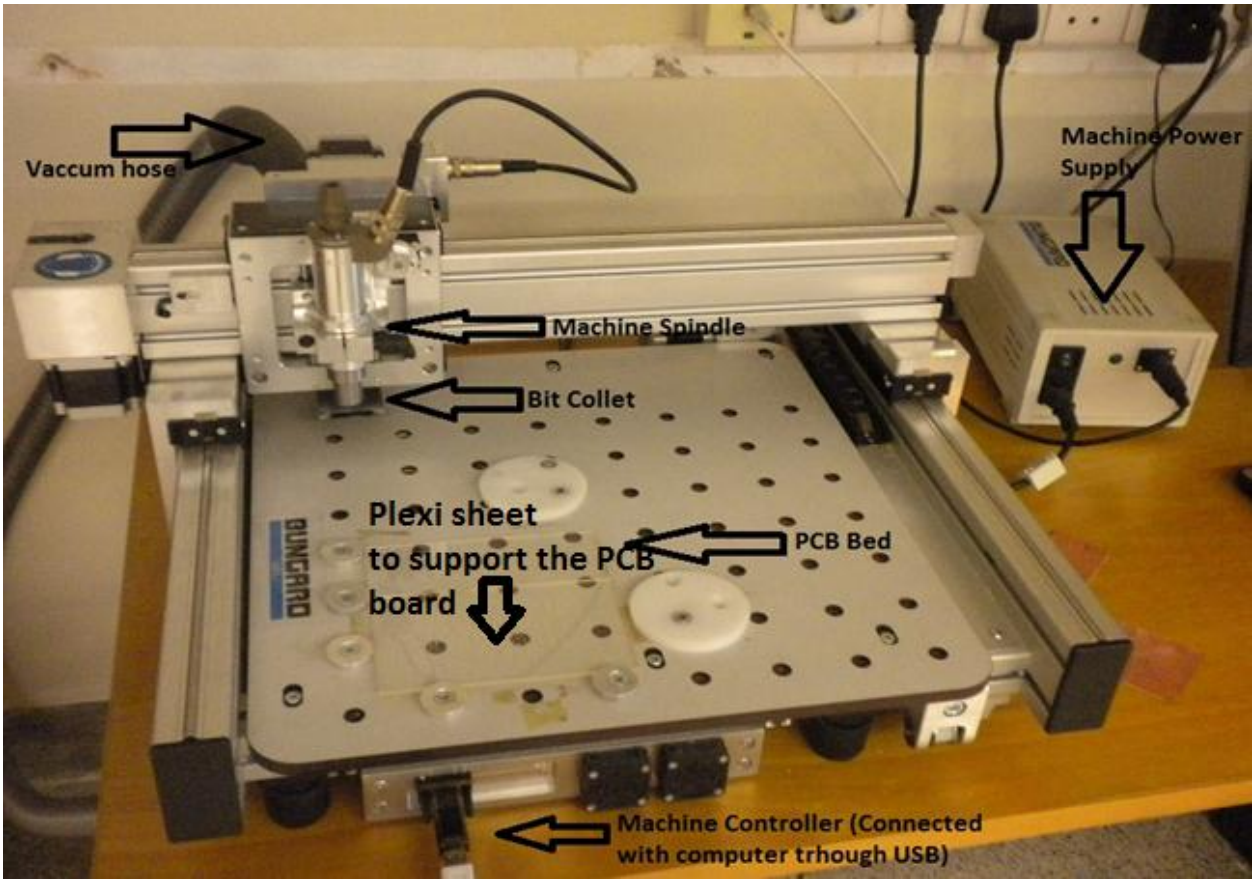
22. If any thing goes wrong repeat above step 2 and also refer to the ISOCAM help menu. Also consult the ISOCAM manual on Physlab website or visit http://www.bungard.de/index.php?option=com_content&view=article&id=5&Itemid=98&lang=english

CHAPTER 4: CNC MACHINE OPERATION

Setting up the CNC machine for operation.

1: Basic parts

- **Machine Spindle**: Machine main motor, having speed upto 65000 rpm.
- **Bit Collet**: Attached to spindle and holds drilling/routing bits.
- **PCB Bed**: Holds the PCB board on which the circuit trace to be made.
- **Vacuum Hose**: Sucks in board dust during machine operation.
- **Machine Controller:** Controls the machine operation with computer interface. Connected to computer via USB.
- **Machine Power Supply**: Machine's main power supply with ON/OFF button. For further details consult CCD2 manual on Physlab website or visit http://www.bungard.de/index.php?option=com_content&view=article&id=5&Itemid=98&lang=english
- **Plexiglas sheet**: The transparent Plexiglas sheet is used to protect the PCB Bed while the drilling is done on the circuit board. The DRILL or routing Bit can go through the PCB sheet which can damage the Machine Bed, so make sure that plexi sheet is always under the circuit board or PCB sheet.



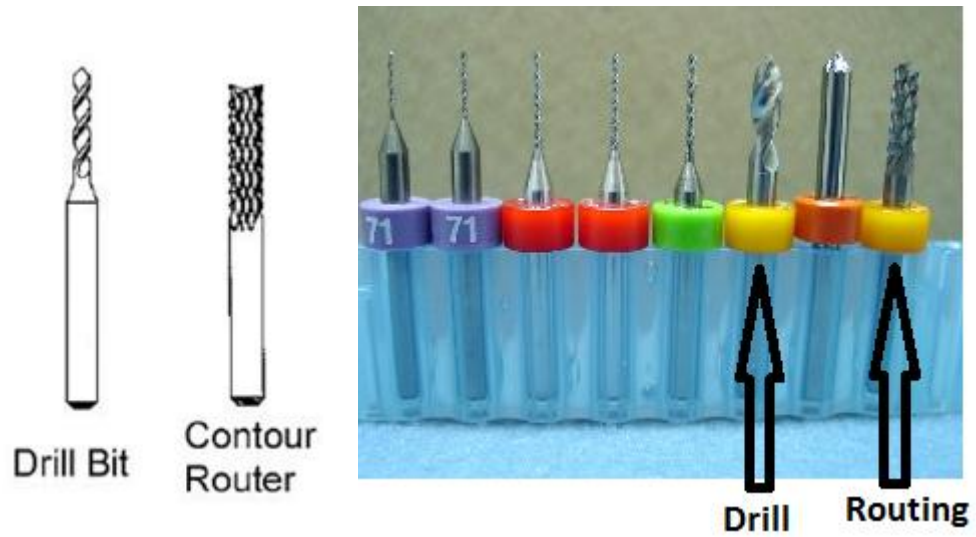
CCD2 PCB MACHINE

1. The vacuum hose is connected to the vacuum pump

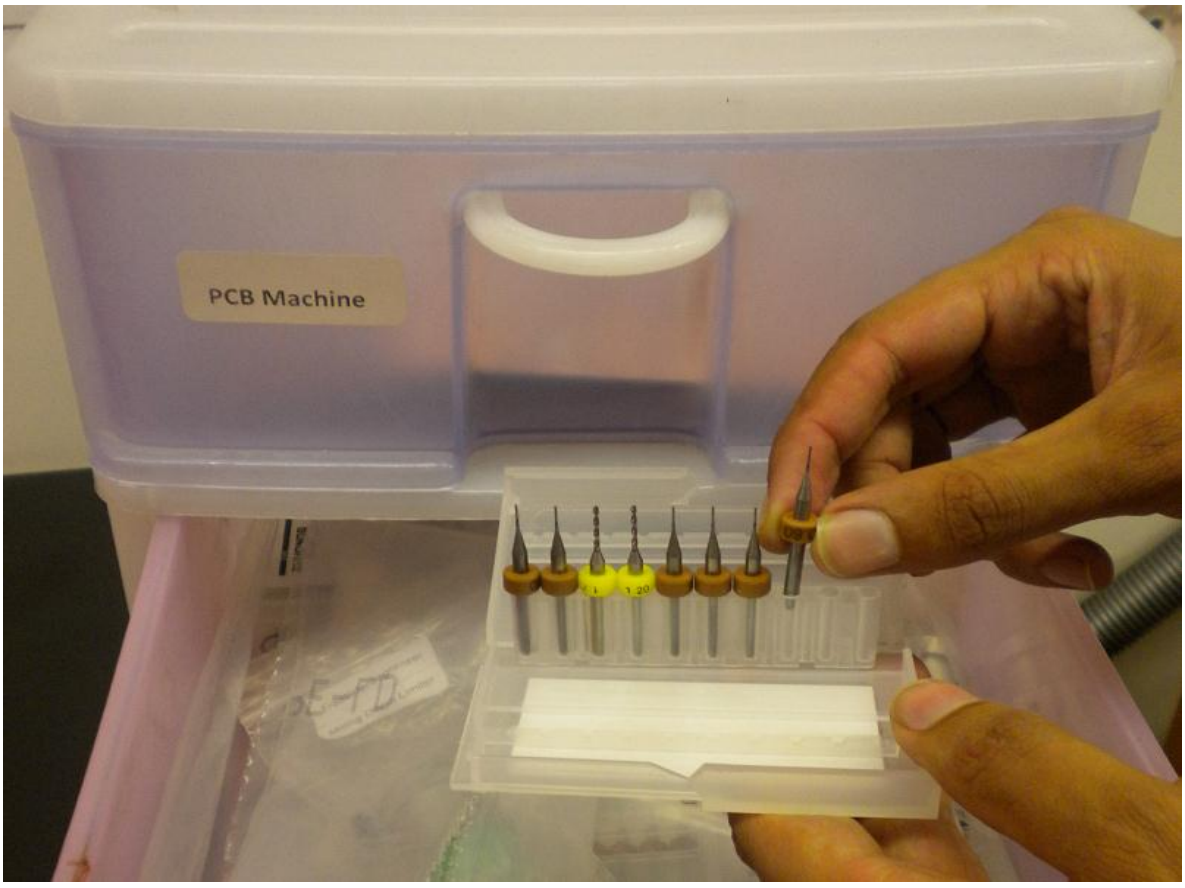
2. The vacuum pump is automated with PCB machine, its turns on automatically when the machine is in operation. Make sure that the manual switch of the pump is turned ON. The vacuum pump turns ON/OFF by pushing the button shown below:

2: Preparing for routing/milling

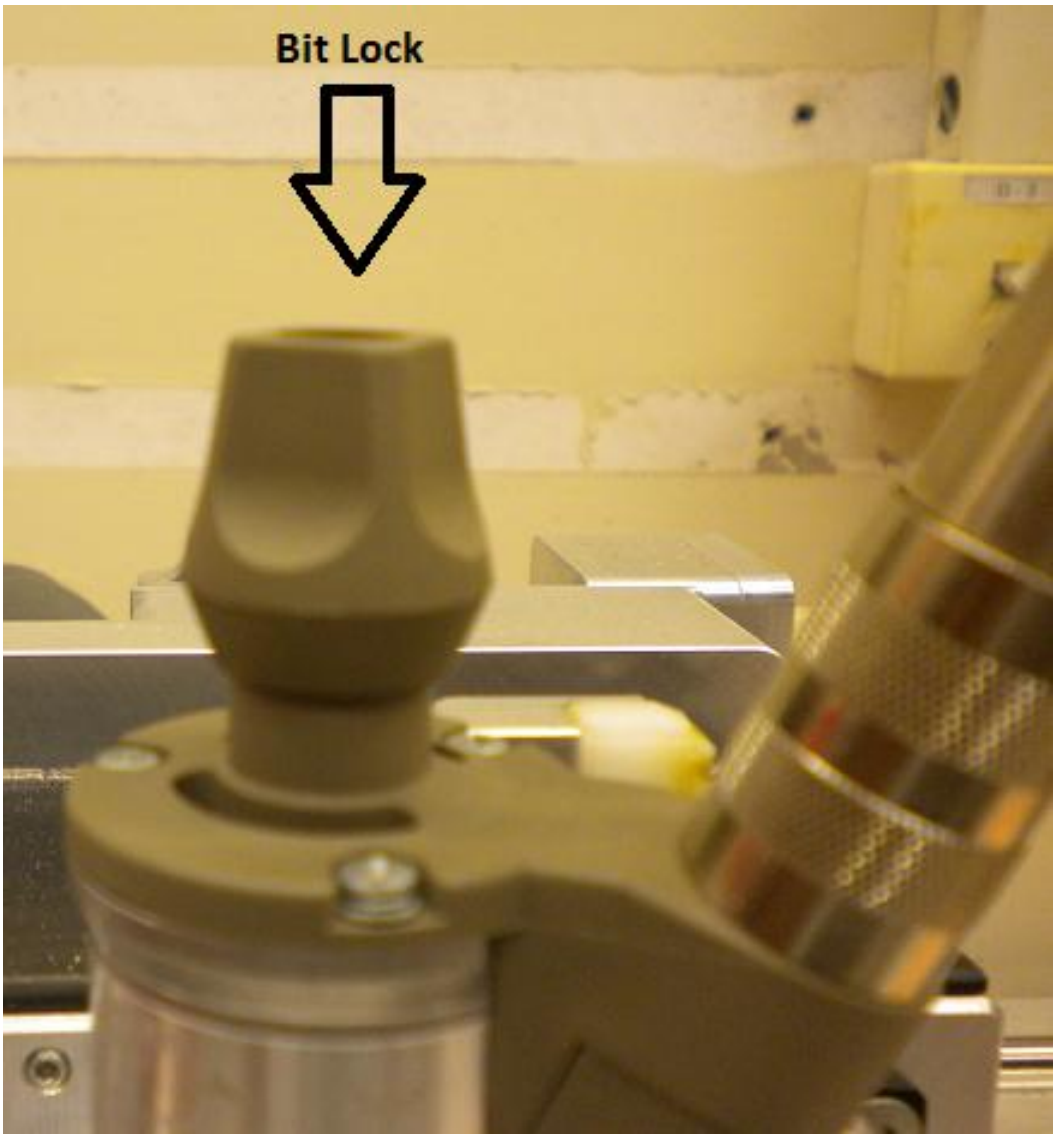
3. To prepare a machine for routing first you need to insert the appropriate routing bit into the bit Collette. The figure below shows a basic difference between a Drill and Routing bit.



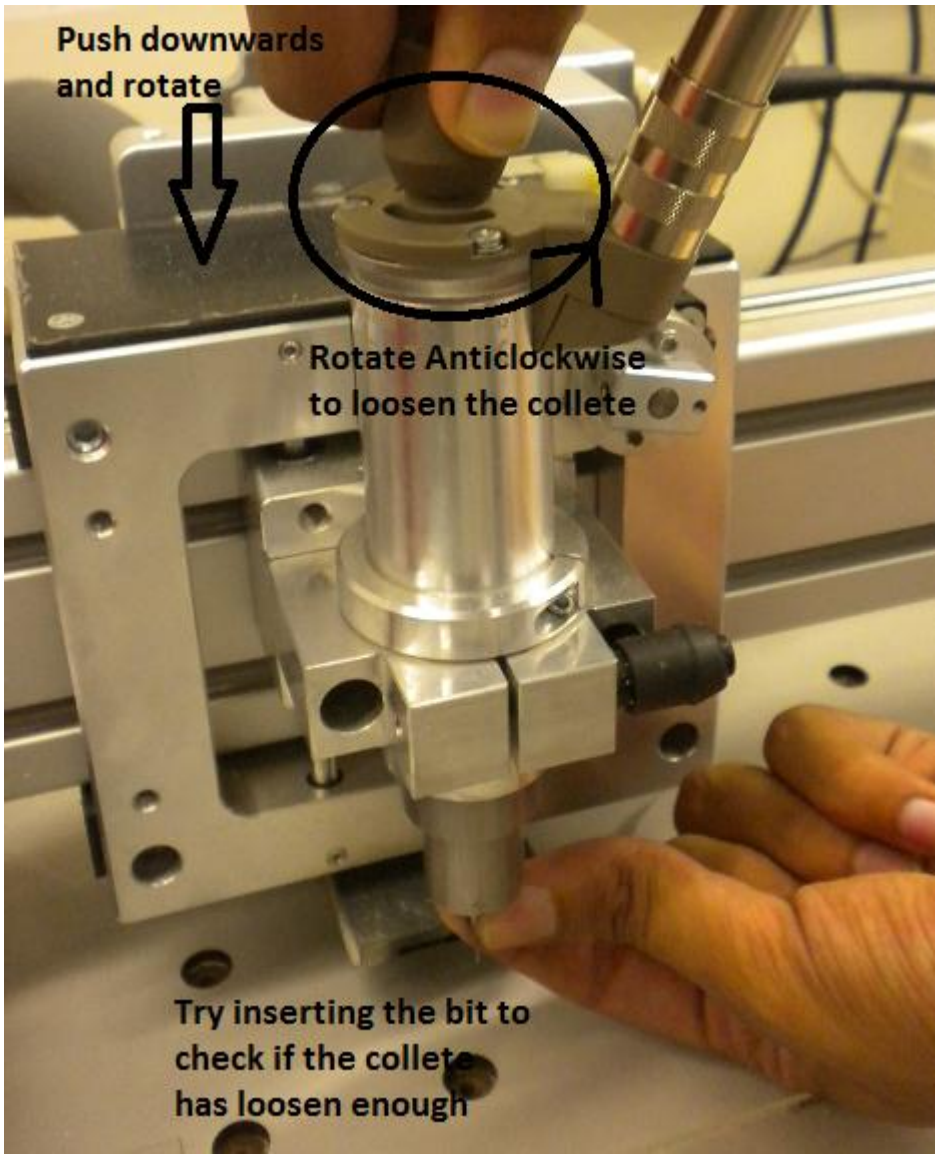
4. In physics lab you can locate the routing bits in a box labeled "**PCB Machine**" located by the side of CCD2 PCB Machine. Only 0.6 mm routing bits are available with brown colored collar as shown below

