DEVELOPMENT OF USER MENU FEATURE ON ZUKEN CR-5000 FOR FINDING CLEARANCE ERROR WITHIN PCB'S COMPONENT PAD

Putra Chandra Erhar[1] , Dian Dharmayanti[2]

Informatics Engineering – Universitas Komputer Indonesia
Jl. Dipatiukur 112-114 Bandung
Email : putra.chandra.erhar@gmail.com[1] , diandmdr@yahoo.com[2]

ABSTRACT

Printed Circuit Board or PCB is the crucial part for developing a more advanced tecnology. The reason behind it was because almost all of technology devices using PCB for its main hardware. While developing some new PCB, the PCB Designer took a vital spot to determine wether the PCB is successful or not. This's because PCB designer need to handle the design to fit all of the component and make sure the board is properly made by following the guidelines rule for the said PCB. By checking the finished design with the guidelines the PCB might have the least possibility chance for occuring flaw or errors. Since the cost of printing PCB is quite much, the developer including the PCB designer must carefully verify the design before submitting the gerberdata to production house. With this being in mind, some method for helping designer's PCB verification job is needed. One of them is verifying the pad to pad distance to make sure both of the component's pads did not short to each other when soldering process by machine.

By using User menu feature on Zuken CR-5000 some function will be developed to find clearance area within component's pad automaticall. Hopefully, with this new function will help PCB Designer finding the errors while checking pad to pad distance which is not met the minimum requirement of 1mm based on the *visual display guidelines*.
The result for this research is an errorlist table which showing component's pads that did not met the minimum requirements of 1mm. With this research being done, writer hope to reduce the weight of the designer work by eliminating the possibility of errors due to manual verification.

Keywords: *PCB Design, ZUKEN CR-5000*

1. INTRODUCTION

Requirement for PCB's development is getting more increasing year by year because of PCB is one of the vital component for developing the greatest technology. Following by high demand, in designing PCB, techcniques that do should be proper. Improper designing able to cause error PCB that can endanger user in comercial use which led to the financial losses in term of the company.

The responsibility of a PCB designer is to ensure error-free of PCB. Error in design can be burdensome development cost, cause it needs high cost to reprint of PCB. Therefore, before sending design, PCB designer must pass through verification step for checking and confirmation that the design has followed the rules in accordance to guidlines and had high competition value design and error free.

During checks on the PCB, there is a moment when designer missed error if verification process has done manually, especially on PCB which has relatively large size and has many components. To anticipate such cases, the use of tools is highly recommended when doing verification process. Zuken CR-5000 software already has tools to assist designers in verification process so that designers able to use the tools easily or creating user menu their own log file.

One of example case is at using flow machine to print PCB according to standard in Visual Display Guidelines checklist, PCB designer must ensure that each pad of components has minimum gap of 1mm. This is to ensure there is no fault or short on circuit when PCB had soldered.

1.1.1 Printed Circuit Board

Printed Circuit Board (PCB) is a board which being printed with copper to wire the suface-mounted and socketed electronic components which ultimately making an useable electronic devices.

PCBs can be single-layer for simple electronic devices. A complex pcb such as motherboard or graphic cards may take up to 12 layer, and a power PCB for server might take up to 16 layer. The more layer PCB had, the higher production cost it will take. PCB's layer is divided as follow:

1. Substrate

The base material used for the board is usually fiberglass with its solid core which gives the PCB its rigidity and thickness. The most common designator for this fiberglass is "FR4". There's also flexible PCBs built on flexible high-temperature plastic[1].

2. Copper

Right after the base material there's thin copper layer which is laminated to the board with heat and adhesive. This layer's used to pass the current electricity within the board. The copper thickness can vary and is specified by weight, in ounces per square foot. While the common PCBs might only use 1 ounce copper, some PCBs with that need to handle high power might use 2 or 3 ounce copper. Each ounce per square translates to about 35 micrometers or 1.4 thousandths of an inch of thickness of copper. In lower cost electronic devices the PCB may have only one side of copper which later called 1 layer PCB. These copper layer may be applied on both side (2 layer PCB) or extended up to 16 layer (multilayer)

3. Solder Mask

The layer on top of the copper foil is called the soldermask layer. This layer gives the PCB its green color. It is overlaid onto the copper layer to insulate the copper traces from accidental contact with other metal, solder, or conductive bits which help to prevent any solder jumper or circuit short in a PCB[1].

