Hand Routing

Hand Routing

What is hand routing?

Interconnecting a printed circuit without the aid of an automated connection tool

No currently available auto router can match a humans ability to plan and connect an electrical circuit

Auto routers always fail when attempting to connect high density low layer count designs

Introduction

This presentation is a collection of board layout techniques I have developed and learned over the last 14 years. Used together these techniques are a design process that reduces wasted time, effort and space. The majority of this presentation will focus on single sided assembly and 4-layer board construction. Multi-layer assemblies and higher layer count will be covered as well. An abbreviated overall design flow will be followed by detailed placement and routing strategy. General placement and routing tips will be covered thought out the presentation and listed at the end for easy review.

Introduction

Basic Concepts

- Starting from the ideal working toward the real

Typical Design Flow

Placement Flow

- Specific techniques for Part Placement

Routing Flow

- Specific techniques for Trace Routing

Questions

- Understand
- To relate
- Off the subject
- Why

Basic Concepts

SPACE MANAGEMENT

Parts (land patterns)

Traces

Vias

holes

Keep outs

Which of these does the physical designer have the most control over

Basic Concepts

- The most direct way to save time is to do tasks only once
- Critical to this objective is have all pertinent data when you start
- Understanding the data is equally as important
- Identify problems early
- Good communication

Typical Design Flow

Review incoming data

Build footprints

Set up Design Rules

Placement

Routing

Power plane splits

Checking

Reviews & sign off

Data output

Revise and repeat

Design Flow Placement

- Read and understand all design rules and guidelines
- Group parts by schematic or connections
- Place fixed location components
- Exploded placement based on connections
- Detailed placement
- Placement review
- Review by assembler
- Build mechanical verification board

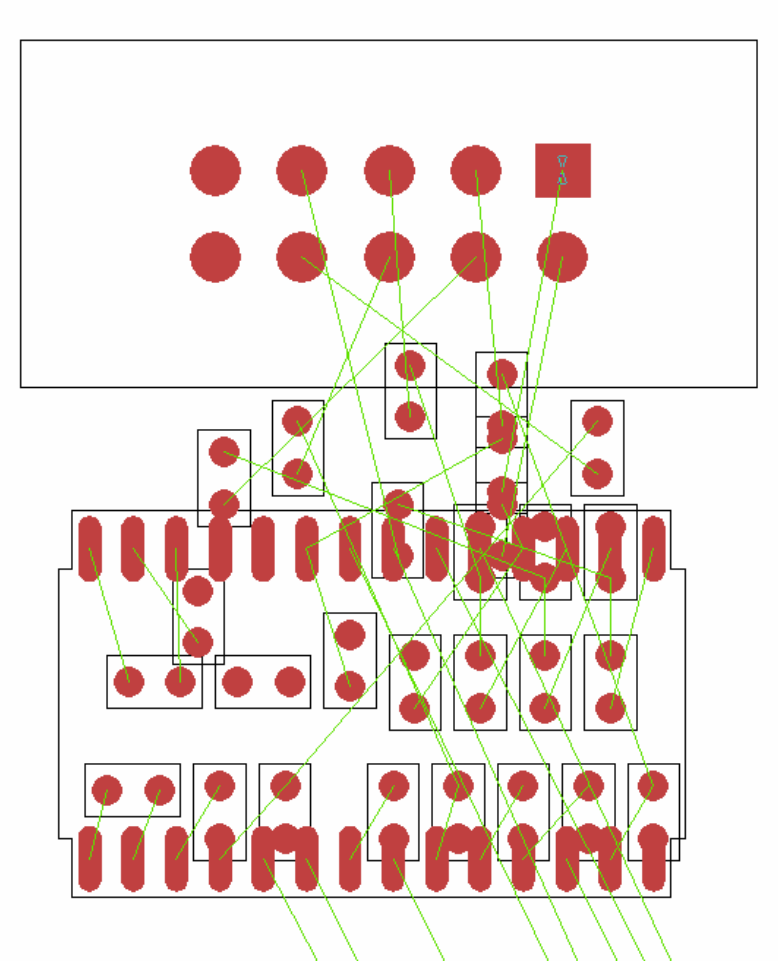
Understand Design Guidelines

- Placement restrictions
- Critical nets or circuits
- Timing rules
- Understand circuit operation
- Assembly process
- Fabrication process
- Point of contact

