CSiEDA
EDA SOLUTION

CSiEDA as the brand.

The goal is to have the No.1 solution and supporting systems.

In 2007, CSiJapan announce the new feature to the CSiEDA world. It is called .Net Express and it corroborates with Internet. There have been the huge issues like collecting data for developing circuit boards, so that it took the boundary out from the application and the network. Web services, modeling, and the fusion of EDA are worked together. This system allows you to work on designing and writing documents, and search various data on the Internet on the same screen at the same time. All databases such as quality control, parts management and the others can be reflected in CAD in real time. CSiEDA made it possible to collaborate all the effective systems to reduce lead-time. Now, we will develop the system, which does electronic design and industrial design. Our goal is to be able to offer you a new total solution system, CSiEDA .NET Express, with our achievements and skilled support system as a backbone.

President Masahiro Kato

Total Solution
CSiEDA

CIRCUIT DESIGN

WinSchematic®

[Electronic schematic designer]

SPECIFICATION

Includes…
- Japanese, Korean, and English language
- Windows Fonts
- VB/JAVA/SQL Database and Dynamic Links
- Frame Wizard
- Pin editor
  Pin shapes, names, and numbers creator and scaling tool
- Symbol designer
  Gate library parts creator
  Pin wizard
  Copying the symbols
  Mirroring the symbol
- Library Manager
- Power parts save-up system
- The function to move the original point arbitrary
- Pin editing function on Schematic Designer
- Annotate function (Auto Update for reference numbers)

Able to…
- Customize with JavaScript VB
- Customize menu bar
- Customize Icon
- Use divided connector
- Set PCB parts for top and bottom side
- Cooperate with CSiEDA Viewer
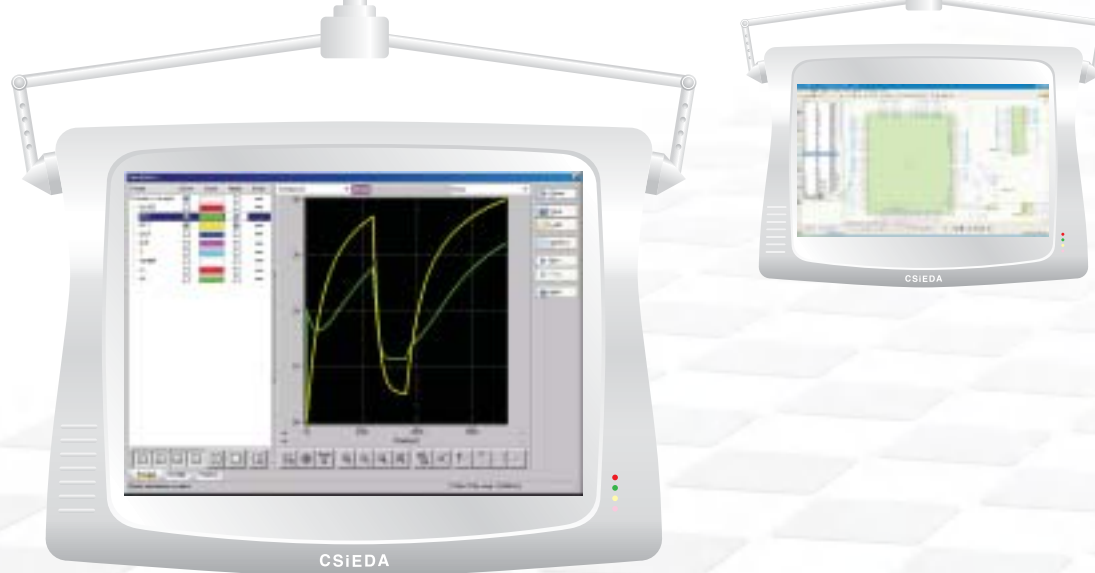
CSiEDA

**All essential tools for EDA
are integrated seamlessly.**

C S i J a p a n

CSiEDA-WinSchematic is the advanced operative software and it offers you the perfect environment for designing circuit diagrams. This software has the powerful project management tool and the dividing system to improve designing. For the theme [-40%Project] to come true, schematic and PCB designer are integrated seamlessly. Until now, it may have taken a long time to edit and check the circuit design. However, many great functions, including the auto-routing system, are now available and it helps engineers to reflect their intentions to their work and accelerate to design.

CIRCUIT DESIGN

WinSchematic®
[Electronic schematic designer]

Changing the parts size arbitrary
When the diagram is too intense to put any more parts, you need to change the size of the part. However, it is not necessary to go back to the Library editor with CSiEDA. It is possible to change the size as you like in schematic editor. Moreover, the width of the line and fonts also can be edited here.

Auto cable cutting function
When a part is necessarily added on the connected cable, just clicking on the cable places a part and connects it to cable automatically. Cables do not need to be deleted and reconnect to put an additional part.

