VII. PCB Design for testing assembly technologies – extended syllabus

Goal of the practice: To design a Printed Circuit Board (PCB) for the investigation of technology limitations of the production. During the practice the students will work in Mentor Graphics PADS CAD system.

Exercise: With the leading of the laboratory teacher, the students must design a PCB with test patterns. The drawing and the apertures are designed in the CAD software. After the physical realization of the PCB it could be used to test the limitations and possibilities of the PCB- and the assembly technologies.

The skills acquired during the laboratory practice: The students will get acquainted with the errors and limitations of assembly technologies in practice. The students will learn the basics of the computer aided design software.

Short definitions required for the laboratory practice:

Trace width and clearance
The minimal copper width and insulator distance on a PCB. With the miniaturization of the components, the pitch of the pins (the distance between the pins) are reduced, so the minimal width of the copper and the minimal distance between the copper lines must be reduced as well. The manufacturing technology limits these minimal values.

Fiducial mark
Marks on a PCB, helping the positioning during stencil printing and component placement. One fiducial mark defines the 0,0 reference point of the design's coordinate system. Two fiducial marks (at the cross opposite corners of the board) defines the positioning (also the angular) error of the board. Three fiducial marks define the distortion of the production films and the PCB.

Schematic diagram
Circuit diagram, containing part symbols and connections. The connection wires define the connection between the pins of a component.

Layout
Physical form of the schematic diagram, representing the actual PCB design. It contains the geometry, the component positioning and the wires of the design. The schematic and layout design process is always connected, even in the level of the CAD software.

Layer
The layout designer programs handle the geometries, shapes, conductor patterns in layer-system. There are physical layers (which will be realized during PCB production) and there are documentation layers, helping the documentation of the design. The layers are present during the work in the layout designer software, however during work the visualization of each layer can be turned off. The different CAD software handle these layers with different names, but the names are always consistent with the function of the layer.
Most important layers are: top/bottom routing layers for copper on the top and the bottom of the PCB, solder mask layers, solder paste layers for stencil apertures… etc.

Stencil
A thin foil made of stainless steel, fixed on a frame. The foil contains apertures, according to the pins of the components. The solder paste is deposited via these apertures during stencil printing. The usual thickness is 75-200 μm. It can be produced with laser cutting (common), chemical etching (outdated) or galvanoplastics (expensive).

Tombstone effect
Soldering failure in the case of discrete SMD components with two pins. If there is a difference of the reflowed solder wetting at the two pins, the resultant vector of the surface tension forces pull up the component, like a tombstone. The root cause of the failure may be non equal paste quantity at the two pads or the non even heating of the PCB.

Introduction

Cost effective approach and miniaturization has an important role in electronics industry. With smaller components, fine pitch PCB traces and precise technology, the same task can be achieved with a smaller PCB. That's why it is important to know the limitations of the technology and to set the parameters of manufacturing to the known limitations. With small components and fine pitch traces, it is important to know the minimum distance between the components (e.g. short circuit investigations). With small distances it is possible to find short circuits or solder bridges. With higher distances the components may cover too much area on the PCB.

1.   Short Circuit Inspection with Small Size Discrete Surface Mounted Resistors

For short circuit inspection, resistors are soldered in pairs. The distance between the pairs gets smaller by each step. It is important to note the longest distance, when the solder bridges (short circuits) appeared first. (Fig. 1.) We can use 0603 size (1,5 x 0,75 mm) discrete surface mouted resistors. The distance between the resistors may decrease from 30 mil (762 μm) to 4 mil (102 μm).

Table 1. – Measurement units

Unit conversion (*mil – mm*)	
1 mil	0,0254 mm
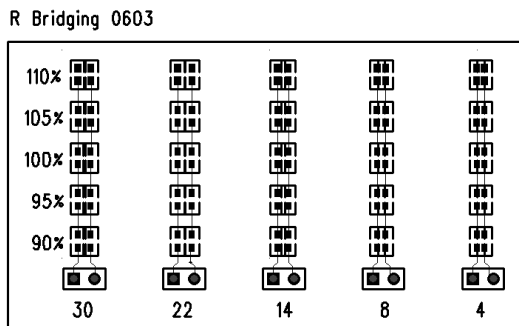
Fig. 1: Pattern for short circuit inspection with 0603 size resistors
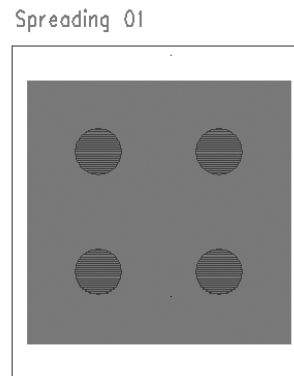
Fig. 2: Pattern for spreading test

The aperture size of the stencil is set to different values for each row of the component matrix. The aperture is 10% bigger in the first row, in the fifth (and last row) the aperture is 10% smaller. The step is 5% between the rows. The test method is also usable for mid-chip balling, while the stencil geometry has square shape. The investigation of mid-chip balling is not the task of this laboratory practice.

Note: PCB designer CAD programs handle the different geometries and apertures in different layers.

2.   Spreading Test

During spreading test, a wettable surface is used for solder printing. The printed solder patterns has circular shapes with pre-set diameter. After the heat transfer (reflow) the spreading diameter is

measured. Wettability is better if the diameter is bigger after reflow. The pattern in Fig. 2 is used for the investigation, where the printed solder paste has a diameter of 5,08 mm (200 mil). The contact surface has square shape with 27,94 mm (1100 mil) side length. The darker areas highlight the geometry of the printed solder paste (the apertures of the stencil).