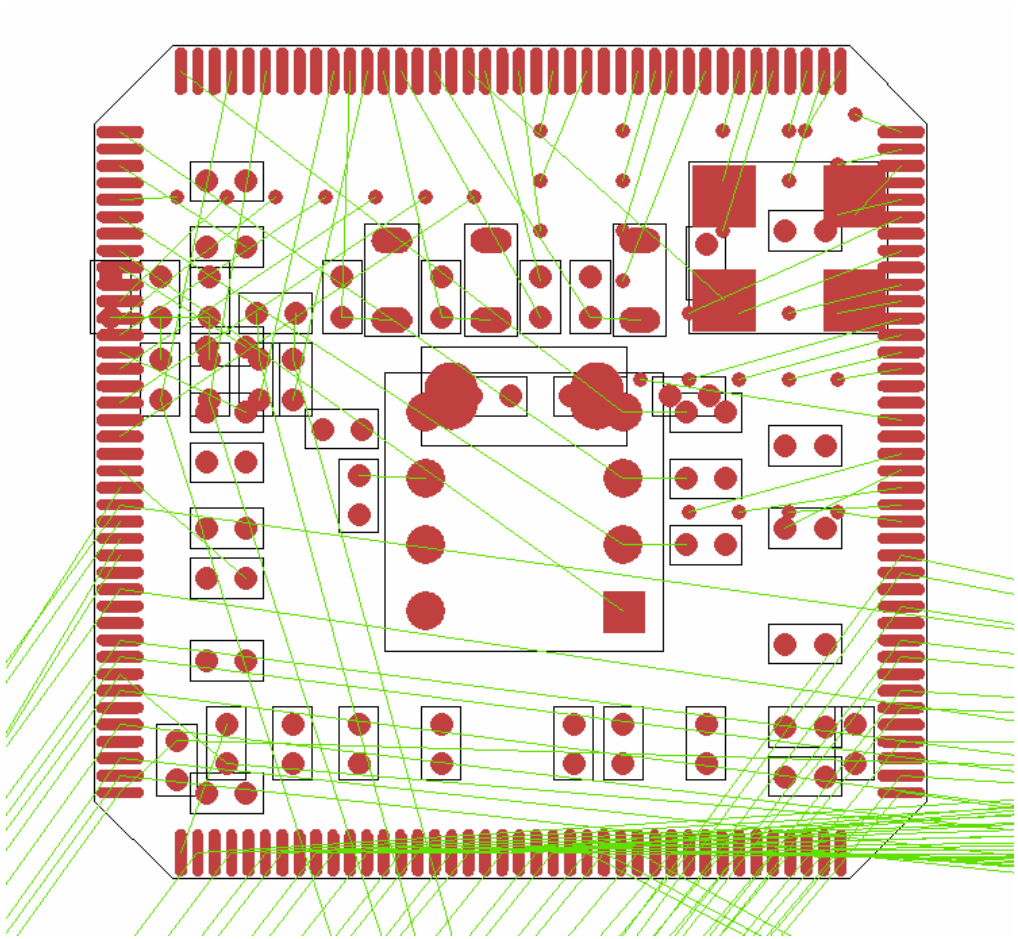
Group Parts

- Group logical components together
- Turn off all power net-lines
- Work outside the board outline
- Don’t worry about clean placement of individual parts
- Keep groups compact
- Identify terminations and place near correct component
- If bypass caps are identified for individual components group at the same time
- If bypass caps are not identified for individual components leave them for later