Creating a new part on the schematic designer
It is possible to create and save a new part and edit on the schematic designer.That means designers do not need to go back to the annoying stages, such as drawing shapes in symbol designer and putting values in library designer. It is very useful and easy to use.
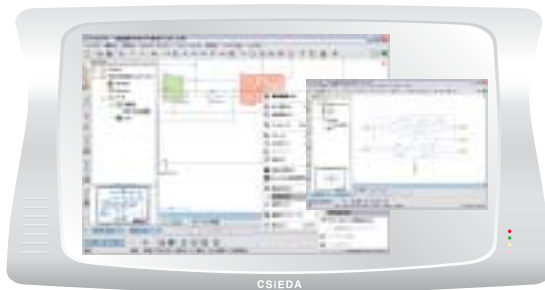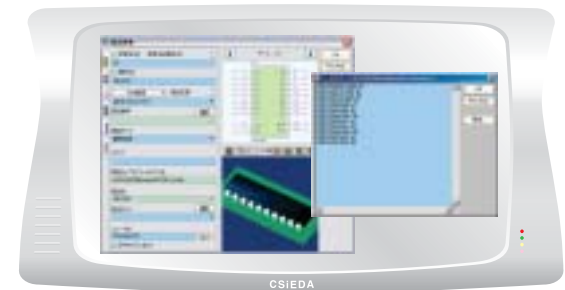
Auto port address creating function
It will auto-create the page number and address to manage a project with multi paged circuit diagrams. It helps to check the schematics in a short time. Searching, Updating the port name, deleting, and printing all ports in all pages can be done at once. The result will be shown in the bottom of the screen and it leads to the points.
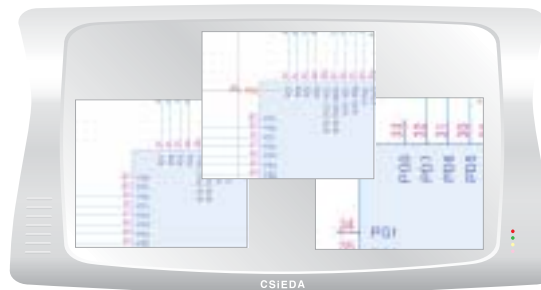
Moving Pins on the schematic designer
It is possible to move any pins that already placed in the circuit diagrams. No need to open the symbol or library editor to fix the position of the pins. You can move, and change the shape, numbers, and name. This schematic editor is very flexible and easy to use.
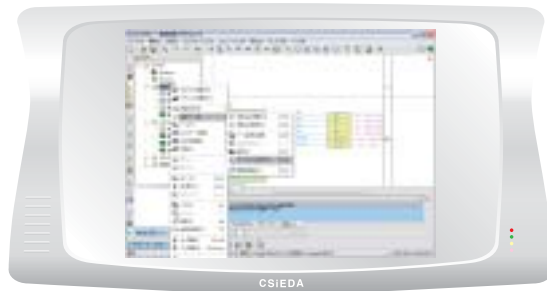
Brilliant part attribute setting
1. While a part for schematic is being created,
   the PCB part can be also specified.
2. Attribute value for Spice also managed here together.
3. It is able to put comments on the parts.
It is possible to specify the shape of PCB part for each top and bottom side.This data of course will be transferred to PCB designer.

The port address auto-create function

Managing the project with multi paged circuit diagrams, it will auto-create the page number and address. It helps to check the schematics in a short time. Searching, Updating the port name, deleting, and printing all ports in all pages can be done at once. The result of search will be shown in the bottom of the screen and it leads to the points.

Great drafting function for circles, arcs, and lines

Circles, arcs, and lines are easy to make. To create a very complicated shapes, use [Bisect selected Items] [Trimming] [polygon] functions.
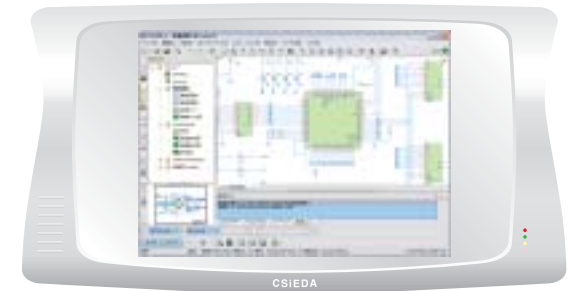
Project management

It is possible to manage the multi-paged schematics with the components. [Search], [print], [Design check] for all schematic files in the project will be done at once. The result of Design check will be shown in the bottom of screen and it leads to the points.

The extension [importing and exporting DXF files]

WinSchematic can import and export DXF files. The imported DXF file can be used as lines. It has the capture function as well, so that it is possible to save the screen image as BMP/WMF. It is effective for the for specifications and so on.

Exporting the list of parts and the Netlist

When the circuit diagram is all finished, the final stages of the designing are to initialize and update references, to check the design, and to export the list of the parts and the netlist. Errors that detected during the design check, the error messages will be shown in the bottom of the screen and if you click on the message, it will zoom in to the points.

Creating the part list

It is possible to adjust the data and export the list as Excel, HTML, XML files. If you have a contract with CSiJapan support system, CSi will create the list with your own list format. By just importing the file to the list format, then it will automatically creates your own style part list.

4

High performance circuit simulation

WinSpice, the powerful analog and digital simulator, has XSpice engine that allows you to analyze the diagrams accurately.
It cooperated with WinSchematic smoothly, so it cuts down the designers' extra work.

Win Spice
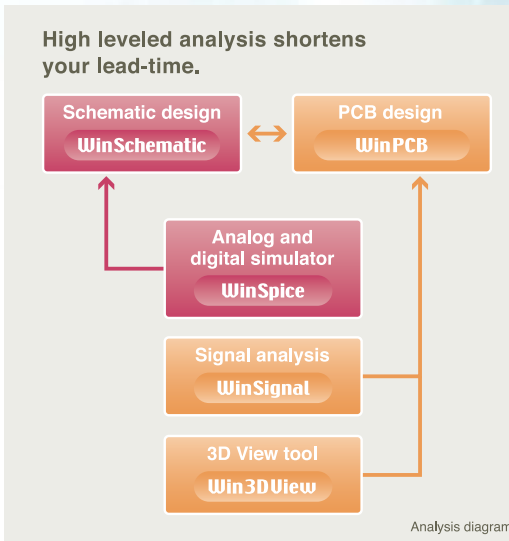
[Analog and digital simulator]

What is WinSpice?