4. Silkscreen

The colored silkscreen layer is applied on top of the soldermask layer. The silkscreen adds letters, numbers, and symbols to the PCB that allow for easier assembly and indicators for humans to better understand the board[1].

1.1.2 Zuken CR-5000

Zuken CR5000 is a software developed by Zuken INC a Japanese Multinational corporation. Zuken CR5000 is used to designing and drawing electronics system and a PCB. The Zuken CR5000 had all the tools that needed to make a proper PCB from scratch.

1.1.2.7 User-Menu

User-menu function in Zuken CR-5000 can be accessed in Placement and Wiring tools that reside in Utilities -> User-Menu or can be directly accessed using key "Ctrl+M". The user menu function contain command that user defined to make designing progress easier.

1.1.6 VD Guidelines Rule

Visual Display (VD) Guidelines Rule is a checklist that must be applied while designing a VD PC Board. Missing one of the rule might lead to error in design. Using this Guidelines Rule as the basics, the writer will later improve the use of User Menu to search errors in design.

1.1.7 SMT

Surface Mount Technology (SMT) uses an assembly process in which the components are soldered to lands on the surface of the board, rather than inserted into holes running through the board[2].

1.1.8 Solder Paste

Solder paste is the combination of flux and solder powder, which plays an essential role in SMT. It offers a solder medium to form solder joints in reflow process with enough electrical and mechanical strength.[3].

1.1.9 Stencil / pastemask

PCB stencils help to apply solder paste to PCB boards during the reflow soldering process, to transfer the exact amount of solder paste to the exact location on the circuit boards. The stencil is placed on the surface of the PCB and then the solder paste is applied [4].

1.1.10 Pad

Pad is portion of exposed metal on the surface of a board to which a component is soldered using solder paste[4]

1.1.11 Residual Resist Guide Line

Residual Resist Guide Line is a portion of a metal that exposed and applied to every pad of a component to guide the residual of a resist and make sure the resiudal resist from soldering did not short circuit to another pad.

1.2.1 Software Development Method

There are five steps to implement waterfall[5] methods:

1. Requirements:
   This step is to find needs that needed for the research by conducting interview from Meister PCB CO., Ltd's employee and analyze company's needs.

2. Design:
   This step is to analyze and design system based on company's needs.

3. Implementation:
   After design of the system has been completed, company's needs will be implemented at user-menu feature in Zuken CR-5000.

4. Verification:
   After implementation of the system was acomplished, user-menu feature that have

been developed will be tested whether it is according to company's need or not.

5. Maintenance:

Last steps, if company's needs not fullfiled or there is additional needs for the company, the new function should be developed or revised to fulfill the newly arranged requirements.

