DEVELOPMENT OF USER MENU FEATURES BY ADDING RESIDUAL SOLDERING CHECK FUNCTION FOR PCB LAYOUT DESIGN USING APPLICATION ZUKEN CR-5000

Saguh Al Hafidz

Department of informatic Engineering, Faculty of Technique and Computer Science, Indonesian Computer University
Jl.Dipatiukur 112-114 Bandung
hafidroid@gmail.com

ABSTRACT

Advances in science and technology have an important role in the welfare of human life, especially to help in facilitating human needs. The need for sophisticated tools such as LED TV, Smartphone and Notebook consisting of electronic components both small and large, be evidence that an electronic circuit can not be separated in human life.

Printed Circuit Board also called PCB is a printed circuit board that is full of metal circuits that connect electronic components of different types and with each other without wires. PCB which has a layer of copper on only one side of the surface is called one-sided PCB (single-layer). While the PCB which has a layer of copper on both sides of its surface called two-sided PCB (Multilayer). Zuken CR-5000 as an application design PCB layout which features the examiner error is still considered less than optimal in examining every element that can cause errors at the stage of printing PCB board, constraints currently owned is on the residual soldering which makes tin accumulate on the components SMD.

By adding functionality on the feature User-Menu Zuken CR-5000 author will attempt to maximize the examiner process errors and provide optimal results, especially on the residual soldering. The addition of residual function in the user-menu soldering applications ZUKEN CR-5000 PCB designers can help to further improve the quality of its PCB layout design so as to minimize the error that caused the failure of the PCB production process.

Keywords : Design PCB, ZUKEN CR-5000

1. OPENING

1.1 Problem Identification

Advances in science and technology play an important role in the welfare of human life, especially to help enable people to meet their needs. The need for sophisticated tools like LED TVs, smartphones and notebooks consisting of electronic components both small and large, to be evidence that an electronic circuit can't be separated in everyday life.

PCB (Printed Circuit Board) is used in the application of electronic components in their manufacture which includes schematic design, manufacture of circuit paths, drilling, component mounting and soldering. PCB circuit manufacture can be done manually or computerized. Make a series of manually it will spend a lot of money, time, and when an error occurs will be difficult to be detected and corrected. Thereby making the circuit PCB with the help of a computer is a prime choice for a designer than by designing the PCB manually, because it can further increase the time of the design of products, the results are more precise and accurate, and easy to check and repair if found fault in the circuit PCB.

ZUKEN CR-5000 is a software developed to address the needs of complex PCB circuit design and addressing challenges such as multi-layer design technology and electromagnetic compatibility. This software is widely used by companies medium to large companies to produce a series of PCB in large quantities in order to meet market needs. This software features the User Menu to check for errors in a single-layer PCB design and multi-layer.

Error checking on PCB design becomes very important because the most common mistakes is generally at the design stage layout. To reduce the loss of production in the PCB layout to be printed, then the error should be more detailed examination and specific. Many companies fail because of production error checking on the PCB layout is still considered optimal.

Based on these descriptions, then this thesis aims to develop friendly features menu on ZUKEN CR-5000 in order to maximize the error checking on PCB layout design stage, especially on the residual PCB soldering.

1.2 Problem Formulation

Based on the background of the problems that have been described, it can be formulated some formulation of the problem How to add an error check function on user features menu ZUKEN CR-5000 and how to create a function to check the residual errors soldering on PCB layout design.

1.3 Objectives

The purpose of this study was to develop friendly features menu on the application ZUKEN CR-5000 by adding the residual function soldering check.

1.4 Limitations Problem

The problem in this paper are limited to the following:

1. Applications ZUKEN CR-5000 can only be used on UNIX and Windows.
2. Development of User Menu feature only covers residual soldering, Top / Bottom referenc and Top / Bottom Component Area.
3. Using Log Files to add and change the error-checking function.
4. Only briefly describes the use of tools and workspace available on the CR-5000 ZUKEN.
5. Examination of error refers to the standard rules of the CIU company Cooperation. in producing a PCB layout design.

