

Micro system Fabrication with Advance Manufacturing Techniques
Prof. Dr. Shantanu Bhattacharya
Department of Mechanical Engineering
Indian Institute of Technology, Kanpur

Lab - 10
PCB Making

Hello, and welcome to this course on advanced machining techniques for micro system fabrication. Today we going to learn a very interesting topic of fabricating the printed circuit board p c b, which used for most of the electronic applications related to this micro system devices typically there are two different modality in which this a printed circuit board is fabricated. One of them is called the body mounted scheme of the printed circuit board, the other one is called surface mounted scheme of the printed circuit board printed circuit board is these days are very common place in all micro electronics.

(Refer Slide Time: 01:03)



And there is the huge potential in assembling the different mems devices are may be even all other microelectronic devices in manners. So, that there is the inter communication between the devices based on some connections extra, and particularly is more. So, important for mems, because the mems devices produce the signals which are the very low strength, and there is always requirement of signal processing ahhh from which you can get to know about what exactly are the different aspects are that signals is it actually a signal or noise what it is able to represent. So, typically the strategic that followed in most of the micro systems are hither acids which is called a application

specific integrated circuit it is terminology used for completely sort of buried or embedded electronic module, which will do the signal processing for the memes are the micro systems ahh which is fabricated on exits typically on the top of the acid the other simpler way of producing this devices are through the pcb outs where you can make, and built on circuit which you, otherwise we have do to on completely semiconductor level on much more simpler form on the printed circuit board.

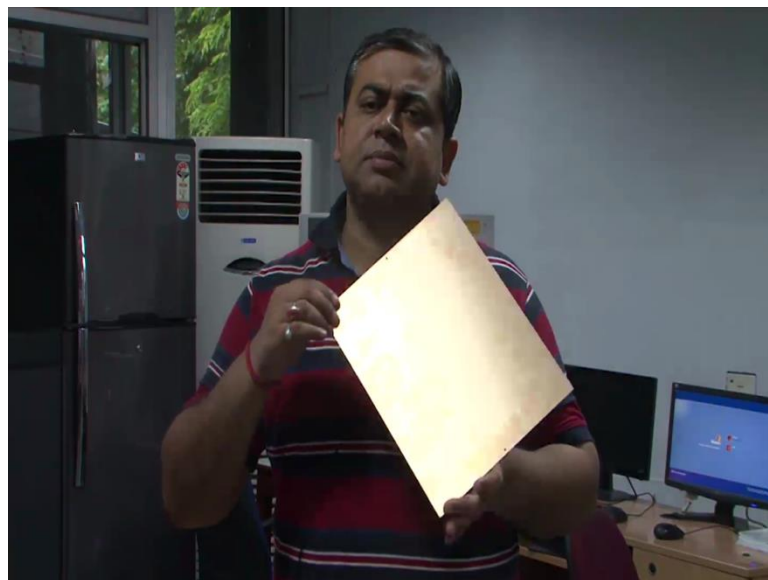
So, the printed circuit board illustration is you can represent know you you can see this is actually how the board would look like before it is started to fabricate this. In fact, the pcb where you can see the lot of the small dots here and. In fact, of holes which are drilled in the pcb which allows the components to be placed heirs crossed cross this board. So, that there can be soldering other can be joining the other side of the board which is the back side of the board. So, typically you make the pcb in a manner. So, that mount the components on this side on this particular side, and the mount the all the connection or the you know inter connection between the components you do on the other side.

So, this called the body mounted fabrication strategic for the pcb similarly you have another option where you can do the even surface fabrication of whatever components electronic components what to layout the inter connect, and that is called as surface mounted design of the pcb I will just show you the illustration of that it typically if you look at this is small tiny dynamic pcb here you can see that there are hardly expect of some portion where you need to probably mounts some kind of the more complex integrated circuit the other portion is not to drill .

And this kind of a system the interconnection all made on the same side as placing the component. So, we are mounting the component of the surface is also interconnecting of same the surface itself. So, different soldering strategy or different of the joining strategy. So, today we are going to give you on illustration how this whole fabrication process of generating microelectronics you know there can be signal conditioning, and processing can be done, and realized on a scale like this using the different you know machines we have at fore I lab in iit kanpur. So, the first thing that is the important for me to tell you the substrate, because that where will be the pcb build up actually happens the excise happens?