This software is to simulate how the electrical circuit works on the computer without making a mock up boards. Spice used to be inconvenient software because schematic design and analysis tools used to work individually as different tools. However, WinSpice is the plug-in system for WinSchematic, so it works on the same screen and it is not problem to go back to design and analyze the data again. This single unified design environment dramatically improves speed and cost to check. No longer money and extra effort are needed as it used to be.
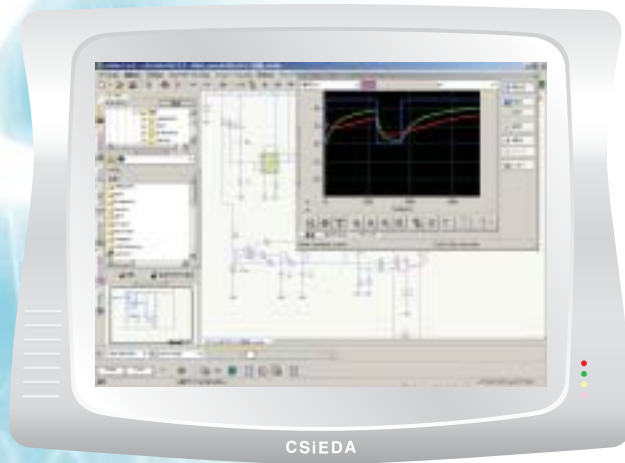
Very easy use

Setting and executing simulation, and displaying the result are all in WinSchematic. WinSpice can simulate both analog and digital analysis and it recognize the interface between analog and digital so that it can insert interface· sub-circuit automatically. It can simulate Excessive, DC, and AC analysis and more. You can design in WinSchematic and set the value in the parts for simulation. At first, you have to set the analysis environment and specify the model of the parts from the library. There are customizable basic models and macro already. At the end, set the parameter for each analysis and set the probes.

Probe

In WinSpice, It is possible to display the result of the analysis with a waveform graph. By placing the probes on the schematic design that you are working on, you can view the waves of voltage levels at that point. WinSchematic and Probe is linked in real time so that by changing the position of the probe will show the changes in the simulation.

High leveled analysis shortens your lead-time.

Schematic design		PCB design
WinSchematic	↔	WinPCB

Analog and digital simulator
WinSpice

Signal analysis
WinSignal

3D View tool
Win3DView

Analysis diagram

Excessive analysis

This analysis is to analyze what kind of waves will be exported as in a certain time when the arbitrary waveform was input. For example, it will search the out put waveform when the wave is set as square, sine, and triangle wave. And more, it will test how it will respond when pulse is entered. It is possible to enter DC power and arbitral wave and it adopts line shaped parts and non-line shaped parts
*It is the same way to measure with the oscilloscope and FFT

Basic analysis functions

Excessive analysis	Operating analysis
DC analysis	Noise analysis
AC analysis	Sensitivity analysis

Operating analysis functions

|	
Monte Carlo analysis	Parametrical analysis
Enviroment setting function	
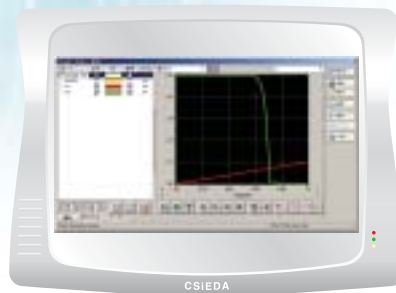
DC analysis

DC analysis is to analyze each voltage and current node when the voltage of DC power device is changed. It is the same thing as the operating point analysis to measure the value of voltage, but the operating point analysis can be done only for once for the circuit. But DC analysis is to analyze direct-current accompanied DC power fluctuation, and shows it with a wave.
*It is the same as Decibel, measuring with X-Y plotter.

Monte Carlo Analysis

It gives the random value to the parts and analyzes them repeatedly. After the data was analyzed several times, the result will be displayed in a density histogram.
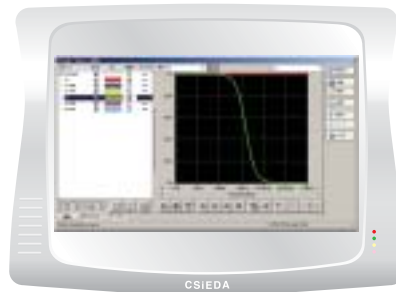
AC analysis

AC analysis is to see the change of the sine wave when the wave was input into the circuit. It means the same thing as the calculating with the spectrum analyzer.
*Same as measuring with the analyzer.

Parametrical Analysis

The value and temperature of the parts become parameters and displayed in several graphs and see how the frequency will be changed.

6

PRINTED WIRING BOARD DESIGN

WinPCB®

[Printed circuit designer]

SPECIFICATION

Includes...
- Japanese, Korean, and English language
- Windows Fonts
- VB/JAVA/SQL Database and Dynamic Links support
- Gate swap function
- Pad stack function
- Via stack function
- 1024 layers
- Auto measuring system
- Outline view mode on each layer

Able to...
- Import IGES and STEP 3D data
- Import/export DXF and DWG files
- Stabilize layers
- Preview printed image
- Put 128 letters for names and numbers
- Cooperate with CSiEDA Viewer

CSiEDA

PRINTED WIRING BOARD DESIGN

C S i J a p a n

This is the brilliant editing function
# that reduces designers'stress.

CSiEDA-WinPCB made it possible for Printed circuit design to be approached easier than ever before. This is the next generation type of PCB editor because it has improved the weakness and it has just full of excellence. It is not only connected to Schematic designer tool seamlessly but also has the great functions to design any kinds of circuit boards, such as analog, flexible, and power boards, and more complex boards and so on. Furthermore, it can be controlled by design rules. Using online design rule check helps to check if error is occurred while engineers are designing.   And also, it has auto-designing system with the detailed design parameter so that it helps to design faster than before. On the manufacturing side, Elemeca integrate designing system is the biggest assignment for any CAD, but it has already accomplished with CSiEDA. WinPCB totally helps to solve all companies' subject, an efficient work.

PRINTED WIRING BOARD DESIGN

PRINTED WIRING BOARD DESIGN

PRINTED WIRING BOARD DESIGN

PRINTED WIRING BOARD DESIGN

WinPCB®
[Printed circuit designer]
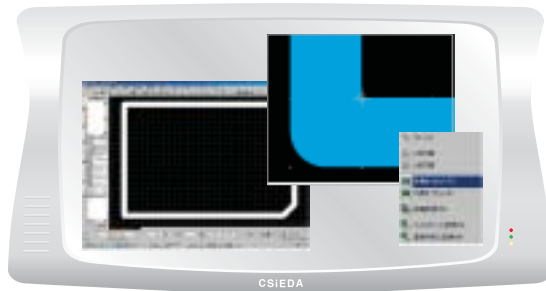
Board Outline
It is very easy to create board outline because there is a scaling system. After creating one, you can change the scale by using this function.