2 LITERATURE

2.1 Zuken

Zuken Inc. (株式会社 図 研 Kabushiki-gaisha Zuken) is a Japanese multinational company, which plays a role in the field of software and consulting services for electrical and electronic engineering. Zuken emerged as a pioneer in the development of the CAD system (Computer Aided Design) in Japan to contribute in electronics manufacturing. Zuken was founded in 1976 in Yokohama, Japan.

Zuken software is mainly used for designing the PCB (Printed Circuit Board), Multi-Chip Modules, and for electrical engineering, cables, wiring harness, pneumatic and hydraulic applications. Furthermore, Zuken offers data management software for electrical engineering and electronics (e-PLM). The company's main market is the electronics industry that includes Smarthome; Mobile Communications Device; means of transport such as cars, special vehicles and rail; industrial equipment such as medical equipment and medical devices; and construction machinery. Zuken also have a strong presence in the aviation industry. Zuken is one of three companies EDA (Electronic Design Automation) big role in PCB design software, the other two are Cadence Design Systems and Mentor Graphics.

2.2 Zuken CR-5000

CR-5000 is Zuken one of the products featured in the provider of electronic systems and PCB design. CR-5000 is made to manage the process of developing and manufacturing on a large scale company, which was developed specifically for medium-sized companies to large companies. CR-5000 offers innovative functionality for multi-layer PCB design and high speed IC packages, addressing design challenges such as signal integrity, power integrity and EMC (Electromagnetic Compatiblity), and ensure a high manufacturability results. CR-5000 can guide the user in the design process. CR-5000 performance, along with scalable design environment, and provide maximum opportunities to collaborate based design team.

2.3 User Menu Feature

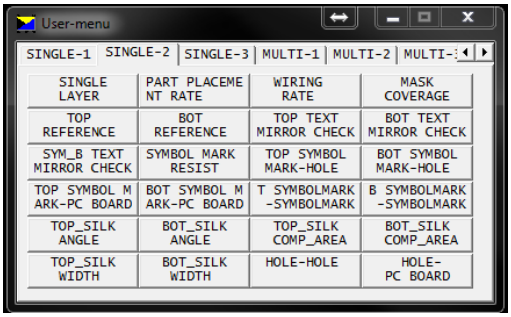


Image 2.1 *User Menu*

User menu in the application ZUKEN CR-5000 is a feature to check for errors in the design of PCB layout for both the design of single-layer or multilayer, so before PCB printed and manufactured PCB design which has been made can be checked beforehand whether there is a path that the error at the stage of placement / wiring, short track (short connected), the distance between the high and low voltage which can cause burning of the track, and so on.

Each company would have rules production standards vary. On the application of CR-5000 ZUKEN production rules for PCB layout design on file extensions: ".rul" that has been integrated with the PCB layout file extensions: ".pcb", so that at the time of the error check feature the menu user error that appears in accordance with the file .rul used.

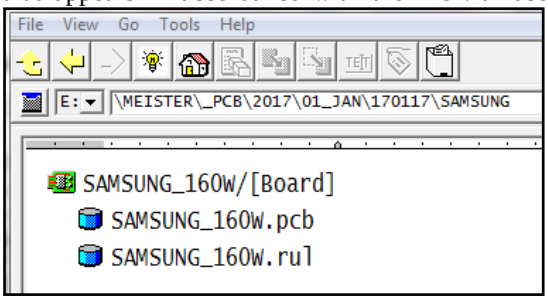


Image 2.2. *File .rul*

Usually each product model have respective production rules in accordance with the standards provided by an electrical engineer who made the schematic design of PCB. Electrical engineer is acting as a client who will always check whether the design results in accordance with the schematic has been created and is ready to be produced. By using Design Rule Editor on CR-5000 ZUKEN a designer can customize the design rules that existed at .rul file in accordance with the client request.



Image 2.3. *Design rule editor*

Part of this editor can specify the type of PCB to be used whether a single-layer or multilayer, adjust the distance between high voltage and low voltage on the PCB track, determine the distance between the component and the perimeter of the components that have a certain height, etc. Design rules that have been made in accordance with the client request to be executed by the user features menu while checking errors in PCB layout design.