And as I already told you that there are two different strategy of mounting of the components one is a surface mounted components, and other this the body mounted components therefore, one needs to first of all see what are the kind of substrate we should able to provide the interconnect between this components if there are surface or on the body on the particular pcb. So, they are different methodologies for doing this ahhh pcb fabrication. So, as I told you there are several type of different type of pcb boards which are used as input material in this particular case I think I show need this double side pcb boards with double sided copper coating similarly you have single sided board which is actually one side.

(Refer Slide Time: 04:57)



You have copper the other side is the plastic as you can see here there haven boards which are related to flexible pcb this for example, flexible board, and you can see, and this is also for the single sided flexible board. So, have copper coating on one side, and the other side, the flexible plastic. So, in depending on the requirements of the different kind of pcb for example, for supposing you are make a body mounted component of the pcb are the solid mounted component of the pcp, we have used to different boards recording this this is the double sided board which would can be use (()) can use probably for case where can use both the smd, and vmd component on the same pcb board. So, one instant make the the interconnects through this copper coating by micro milling this coating on top as well as the bottom side of the board using the micro mill cutter I am going to demonstrate whole the simply line for the pcb process here of the

fore I lab where will see that the is the ahh very specific tool, which works on a (()) data file able to print or able to sort of machine micro machine the copper coating. So, that we can formulate in the interconnect. So, supposing you have to place the component on the top side of particular plate it had a dual smd, and dmd kind of the pcb system. So, some instants you would like to retain the connection of the top side in some other distance like to the retain the connection of the bottom side on connectivity on this particular case is made through micro drilling of sudden holes we should able to take the transport the connectors of the different across the pcb board from one side to the other. So, if supposing the connection on the back side for the vmd case the connector. So, typically go in such drilled holes in the pcb, and the connector would be are the the copper strip connection should be done at the bottom surface.

So, that even interconnect, and similarly on some cases the solid mounted component, and surface mountain component was present you would do whatever connection on this particular side. So, this is the very flexible kind of system where both side the machining can be done, and both kind of components can mounted the other one is the single side where; obviously, assume a thing that it can be either surface mounted or a body mounted in case of surface mounted may be you will use front side, because it is copper, and the body mounted case will use this front side which highly insulating, and connector would be done on the bottom side based on that. So, there are also even multilayer pcb this, because naturally the requirement of micro electronics imposes a lot of yield, and utility issues about the various components of the circuit. So, there can be the possibility that instead of single layer pcb you have many layers as many as about twenty eight to thirty two layer of pcb where each layer is having a circuit interconnect, and the idea is that components are mounted on the only one side the top side of this kind laminated layer of pcb boards where each layer which would, then have circuit which would join the component in a specific manner. So, in the layer one may be there may be the different circuit plat the layer two may be the different circuit plat each layer of the interconnect is separated from the each other through a insulated material insulator material like this. So, supposing you have to draw some lines here for using the milling cutter which would be connector.

And then would to place another pcb board on top of this layer. So, typically you will cover this layer through this dielectric, and then you would like to put this on top of this

layer on this way you can have a press for me of this different layer where each layer would have some circuit, and this circuit can work either in series or in parallel series circuits are sometimes you know designed in the manner. So, that the voltages at one layer the output voltages, and input voltages of the various layer of the pcb boards. So, one layer probably the output voltage whatever is commensurate with that the input output characteristics commensurate with that on a second layer.

So, in that you can to probably a series connection supposing the difference in the voltages between the various layer, then you might to have to use a parallel connection system. So, I am going to help you to sort of demonstrate protocol where we can actually fabricate this design on the pcb which is very critical copper wires which is comes out of this material, and this is of course, the dry process doing the pcb pcb can be. In fact, also exclusive wet process there where can be a photo lithography steps there can be a masking step there also can be a radix reaction for etching steps.