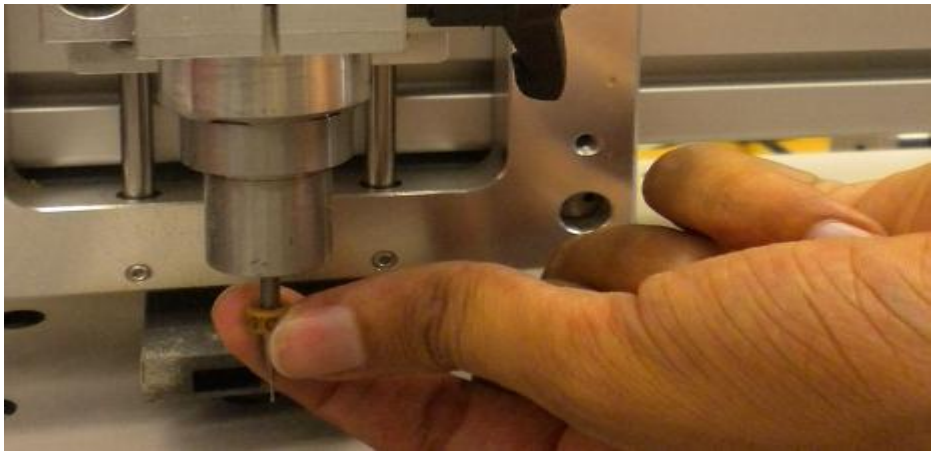


5. To insert the routing bit, locate the bit lock, located at the top of the spindle

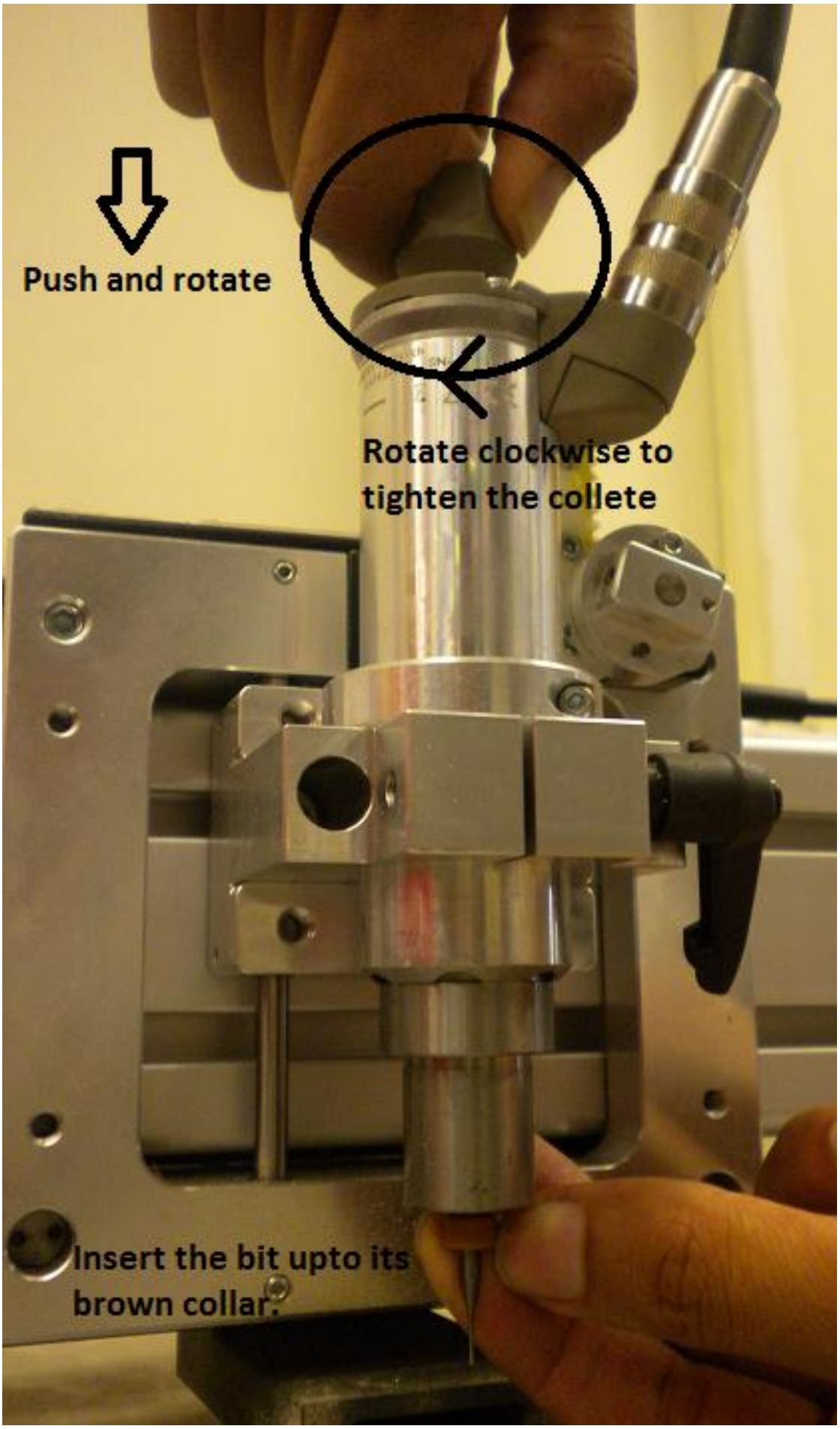


6. First you need to loose the Collette so that you can insert the bit, to do that, push the **bit lock** downwards gently and rotate it counter clockwise, with two or three rotations the Collette will be loosen, to check whether the **collate** has loosen, try inserting the bit in the Collette gently if you unable to do so then try repeat the above steps again until the Collette is loose.



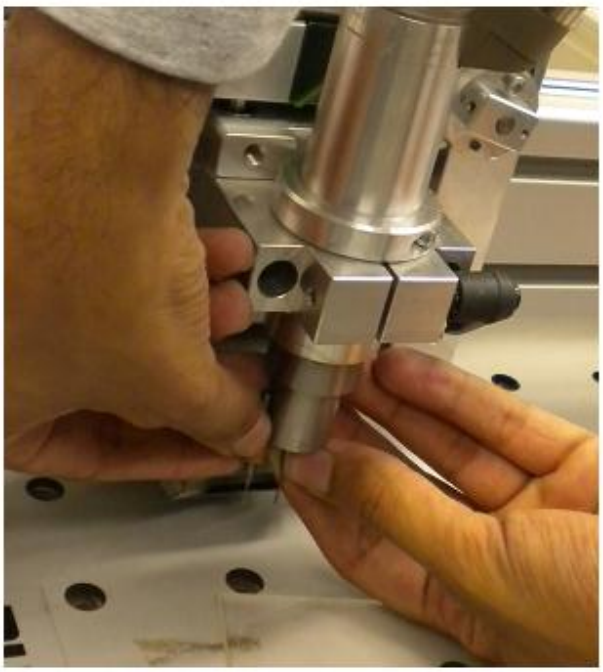


7. When the Collette is loosen enough and the bit can easily slide in, insert the bit upto its brown collar and tighten the collete for which you now need to rotate the **bit lock** clockwise as shown in the figure below



8. To insert the drill bit use the same above procedure. There are different sizes drill bits available in physics lab, but the most suitable commonly use size is 1mm. As shown in figure below

9. Unfortunately the some of the drill bits available in physics lab does not have an indication collar. To make sure that you have inserted the bit to an appropriate depth use the collared routing bit to make a good guess and tighten the collet at that location.

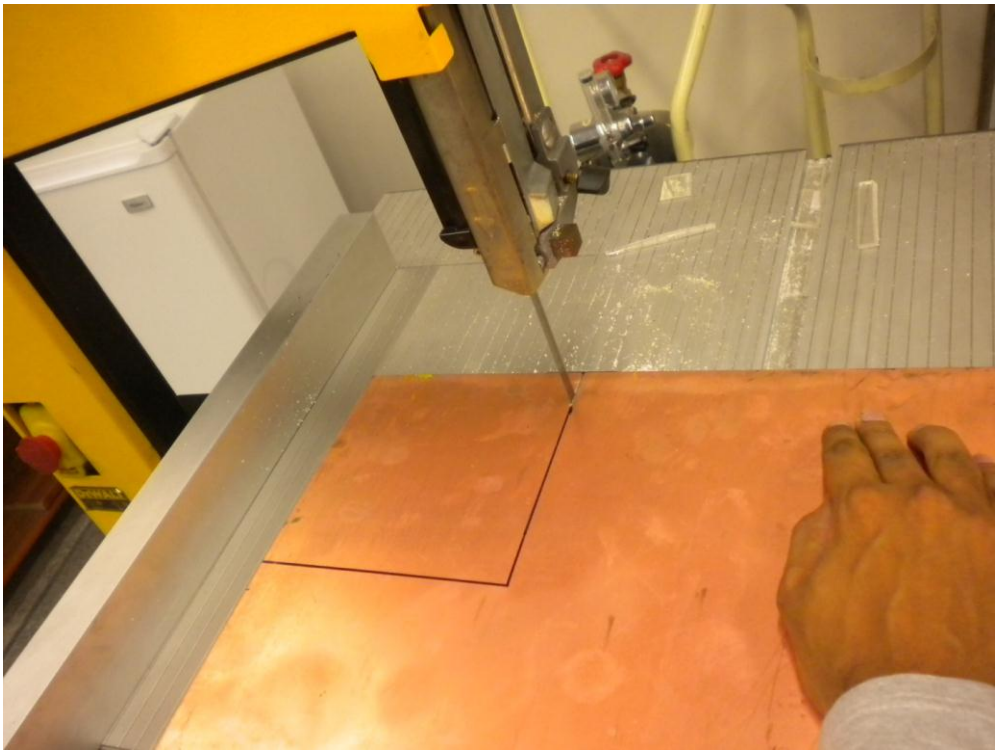


Note: The drill and routing bits are very fragile and expensive, take extra care while handling them.