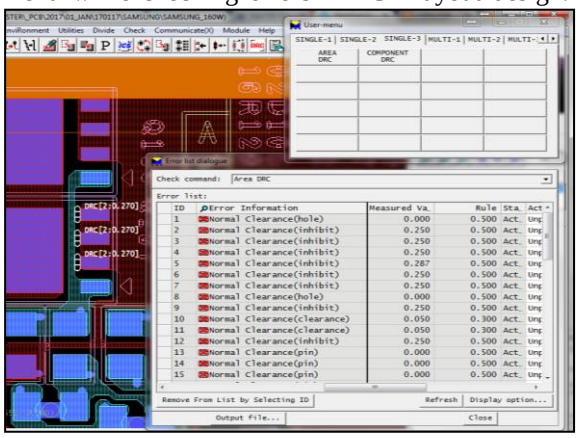


Image 2.4. Error Check

2.4 PCB (*Printed Circuit Board*)

Printed Circuit Board or called PCB is a printed circuit board with a circuit full of metal connecting different electronic components as well as the same kind with one another wirelessly. Generally, the circuit board is made of ebonite or glass fiber in which one or both sides coated by a layer of copper. For PCB having a copper layer on only one side of the surface is called one-sided PCB (single-layer). While the PCB has a copper layer on both sides of its surface called double-sided PCB (Multilayer).

2.5 DRC and MRC

DRC / MRC (Design Rule Check / Manufacturing Rule Check) is an electronic automation in the area of the design to optimize the product design process by considering the manufacturing requirements and determine whether the physical layout of components has met a series of parameters recommended by the PCB design rule or not.

The circuit parameters provided by semiconductor manufacturers that allows designers to verify the correctness of the set-mask. A set of design rules specify certain geometric and connectivity restrictions to ensure sufficient margin to account for the variability in the semiconductor manufacturing process, so as to ensure that most of the parts are working properly.

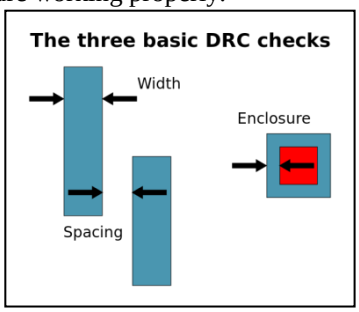


Image 2.5. DRC *Checks* 1

The most basic in design rules is shown in the image above. The first is the rule layer. Wide rules specify a minimum width of each shape in design. Spacing rules specify a minimum distance between two adjacent objects. These rules will exist for every layer of the semiconductor manufacturing process, with the lowest layer has the smallest rule (typically 100 nm in 2007) and the top metal layer has a greater rule (perhaps 400 nm in 2007).

2.6 Residual Soldering

Residual Soldering is a line that is located on the component pin SMD (surface mount device) berfugsi as discharge lines remaining tin on the PCB during the printing process. This section is often considered trivial but the error on the part of lead attached to quite a lot of SMD components, which causes lead to accumulate on the component pin and cause the pin on fire.



Image 2.6. *Residual Soldering*



Image 2.7. Kesalahan *Residual Soldering*

3 MODEL, ANALYSIS AND DESIGN

3.1 Problem Analysis

Zuken CR-5000 as an application design PCB layout which features the examiner error is still considered less than optimal in examining every element that can cause errors at the stage of printing PCB board, constraints currently owned is on the residual soldering which makes tin accumulate on the components SMD. As for as the additional error checking to better minimize failures during the production error checking will be performed on the reference component top / bottom and check the component area top / bottom. Parameters that can be used as indicators of solving the problem is: resist section on SMD components, the distance between resist- resist, resist SMD components, top / bottom reference, the component area top / bottom, and the presence or absence of each resist SMD component that can automatically be checked by examiner system errors.

3.2 Analysis of Errors in the area of MRC Examiner

At this step, the data that will be required to check the residual errors exist in the arrangement soldering DRC / MRC is available at Zuken application cr-5000.