So, copper can be pattern remove selectively in the particular case we not going to discuss that here more of more of a dry pcb fabrication run that we are going to illustrate today and. So, talking about various equipments which are in the laboratory for using this whole for trying to fabricate the pcb first important tool. Of course, is the computer which can used to draw the cadd file of the design that the circuit design which is there where using couple of software here one of them called the the eagle six point one point zero version where we can actually layout the component interconnect them we also have several other software like. For example, the the board master are sometime circuit came six point one version, where we can actually after the components are select together, and after joint together we can make sort of layout design on the pcb with the option of drilling the pcb either on the. So, either also drill plating on the pcb back side or make a un plated condition where there is no plating needed, and its for the smd case on front side. So, this software use to program the basic drawing the file of the design, and then its imported as the (()) data file in to this particular machine write over here.

(Refer Slide Time: 11:07)



And you can see this is the micro tool, where there are couple of controller which would be controlling very precise manner the action of a rotating tool the tool heal the tools particular case that had been utilized are represented are they they can be seen particularly here

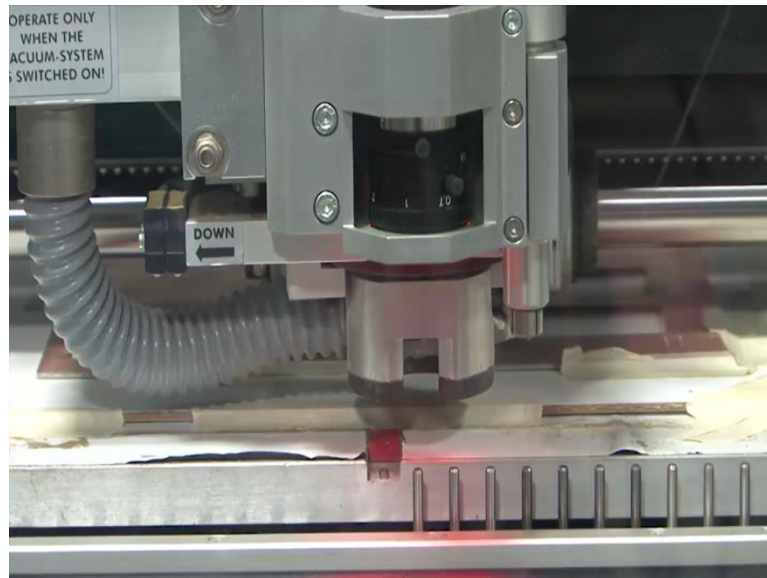
(Refer Slide Time: 11:20)



So, the tools size of about close to two hundred microns about point two mm minimum size all the way to about closed to two mm all sort of are the possibilities are here in this particular tools all you need to select the tip commensurate with your machining activity

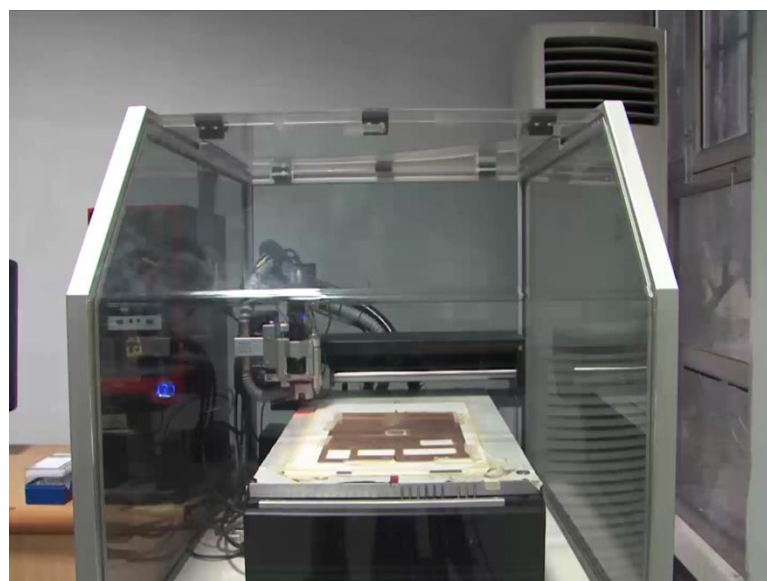
that you would like do an mounted on the machine, and then with the (()) file you basically operate the x y z stage on the machine in order to take care of the the machining of the copper in a selective manner. So, where ever the going to be an interconnect, there is no machining where ever you wanting to remove the interconnect.

(Refer Slide Time: 12:07)



And create a gap between several such lines, and several such interconnect to the copper in that particular area .