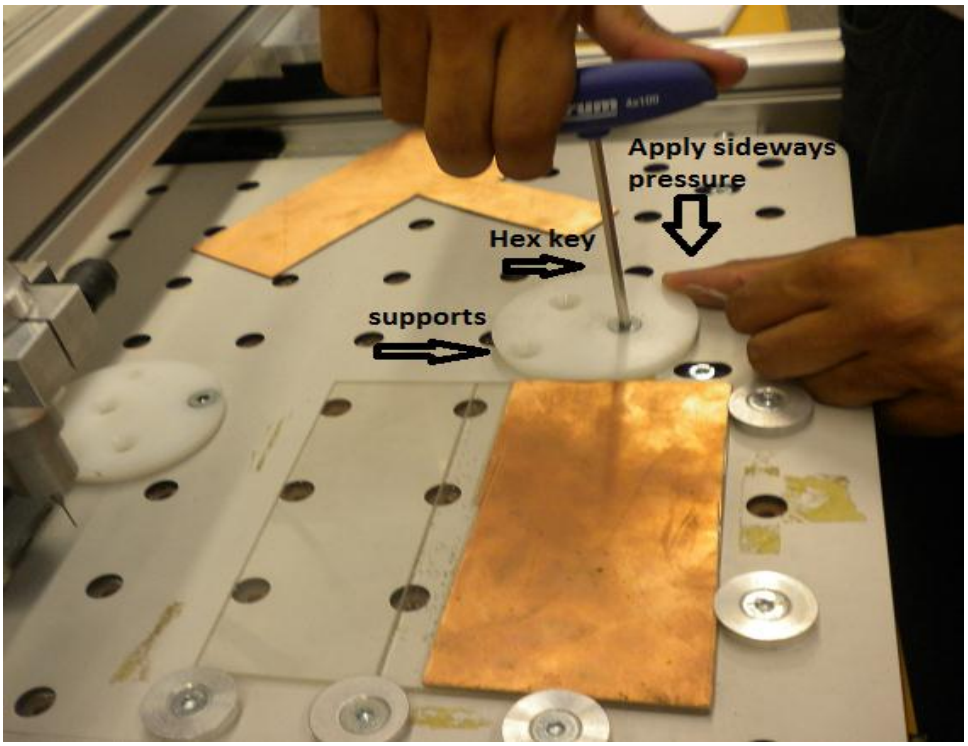
4: Preparing the PCB board

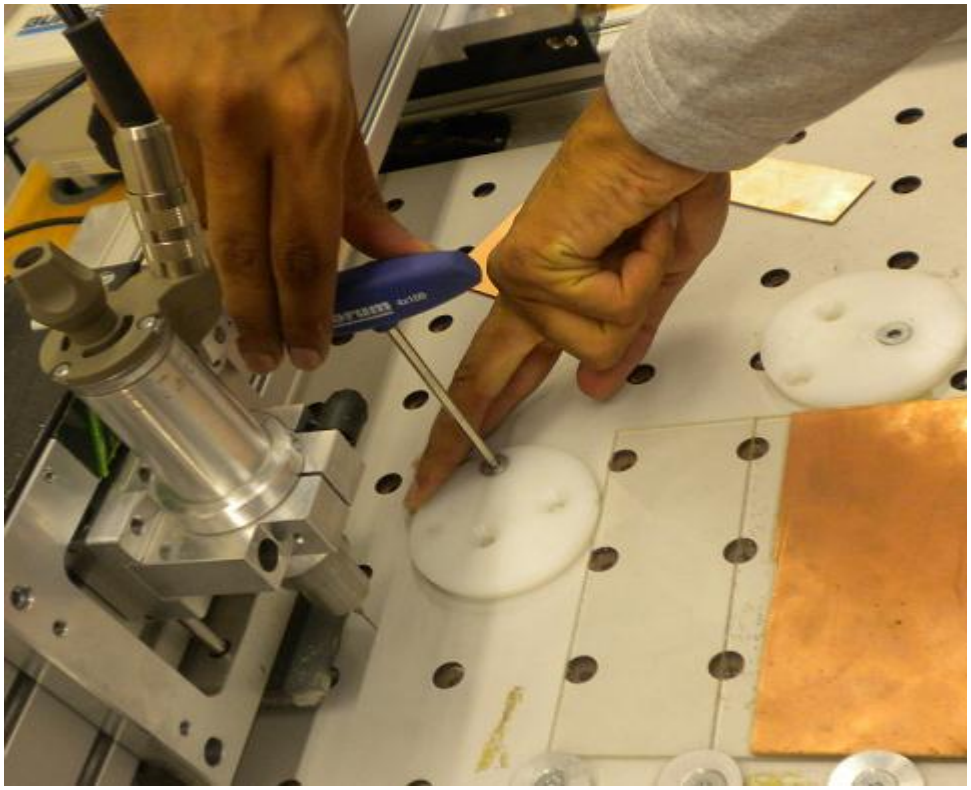
10. Get a PCB board from physics lab support or ask the Lab Incharge, mark the board to cut it into the appropriate size, check the design/ PCB bed dimensions first.

11. Use the bandsaw cutter to cut the marked PCB board. Take all necessary precaustions before using the bandsaw, wear eye protection glasses and wear gloves. Also ask the Lab Incharge before using the band saw to guide you.



12. Now place the board on the machine's **PCB Bed** and use the hex key and available **supports** to fix the PCB tightly, make sure that the board isn't slack.





CHAPTER 5: ROUTE PRO

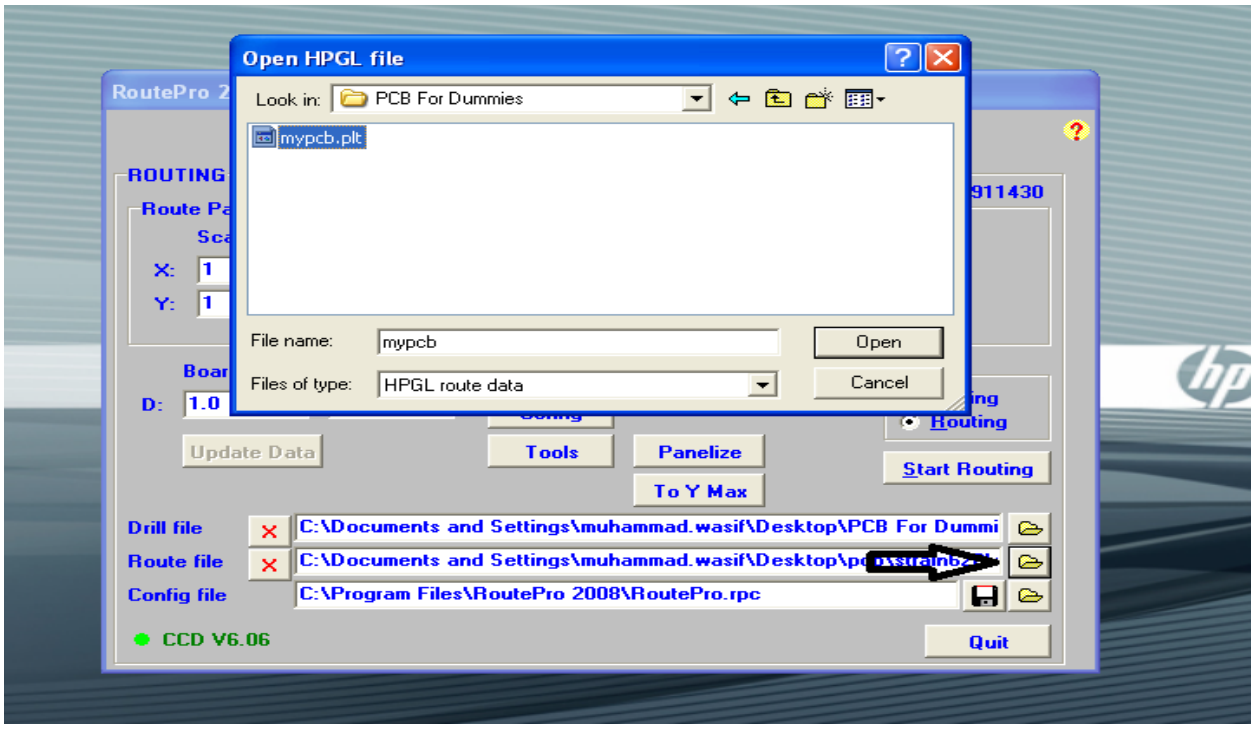
RoutePro is a software that makes this machine work. You have to turn on the power button of machine first in order to run this software. So switch on the power button at the back of the power supply.