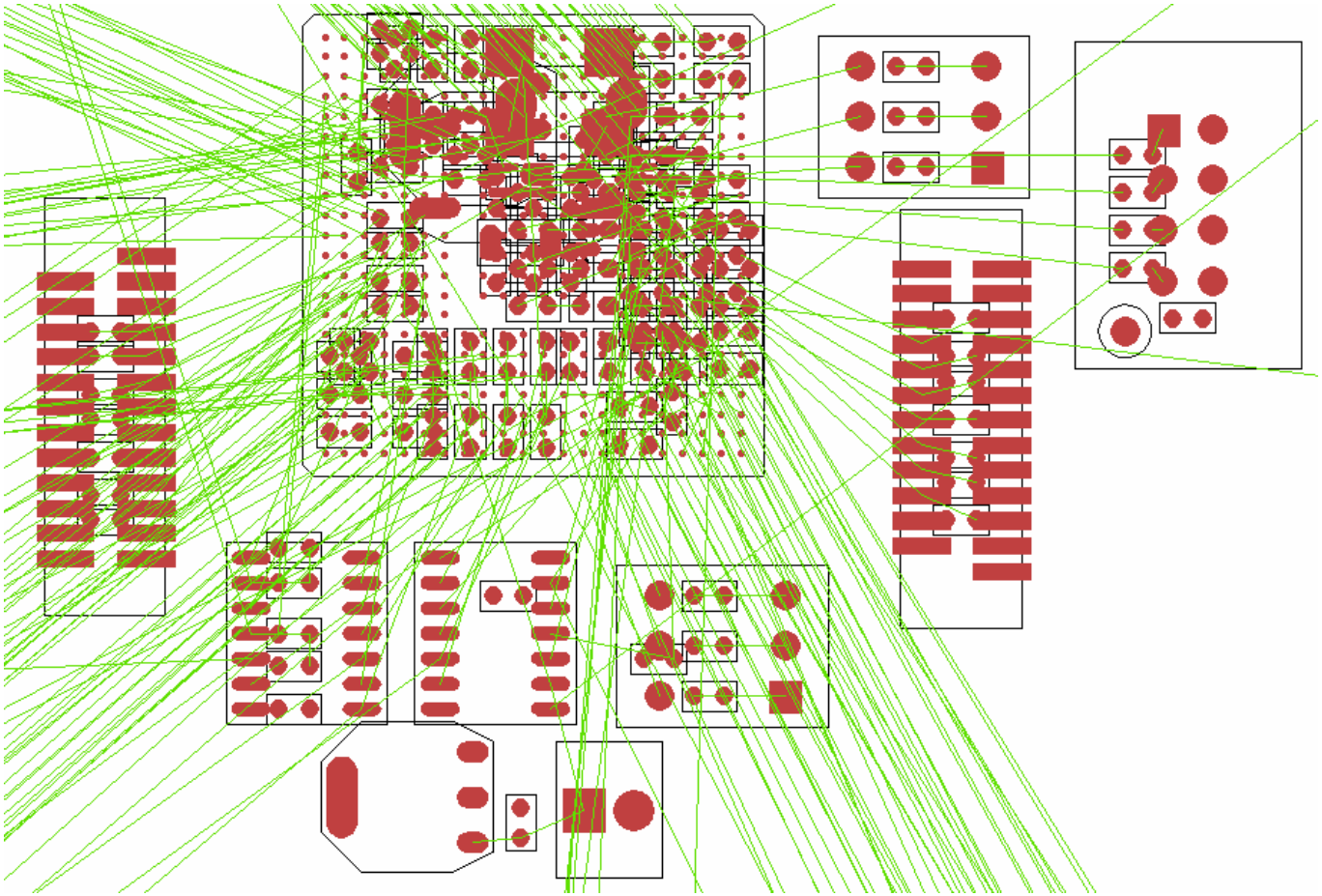
Group Parts



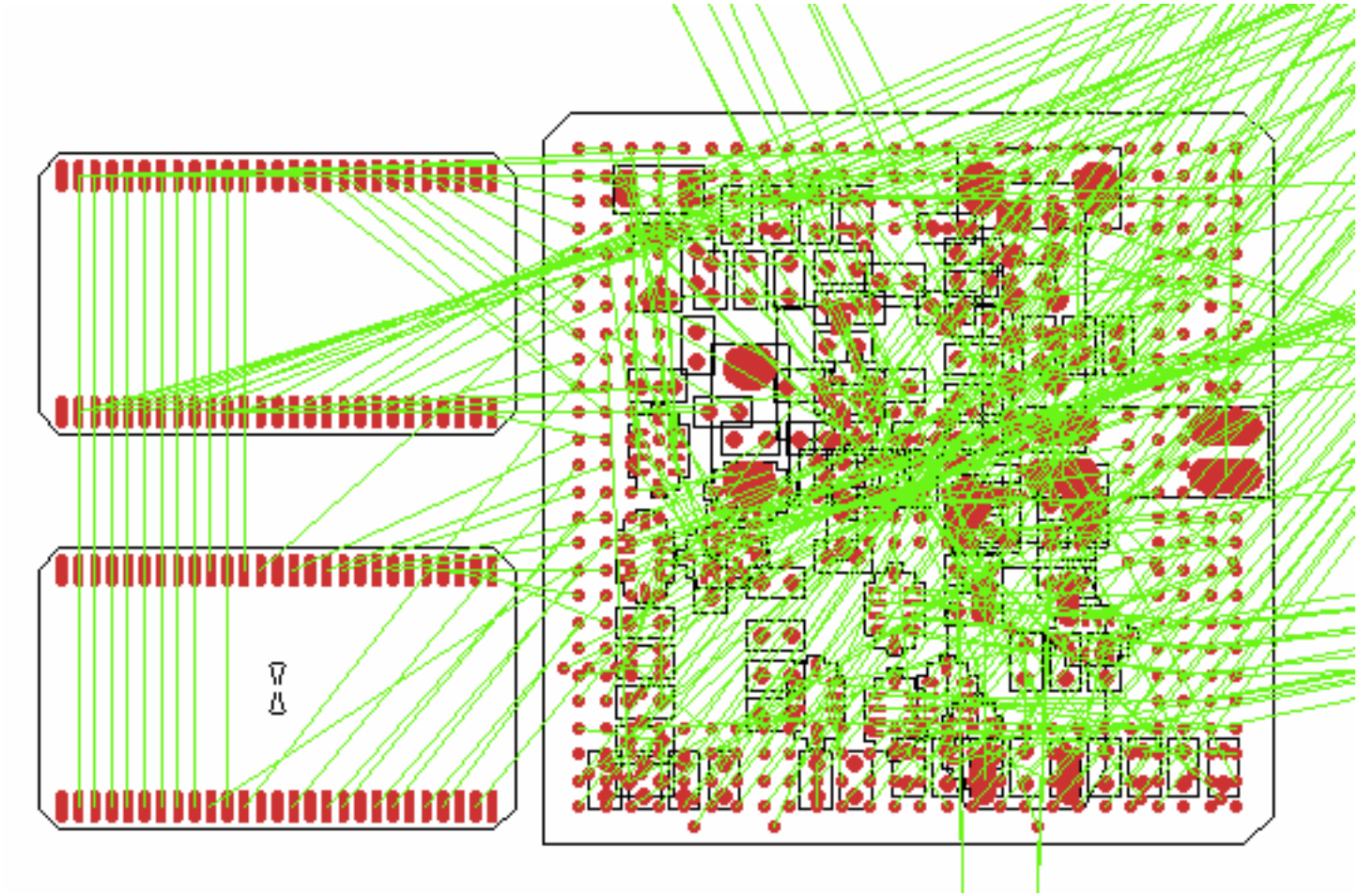
Group Parts



Group Parts



Group Parts



Place Fixed Parts

- Find out if fixed location are set it stone
- Locations are often set before design begins simply for packaging concept
- If fixed location cause serious routing issues feed this back to the engineer
- After locations are identified these parts can be taken off to facilitate the next step as long as you maintain there relative location