2. Research Contents

2.1 Problem Analysis

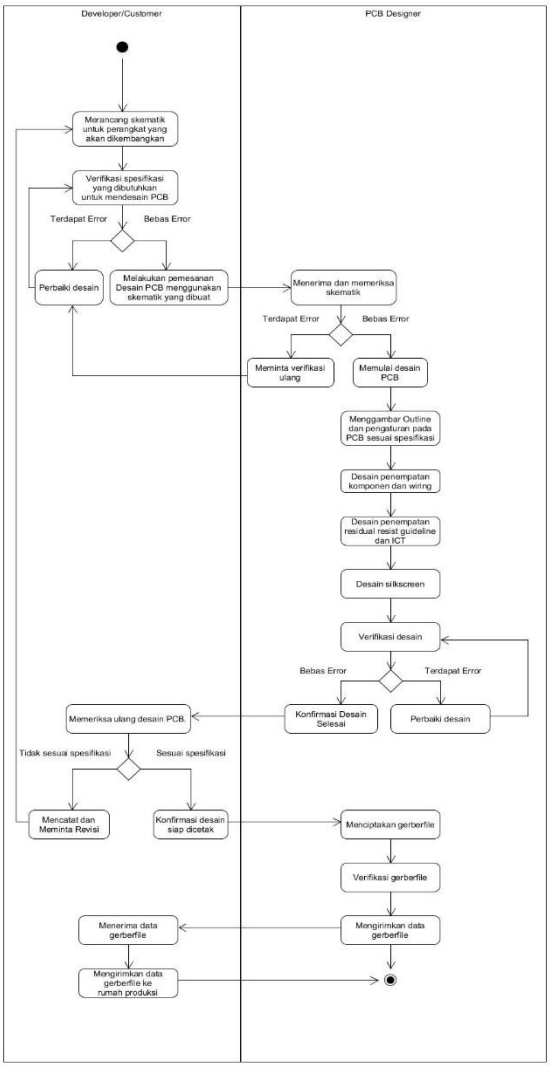
Problem analysis shows overall description of those problems that exists about PCB's designing.

2.1.1 System Analysis

System analysis is to identify and get problems that been in system and determine what needs that needed for constructing the system that will be offered.

2.1.2 Ananlysis of Current System

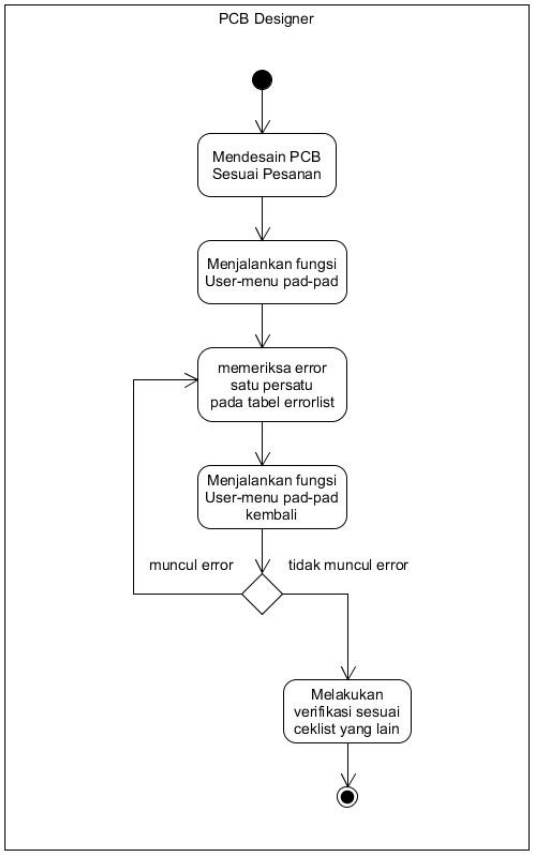
Analysis of Current System has done by doing observation at Meister PCB Co., Ltd in the course of designing PCB.



Picture 2-1 Current Diagram System Activity

2.1.3 Analysis of Offered System

Analysis of offered system is a general description of the system to be built to overcome problems found in current system.



Picture 2-1 Offered System Diagram Activity

2.1.4 Steps of Development System Offered

To achieve the development of system in accordance to analysis system offered, so it should do steps as follow:

1.Setting customization of new function on user-menu.

2.Analysis code on available function.

3.Compare code on available function to know general picture towards that codes and its function.

4.Designing code for pads new function

5. Implement those codes into log-file

6 Testing new function that have been created.

2.2 Analysis of Function

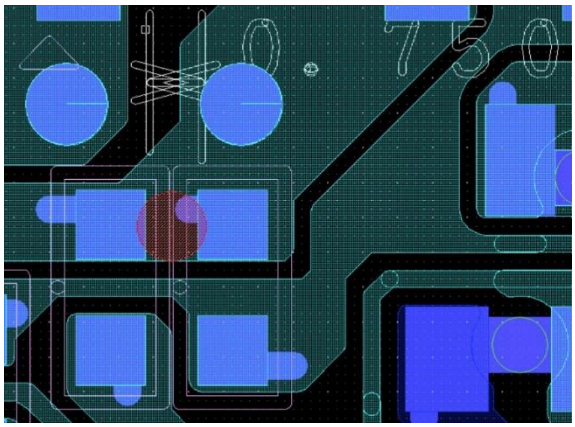
Analysis of function aimed to identify problems using user-menu feature of Zuken CR-5000 including function analysis, comparation function and function that will be developed.