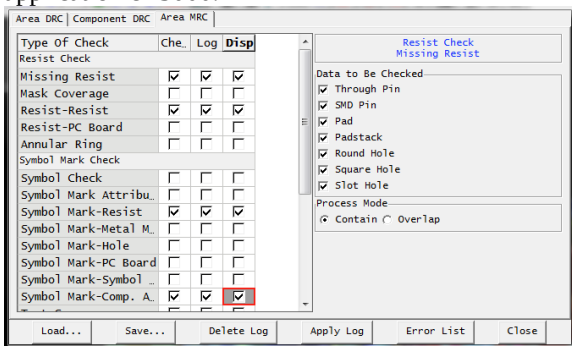


Image 3.1. *DRC/MRC* Configuration *1*

Here are the characteristics of the data used :

Table 3.1. *DRC/MRC* Configuration 1

Type Examination	Information
Missing Resist	Checking and displaying resist is lost on any changes made when the user-menu features run
Resist-Resist	Check if there resist is stacked with another resist, or the distance between the resist is too close to the pin component and will be displayed on the error-list.
Symbol Mark-resist	Check if there are symbols of PCB or PCB design marker lines covering resist.
Symbol Mark-Comp	Check if there are symbols of PCB or PCB design marker lines covering components.

Furthermore, the data required to check for errors in the reference section top / bottom so that there is no reference component names overlapping or missing, then the following are the parameters DRC / MRC needed.

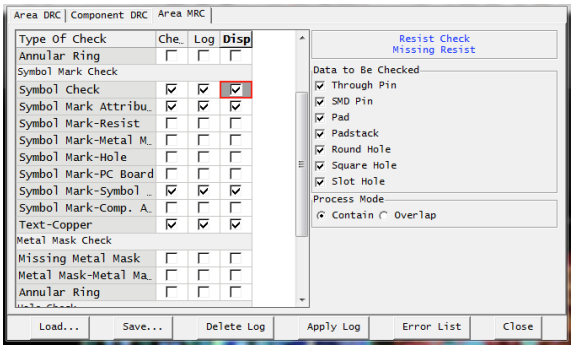


Image 3.2. *DRC/MRC* Configuration 2

Table 3.2. *DRC/MRC* Configuration 2

Type Examination	Information
Symbol Check	Checking all electrical components symbol of both components and non-electrical
Symbol Mark Attribute	Examine the attributes on which there is no symbol on the correct layer and whether having a thickness in accordance with the rules
Symbol Mark-Symbol	Check if there are symbols of PCB or PCB design marker lines that cover the other symbols.
Text-Copper	Check if there are texts that touch the copper layer as a conductor of current.

Furthermore, the data required to check for errors in the reference section top / bottom so that there is no reference component names overlapping or missing, then the following are the parameters DRC / MRC needed.

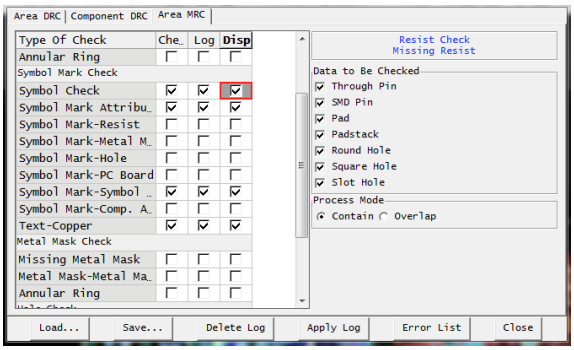


Image 3.3. *DRC/MRC* Configuration 3

Table 3.3. *DRC/MRC* Configuration 3

Type Examination	Information
Symbol Check	Checking all electrical components symbol of both components and non-electrical
Symbol Mark Attribute	Examine the attributes on which there is no symbol on the correct layer and whether having a thickness in accordance with the rules
Symbol Mark-Symbol	Check if there are symbols of PCB or PCB design marker lines that cover the other symbols.
Text-Copper	Check if there are texts that touch the copper layer as a conductor of current.

Next is the data required to check for errors on the part of the component area top / bottom so that no parts are colliding with other components. Parameter DRC / MRC needed.