Operating the machine using RoutePro

1. Turn ON the PCB machine from the power supply

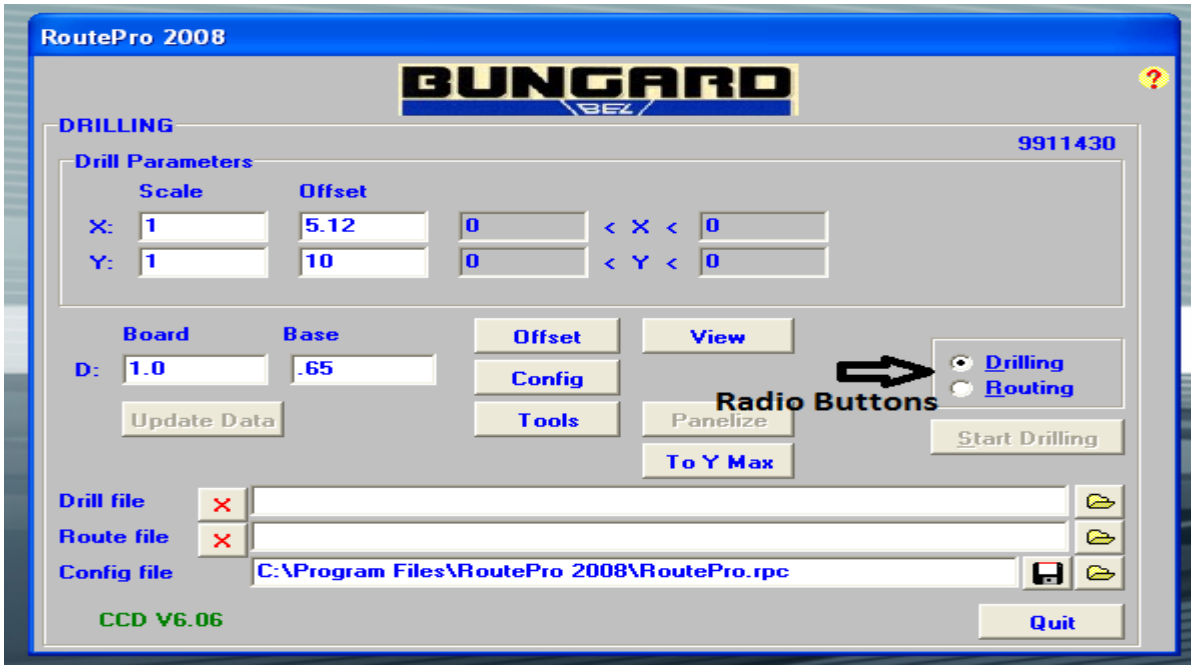
2. Open RoutePro, located on Desktop. Or you can go to C:\Program Files\RoutePro 2008 if you find any problem.

3. First load the **Route File** by clicking on the browse bottom icon shown below. From the browser button navigate to .**plt file** that you have previously created while saving the milling data in **ISOCAM.**



4. Similarly load the **Drill** file by browsing to the file location with extension **.ncp**

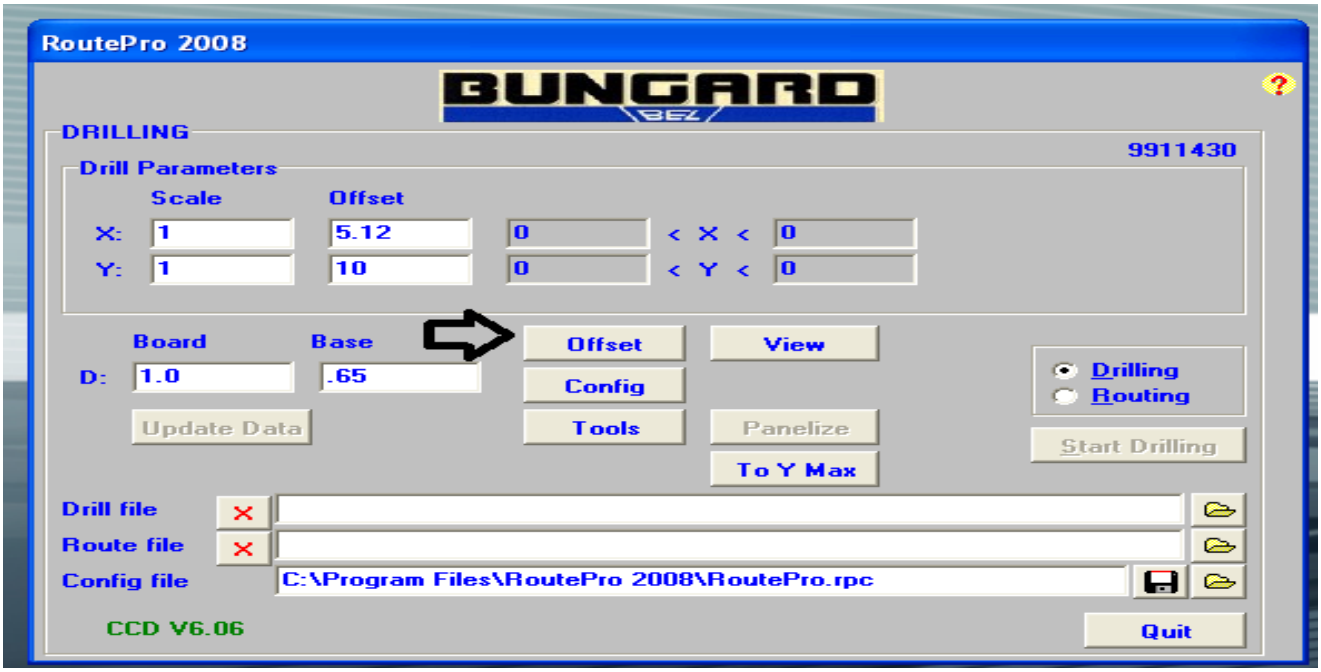
5. The **Drilling** and **Routing** radio buttons shown below are used to configure and operate the machine for Drilling or Routing operation at a time. So when you this machine to route for you, click on the **Routing** option; and if you want to drill on your PCB, choose the **Drilling** option.

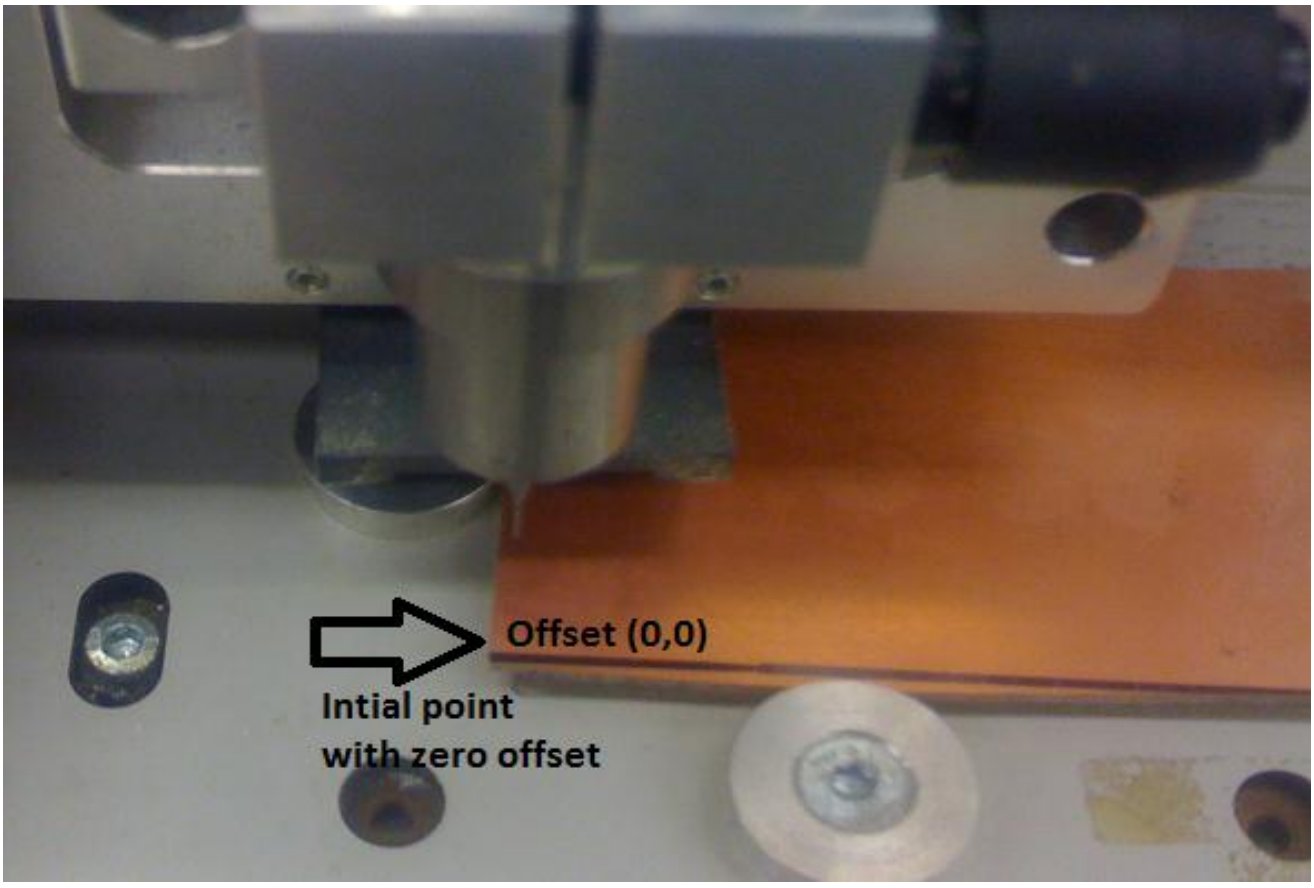


6. **Offset** defines the point on PCB from where you want this machine to start its work. By setting the **Offset**, you set the origin of the machine. Let's configure the machine offsets for drilling operation.