Package Builder
Usually, it is very hard to make PCB parts. CSiEDA has a tool that creates PCB parts by just entering the numbers of components size. The types are [DIP] ,[SOP],[QFP],[PGA/BGA],[PLCC].


Padstack [customizable]
Every pad can have information in each layer. A very complicated shaped pad can be created as a customized pad. It just has to be drawn as a copper and converted the shape to a land shape. It is very easy to make. And more, a strange shaped pad can be made automatically by a numeric input because it is cooperated with Java.
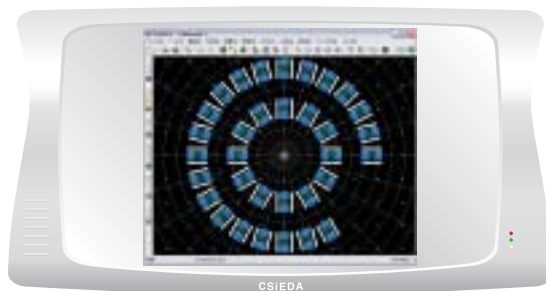

Brilliant arrangement function
Parts, placed arbitrarily, can be coordinated in specific pitch automatically by entering the values for X and Y. The polar grid is available if you want to place the parts radially. The parts will be pulled into the grids and automatically change the angle to fit. Custom grid also can be created, so that, it is very effective when a part must be placed in a very intense design.
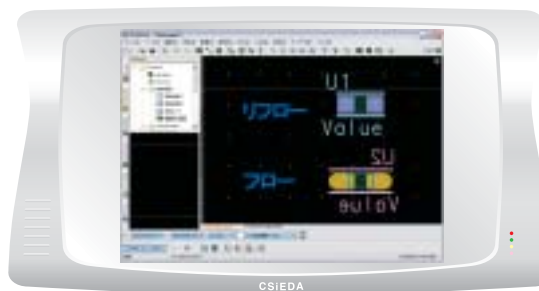
Several parts can be set in one library.
Since there are some parts that need several shapes, this software has this function that store several parts in one library; alternative package. Not only alternative package, you can also set different parts for top and bottom side. When the part is moved to the bottom layer from top layer, it will automatically switch to the one that has been set.
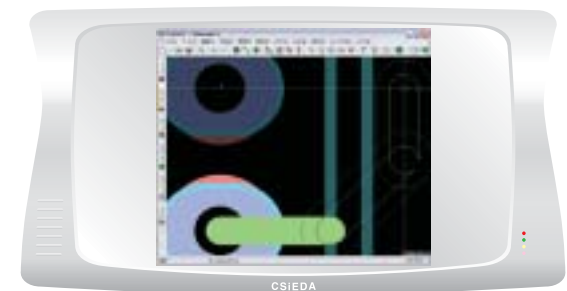
Routing
It is possible to see the clearance around the route, which was set in the design rule. It shows the clearance with dot line, so that it will be very effective to route around BGA and sensitive parts. Clearance for copper, pad and pattern can be shown.

9

Route shield

It allows you to shield for the noise easily. It is just select the route which you want to put the shied to and set the value for clearance, width of the shield and the net name and then it will automatically create shields.

Parallel routing and drawing lines

Parallel routing is very useful to route bundle on a board or creating a flexible boards. It is possible to set the clearance to each routes before routing, and running with the same distance between them at every point. And also, Import the DXF as lines and convert the line to patterns.

Routing with the same length as other route line.

While routing, it not only checks the length of the route and also changes the length after routing, so that, you can route in a very little time. You can see the route length as [Segment],[Pin to Pin],[Net]. And more, by setting the number and length of the patterns, the patterns will be created and adjusted automatically as they are set.

Divisional Designing

WinPCB has the environment that several people can work on one board design. When the board design is big but it must finish in a little of time, this function will help. Several people can work and those designs can be combined at the end.
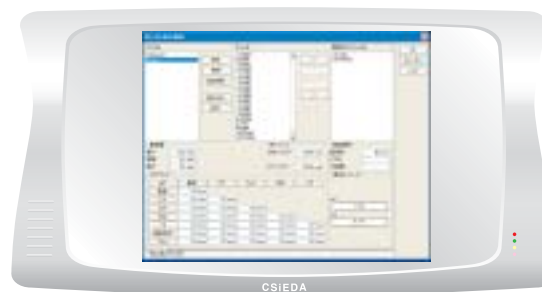
Brilliant DRC function

It is possible to set the area of DRC (Design Rule Check) dynamically in WinPCB. The rules for width and clearance of the route can be set among each area and vias and patterns can be change by the rules. In addition to the former DRC check function, it can also check designs and manufactures. The Design rule can be set in each component and also the net.
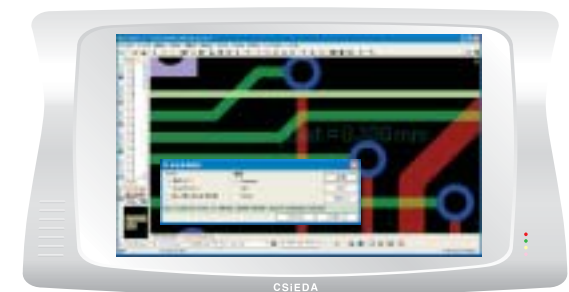
### The shortest distance between two patterns

There are tools called [check minimum distance] and [check minimum distance batch]. Of course, you could check the shortest distance with DRC, but these tools show the shortest distance between two patterns with measure. You can measure between center-to-center or edge-to-edge of the patterns. For patterns, you can select [Segment],[Net],[Pin to Pin].