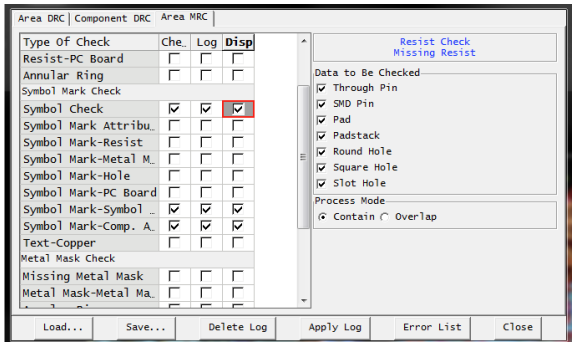


Image 3.4. *DRC/MRC* Configuration 4

Tabel 3.4. *DRC/MRC* Configuration 4

Type Examination	Information
Symbol Check	Checking all electrical components symbol of both components and non-electrical
Symbol Mark - Symbol	Check if there are symbols of PCB or PCB design marker lines that cover the other symbols.
Symbol Mark-Component Area	Check if there are symbols of PCB or PCB design marker lines that cover the other symbols.

4 RESULT AND IMPLEMENTATION

4.1 Log File Implementation

The implementation phase is the implementation phase of the design stage. At this stage the log files that have been made should be saved in the user folder menu at address C: \ bd_setup \ USER-MENU \ Single-Layer.

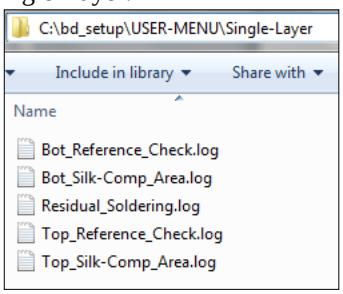
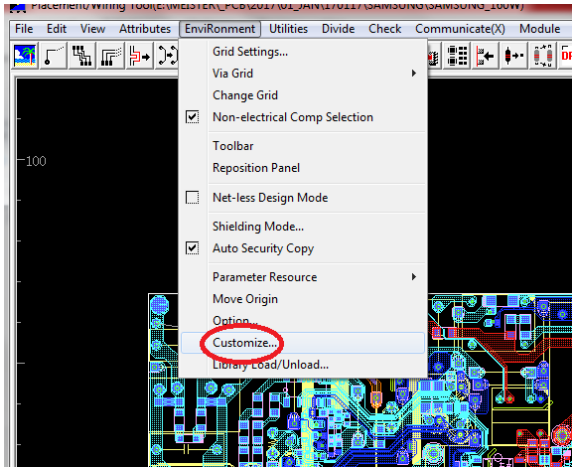


Image 4.1. log file storage

Bd_setup folder is a folder that has been integrated with the application Zuken cr-5000 is a place where the log files to run the application menu on the user features Zuken cr-5000. Once the file is saved then open cr-5000 Zuken application of environment-click and select customize.



Iamge 4.2. Implementation log flie 1

After that will come a customized dialog box and select the tab "User-menu".

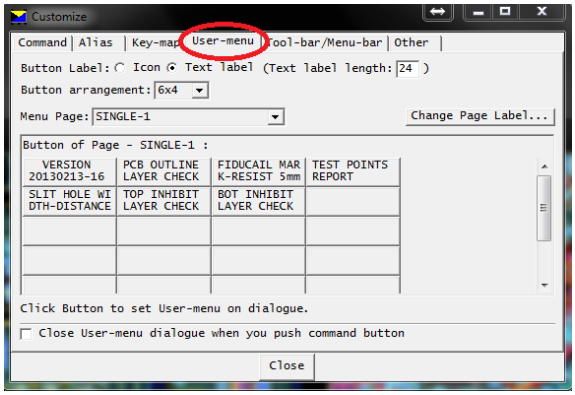


Image 4.3. Implementation log file 2

Zuken cr-5000 provides many kinds of settings for changing the look-friendly features such as menus.

1. Button label: User can change the look of the button to activate the function of both the user-menu icon or the text whose length can be determined as needed.
2. Arrangement Button: Users can change the size of user-menu table with length x width as 6x4, 24x1, 8x3 etc.
3. Menu Page: Users can choose on which part of the function log file will be saved, in this section there is the option Single-1, Single-2, Multi 1, Multi 2 wherein the single means for single-layer PCB inspection and meaningful Multi examination will carried out on double-layer PCB.