2.2.1 Analysis of Available Function

Analysis of Available Function done by directly interviewing PCB Designer employee at Meister PCB Co., Ltd to understood more about each function. So, by knowing how each function works in the user menu feature on Zuken CR-5000, the new function can be easily designed.

2.3 Designing User-Menu Function for checking gap between pad component.

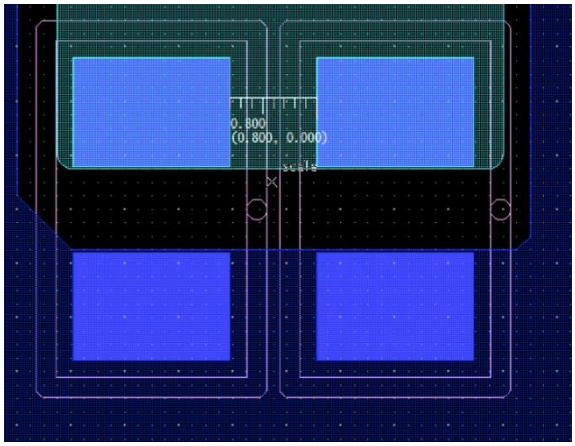
In the guidelines VD rules, placement of component must qualify the rules each one component pad to others with 1mm minimum gap. Therefore many components into a VD PCB, can make error easily and probably, PCB designer missed some faults when doing verification, since doing manual verification eventhough many components should be cheked.



Picture 2-3 Error caused by gap between component pad less than 1mm.

The picture above shows an example of the errors caused by the gap between pad components is less than 1 mm. It should be 0.75mm. When PCB designers find out such errors, they have to do replacement of that component in order to make a condition which minimum gap between pad should be 1mm.

Eventhough keeping a gap between pad with minimum 1mm is a must, sometimes there is a condition which space of PCB ia very narrow and hard to do component placement. In that case, pads that has similar net able to be tolerable until 0.75mm minimum gap, yet for different net has to keep 1mm gap without tolerant.



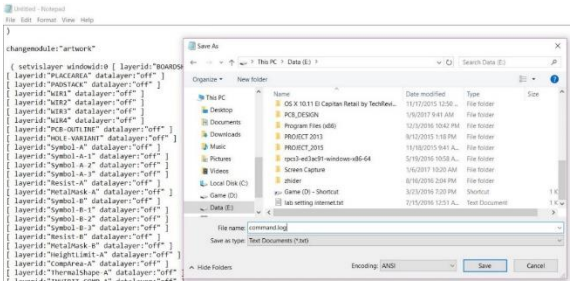
Picture 2-4 Tolerable Error cause of similar net

2.4 Function Implementation

Function implementation is application of function that been designed into user-menu feature with using logfile as command storage which will be operated through user-menu fitur for running comand that been used to find clearance error between component pad.

2.4.1 Creating Log File

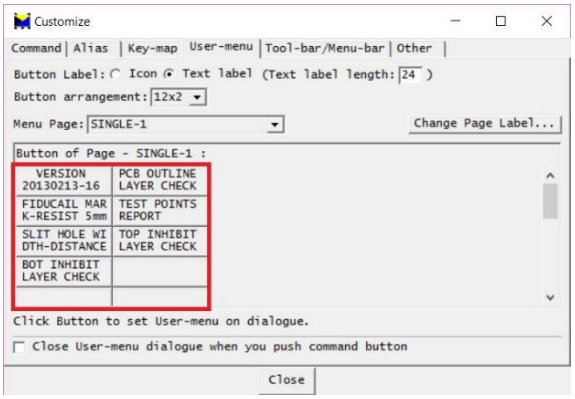
For ease comand writting that will be used in user menu at Zuken CR-5000 software, command line can be written into a log file that been saved at drive. This log file will be operated and that written command in it will be executed by Zuken CR-5000.