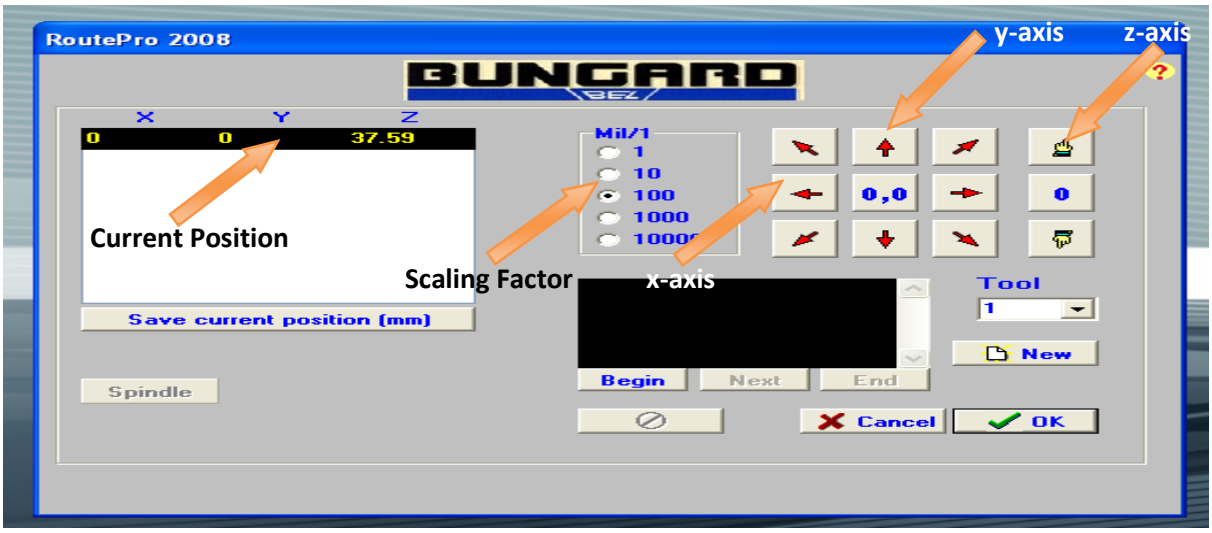
Note: RoutePro directly operates the machine so make sure that PCB machine bed and the area around the machine is clear.

7. You can set the machine offset by clicking on the **Offset** button. The machine will relocate its position to its initial (0,0) location as shown below. Note that the radio button is set on **Drilling.**



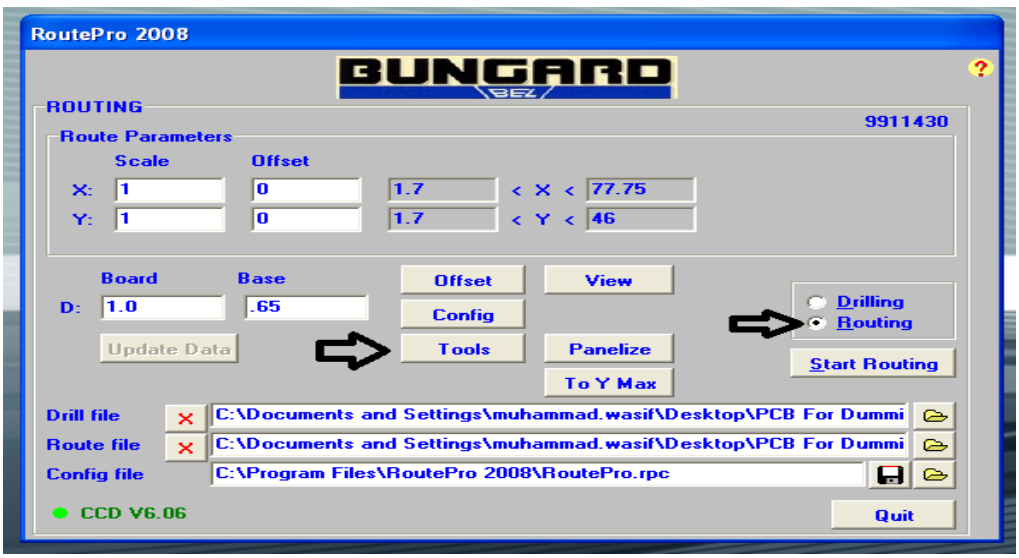


8. After clicking on the **Offset** button a new will appear as shown below:

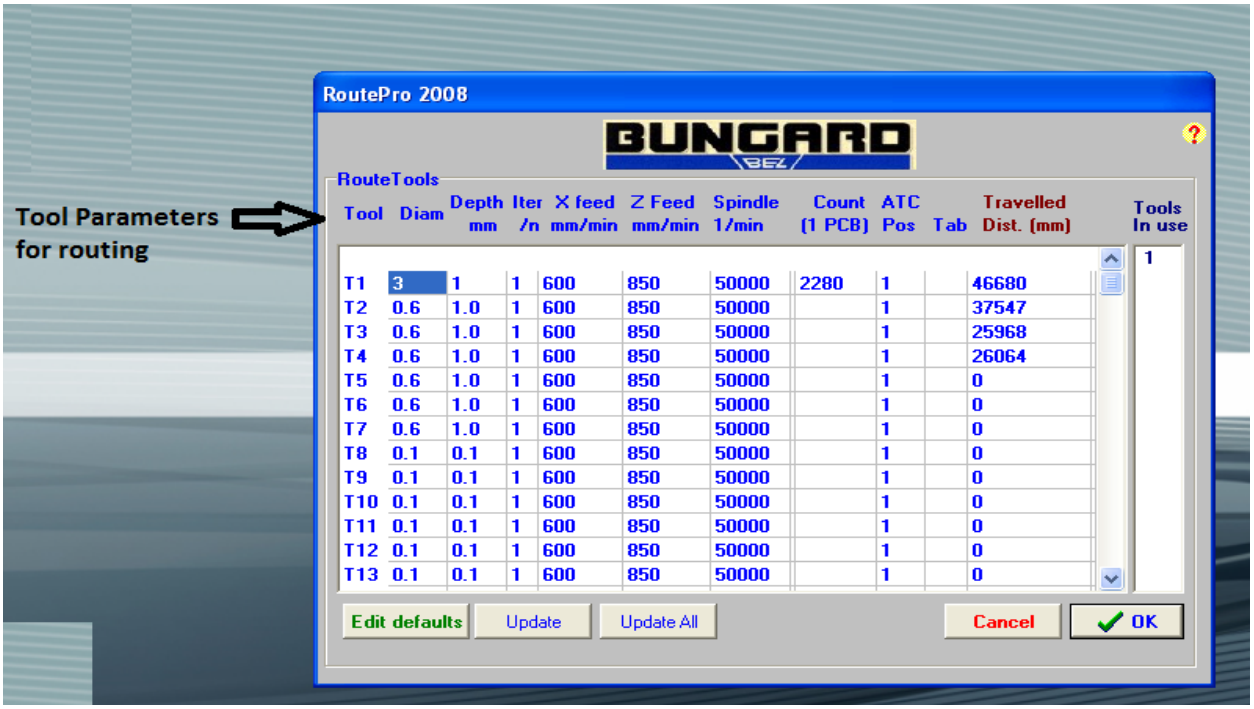


9. By clicking on the "**red arrows**" you can adjust the machines initial position of operation. Adjusting the machine's initial position by putting the offset means that the machine will start making the circuit from that point. Once you are satisfied with the initial position (Offset points) click **OK.**

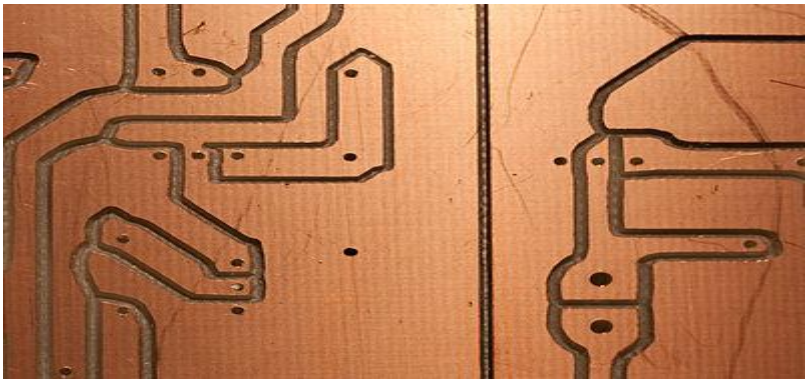
10. The same way you can set the offsets for **Routing** by selecting the radio button on **Routing**. Do make sure that the Routing and Drilling offsets are the same otherwise the drilling and routing won't be synchronized and you will have a circuit board with erroneous drilling locations. For further information check the online manual on physics lab website.
11. From the scaling factor, you can control the scale of the moving spindle. More the scaling factor, greater will be the movement of spindly in that particular direction.
12. Note: Do not bring the spindle/drill bit near the PCB. It means you do not have to adjust the Z-axis. Just configure the x and y-axis. Machine will figure the values of z-axis via **Tools** configuration.
13. Now let's configure and finalize the routing parameters , make sure that the radio button is set on Routing and then click on **Tools.** Tools option allows you to configure the setting about the tools you are using. You will give machine the information about the routing or drilling bit through it. For example, you have to drill holes in your PCB of diameter 0.3mm and 0.6mm. So you Tool1 (T1) will be 0.3mm diameter of drill bit and Tool2(T2) will be of 0.6mm diameter of drill bit.