To enter the log file that was created in order to run the application Zuken cr-5000, then the first

thing to do is to choose on the Menu Page types of PCBs in accordance with the error-checking function has been created. In this case the log file is created utuk single-layer PCB then click on an empty field to add new functionality to the user-menu features.

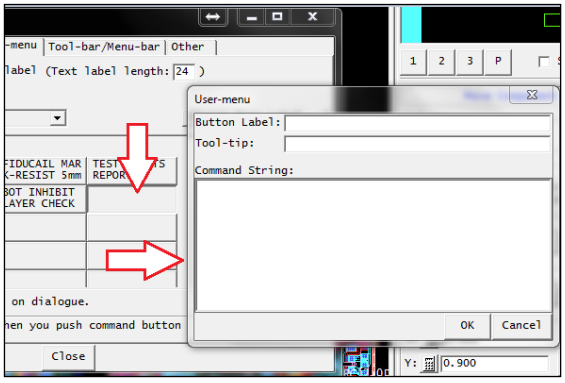


Image 4.4. Implementation *log file* 3

After clicking on an empty column will display a dialog box "User-menu" then on the Button label enter the name of an error check function to be added and on the part of the command string enter the command to open the log file that was stored on the computer directories by writing the command:

)(playbacklog filepath:"C:\\bd_setup\\USER-MENU\\Single-Layer\\Residual_Soldering.log")

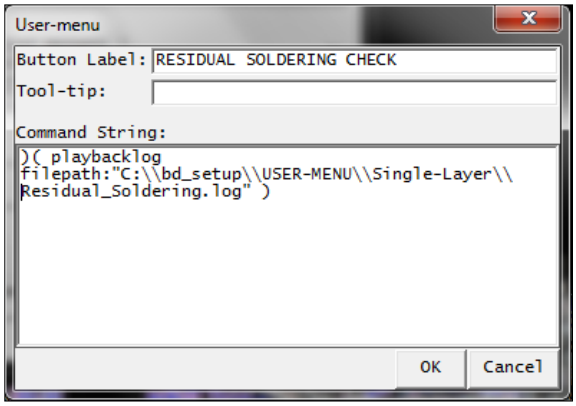


Image 4.5. Implementation *log file* 4

4.2 Log File Test

Test plan log file is a strategy of the documentation used to verify and ensure that checks errors in PCB layout design which has been made in accordance with the specifications that have already been designed. In the test plan log files, there are two stages of the testing plan is a plan of alpha testing, and beta testing.

1. Alpha Test Plan

Functionality testing plan is a process of testing the functionality of systems using alpha method, where the testing focused on the outputs generated in response to the production and design rule parameters selected conditions. Alpha testing plan itself can be seen in Table 4.1.**Tabel 4.1**. Rencana Pengujian *Alpha*

Parameter Test	Testing Details	Test Type
Symbol Mark - Resist	1. *Residual soldering on the body of the component symbol* 2. *Residual soldering the components name reference*	*Blackbox*
The absence of residual soldering on SMD component pad	1. Removing any of the existing residual soldering in the SMD components	*Blackbox*
Placement of residual soldering wrong	1. *Residual soldering resides in the body of the component*	*Blackbox*

2. Beta Test Plan

Beta testing plan will be carried out through interviews to users of Zuken CR-5000 applications are employees UKI Cooperation. This interview consists of several questions that support the ultimate goal of examiner error in the log file created.

4.2.1 Alpha Test

Alpha testing is done using black box method. To determine the alpha testing. As seen below.

1. Residual Soldering Test against Symbol Mark

Image 4.6. Testing residual soldering 1 and Image 4.7. Testing residual soldering 2 is a testing scenario design rule manufacturing or called MRC were performed on the residual soldering with the symbol mark.