(Refer Slide Time: 16:46)



So, you can do later on the silver paste extra the particularly in the surface mounted phase and. So, what essentially this step is about once you have the kind of the ahhh drill sample with you with some kind of a copper tracks on the other side. So, one of the purposes why we need protection layer, and the top of the copper seeing that the copper layer in the last steps is that you want prevent the atmospheric corrosion to damage the copper interconnect copper which have we already created by the micro million as you learn you know the layer of copper which is involved in making this connects the interconnects quite thin.

And there also the possibility they are, because several wet steps between there is going to be a application of temperature particularly for soldering extra there always the tendency of the copper to go away, because of this corrosion action. So, at least one aspect that we need to take care of the pcb manufacture is somehow be able to protect, and re protect selectively. So, that the interconnect parts which really do not do any connectivity between the components as such, and the copper tracks, and they are protected.

And there are only the pad part where there is really the connection happening should be the re protected. So, one there is the small zone of the pad which actually slightly bigger in size well where you would actually eventually, where would you would well need to connect the connectors of the electronics that portion has to be exposed the remaining portion has to be covered. So, there are several steps which are followed in this process basically there is a the lacquer coating for the pcb manufacture.

(Refer Slide Time: 18:56)



There is a lacquer coating, and there is a particular product box which is given us can be came here called the quick start pro mask pro legend kind of you know different kind of you know you know the protocol where what you would do is actually you will start with this pcb board, and there always the design which is already their of the pcb board for remember the last step done this design.

So, we would add the outside printer mask, and this mask should be made in transparency printed by a local laser printer, and going to be selective deposition of printed, and un printed zones in this mask, and the idea this lacquer which we are going to use here is auto curable, and when there is exposure uv radiation of the frequency there is going to be change in the lacquer in some zones where there is no exposure as goes to exposure and. So, I think I earlier illustrated about negative, and positive photo resists, and let me just recall what I have said is that whenever in a negative tone results whenever exposure there is hardening of results on the positive zone research whenever there is exposure there is going to be a removal selective removal of the research from the particular portion.

So, in this case the research is where wherever there the more exposure is you can easily develop it of, and the exposure makes set have hard, and out or cross bran. So, it is some kind of a replica of the negative tone research is also known as the photo research are basically in some kinds of the lacquer by itself, and basically there are different

chemicals which come with the sketch some of the chemicals that we use are this for examples jelly like materials, and the coming two different parts which has used to premix, and they are what formulates the after hard baking lacquer fill on the top of the pcb material. So, you basically mix this two packets together, and then soft manner you apply the lacquer on the surface of the pcb board using this roller, and made up of the soft materials, and if you work to really to the hard coating there is the possibility whatever the drilled the holes are there the lacquer may go inside of holes that may create the problems.

So, you want to avoid that we the first place, and you want just use the soft polishing mechanism. So, that the lacquer can be applied over the whole pcb board several steps of processing ahhh where you can first probably hard bake, and the lacquer. So, it is done in a gravity conversation of oven of proto flow which actually heated to the temperature of eighty degrees Celsius, and the the material which coated is kept in size about the oven close to thirty minutes. So, that the initial solvent which is there in that paint would evaporate, and there is going to a hard, and frame on the surface left over, because of that.

And other aspect that this lacquer has is that it actually a incurable (()) lacquer is told you, and need a part of mask for the purpose of selective exposure, and supposing draw a pcb something like this where has be copper line as well as there the certain place of the copper pads there would be the inter connect you have to... Now what you have to expose the pads, and cover the lines for production sale for doing that you print mask like this if you look into very closely this particular elastration in this particular area.

You can find out lot of black, and white areas are regions the black regions are typically ones the black the uv from going all the way into the leaker, and transparent region are one which process the uv light. So, it is all photo lithography which has been on illustrated, and also video graft, and shown to you before, and basically the idea that whenever there is light transmission across this is would be a masking layer over the lacquer there. So, some zone of the lacquer not expose or some are exposed. So, which ever zones are not exposed in this particular case there would be a tendency thus zones the come off are get developed, and exposed the pad in this zones where as the area which are exposed to light would harden, and they remain as such. So, then the question of development comes.