10

-40%project_CSiEDA

# Designers' thoughts will become real with vivid editing system.

CSiEDA has a lot of very easy-use editing functions for designers ideas to become to shape. It is not only for one-side boards but also complex multilayered boards. Since it has the user-interface that you can create the boards' outline and edit coppers and routes in one place, you can work faster and more effectively than ever before.
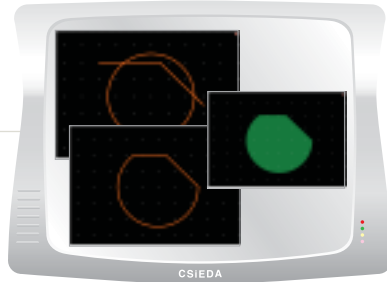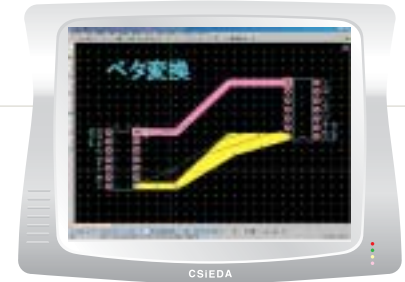
Shape break up function

Copper And /Or combine function
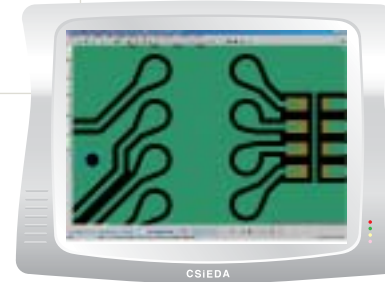
Poured copper

Enhancing and editing coppers

VARIEGATION

Editing coppers
By using the several segments and editing those few shapes, you can develop a shape that you imagine. And the shape you created can be converted to a copper, so it makes easy to edit the lands and copper. Moreover, there are functions that make copper to combine, cut, and the extract easily. Offset tool also makes it easy to create the lands and coppers for the complex design.

Enhancing
There are great functions to enhance coppers for the analog and power circuit boards. It includes "poured copper" which allows you to fill inside of the shape and make power and GND to be stronger. This copper, of course, connects to net by setting and also it automatically removes the island coppers. Furthermore, pattern can be converted to a copper, so that it free to thicken the lines and change the shape.

Free to adjust
There is a function that changes the shape of line, patterns and copper free, so that designer can easily make the design the way they want. For example, if you use break-up tool and offset tool then it is possible to make a very complicated shape pad and save it in the pad stack to share. It actually helps to short your time to create a very complex shape such as dotted lines or bunch of lines together by using the trimming function.

Adjusting
To adjust after creating the route line and normal lines, you can use the function that adjusts the shapes at once. To route on a flexible board, Parallel routing function will be very efficient. You can use the Trimming function on parallel lines and take out the corner of the lines at once. You could create the custom snap lines as well, so that it will help you to place parts in a specific position and drawing the lines.

Routing/drawing
In WinPCB, it is the same way to create all board-outline, pattern, and copper. Align function allows you to design with great balance. Trimming and miter tool are effective for one or many lines so it is no problem to create a very complicated shape for a board outline. For routing on a flexible board also is no big deal since radius, angle and arc can be set.

- Select all component function
- Align function for points of lines
- Align function for lines
- Line trimming function
- Shape break up function
- Muter function
- Parallel routing function
- Customize snap
- Pattern to copper, copper to pattern function
- Copper AND/OR function
- Copper offset
- Copper editing function
- Poured copper
- Power plan function
- Auto island copper deleting function

11

-40%project_CSiEDA

Easy to create the manufacture drawing

It has been told that it takes a long time to create the layouts. WinPCB will compose all kinds of layouts automatically so that it does not take such a long time. We want you to work really easily, so that we ask you to request for the new function.

Height Diagram/Dimension
It will automatically create the height diagram by entering the information. For the board outline, the dimension can be input easily. And more, it has OLE function so that you can insert a data like Excel file on the CSiEDA.

Enlargement Drawing
The detailed draft can be created in the PCB design. You can place the specific area of a diagram, magnified, anywhere you want, so it helps you to write a note in PCB file.

AUTOMATION
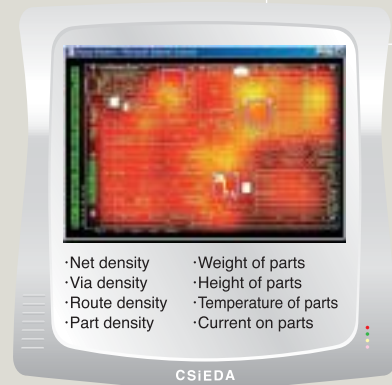
Density Analysis
It is possible to input those data such as density of the Net, Route, Via, Part and so on. Based on these data, it will show the information like the picture above. It is also possible to set the temperature for the parts and show it here.

·Net density   ·Weight of parts
·Via density   ·Height of parts
·Route density ·Temperature of parts
·Part density ·Current on parts

Outline View Mode
It is possible to show everything in outline view mode instead of solid lines. And it can be printed out in this mode. It is very useful to see connected point of pads and patterns. It is completely displayed as outlines.

-40%project_CSiEDA

Easy to export & edit very complex CAM

It is possible to edit the manufacturing data, which was created with any CAD systems. It has the great function such as paneling, mirroring, searching for the aperture data, measuring, deleting unused pads and so on. It is no problem to import this edited file here to the PCB editor, WinPCB. This feature totally helps you to use the old data even though the data was made with the other CADs. It exports in 274D and 274X format.

Able to change Gerber data to PCB data.
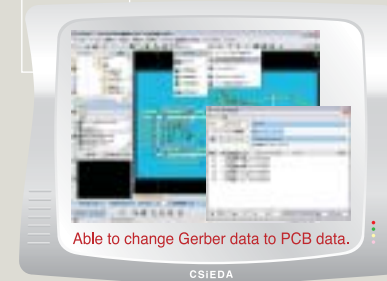
Exporting CAM
WinPCB and WinGerber are collaborated so the manufacturing data can be checked in WinGerber. There are two ways to export the data. One is exporting the data individually, and another is exporting batch like one side, both side, 4 layers, 6 layers... You can export the many data at once.