3.   Surface Insulation Resistance (SIR) Test

For measurements with high impedance measuring devices, the copper traces are formed in a comb shape. (Fig. 3) High voltage is applied to the pattern (100 V–1 kV). With TeraOhm measuring device it is possible to measure the surface insulator resistance. After soldering, the flux residues may reduce this value. If this effect is smaller, the qualification of the paste is better.

The figure shows copper tracks with 406 μm (*16 mil*), with a distance of 508 μm (*20 mil*) between them.

4.   Bridging test

During bridging test copper traces are positioned under each other. Each trace has same width. Paste is deposited perpendicularly to the traces in pairs, with decreasing distance between them. (Fig. 2) The figure shows the traces horizontally, the vertical shapes are the deposited paste tracks (stencil apertures).

After soldering it is important to note the longest distance where a solder bridge appeared. Wetting of the solder is better if the solder bridges are forming in longer distances.

The copper traces has 0,762 mm (*30 mil*) widht, their length is 43,18 mm (*1700 mil*). The tracks of the apertures has width of 1,02 mm (*40 mil*), the distance is decreasing from 813 μm to 102 μm, with steps of 101 μm (*32 mil* to *4 mil*, with *4 mil* steps).
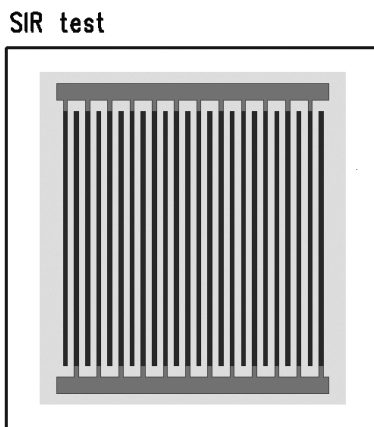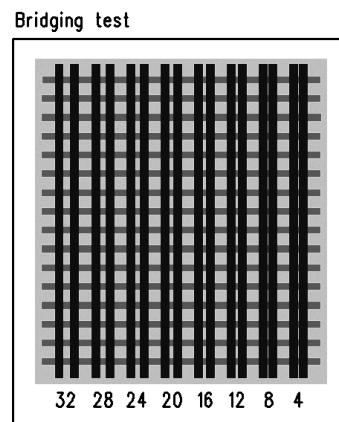

Fig. 3: Pattern for SIR tests

Fig. 4: Pattern for solder bridging tests

The flow of the work

General note: it is suggested to read through each part of the exercise before starting work. This way it is possible to get a thorough point of view on the current problem.

Create a directory, which will be used as a working directory at the following place with own Neptun code. (If D:\ is not available, please refer to the C:\ drive)

D:\User\ET_VII\NeptunCode

Note: Do not use any special symbols or characters with accents in file names, directory names, or any CAD database/library names.

Open PADS Logic and PADS Layout from the desktop or Start menu. Logic is capable of designing schematic diagrams, while Layout is capable of designing layout diagrams. The connection between the two software is handled via a link (.eco file), and a common database directory (libraries).

Let's search for the Library Manager in the menu of PADS Logic (*File/Library…*). Let's create a directory with the button <u>Create New Lib...</u> (*Library*). The component library is not a part of the windows directory-system, but a part of the PADS database. Save the new library in our directory and choose a name for the library according to our Neptun code.

D:\User\ET_VII\NeptunCode\Lib\NeptunCode.pt9

If all goes right, our library can be found in the drop-down menu of the Library Manager.

Note: The Manage Lib. List has an option to set the order and the precedence of the libraries. It is possible to delete libraries as well.

It is important to set the new library in the PADS Layout as well, so it is important to add the new library in PADS Layout with the same manner as shown above. Find the Library Manager in Layout, use <u>Manage Lib. List.</u> , then <u>Add…,</u> , then search the library, click OK, and check the presence our library in the drop-down menu.

During the practice, we will use mil units, so set this parameter PADS Layout software.

In PADS Layout:

<u>Tools\Options\Global-General\Design Unit:Mils</u>

I. Short Circuit Inspection with Small Size Discrete Surface Mounted Resistors

Let's make sure if Pads Layout window is active. Open Library Manager (if it was closed before), then use the search function. Make all libraries active in the drop down menu (<u>All Libraries…</u>), make the <u>Parts</u> ( ) button active at the *Filter*. Search for *0603* string to find 0603 resistors (<u>Apply</u>). * characters are used as a general searching character in the search syntax. Finally choose "misc:RES0603" with double click. Due to the double click selection, <u>Copy</u> becomes active on the right. Copy the selected component to our library, so the default component in PADS will not be modified only a backup copy of it. It is possible to go back to the default if something goes wrong. If the copying was a success, then we are able to find our component in our library.

If the Library Manager window is active, the component in the library (make sure our custom working library is active) can be edited in the following way: double click on the name of the

component, or <u>Edit</u> button on the right. In the popup window (*Part information for Part 0603*) the custom settings must be neglected right now, and we save the file with <u>Save as</u>, with the following name:

RES0603_100

This way the original size label is given to the part (100%), the other sizes will be modified according to this original geometry.

During editing it is important to open the original footprint (physical drawing of the component pins), which contains the basic information for the drawing. Let's make <u>Decals</u> active in the Library Manger's Filter (<u>Decals</u>). Then let's search *0603* string in all directories, then choose "common:0603". Activate it with double click, then click on Edit. Close the *Part Type List* with <u>Close</u>, then put away the Library Manger window. Note that the main window of the software has changed to a new view, where two pads and some other graphical elements are shown, according to the 0603 footprint.