Picture 2-5 Creating logfile with notepad program

2.4.2 Creating New User-Menu

By going towards User-Menu tab at cuztomize dialog, available command button can be filled with new command by clicking desired command button.



Picture 2-6 Choose the comand button that would put new command line

After clicking the command button, new dialog will apear on the screen and user should put command as follow at field command string:

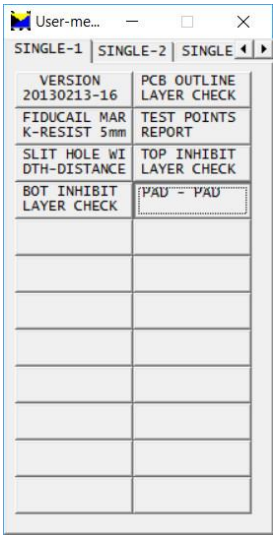
*) (playbacklog filepath:"*file location*")*

2.5 Testing New User-Menu Function

After the log file has finished and saved into drive and user menu able to call log file directly, so testing can be done to ensure that user-menu function already in accordance to the purpose of the development function.

2.5.1 Alpha Testing

Alpha testing is a test from the side of developer that making situation as if developer was viewing from user's side, that is PCB designer. Then record all of faults on new function and ensure its functionality has already according to the design.



Picture 2-7 Display Command Button PADS to Check the Gap between Pad on PCB

2.5.1.1 Scenario of Testing Pad – Pad User-Menu Function

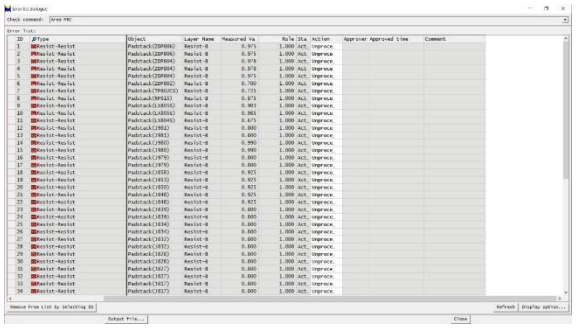
This following is a scenario which will be used for testing in pads function at user menu that has been built.

Table 2-1 Draft Alpha Testing

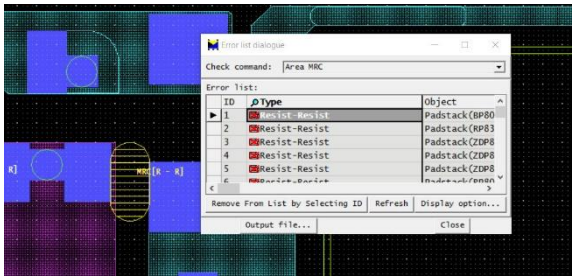
Test Parameter	Scenario	Kind of Test
Pad - Pad	Placing components pad less than 1mm. Placing components pad more than 1mm. Placing components pad within same net in less than 1mm.	*Blackbox*
Pad – Residual Resist Guideline	Placing *Residual Resist Guideline* on *pad*	*Blackbox*
Thermal Resist – Thermal Resist	Placing *Thermal Resist* on *pattern*	*Blackbox*
Pad - ICT	placing *ICT* near pad	*Blackbox*

2.5.1.2 Case and Test Result on The Pads Function

This following is the result of testing to pads function according to the scenario that has been created.