And once the exposure has been done there is another kit which is there provided in this pro mask strategic where basically dissolve this is like a powder it is a powder material. So, dissolve it in a sudden amount of ml of water the, and immerse the exposed pcb on the solution. So, that portion which are now immersed exposed; that means, pcb on the solution that portion which are un exposed covered by black on a mask, and uv did not hid that portion. So, those the portions would be slowly dissolved into the solution thus exposing the bottom plate the copper plate in those the small zones now if you the power of cadd work at a much more magnified scale, and able to print very small region.

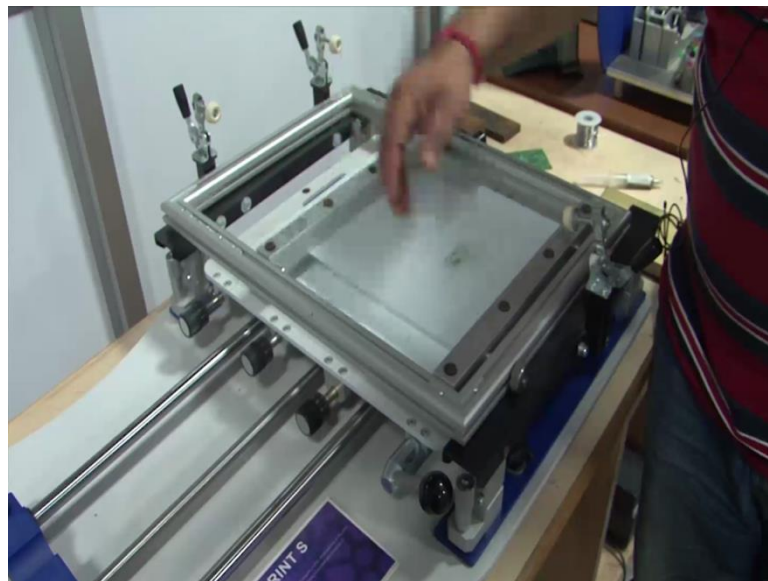
Which you want to re protect, then the idea is that only a small region of copper would be re protected which would be helping as we do not want to expose the whole copper, because of production, and, because the life of the pcb. So, once this developed is happens you now have small opened in that particular area where you can made the contact pads, and once this process has been done need the heat hearing of one more steps where put this particular expose, and developed pcb into the proto flow again for heat for a temperature of hundred, and sixty degree Celsius.

So, now for thirty minutes, and the intension for harden out the materials which is remaining. So, the portions which are not coming for the portion which are already there on the the uv exposed regions of the mask they need to cross bond well. So, that we can harden the temperature this may be the catalyzed there the same thing put happen put this in a room temperature condition for in two three days, but the process of hardening. So, slower in the particular case, but this case we want to be accelerate the process. So, we heated, and act the temperature act as the catalyst this more cross bonding action, and just about the thirty minutes whole lacquer which is expose to the uv now has completely hard, and structure, and those region which are now which had been developed of or now exposed, and this gives you the complete processing related to the pcb board cut magnitude.

So, once the hard baking steps has at one sixty degree Celsius as completed, then when need to clean the pcb substrate before the electronics soldering extra, and component placement we can start to take place is start. So, also before apply the solder paste typically through the another masking strategy that I will show little bit this. So, there is another pouch which pro mask kit is again which dissolved in cleansing action. So, we can either spray them the solution of the surface the completely harden pcb. So, for some

time you can remove of all small dust particle on other organic impurity surface during this whole process of lacquer application, and exposure, and development. So, that you have completely clean board again fit for (()) for next step of the component we can place soldering. So, the next step that is there in the pcb fabrication after particularly the cleansing went on the ready for the component mounting scale is the silver paste application it is as told you earlier. So, that the solid mounted, and surface mounted case of the components there is always a tendency to interconnect between the components using some kind of the silver paste on the same side of the mounting of the components. So, interconnect through copper the paste is typically applied for joining the component to the copper interconnect through the pad, and it is applied on the pad you already know that the pad sort of exposed to the pad to the opened requirement, and pcb board top to bottom here you can see the pcb board that we had talked about earlier.

(Refer Slide Time: 27:29)



Now, need to apply particularly those where you have the copper pads visible in this particular case in those areas you have to applied the silver paste. So, for that have to do another masking process are masking strategy where you have to take some kind of the self an paper or butter paper like this using the same drawing the file illustrated earlier.