Make sure, that with a normal mouse click we select the required element. With a right click on the blank working space of the software, we can choose what we want to select with the cursor. Let's select <u>Select Terminals</u> from the list. The datasheet of the pad becomes available with a double left click on the pad. The <u>Pad Stack</u> icon will lead to the editing window.

The *Pad Stack Properties for Pin* window helps us to edit the shapes of one pad. We do not need the complex settings right now, so let's focus on the main task: we should modify the stencil aperture layer parameters for the pads. This layer contains the geometry data for the stencil apaertures. The conductive copper layer does not need any modification.

Note: The stencil-aperture layer data is basically the same as the data in the copper layer. The data for the stencil can be modified in "PASTE MASK TOP/BOTTOM" layers, while the data for the conductive layers may be modified in the"TOP/BOTTOM" layers. TOP/BOTTOM always note the top and bottom size of a PCB.

We should modify data gathered from <u>Mounted Side</u> (TOP) in the window of *Pad Stack Properties for Pin*, with values according to calculated percentages, however now a new layer (PASTE MASK TOP) layer must be set.

For this, we should do the following: "Add", Paste Mask Top. A new layer appeared in *Sh. Sz. Layer* box. This new layer must be set according to Fig. 5, and the parameters shown there. "Apply", then "Ok" and again "Ok". This parameter setting must be done on the other pad as well. Let's compare values of Table 2. with the values of Fig. 5. It is now clear that we done a 5% aperture reduction, compared to the original sizes.

If both pads are completed, then *File/Save Decal as…/* , where we should set our own (!) library, then give the following name to the new Decal:

0603_95

The software will ask "Would you like to create New Part Type?", and the reply must be "No". (Thus far, only decals are created, no new part types were defined). Before creating actual new part types, it is important to create all Decal shapes in the way described above. Let's create the other Decals according to the parameters of Table 2. (0603_90, 0603_105, 0603_110)

Hint: The three remaining decals can be completed from the fresh Decal, only by re-setting the parameters, then using Save as… function. It is important to save our work in our own library!
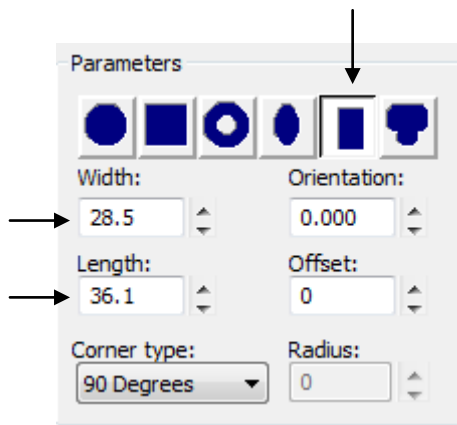
Table 2: Stencil aperture parameter setting

Size, %	Width, mil	Length, mil
90%	27	34.2
95%	28.5	36.1
100%	***30***	***38***
105%	31.5	39.9
110%	33	41.8

Fig. 5: Paste Mask Top parameter setting

We will prepare new components for the new footprints in the next step. With the help of *File/Exit Decal Editor* exit the decal editor tool. Get back to the Library Editor, then highlight our own library from the drop-down menu, then we filter for parts in the proper part of the window (see above).

Let's use the Copy function to copy four instances of the ***RES0603_100*** component in our own library, with the following names: RES0603_90, RES0603_95, RES0603_105, RES0603_110. These components will be shown in the library list, if we refresh the view. (eg. let's click on the Decal filter, the again on the Part filter). The new components should be reconfigured by us in the following way:

Edit RES0603_90 (Edit) then click on the *PCB Decals* tab. Unassign the "0603" original decal, then assign 0603_90. Click on Ok, then do the same routine on the remaining three components. (95,105,110)

The components are ready for short circuit inspection, so these will be used in our schematic.

The following part of the lab practice will use geometry elements, not library elements, so we should continue the work with the circuit design of the resistors.

II. Schematic and layout editing

The PADS Logic basic view consists of a drawing sheet, where the circuit must be designed. Let's make sure, that we can see the editing toolbars below the File, Edit, View…etc. menupoints. This can be defined in *View/Toolbars* menupoint. We should set <u>Standard Toolbar</u> and <u>Schematic Editing Toolbar</u> active.

The blank page should be saved in the working directory according to our Neptun code:

D:\User\ET_VII\NeptunCode\NeptunCode_schematic.sch

Let's do the same in the PADS Layout software, save the blank page.

D:\User\ET_VII\NeptunCode\NeptunCode_layout.pcb

If both files are saved, return to the Logic software. We can add parts to our design with the Add Part icon (). Add 10-10 resistors from every new component, that we have just created. (In Add Part window highlight our own library, highlight the component, Add, then click on the blank page one by one.)

Hint: Save the design continuously, to avoid losing data.

It is a logical step if we would add the resistors row by row, starting from 110% resistors. We should sort the resistors in pairs, according to Figure 6.

Hint: When we select the proper resistor in the Add Part window, lower the window, then rotate the component with CTRL+R combination to +90°. Then all added components will be in proper orientation.
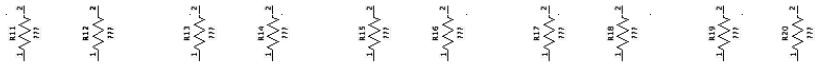
110% - first row
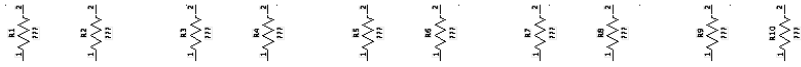
Then the next row should be set, with the 105% components. Similarly to the row above, let's sort the components in pairs. Now we should see something like below:

110% - first row
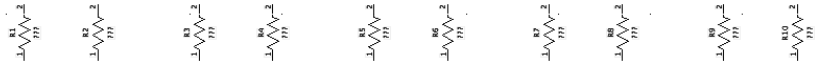
105% - second row

Then we should add the following components continuously: 100% (3rd row), 95% (4th row) and 90% (5th row).