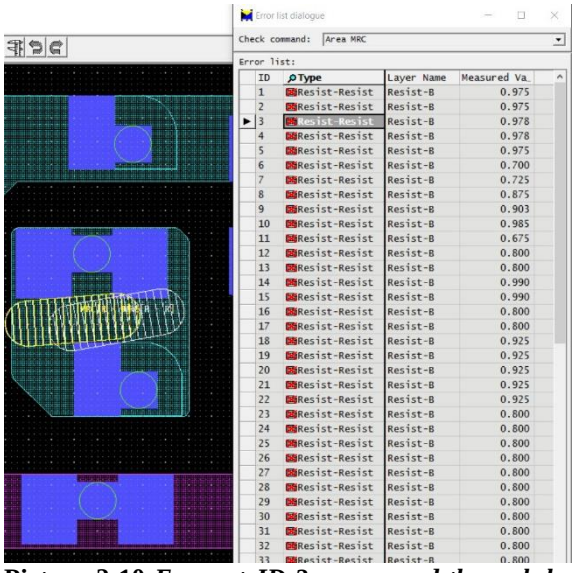


Picture 2-8 Errorlist interface whichis showing error if gap between resist is less than 1mm



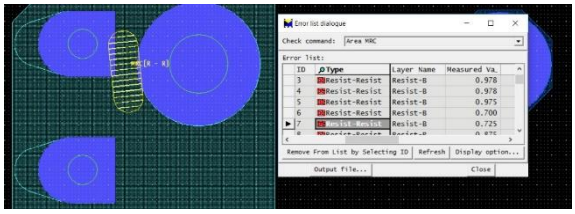
Picture 2-9 *Error* on *ID 1* can be seen by yellow DRC line

At two pictures above, there are yellow lines that indicates error that were highlited when type column on errorlist was clicked, and the canvas will automatically zoom and search for the area of error.



Picture 2-10 *Error* at *ID 3 can passed through by signify false error*

At that picture above shows that error happens between component pad that has net similarity. Because of there is a tolerancy at pad which has net similarity until 0.75mm, so that error can be denied by giving false error status if there is no possibility to move component by dint of narrow space on the board.

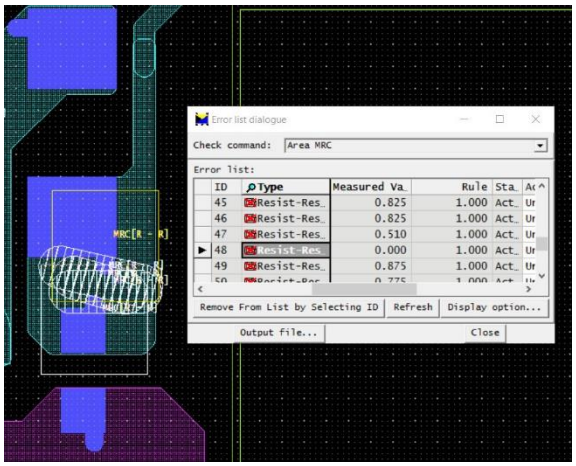


Picture 2-11 Error* at *ID 7 can't be tolerant eventhough has net similarity

At that picture above, pad component which has net similarity yet not met the required rule of 0.75mm should be moved either one of component so that the gap will be able to reach 1mm minimum standard or 0.75 minimum tolerance.

Table 2-2 Testing on test parameter of pad-pad

Case and Test Result			
Test Parameter	Scenario	Observation	Conclusion
Pad - Pad	placing component pads more than 1mm	not showing error on errorlist	[√] Accepted [ ] Rejected
Pad - Pad	placing component pads less than 1mm	shows *error* on *errorlist*	[√] Accepted [ ] Rejected
Pad - Pad	placing component pads on same net less than 1mm but more than 0.75mm	Showing gap information on errorlist and later be given false error status	[√] Accepted [ ] Rejected



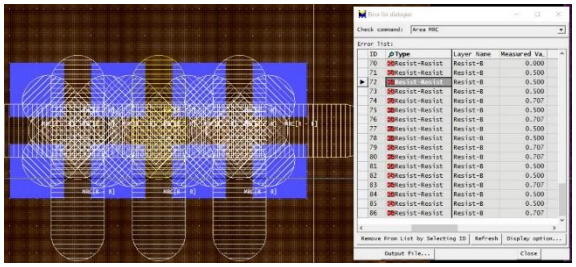
Picture 2-1 Error on ID 47 and 48 because resist besidde the component pad.