Exploded Placement

- Reduce major bus crossing by moving the component groups around
- Visualize the routing
- Visualize the plane splits
- Always have a reason for placing a component group
- Collapse the groups until all are inside the board outline
- Repeat until you feel you have the best routing solution
- This method allows you to see how the circuit best fits the shape of the board

Exploded Placement 1

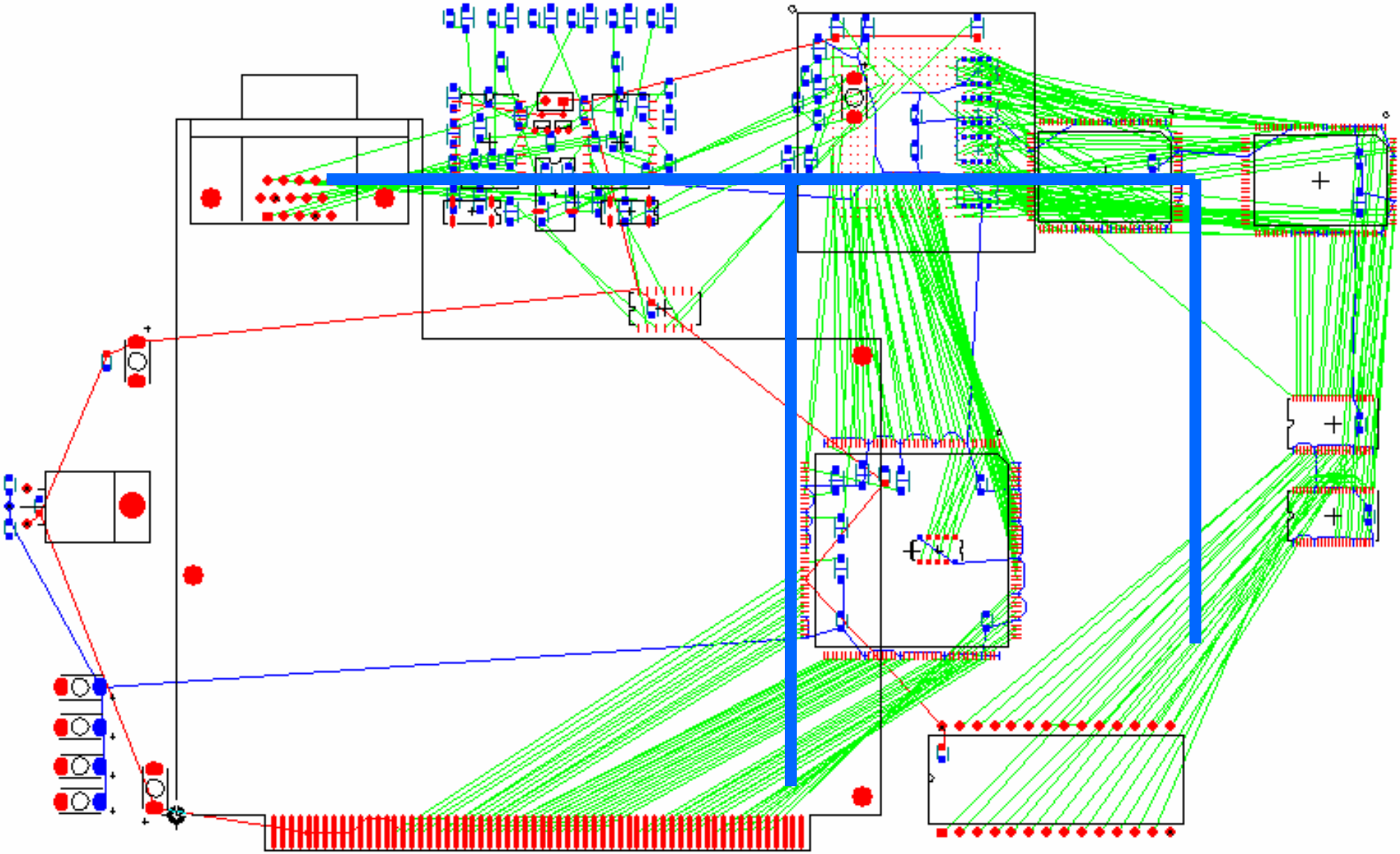
All small devices grouped with associated large component