Hint: Pay attention to save your work in once a while!

Test pads are needed to close the connections. We should head back to Add Part function, then search in All Libraries (highlight All Libraries), then search for the following string: *CON-SIP-2P*. Let's put two symbols of CON-SIP-2P below each double-column of resistors. The two symbols are physically the two pins (J1-1, J1-2) of one component (J1). Do this according to Fig. 6.

The added parts should be connected with each other. The connection (wiring) should be activated with F2 button, the deactivation of this mode is done with ESC. The wiring should be done by clicking on the pins of the components. The wiring is not done by drag and drop drawing with the cursor, but each click on the page will add a corner to the wire. Pay attention, not to pull a wire trough a symbol of a component. We can check the connections by moving the component around. If the wire is attached to the pin, it will follow the component. If the connection shows a small rectangle at the pin, the connection was not completed. In this case move the component to a slightly different position, then finish the wiring. When we have finished the connection of the components, we should <u>Create Text</u> ( ) and write the values on the left side of the circuit. Let's set 20 points for the font size (Size pts.), to have a better visibility.

Hints: To zoom, let's use CTRL+mouse wheel, or CTRL+W combination.
If we want to manipulate the positioned components, we should use the selection (see above) – right click on the blank part of the page, then <u>Select Part</u>.
It is recommended to set <u>Select Connections</u> if we want to manipulate or delete any already complete connection.
To highlight or select a text use <u>Select Documentation</u>. If we're ready it is recommended to set back the cursor to <u>Select Anything</u>.
If we have a spare component, wire piece, text, deletion can be achieved by highlighting the part, then pushing the DEL key.

Note: This part of the work will let you know the general basics of the schematic editing and library handling in a PCB CAD software.
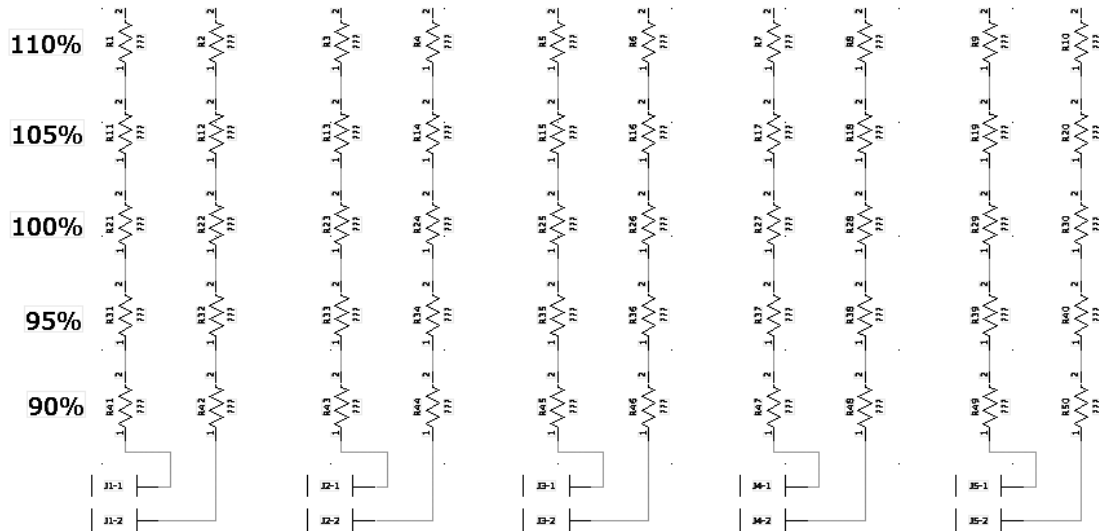
Fig. 6: The ready schematic with rotated resistors, texts and two symbols per each two-pin testpads

In the next step we should move the design to the blank page of the Layout software. Let's do the following in the PADS Logic: activate *Tools/PADS Layout*. The Document tab will show if our file is active or not (*D:\User\ET_VII\NeptunCode\Neptuncode_layout.pcb)*. If it is not active, open the previously saved .pcb file from here. Then click on the Design tab, click Send Net list, then ECO to PCB. (The pop-up warning and log files should be neglected for now.)

The components will appear on the blank page of layout software, but positioned on the Origin point of the design, on each other. Let's disperse the components with *Tools/Disperse Components*. The connection between the pads are not actual wires, only "rubber bands" representing the logical connection which was defined in the schematic.

Hints: The positioning must be done on a working grid. A proper grid is the basic of a precise design. The distance between the grid points of the design can be modified with the "Gxx" modeless command string, which must be written on the working page, then we should set the value with Enter. XX means the actual numerical value. For 20 mil grid point distances, "G20" string must be set. The visualization grid (white points in the background) can be set with the"GDxx" string, meaning Grid Display.

The zoom must be done with CTRL+mouse wheel combination or CTRL+W combination. Zomming to a full view can be achieved with CTRL+B.

If we want to manipulate the components with the cursor, we should choose selection (as above – right click on a blank part), then "Select Part". If the component is not highlighted by the cursor, we can't manipulate it. Selecting a piece of a component footprint (eg. only one pad is highlighted) does not mean that the whole component is selected and we will be unable to manipulate the whole component. In order to do this we must highlight/select the whole component.

If we route the wires, it is recommended to set "Select Traces/Pins". If we manipulate a text, it is recommended to set "Select Documentation". If we manipulate the board outline, the "Select Board Outline" must be set.