14. The figure below shows the configuration parameters for routing tool. You only need to set parameters for tool **T1** because we are only using one tool for routing. Now below on **Diam** column enter **0.6** and press **Update**. Diam represents the diameter of the routing tool bit, which we know is 0.6 mm. Under the **Depth** column enter 0.1 and press Update. For the **Iter** column enter 1 and press Update similarly for **X feed** and **Z feed** enter 600 and 850 respectively and press Update. Finally on the **Spindle** column you can enter any value between 50000 to 60000 which is the spindle speed in rpm. After you are done with **T1** row press **OK .**



The **Depth** parameter in the above figure represents the depth of the routing tool inside the PCB board or you can say it is the depth of engraving done on the board while making the circuit tracks as shown in the figure below. If you see that the engraving depth is too much that the bit is passing through the entire thickness of the circuit board than reduce the entered **Depth** to 0.5 or essentially to 0.1 or if you see the engraving is to not done and the bit hovers on the board without touching it then increase the depth. For further understanding refer to the manuals uploaded on the physics lab website

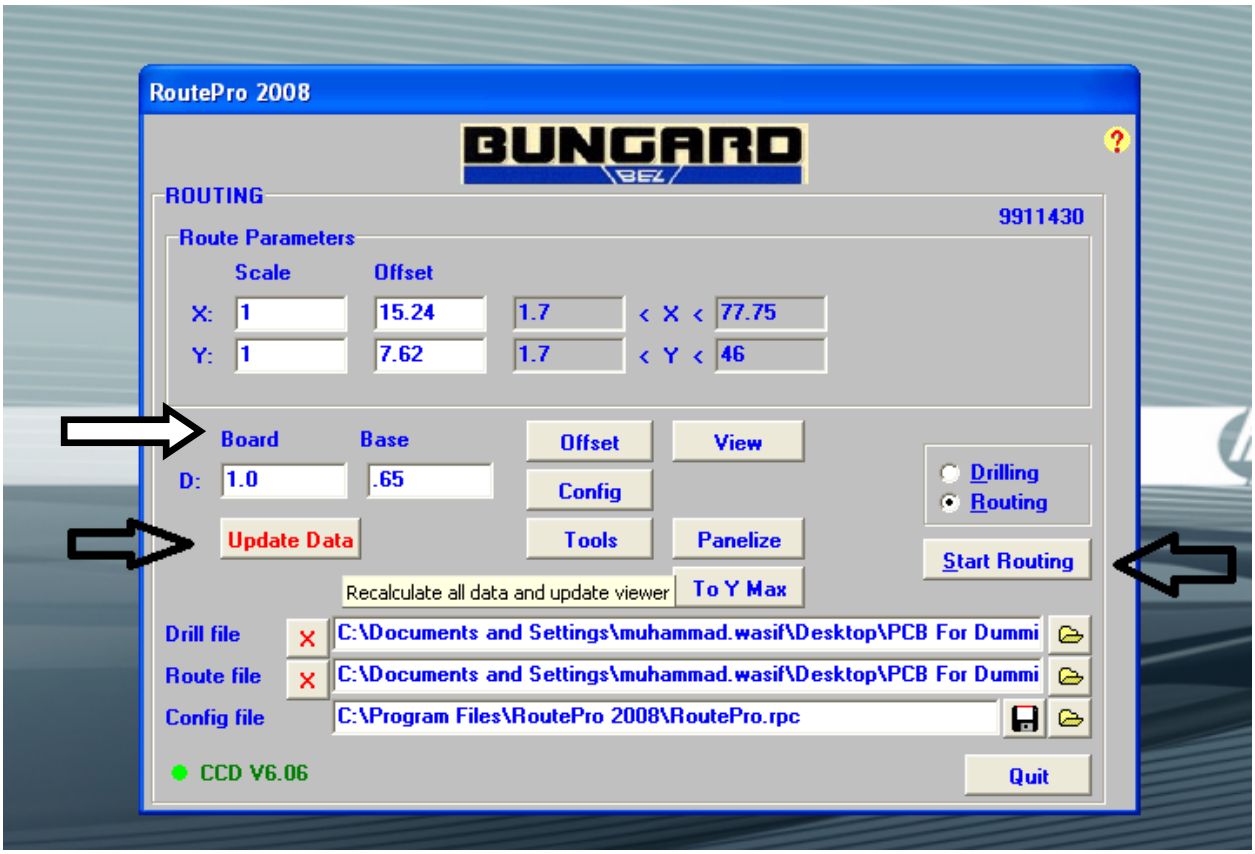


15. You have to provide the value of Thickness of your Board in **Base** option. Its units are in **mm**. Measure the thickness of your Board (on which you have to draw your circuit) and write this

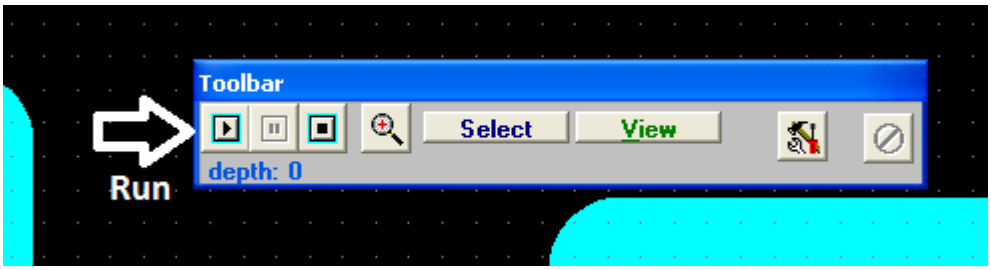
value in the given space. Similarly write the value of thickness of Base of you are placing any board/base for the protection of your machine.

Note: Through these value, machine will figure out automatically how much it has to drill and how deep it has to go for drilling.

16. Now press **Update Data** and then press **Start Routing** as shown below:

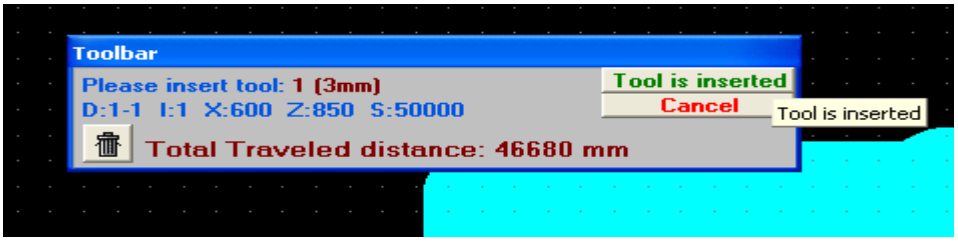


17. Press the RUN button as shown below

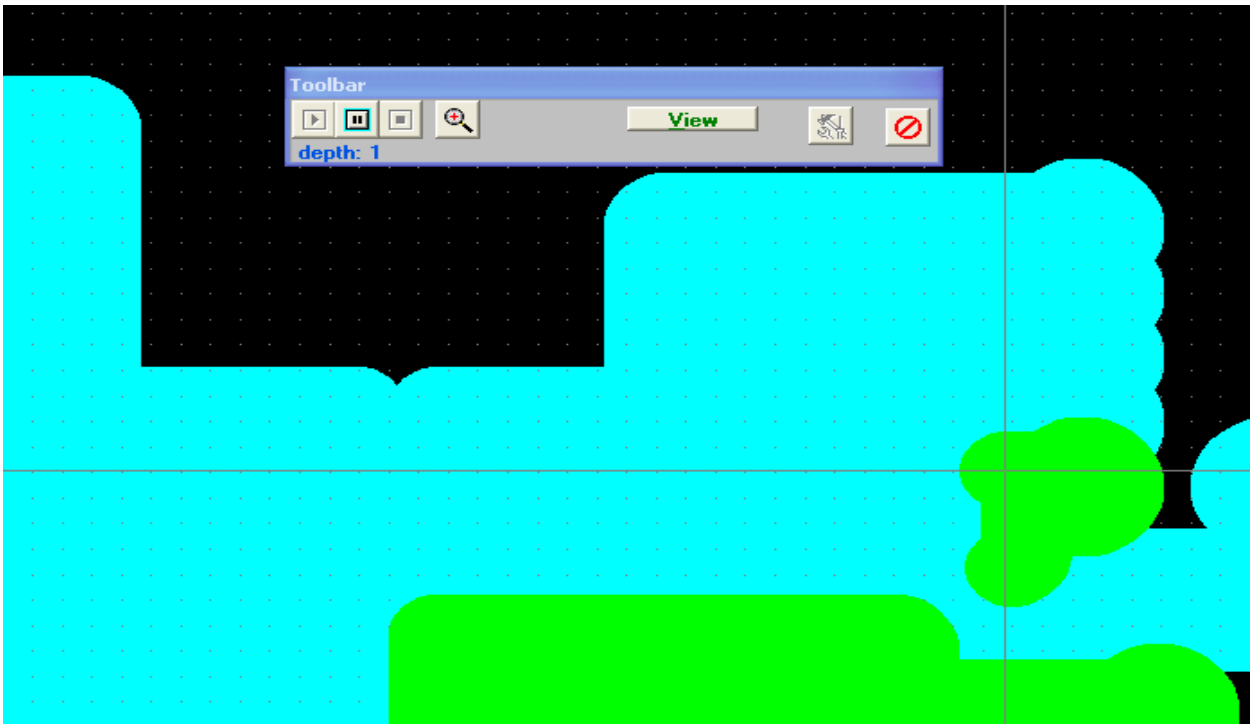


Make sure the appropriate tool is inserted in the collet and the collet has been properly tightened. Press **Tool is Inserted** and then **Start Spindle**