Nice flow of major busses (no crossings)

Voltage regulator and bulk caps together

2 voltages red and blue net-lines

Exploded Placement 1

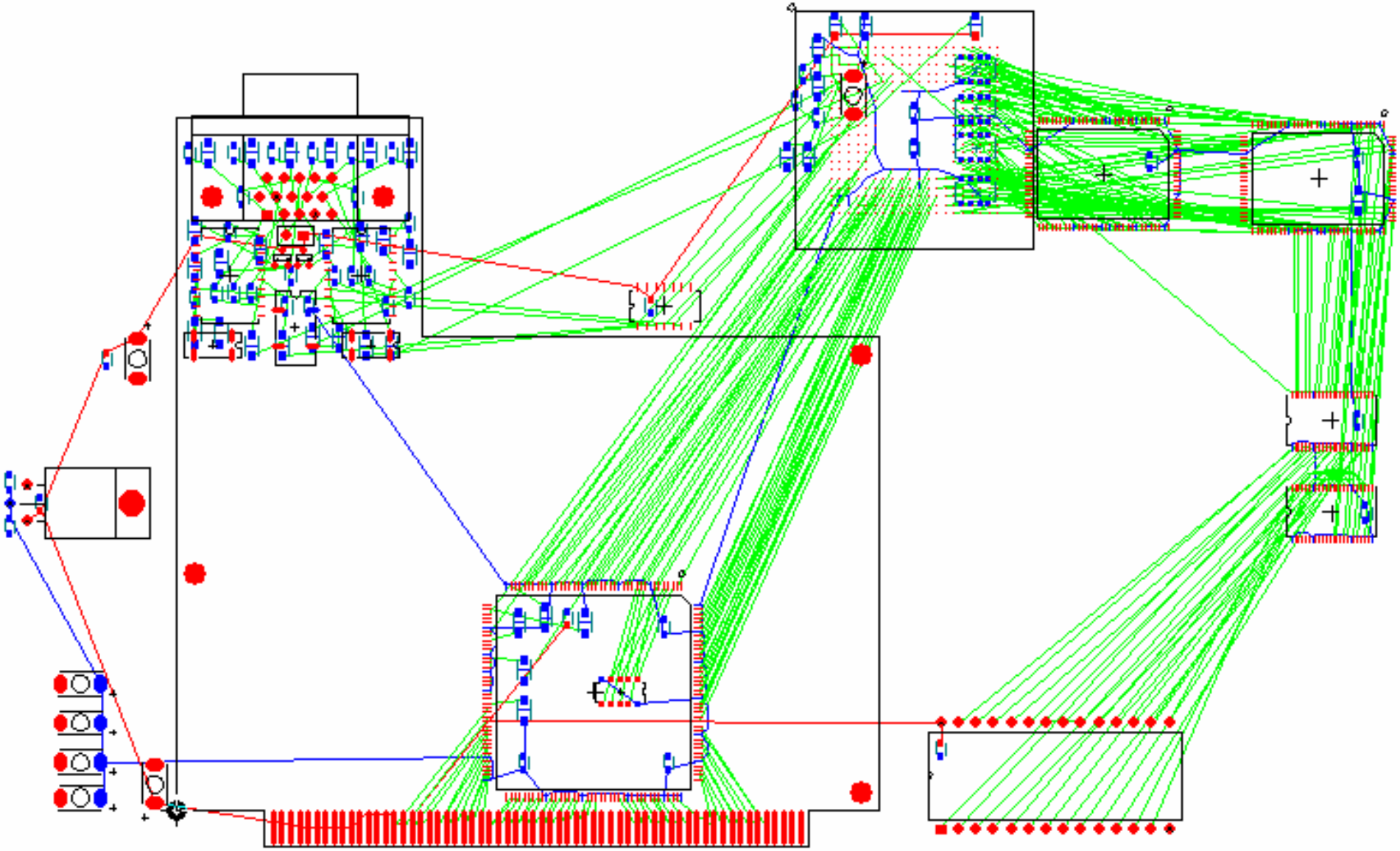


Exploded Placement 1

QFP placed near edge connector to reduce routing length

Analog circuit placed near output connector to reduce routing length and digital noise