Set the grid to 40 mil. This should be achieved by pressing the "G40" string on the work page, then pressing Enter. The components must be positioned in the same manner as we did in the case of the schematic design. The dispersed components can be positioned according to their numbers, which can be traced on the schematic design as well. Figure 7 will also help the positioning. The selected components can be rotated with CTRL+R, the rubber bands (white lines) should connect the neighbouring pads. It is recommended to set the components to the same distance in the vertical dimension as well.

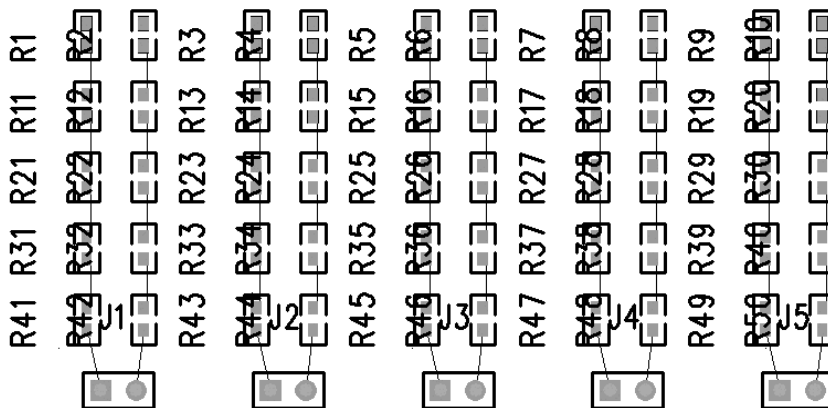It is not hard to position the components on a 40 mil grid, so please take care to present a clean design.

Fig. 7: Positioning after dispersion

There is no need for the additional component names, so it is suggested to delete them. The names can be selected with "Select Documentation", deleting them works with Del key.

The next step is fine positioning, where the components must be set to a given distance from each other. For this we should set the grid to 2. This should be achieved by pressing the "G2" string on the work page, then pressing Enter. Figure 8. may help completing this task.

For the fine positioning we should move the top right component of a double column, then move it to the edge of the left component. The edges of the pad shapes should just contact each other, then with the step of 2 mils, the distance must be set between them, by moving the right component. The fine positioning can be done with the cursor keys on the keyboard.

In practice: Pick up the first double column's upper right component. The edges of the pads should be set next to each other, just make sure they do not overlap. With a grid step of 2, it takes 15 moves with the cursor keys to move 30 mil. Do this with the component by pressing the right cursor 15 times. Make sure, that we do not interfere with the mouse. The next component underneath will only need a vertical alignment to the upper component - while doing so, the connecting white rubber band will set to a straight line. For 22 mils we need eleven steps and so on.

When we are ready with fine positioning, it is recommended to reposition the test connectors on a 20 Grid.

After positioning, we should write the text notes on the design. A click on the Drafting Toolbar (🖋) will reveal some new tools on the toolbar. We can put Text on the design with (ab|) tool. Let's write "*R Bridging 0603*" and do not forget to highlight the percentage values as well (Fig. 8).

Finally draw a line around the work area with 2D Line (⬆️) tool. Each corner takes one click. Finishing a line takes a double click.

Important: The 2D outline can be sized according to the coordinates, which are shown on the bottom line of the program, above the bottom Taskbar of the Windows OS. The size can be set via a parameter window as well, which can be activated by double clicking on the line.

The next figure should be achieved:

R Bridging 0603
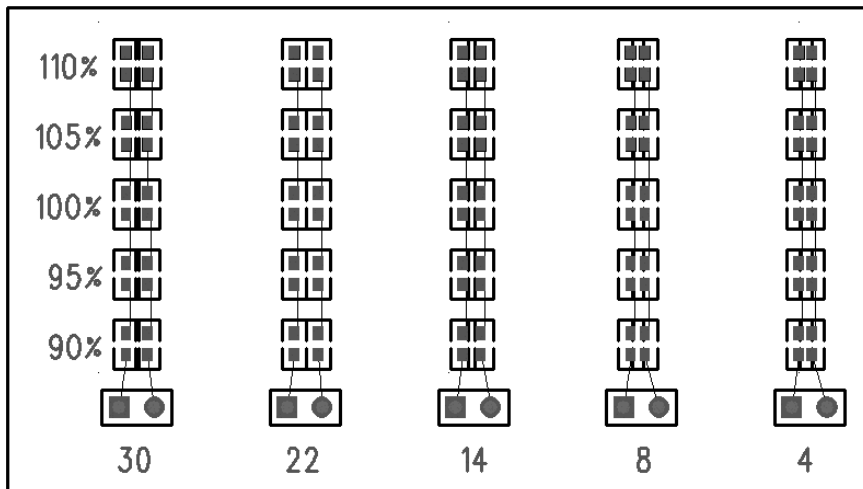
Fig. 8: State after fine positioning

The final step is completing the wiring, which must be done in the selection mode of "Select Traces/Pins". Thus the wiring can be started by double clicking on a pad of a component. A corner can be added to a wire with a click, the final connection can be achieved by clicking on the pin of the other component. The wire must be drawn according to the white (rubber band) lines. It is recommended to use 20 Grid, and to avoid non-obtuse angles such as a perpendicular corner.

The wire thickness should be 6 mil. This can be set in:

Setup/Design Rules/Default/Clearance/Trace Width. Each values must be set to 6 (<u>Minimum</u>, <u>Recommended</u>, <u>Maximum</u>), then Ok, Close, Close.