Paneling (different kind/individually)
After you finish designing the PCB, the data can be flipped over and copied. Auto panelize function helps you to put the panels in order immediately. The data for the panel is saved as a link so that the size of the file would not be so as big as it might be, so it does not take a long time to load
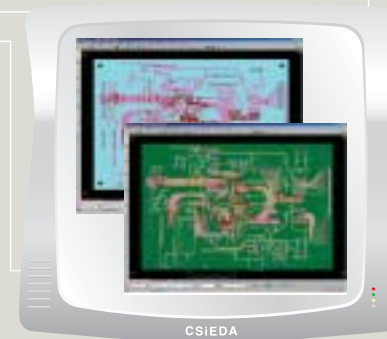
Gerber In   *WinGerber*
Not only the Gerber data, exported from WinPCB but also the data from the other cad also can be read in this software. Paneling, editing the D code, and Gerber can be done here, and also the Gerber data can be converted to the PCB data with this tool. This is very efficient that the old data from the other cads can be used.
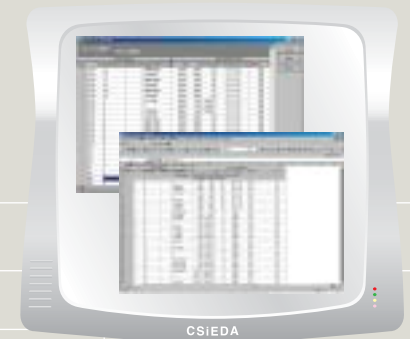
Many kinds of the report can be exported
During designing, the information of the parts, Pads, Vias, and routes can be exported and make it as a list. The list is customizable and sorted out as you like and it can be exported as Excel or HTML file. There is an original point manager so many points can be placed.

SIMULATION SYSTEM

PRINTED WIRING BOAD DESIGN

At the first step of designing, solve the EMC/EMI problems.

Those problems during designing such as high dense, high crock, slowing down because of high speed, impedance, reflection, crosstalk, are not only for special design. It is for all the designs. With WinSignal, you will be able to check and analyze the EMC, EMI and microwave problems from early stage Since you do not need to create a mock ups so you are able to produce in a short time before other companies does.

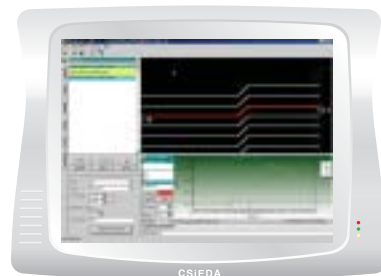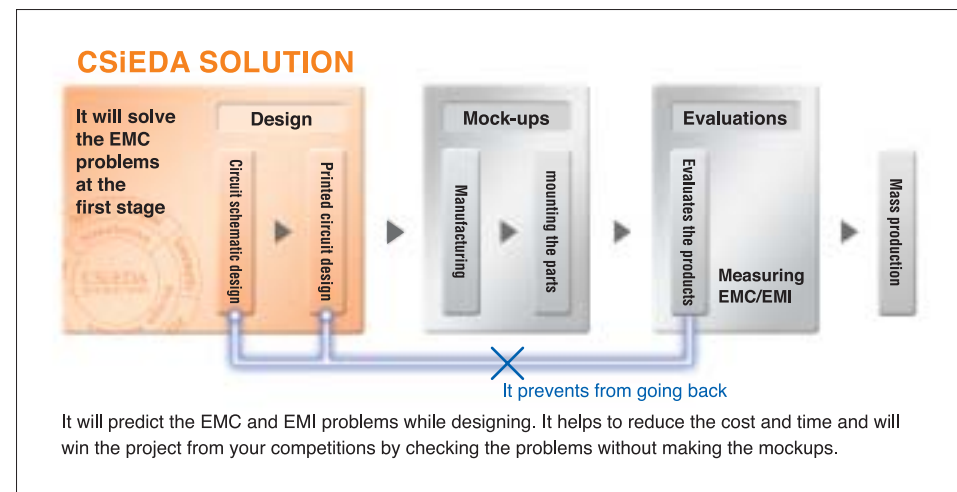## Win Signal
[Electric flow analyzer]

Analysis
WinSignal can create the situation of the pattern on a board, calculates the impedance, and use the information of I/O buffer macro level. This simulator supports IBIS model, which means any models from any device maker, can be used. Furthermore. It can test any kind of terminal environment, and it will find the best terminal resistance by considering waves on the specific node.

Displaying the waves
WinSignal has the function that shows the reflection and the result of the crosstalk analysis in waveform. Not only it shows the result as a wave but also the wave is free to scale up and down, so that it is very easy to see the details. Moreover, it is possible to change the colour for each 14 channels probe and the result can be pasted to other applications.