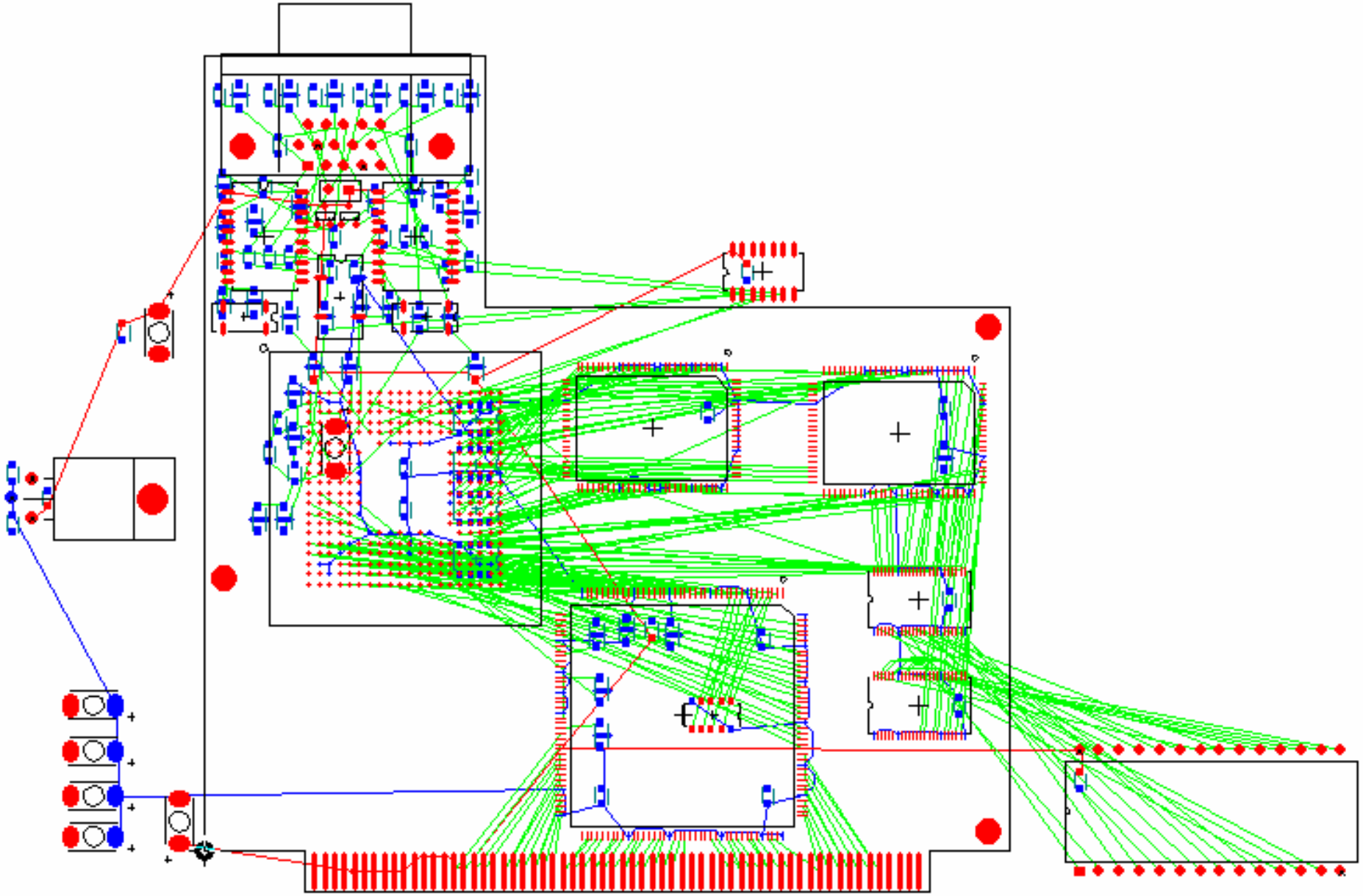
Exploded Placement 1



Exploded Placement 1

- BGA driving analog circuit placed adjacent to analog circuit
- Other devices placed according to circuit flow