At residual resist guideline installation each pad to SMD component pad, residual resist guideline placement also using Resist-B layer that's also part of resist which program do check. Therefore, an error in id 48 has a gap of 0mm because component pad and residual resist,

Error in ID 47 also occur due to residual resist guideline of different component pad. Because of main purpose of checking error is checking each pad component in addition to error that occur between component pad can be denied and given false error status.

Table 2-3 Testing on test paramaeter pad – residual resist guideline

Case and Test Result			
Test Parameter	Test Parameter	Test Parameter	Test Parameter
Pad – Residual Resist Guideline	Placing residual resist guideline on pad	Showing information kind of resist which is "line" and later be given false error status	[√] Accepted [ ] Rejected

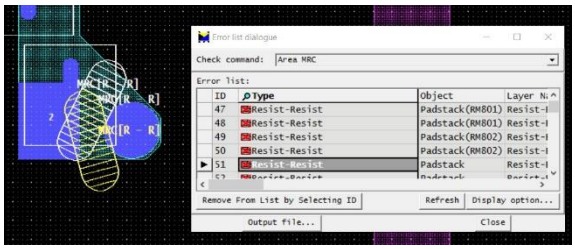


Picture 2-13 Error that happen because of *thermal resist.*

Installation of thermal resist also into checking condition of the program. Thus, the ID 72 to 86 in the figure above, the entire thermal resist will be calculated as error by the program due to installation resist thermal standard is 0.5mm for each thermal resist. Therefore thermal resist is a seperate object of pad component, then the error can be ignored and given false error status.

Table 2-4 Testing on test *Thermal Resist – Thermal Resist*

Case and Test Result			
Test Parameter	Test Parameter	Test Parameter	Test Parameter
Thermal Resist – Thermal Resist	Placing *Thermal Resist* on *pattern*	Showing information kind of resist which is "area" and later be given false error status	[√] Accepted [ ] Rejected



Picture 2-14 Error that happen because of ICT

Error that showed in ID 51 is an error that caused by ICT which is too close to the pads or another resist object like residual resist guideline or even thermal resist. Eventhough there are rule for placing ICT near pad, but checking ICT is not the function that currently needed. Therefore, this error can be given false error status

Table 2-5 Testing on test parameter pad-ICT

Case and Test Result			
Test Parameter	Test Parameter	Test Parameter	Test Parameter
Pad - ICT	Placing ICT near Pad	Showing information kind of resist which is "padstack" and later be given false error status	[√] Accepted [ ] Rejected

2.5.1.3 Alpha Testing Conclusion

Based on testing that have been done on new function, the conclusion is that this new function can do the work to find gap between component pad accurately.

2.6 Beta Testing

Beta testing was done by testing the target of users who are PCB Designer in Meister PCB Co., Ltd by interviewing them about the new function whether its functionality was done properly as it is, and the user can give feedback or input to help repair and improve the function.

2.6.1 Interview Data

The questions that will asked when doing the interview will be shown as table bellow:

Table 2-1 Question draf that will be asked during the interview

No	Question
1	Does all this time you have difficulty in checking the distance between pad components?
2	When verifying the distance between pad components, how often do you find an error?
3	Do you have a confidence when confirming the verification shortly after verifying the distance between pad components that there will be no more errors will happen?
4	Do you feel more helped by using this newly add funcion on the User-Menu feature?
5	Right after using this new function, do you have confidence to confirm that there will be no more errors will happen later?

2.6.2 Interview Results

. With question that have been prepared before, the result of interview with PCB Designer employee on Meister PCB Co., Ltd will be explained as follows:

1. Does all this time you have difficulty in checking the distance between pad components?

Answer:

Checking clearance between pad's component is something that consumes time especially on large size PCB with many components within it.That being said, i found it difficult for doing the verification since i must memorize all the location that i have already checked.

2. When verifying the distance between pad components, how often do you find an error?

Answer:

When doing component placement, me and maybe our other PCB designer staff only depends on hunch when placing the components. Because of that, while doing verification process there are many pads between component that did not met the requirements or almost met the requirements. For example 0.95mm or 0.975mm which is very close to 1mm, but is not allowed by rule.