CSiEDA SOLUTION

It will solve the EMC problems at the first stage

Design
- Circuit schematic design
- Printed circuit design

Mock-ups
- Manufacturing
- mounting the parts

Evaluations
- Evaluates the products
- Measuring EMC/EMI

Mass production

It prevents from going back

It will predict the EMC and EMI problems while designing. It helps to reduce the cost and time and will win the project from your competitions by checking the problems without making the mockups.
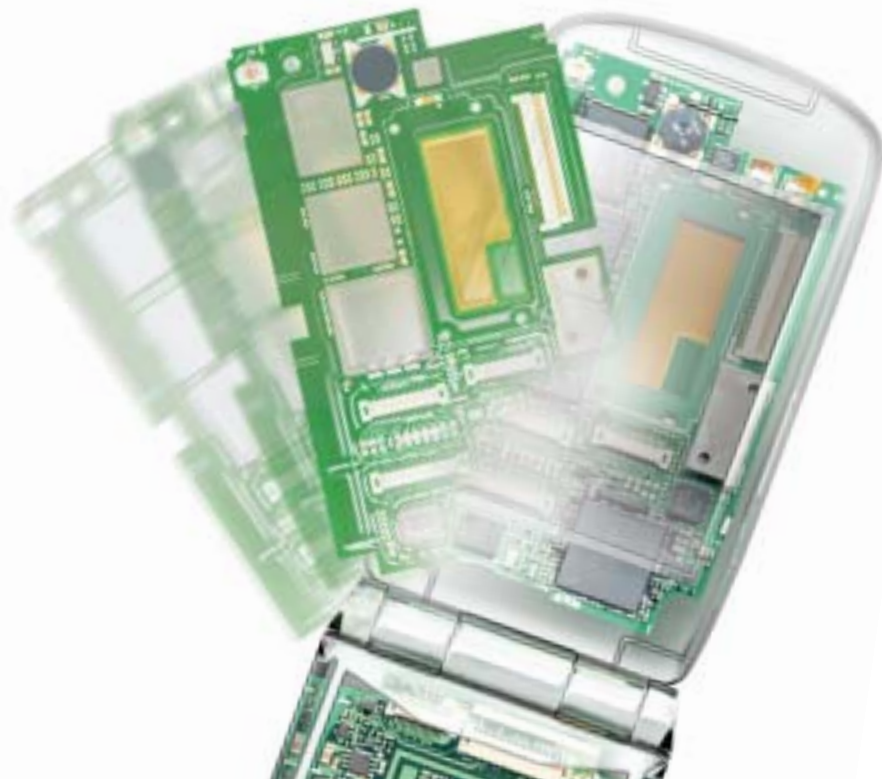
13

-40%project_CSiEDA

Collaboration of electrical and mechanical design

You will be able to see the lead-time is shortened dramatically.

SolidWorks Solution Partner

CsiJapan and SolidWorks Japan have solution partnership for Elemeca design.
*Elemeca : electrical designs and mechanical design.

Win3DView®
[3Dimensional viewer]

It seamlessly integrates the printed circuit board design and case design. The circuit design will be automatically become 3D harness design and the data is possible to be exported as I-GES/STEP and DXF. Those files are easily read in the 3D tools. It is no problem to see how the final design of the board and the cover interact. You can test all in 3D.
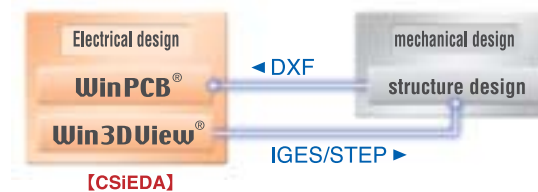
The mission
・No more handing in all the information like size, forbidden area and parts' height.
・No more going back and fixing the data, and make the mock-ups

If you use this feature then you will be able to...
1. submit the DXF data of the board when it is structuring.
2. submit the IGES/STEP data to see if the boards do not touch to the covers.

Cooperate with the 3D CAD system
By using Win3Dview, it allows you to submit the physical information as solid information to the 3D cad tools. It means that you can do till structure analysis on the 3D CAD, so that it allows you to save money to make trial products or shorten the lead-time.

Electrical design
WinPCB®
Win3DView®
【CSiEDA】
◄ DXF
mechanical design
structure design
IGES/STEP ►

【CSiEDA】 + 【3D CAD】 = 【combined】

14

This is the system that manages the entire design file and parts with the enhanced security system.

CSiEDA.NET Express
[Data managing system]

Net express is the data management system that constructed with those 5 systems.
LMS(Library Management System)
FMS(File Management System)
PMS(Part Management System)
DMS(Data Management System)
BMS(Bom Management System)

It is very effective to manage the very important information to process designing and manufacturing. The information of parts, libraries, design, and data can be checked in real time so that it will reduce the cost and speed. Till now, database of the material and electric design department was not connected, so that there was a time-lag. .Net express collaborate with the system that Microsoft supplies. It is very efficient to use the windows tools for designing. CSiJapan will support you with the idea to construct the system that very useful for you and construct the great database systems very cheap.
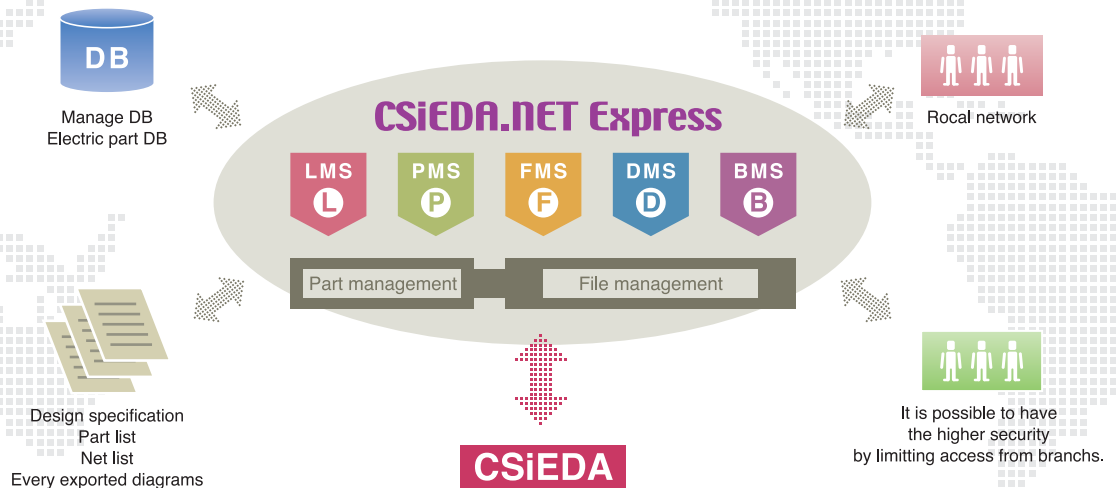
■Commutation with Chat is not on the program
The chat window is on the CSiEDA window and you can communicate with colleagues with instant message and talk about the project. It does not matter where you are. It will help for the cost down and you to concentrate on the project.

■User Customize using java *Visual Basic
Cad is not perfect for everything. However, it is possible to customize the function the way you like. BY customizing the program, it is possible to enhance the speed of designing.