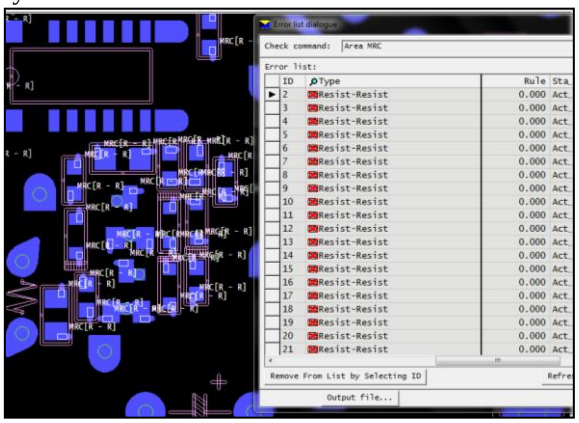


Image 4.6. *residual soldering test* 1

The test is continued by placing the reference name that touched residual component or element

soldering resist. Then the error-list display data with which to design errors occur because there is a component symbol or reference name that is covered by the residual component soldering.

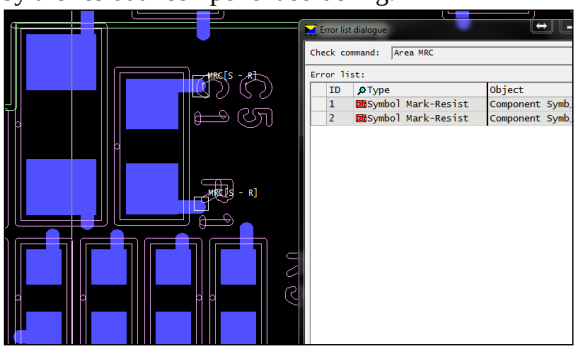


Image 4.7. *residual soldering* test 2

Table 4.2. *symbol mark - resist* test

Cases and test results (normal data)			
Parameter Test	**Be expected**	**Observation**	**Conclusion**
Symbol Mark to *Resist*	Can display an error on the part resist is about symbols in the design of PCB	MRC-error can display on the resist is in contact with a component symbol	[√] Accepted [] Rejected
Cases and test results (wrong data)			
Parameter Test	**Be expected**	**Observation**	**Conclusion**
Reference name of the component to Resist	There are errors due to residual soldering components so that reference cover the reference name should be moved	Showing errors on the error-list	[√] Accepted [] Rejected

1. Test The Absence Of Residual Soldering On SMD Component Pad

Image 4.8. 3 soldering residual testing was scenario testing done by removing the residual soldering pad SMD components.

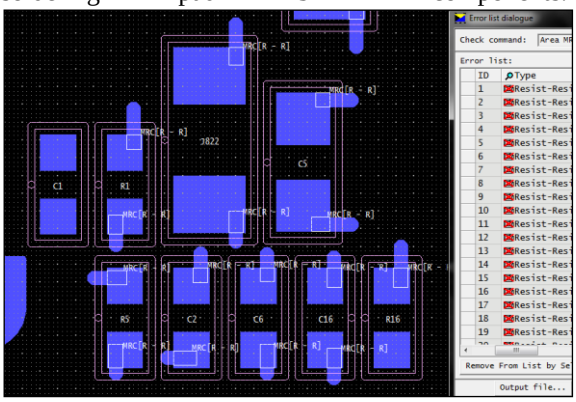


Image 4.8. *residual soldering test* 3

Table 4.3. Testing absence of residual soldering

Cases and test results (normal data)			
Parameter Test	**Be expected**	**Observation**	**Conclusion**
Remove the residual soldering pad SMD components	Can provide information on the components that do not have residual soldering	Residual component by soldering shown as error (false error) and no residual components that have no visible soldering MRC-error. The results can be used as an information and in accordance with the diharpkan	[√] accepted [ ] Rejected

2. Wrong Placement Of Residual Soldering

Image 4.9. Testing residual soldering 4 was scenario testing conducted by placing the body in the residual soldering SMD components.