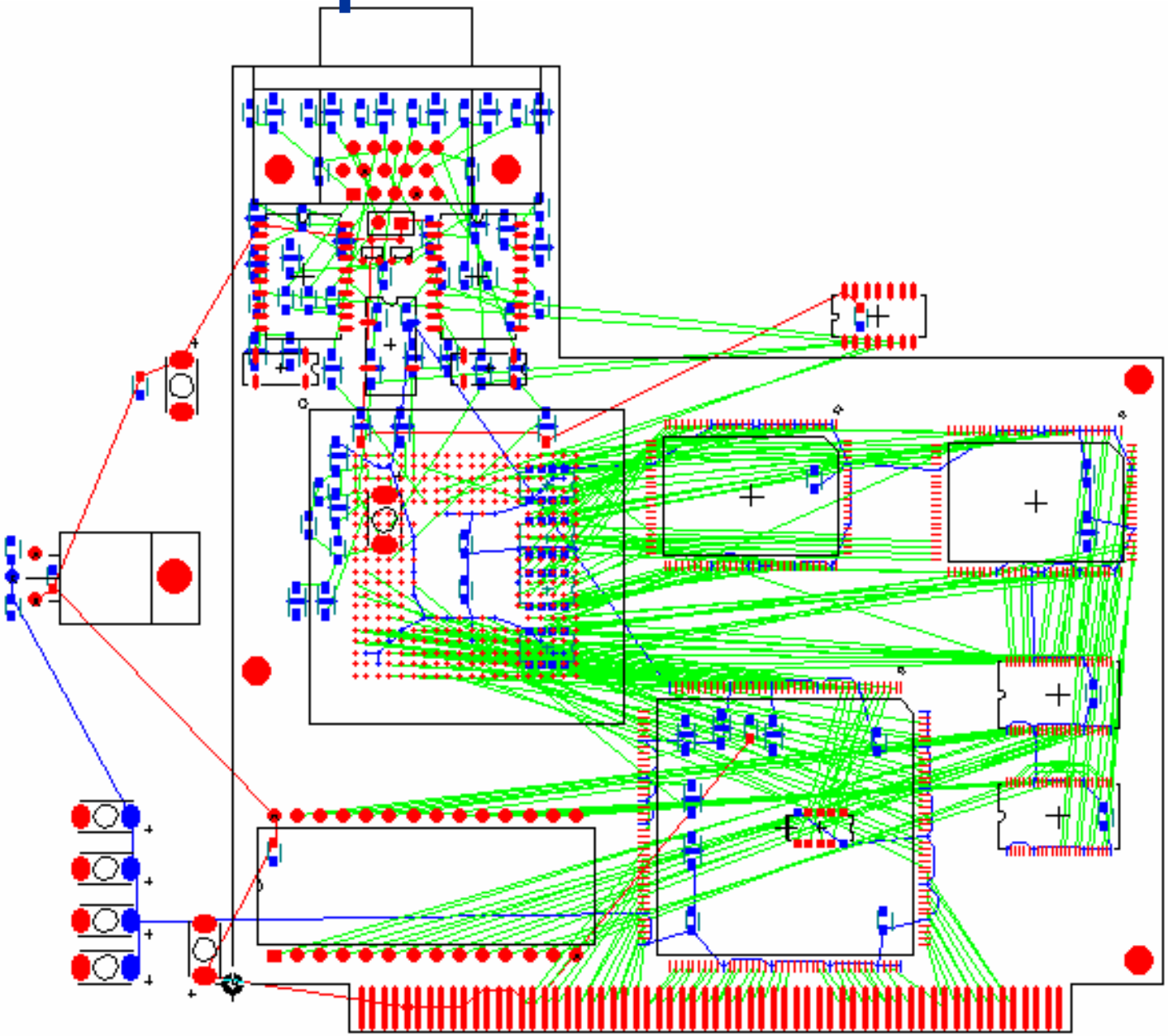
Exploded Placement 1



Exploded Placement 1

- Remaining major component placed in only available space
- Works well with voltage distribution