■Access to the every kind of data base directly.
CSiEDA .NET Express allows you to access to the database related to materials. The parts can be added to the circuit design by looking at the information, such as stock, price, and priority. So, it will help the product to not cost much by allowing it to work with the part list, material information and circuit design all together

DB

Manage DB
Electric part DB

CSiEDA.NET Express

| LMS | PMS | FMS | DMS | BMS |
| L | P | F | D | B |

Part management   File management

Design specification
Part list
Net list
Every exported diagrams

CSiEDA

Rocal network

It is possible to have the higher security by limitting access from branchs.

By using the web server, material and design departments can communicate in real time. It is possible to access to the part library anywhere in Japan.
By this access and exclusive managing system, it is possible to manage the data from anywhere and data are able be edited by many people. .NET Express allows designers to work on the files, downloading the data and look at the latest library from anywhere and whenever. Part managing system, LMS and PMS, is cooperated with material database, so that just edit on the database can automatically updates the part information in the circuit designs. File management system, FMS, DMS and BMS, has the searching system to see when who is working on which files. This has a great security function so that it prevents from the data leaking to the outside of the company. Until now, it was very difficult to solve the problems, such as missing data, missed to be updated and so on, by managing the data and files. However, even though .NET Express is very simple and low price, it will help you to solve all those problems.

Electric part management system

LMS   Library Management System

You will be able to manage the entire libraries on the database in the company. If there is a change in database after you design, it will update the data in both design data and the database. There is the idea of "database = design data." For example, Part database and the library automatically compared, and update the information into the design data. By that, no longer it will take time to manufacturing.

PMS   Part management system

This is the management system that database and LMS are cooperated so that the data of parts can be added and searched here. You can also just pick up parts by its value, so you can check the cost, stocks and the priority of the parts while designing.

CSiEDA and database are linked in real time

If there is a change in the parts while designing, it will update automatically.

Part DB   Real time autoupdate   CSiEDA

WEB   SERVER   WEB

NewYork

L.A.   Boston
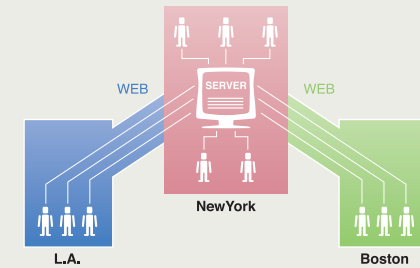
■PMS main features

- ・Access management
- ・Look at the database in the browser
- ・Edit, add and delete function in the browser *access pass is needed

- ・The data will be updated automatically
- ・Search several parts that have the same value in browser.
- ・Setting related file
- ・Setting security

File management system

FMS   File management system

Have you ever had problem managing the design files? All the design files will be saved in the server and all the files have attributes, such as the name of designer and project, and the date, so that it will be easy to look for. Moreover, the data will not be able to save in local without the approval from the manager. It will enhance the security in the company.
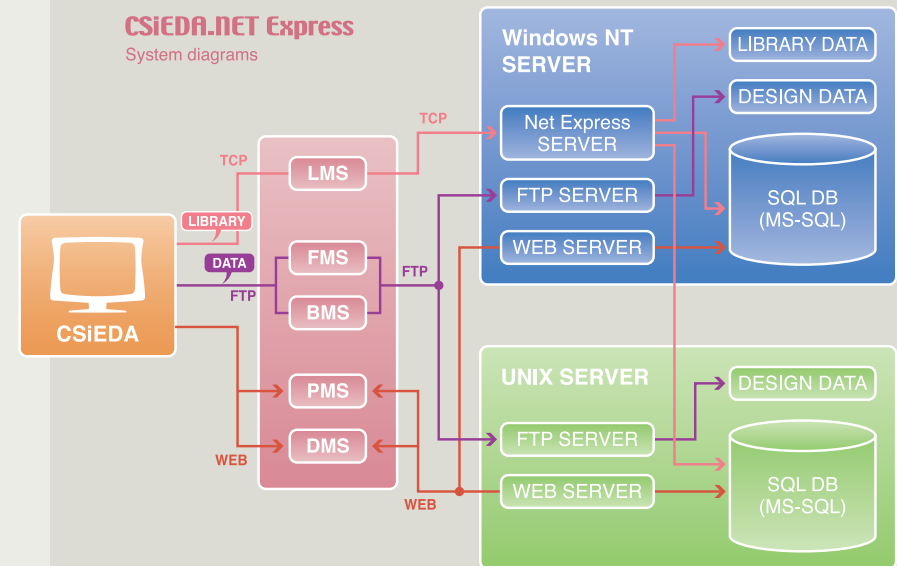
DMS / BMS   Data Management System / Bom Management System

Design data can be searched, added and checked in a browser. It allows you to check the data anywhere and whenever you like. It also has the very advanced security system that limit the access, so that it is not scary to share the data with the cooperate companies. It is also possible to manage the documents like part list in BMS.
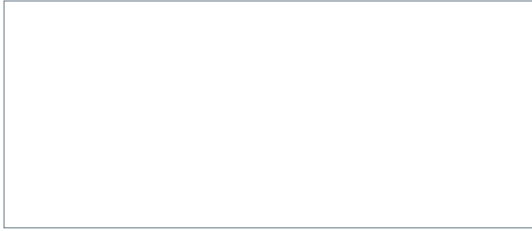
■FMS main features

- ・File access managing system
- ・Exclusive design file management system
- ・Exclusive project file management system
- ・Checking system of who is looking at the files.
- ・Time management for working on a file
- ・Security setting functions.

CSiEDA.NET Express
System diagrams

CSiEDA

LIBRARY
DATA

LMS
FMS
BMS
PMS
DMS

TCP
FTP
WEB

Windows NT SERVER

Net Express SERVER
FTP SERVER
WEB SERVER

LIBRARY DATA
DESIGN DATA
SQL DB (MS-SQL)

UNIX SERVER

FTP SERVER
WEB SERVER

DESIGN DATA
SQL DB (MS-SQL)

TCP
FTP
WEB

16

CSi Japan