If the default setting does not allow to visualize a wire with 6 mil thickness (a thin line is visualized only with the corresponding color) then it is suggested to set the following: *Tools/Options/Global/Drawing*, where "<u>Minimum Display Width</u>" should be 6. If we zoom in, we could see now the actual thickness of the wire.

III. Spreading test

The spreading test pattern is composed of a full copper area. Above it an aperture is opened on the solder mask layer, then four apertures are designed on the solder paste layer (the actual apertures for the stencil printing). (Fig. 9.)

To draw a copper area, we should choose the <u>Copper</u> ( ) tool on the toolbar. To draw a copper area we should define its edges first. We should draw a rectangle on a 20 Grid, with 1100 mil edge length. The dimensions are still visualized on the bottom line of the screen. After the first click we are able to trace the length of the given line, what we are drawing with the cursor.

If the rectangle is completed, *Add Drafting* window pops up. Make sure, that the type is <u>Copper</u>, let's click on the <u>Solid Copper</u> checkbox (make sure it is marked with a tick), then assign the layer to the TOP *Layer.* Apply, then OK.

Back on the work window, highlight the shape (right click on a blank part of the page, let's make the appropriate selection), then CTRL+C, CTRL+V. We will receive a copy of the original shape. Let's put it on the other shape, and when the *Add Drafting* window pops up, we should set "Solder Mask Top". Apply, then Ok. We have opened an aperture above the copper layer, on the solder mask layer. For the last step, it is necessary to put four circle shapes on the rectangular shape.

To draw a circle, choose Copper tool on the toolbar. Then with an active cursor, pull off a right click, then choose <u>Circle</u>. The circles must not be drawn on the rectangle first, but on a blank part of the page. With the first click we define the centre point of the shape, then with the second click we define the radius. Let's make a circle with a radius of 100 mil. (Again we can trace the distances on the bottom part of the window. It is suggested to work again on a 20 Grid.) We should choose

"Paste Mask Top" layer in the Add Drafting pop up window. Do not forget to set Solid Copper, Apply then OK. After the first shape, the others can be copy-pasted, to make the job faster.

On the toolbar of the software, we can find a layer selecting drop-down tool.

Let's set "Paste Mask Top" active, to see the circular shapes. The other way to make all the shapes visible is to press the "T" button on the working sheet (like the modeless command of "G20"-grid setting) and press Enter to set every layer Transparent. Pick up each circle with CTRL+E, then put them on the rectangular shape in a symmetrical way. If we want to pick up a circle, which is already positioned on the rectangle, we should use Tab key, to cycle trough the layers, until the circle is selected.

After we are ready we should set the Top layer back to its active state, and turn off the transparency with pressing "T" and pressing Enter again. Draw a shape of 2D line around the rectangle and let's write a label *"Spreading 01"* above the outer line.
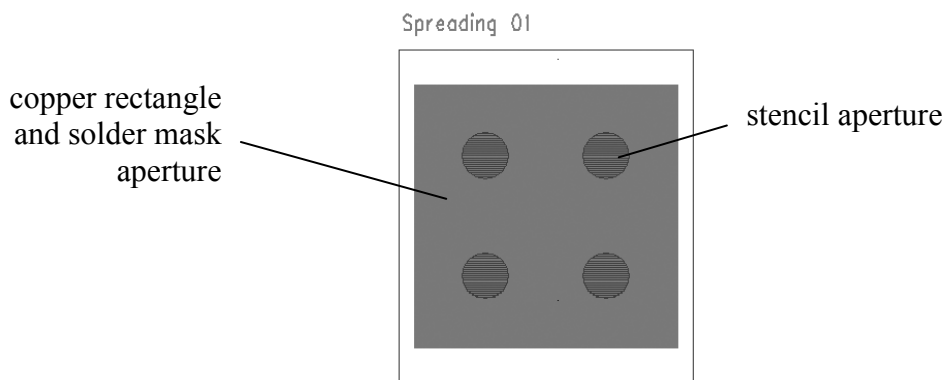

Fig. 9: Spreading test

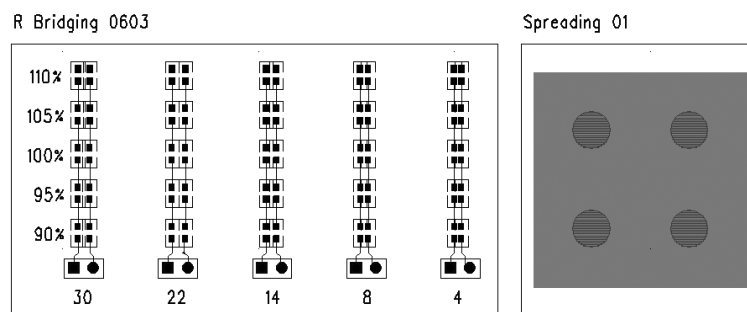Our board should look like this at the moment (Fig.11).

Fig. 10: The board after completing the bridging and spreading tests

IV. Surface Insulation Resistance

To measure surface insulation resistance, we must design the pattern of Figure 11, with the already known methods, just below the bridging test. During the process we will draw shapes on the Top layer, then open apertures on the solder mask and the solder paste layers.

First we should draw the two horizontal rectangles on the Top layer. Copper tool, 20 Grid, length X=980 mil, height Y=60 mil. Activate Solid Copper in the *Add Drafting* window, assign the shape to the Top layer. We do an additional step in the Add Drafting window. Set Line settings/Width to 1, then copy and paste the shape. Let's position the two rectangles from 1000 mil distance from each other according to the inner edges of the shapes. The coordinates on the bottom of the screen will help the positioning.

Hint: Pressing "q" on the working sheet (and pressing Enter of course) we can pull out a measurement tool, which can measure the distances according to the pre-set grid.