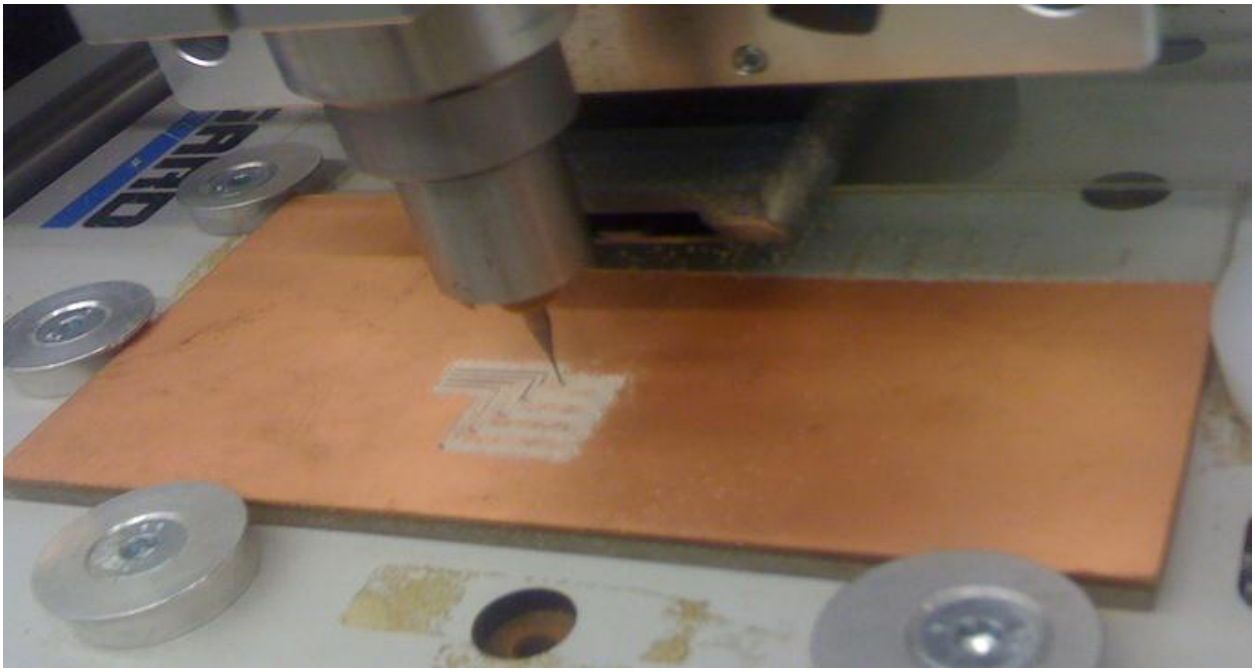
Precaution: **Wear safety glasses all the time.**



18. Now the board routing will start and you can see the circuit tracks being made by the machine, if anything is not going right, for example the routing depth is not right or machine offset is not properly set, you can press the **ESC** key to stop the machine.



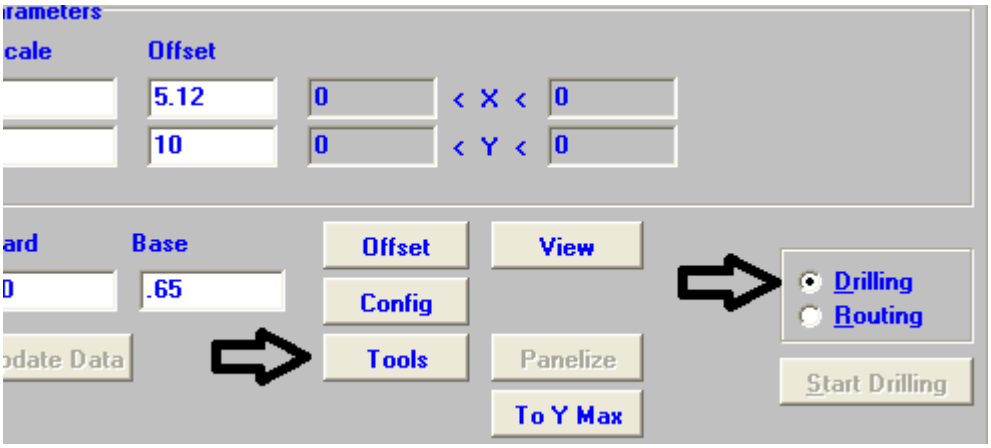
Software view while the machine is in operation, press ESC to stop at any moment


The machine is operating in routing mode.

Once the machine is finished with the routing, a message will appear on the screen as shown below and the machine will stop. Press OK.

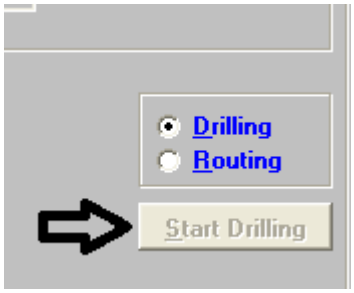
19. Now replace the routing bit with the 1 mm **Drill bit** as described in the previous section.

20. In the RoutPRO, now select the **Drilling** option from the radio buttons and press **Tools**

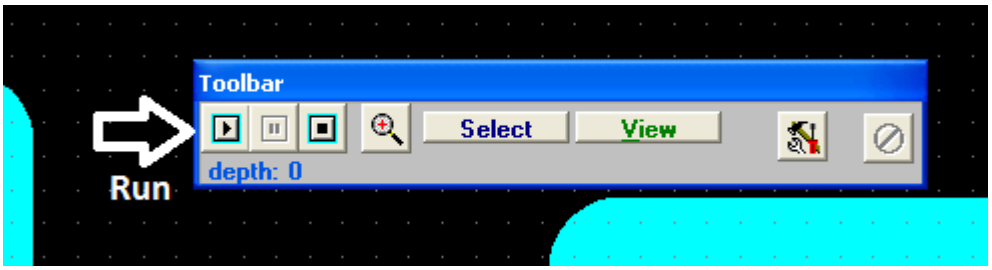


21. Now in the Drill tool options, enter the drill depth **2** under the **+mm** column for all the rows including T1, T2 ……. Tx, press **Update** and then **OK**

22. Now on the main window of RoutPro press **Start Drilling**

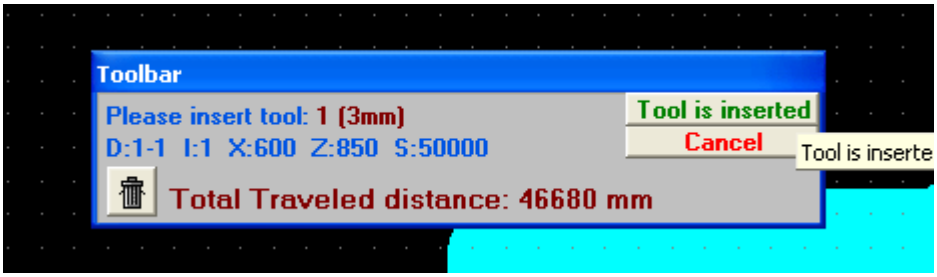


23. Then press the **Run button** followed by **Tool is Inserted** and **Start spindle**



24. Now the machine will drill holes through the circuit board, remember if the holes haven't been drilled thoroughly through the circuit board, then increase the drilling depth under the **+mm** column shown in the previous step.

25. Normally in PCB circuits, there are multiple drilling tools used to drill different size of holes and the machine will stop after drilling one size of holes and will ask you to insert another Drill tool of xx size as shown below in the figure. We suggest that you should continue with the same tool and drill all holes using the same **1 mm** drill bit. To do that just continue by pressing **Tool is inserted , Start Spindle** and just keep repeating the steps every time the machine stops and the software ask you to insert a new size drill bit.
Note: Drill bits are fragile and expensive kindly make proper use of them.



26. After drilling your PCB circuit will be ready.

Note: After you are done turn the machine off and put back everything at the proper location. Cover up the machine and turn off the computer and **don't forget to return the Dongle USB Key.**