You develop a mask like this you can see some of the purpose region which has been done by the laser machining the laser machining the illustrate one of the earlier modules, where we show that drawing data with drawing data you can actually print with laser,

and try to plate with laser. So, selective machining can be done. So, this actually selective machining case in this particular zone here as you can see of the specific kind of the paper use for the masking strategy, and it has been framed up nicely in this particular metal frame which is now used to mount on this machine.

So, this is again mounted on this reel which is meant for this paper, and you have on both sides sort of wheels which would be able to very easily move, and mount this frame. So, for selective coating of the silver paste over such a mount. So, we have shown how this mask is mounted the other aspect is to sort of align this pcb over this stands which are mounted through this (()) boards into the stage the stage is mobile in a way it goes all the way down here. So, that it can go in a line with the mask itself. So, as you can see here the mask has now been aligned from very well pcb which is actually at the bottom side, and through the x y napes extra which actually here in this particular zone you can shift over this plate here in manner.

So, you can do proper alignment between the mask at this is centre the mask, and the pcb also what is important for me to tell you that in this particular pcb you can see now only the copper pads, which had been fabricated done exposed in last step there coming out through this mask, and the drawing file shade by the machine which would copper pad milling, and a mask making laser by other the same file. So, therefore, overlap them the very well, and the copper pads starts come out of this butter paper the mask the paper, and they get exposed selectively the outside in one now there is a scope for introducing the silver paste which actually comes in a pre mixture form, and the comes the (()) needle like this. So, basically the idea is to be able to apply this paste on the hole of this region, and then try to cover the surface properly with this ahh small flatter, and you can actually this can be something like you know spread maker. So, whatever silver paste has been dropped you can spread, and hole surface, and cover the whole area ahhh enabling the copper pads to be just coved with silver paste in the remaining area which is coved by butters paper to be not having in a silver paste.

So, now selectively coating the silver paste on the top of the pcb, and one of the reason why you do that the silver paste used in a next stage of the component getting connect electrically to the copper circuit. So, silver paste is sort of conducting between the component terminals, and the copper interconnect which is there on the pcb by means of the pads. So, you already saw how to do the surface mounted component parts with

silver paste on same side of the interconnect, and sharing the same side the component copper the component of the surface the other body mounted components can be actually done using not using the silver paste, and the masking methodology solder wires, and there the machine have subsequently we can actually solder tips of electric of electronic components on the other side on the micro milled pcb board. So, that connection can be the bottom side component can be mounted on the top side.

(Refer Slide Time: 31:50)



So, now next step basically the small (()) component that would on the top of the pcb board, and for doing this we use this machine called proto place which is right round this place the idea behind this machine is there is the small section created at a tip of this particular you know you can see coming down all the way. So, this tip is actually se for going up all the way to a component, and applying the certain pressure. So, that you can pick up this particular component. So, now, we can see that this component has been sod of you know dust by the vacuum tape, and the with the pressure on condition the would be the possibility of component to get lifted as you can see now the component has been held by the section, and this particular tips, and you can bring the tip all the way to the pcb board. So, the component broad all the way of pcb board.

So, that you can actually place it on the top of the particular area that you need this component will placed. So, typically you can rotate the component as per the as per the orientation of the pcb itself, and then you can actually go ahead, and place this

component on the top of the portion where would like this component typically set. So, the idea behind this whole exercise the precise nature of the pick, and place of the individual components from the stage which is mounting the components through vacuum section, and placing it over the small region of pcb very would really need the component, which sold already as you know the copper pads or pre exposed to the solder paste, and then the picking, and placing of the component itself should ensure the components in connected in the particular case this is the case of the smd are the surface mounted components a different protocol a slightly different protocol would be the body mounted where next step are going to the some of the future photo placed machine is that you know you look at the different aspects of this machine you have this display here the lcd display here his talk about various commends ok.

In fact, you know there are there are commends like manual we can place turn table extra the typically turn table in this particular zone you can see the various chamber of the are filled with the microelectronic components you can actually easily go all the way to the turn table. So, we go to the menu, and we go to the turn table, and we can either go clock wise direction or counter clock wise there is the set of toggle keys which are there are on this side of the proto flow machines right over here, and with this toggle key we can select the various comment options in the turn table in the in the in the display in the lcd display like turn table counter clock wise clock wise.