SIR test
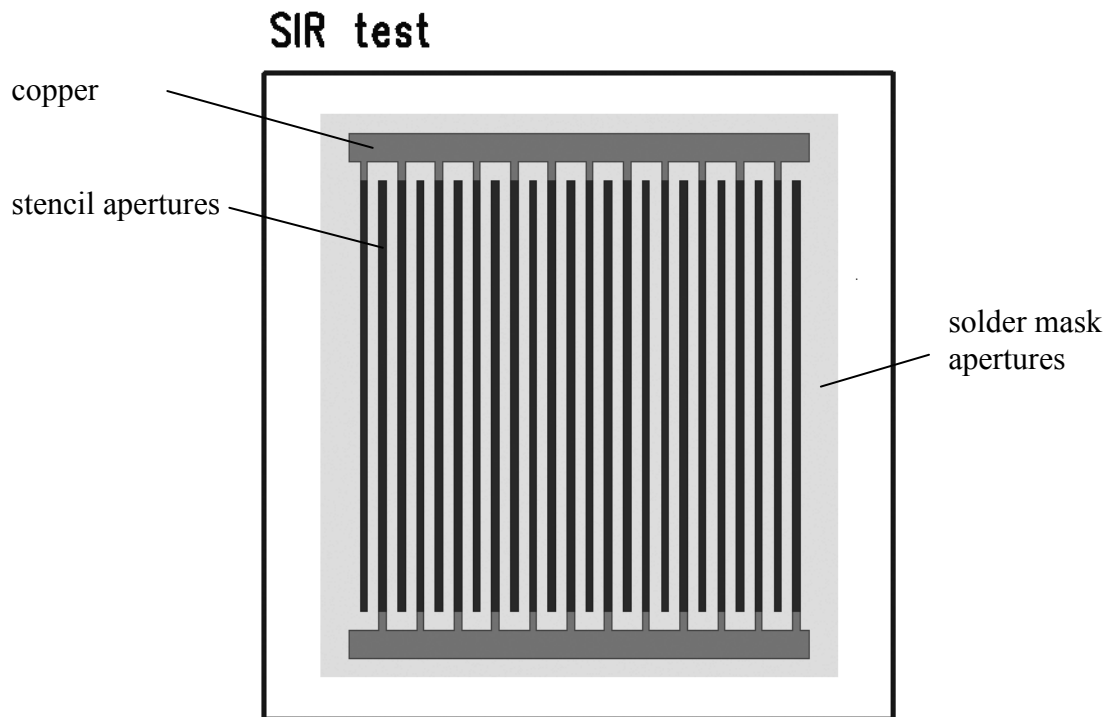
Fig. 11: Surface insulation resistance pattern

Finally we should draw the comb lines. The height is Y=960mil, the thickness is X=16mil. It is recommended to set a grid of 16 with "G16" modeless command. Take note: by continuous setting of the grid, we can adapt to the sizes required for the task. In Add Drafting, we should set the Width again to 1. Do not forget Solid Copper and assigning to the Top layer

If one vertical line is ready, the grid should be set to 20, then copy paste, and make copies of the vertical line, place them according to 20 mil distance between each other.

Hint: It is suggested to use the "Select Shapes" selection, by right-clicking on a blank part of the sheet. With other selection, we will activate only the edges of the shapes, not the full shape itself.
It is suggested to use CTRL+E combination for moving highlighted/selected shapes.
With group selection, and copy pasting, the task could be achieved in a short time.

If the comb is ready, let's prepare the apertures. We should do apertures on two layers. Let's draw a bigger aperture, (a rectangle) around the pattern with the Copper tool. The sizes should be: X=1100 mil, X=1200 mil. Let's assign the new shape to the Solder Mask Top layer, with activated Solid Copper in *Add Drafting* window. Then switch back to a grid of 4 ("G4"), then draw a rectangle with the following dimensions X=16 mil, Y=920 mil. It should be then assigned to Paste Mask Top with Solid Copper! We should copy paste another instance of this shape, then position it on a comb line. Do this until one paste mask rectangle is positioned over each comb line. If the Graphical User Interface is not visualizing the layer, we should choose Paste Mask Top layer in the drop-down layer menu of the top toolbar.

Hint: If three shapes are positioned over each other, with appropriate selection, we can choose between them using Tab key on the keyboard. This may help if the cursor does not find the appropriate shape in the layered structure.

Finishing the job we should draw a 2D line around, and add a text above the line: „*SIR test*".