3. Do you have a confidence when confirming the verification shortly after verifying the distance between pad components that there will be no more errors will happen?

Answer:

Maybe I will have confidence to say so if the PCB's size is small. But for a large size PCB, even doing it many times, sometimes there's always clearance errors found. Because of that, I think i would not have any confidence for it..

4. Do you feel more helped by using this newly add funcion on the User-Menu feature?

Answer:

By using system that have been computerized for checking, of course i feel more helped while finding clearance error between component's pad. It is because the new function on User Menu can display where the errors happen and shorten the time needed for the process.

5. Right after using this new function, do you have confidence to confirm that there will be no more errors will happen later?

Answer:

Different than checking the errors with manual way by using ruler feature that available on Zuken Software. Verification with computer and MRC system as its basics of course i have more confidence to confirm there are no more errors and believe that there will be no more errors beside the one that is displayed on errorlist table on that new feature.

2.6.3 Beta testing conclution

After interviewing one of the employee of Meister PCB Company, can be concluded that this newly developed function was helping the PCB Designer employees to finding clearance error between component's pads based on VD Guidelines rule and reduce the time needed for verification process thus making work more efficient.

3.Closing

3.1 Conclusion and Recommendations

3.1.1 Conclusion

Based on examination result in the previous chapter, the conclusion will explained as follows:

1. By developing this new function, pad to pad clearance error verification process can be done with more accurate than doing the process manually. Hence, lower the possibility of error on the design.

2. By using this newly developed function, PCB Designer find out it is helpful to finding pad to pad clearance error based on the rules and shorten the time needed for the verification process.

3.1.2 Recommendations

The recommendations offered for this project will be described as follows:

1. While applying the pad-pad function it is recommended to do it before placing ICT, Thermal resist, and residual resist guideline. This is purposed to reduce false error result because of these component.
2. More advanced development of this project might be done if there's another needs from the company or if in the future a VD Board require 2 layer. Therefore resist-A layer should be included as input.

THANKYOU NOTE

This research project can not be done without support, assistance, and input from various parties. Hence, the author would like to give thanks to:

1. Allah SWT for His mercy by giving health and knowledge
2. Both beloved parents who always support all my ideals, and always provide support and prayer to make me a better person.
3. All of my sisters and brothers and my main family who always give input and support.
4. Mrs. Dian Dharmayanti, S.T., M.Kom. as lecturers who have provided guidance and direction in the work on this project.
5. All lecturers Universitas Komputer Indonesia, especially the Department of Informatics Engineering that has provided knowledge and motivation during the lecture.
6. Mrs. Prof. Dr. Hj. Umi Narimawati, Dra., SE., M.Sc. who have give me the opportunity to go to college in South Korea by scolarship and for guiding us during the period of education there.
7. Mr. Lee Sung Hoon as director of Meister PCB Co., Ltd who have give me opportunity for taking internship on Samsung Electro-mechanics in South Korea for 1 year.
8. Mega Ayu Puspita who always patiently waiting for almost 2 year while I was studying aboard and always give me support and motivation to complete this project.
9. All of double degree program college friends for helping out this project.
10. All of IF-1 2012 classmate for helping out while lectures is on take.

4. REFERENCES

[1] Hord, Mike. "PCB Basics" [Online] https://learn.sparkfun.com/tutorials/pcb-basics/ Accessed on 3 March 2017

[2] Sibley, Jason. "Suface Mount Technology" [Online]http://www-personal.engin.umd.umich.edu/~jwvm/ece539/W99Presentations/SurfaceMount/SurfaceMount.PDF Accessed on 3 March 2017

[3] allpcb.com [Online] http://www.allpcb.com/solder_paste Accessed on 3 March 2017

[4] allpcb.com [Online] http://www.allpcb.com/solder_paste/stencil.html Accessed on 3 March 2017

[5] Hughey, Douglas. "The Traditional Waterfall Approach" [Online] http://www.umsl.edu/~hugheyd/is6840/waterfall.html Accessed on 5 November 2016