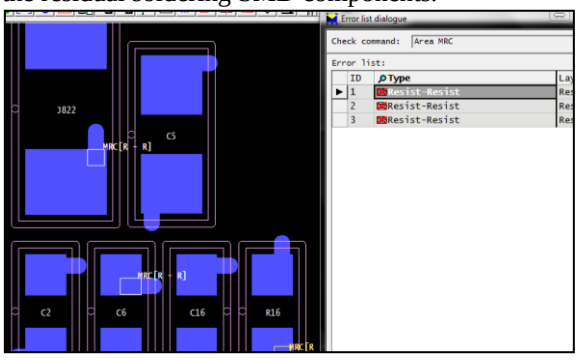


Image 4.9. *residual soldering test* 4

Table 4.4. Placement test of *residual soldering*

Cases and test results (normal data)			
Parameter	**Be expected**	**Observation**	**Conclusion**

Test			
Placement of residual soldering wrong	Can provide information on the residual component by soldering wrong position	Having traced and disable MRC-error by changing the action on the error-list to false error then do the filtering of the position *residual soldering* yang salah	[√] Accepted [ ] Rejected

4.2.2 Conclusion Alpha Test

Based on test results logfile residual soldering has been done on the whole, it can be deduced that the process of inspection of an application bug ZUKEN CR-5000 has experienced an increasing process, and already maximize these processes. Functionally logfile can already be used, and produce output as expected.

4.2.3 *Beta Test*

Beta testing is done on a user who is a PCB designer to use applications ZUKEN CR-5000. In beta testing, conducted research directly to the user application by using the interview to the PCB designer part of the function of residual soldering logfile that have been made.

Interviews were conducted to one company employee ZUKEN PCB design with CR-5000. Here are the results of the interview can be seen as follows:

1. Usually any errors that often occur at the design stage?
   Answer: "Typically less accuracy when checking, so a lot of things that disrupt the production process."
2. Are additional user-menu functions on ZUKEN CR-5000 PCB design adds to the performance results you?
   Answer: "Yes, I think the more complex the system checks for errors then the resulting design will get better."
3. Is the error on the residual functionality soldering inspection is in accordance with the needs of PCB design?
   Answer: "I think it's very pretty to maximize the error-checking process. "

4.2.4 Conclusion Beta Test

Based on the results of the beta test, it can be concluded that the addition of residual functions on the user-menu soldering applications ZUKEN CR-5000 PCB can help designers to maximize the work of the design so as to minimize the failure of the PCB production process.

5 CONCLUSION AND SUGGESTION

5.1 CONCLUSION

Based on the analysis and testing that has been done in previous chapters shows that the complexity of the examination error on any part of the PCB layout design is influenced by the use of the MRC parameters and grouping the data layer is activated or deactivated so that the inspection process can be optimized. Functions created have been able to meet the objectives of the research are as follows:

1. Residual Soldering Function Check can maximize the results of the PCB design and reduces production failures.

2. Knowing the manufacture logfile and create error-checking function more optimally.

5.2 Suggestion

The author gives suggestions for the development of user features menu on the application ZUKEN CR-5000 in the future, ie add-checking function fault on the part resist which has a range too close to the test point and can display an error if the distance between the residual soldering too close.

BIBLIOGRAPHY

[1] Zuken Incorporated, *Beginner's Training PCB Design*, 2007.

[2] *CAM (Computer-aided manufacturing)*, http://ahtovicblogs.blog.ugm.ac.id/?p=80, 17 Maret 2017 20.25.

[3] Dadang Fauzi, *ARTI DAN DEFINISI SERTA PERBEDAAN CAD, CAM DAN CAE*, http://www.artikelcad.com/2014/09/arti-dari-definisi-dan-perbedaan-cad.html, 17 Maret 2017 19.30.

[4] SELC - Sumber Elektronik, *Printed Circuit Board ---- PCB ---- Prototype*, http://www.selc-komponen.co.id/index.php?route=information/information&information_id=7, 18 Maret 2017 20.45

[5] Altium, *Module 12: Design Rules*, https://www.ece.ufl.edu/sites/default/files/pictures/Module%2012%20-%20Design%20Rules.pdf, 18 Maret 2017 21.57

[6] Jason Howie, *Design Rule Checking*, *https://techdocs.altium.com/display/ADOH/Design+Rule+Checking*, 19 Maret 2017 19.07