V. Bridging test

In the previous steps we have learned to use the basic elements of PADS Layout, and the possibility to create shapes on different layers. Combining the knowledge what we have gained before, we should prepare the pattern on Figure 12. The track width of the copper tracks are 30 mil, the length of the tracks is 1328 mil, the distance between them is 60 mil. The stencil apertures have 40 mil width, the distance between them is reduced from 32 mil to 4 mil, with 4 mil step. The stencil aperture pairs should be positioned with 60 mil distance between them. It is recommended to use a grid of 4.
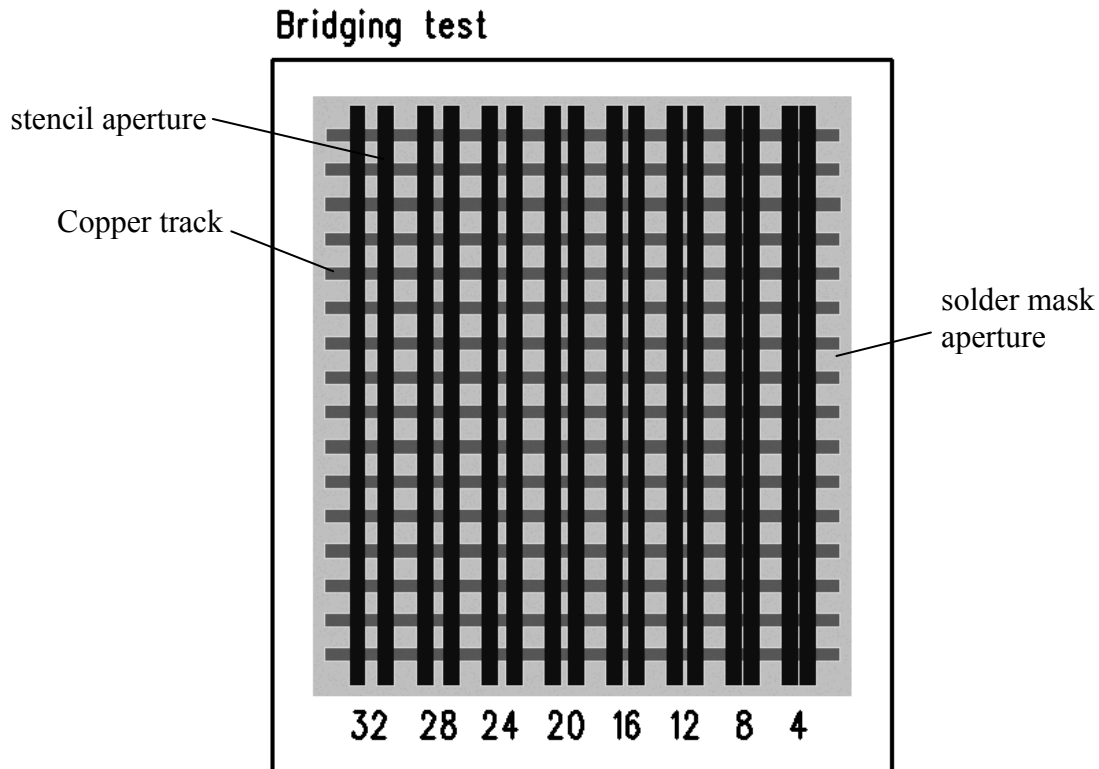
Fig. 12: Bridging test

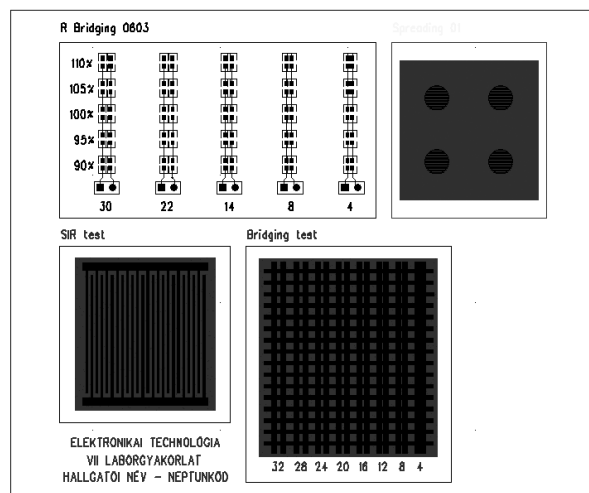Figure 13 shows the ready test board.

Fig. 13: The test board

VI. Drawing board outline and positioning the fiducial marks

The Board Outline should be drawn around the four complete patterns. This outline will mark the edge of the board. It is possible to give exact coordinates for the board, however to make our work more simple (and to adjust the outline to the space requirement of our design) just draw a line around our test patterns.

Search Board Outline and Cut Out (⬚) icon on the drafting toolbar, activate it, and draw the outline, with attention to leaving some space at the corners for the fiducial marks. To select and modify the drawn outline, right click on the blank workspace, *Select Board Outline*.
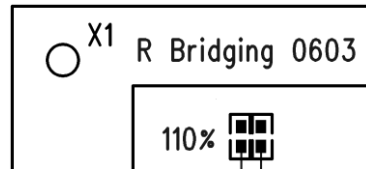

Fig. 14: Upper left corner with the outline and a fiducial mark (X1)

To add a fiducial mark, activate BGA Toolbar (🌼), search for Add Component (🔳) , search for *moire* string, then add "common:MOIRE" to the four corners, within the board outline. If it is not possible due to the lack of space, modify the board outline.

VII.   Finishing the work, generating manufacturing files

The last step is to generate the manufacturing files. These will be the initial starting points for the PCB manufacturer during the processing of our design. Do this with *File/CAM* menu point, *Define CAM Document*, then define a path in the Create Directory drop-down menu:

   D:\User\ET_VII\NeptuneCode\CAM

Then use Add…, and select in Document Type the Routing/Split Plane, and Top. The document must have a non-default name (Document Name) like *NeptuneCode_TOP*, then the Output File should be named as *NeptuneCode_TOP.pho*. Afterwards click Customize Document/Layer, then in the *Select Items* window activate Other/Board Outline. This will store the board outline on the top layer.

This whole process should be done on two other documents: Solder Mask and Paste Mask. There is no need however to activate the board outline in these layers.

The three documents should be listed in the window of *Define CAM Document* in the CAM Documents section. If all documents are selected, then click Run. The document files are stored in the folder defined above. The files with .rep extensions are only report files, they are not needed by the manufacturers. Right now is no need for drill files, while we do not need the through hole vias for the test pads either.

VIII.   Finishing the work

Clear our Library from both PADS Logic and PADS Layout software in the Library Manager list (Manage library list). Make an archive file (zip/rar) of our own working directory to the following place:

   D:\User\ET_VII\GroupNumber

After a discussion with the laboratory teacher, turn off the computers.