So, here for example, we want to select the clock wise option the clock wise option by moving the toggle on the left, and you can see that there is going to be a movement of the stage in a manner. So, that by the repeated motion of counter clock wise, and clock wise you can place particular stage containing the electronic component just underneath the picking of bed which is actually the perforation right here (()) sheet. So, once this is done the other option the other option use to x y stage position this all the way and. In fact, in video you can see here position this all the way you go to the selective location where this virtualizes component, and just before you can probably pick this component using the vacuum, and then drag it all the way to the pcb as has been illustrated just few moments earlier.

So, for picking of components of various sizes you can see there are different sizes nozzle in which are around the particular place, and you can vary the nozzle tip diameter in a manner. So, that the big bigger component thus, because there would be a larger area

to graph the vacuum which surface of the component the surface of component need larger area for creating in the force good enough for dragging a component all the way stage on the pcb the pcb is you can see kept right on here in summery the nozzle this right around the corner, and the nozzle goes to top particular zone here.

Which can be found out in this camera the web camera here, and you can actually able to position in nozzle (()) as you know what is also very important me to tell you there is another line which there in this particular zone where we we are talking about you know there is one nozzle here which lifting the component now the purpose of the other nozzle which you can see here actually put back into the (()) at any point of time is to draw up the excess silver paste which has been applied in the last masking by the chance there may have be spread over by incorrect masking strategy.

So, from a area which otherwise not needed to have the silver paste if you want to draw use as well as draw same vacuum of the system.

So, all you need to sod of the game once you have put this nozzle in this place you can actually illustrate which actually take the particular stage all the way the pcb as can be seen here. So, that know you can now actually dropped the excess paste from the region that you would like you know something like that by going to different places, and drawing of the excess silver paste only reason why the excess silver paste needed to be removed is that sometime connectivity problem, because of this fill over, and the area which not needed to coated to silver paste gets coated with silver paste as the result of which always try do this alternate action the drawing of paste from some zone, and try to pick, and pick the components of other zones for the proper electrical connectivity between pcb on the components. So, now, we have a pcb which is mounted with some component some this component are now setting on the silver paste we need to dry of the silver paste. So, that there is a proper region of the electronic component with the pcb board.

So, as I told you the warming tool is reflow oven which actually gravity kind of the conversion kind of the system which is called the proto flow, and there is a controller which is there on the oven which ensures the door can opened, and closed in automatic manner there is of course, stage of setting of pcb which now has the pasted component from the earlier step, and we need to. So, reflow. So, there are three steps in this

particular process one is actually warning of the oven where with temperature has to go all the way to about hundred, and ninety degree Celsius before you know the the sample can be put in that is typically curing temperature the silver paste which is applied here the other aspect is the should only sort of go to that also reflow into the various gap which are created, and ahhh even priority that important.

The whatever oxygen whatever air bubbles on the silver paste which comes doing the time of application of pasted have been removed therefore, this first step of the program as I told you first step of the program is to sort of heated hundred, and forty second hundred ninety degree Celsius. So, that gets the rate of the air bubbles in which are closed within the paste the second step is to sort of able to reflowed reflow condition happens at two hundred seventy degree.

So, difficulty solder of melting of, and then goes around those places where the leads of components the input output of the component are being placed some kind of the fusion between the reflowed those particular places are gap, and once that is done actually automatically cool the system down the oven automatically for doing after doing all the processing. So, that whatever component have been laid out silver paste in the previous step has been actually now completely hard, and solidified electrically connected to the copper connection which are there interconnected interconnects, which are there charges.

So, let us look at the oven how it is done controller right here has, and option of a profile selection. So, need to sod of inter the profile selection, and then go to the open close option for the oven, and then we you can see the stage of the oven comes out automatically, because of the controller the controller fully programmed of course, then going to do this to take on the pcb component which had been earlier made by the proto flow system with the picking, and placing operation, and place this pcb components all the way to the oven right about here, and after doing this basically close the oven, and start the program. So, that now whole curing action here can happen of the silver paste in the electrical connection between the component on the pcb can be established in this manner after the certain amount time is spend.