Exploded Placement 1



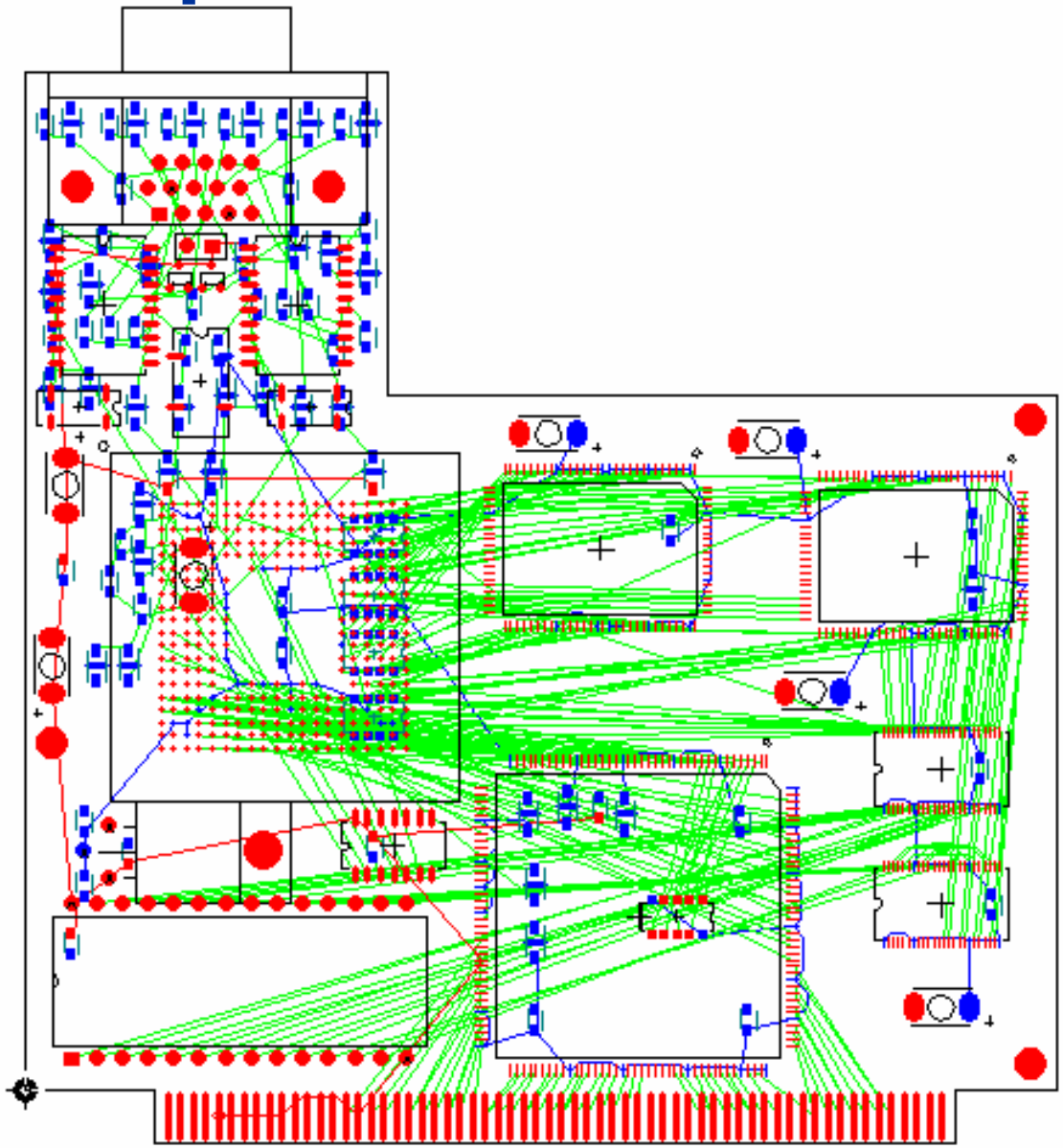
Exploded Placement 1

Place voltage regulator

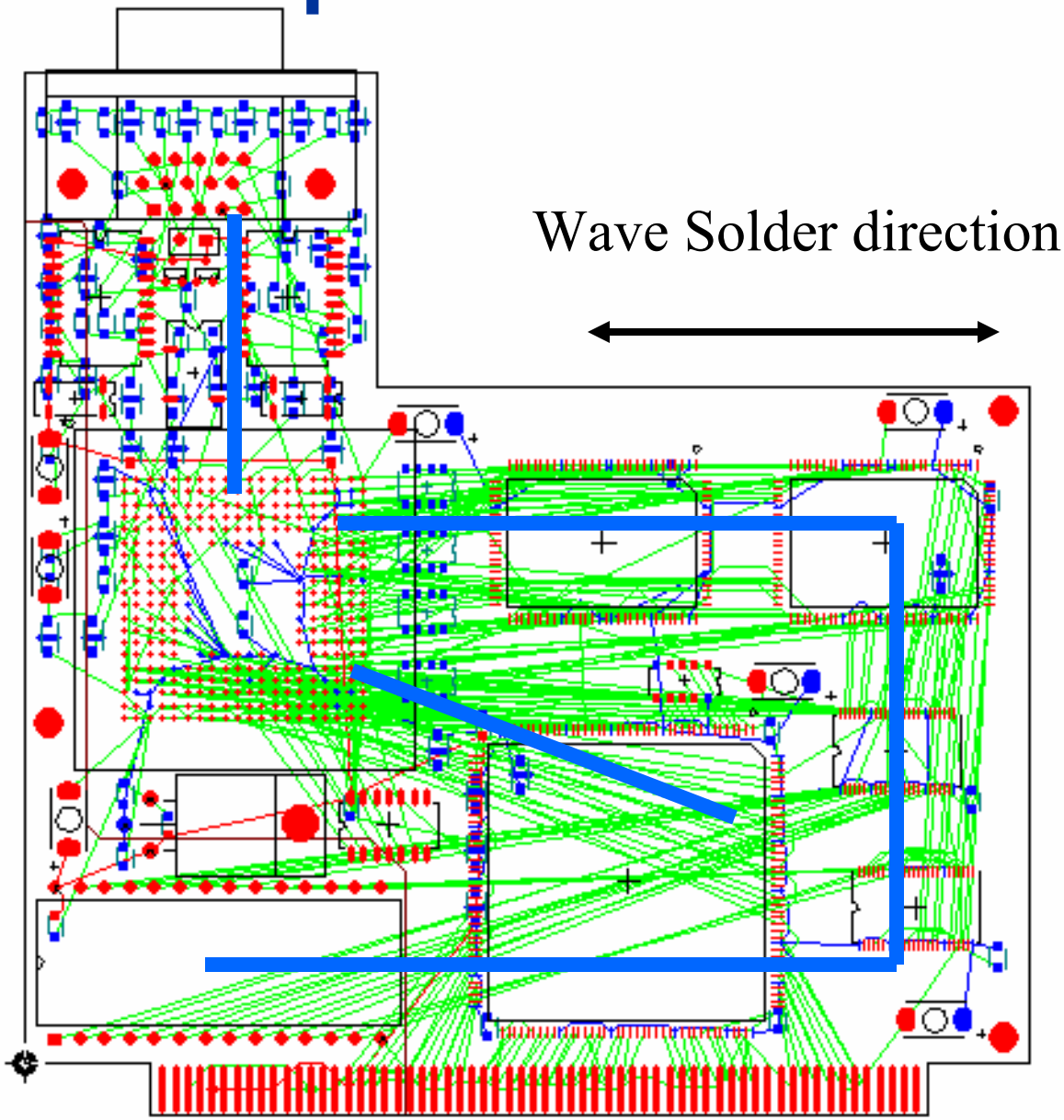
Place bulk capacitors

Place remaining logic device

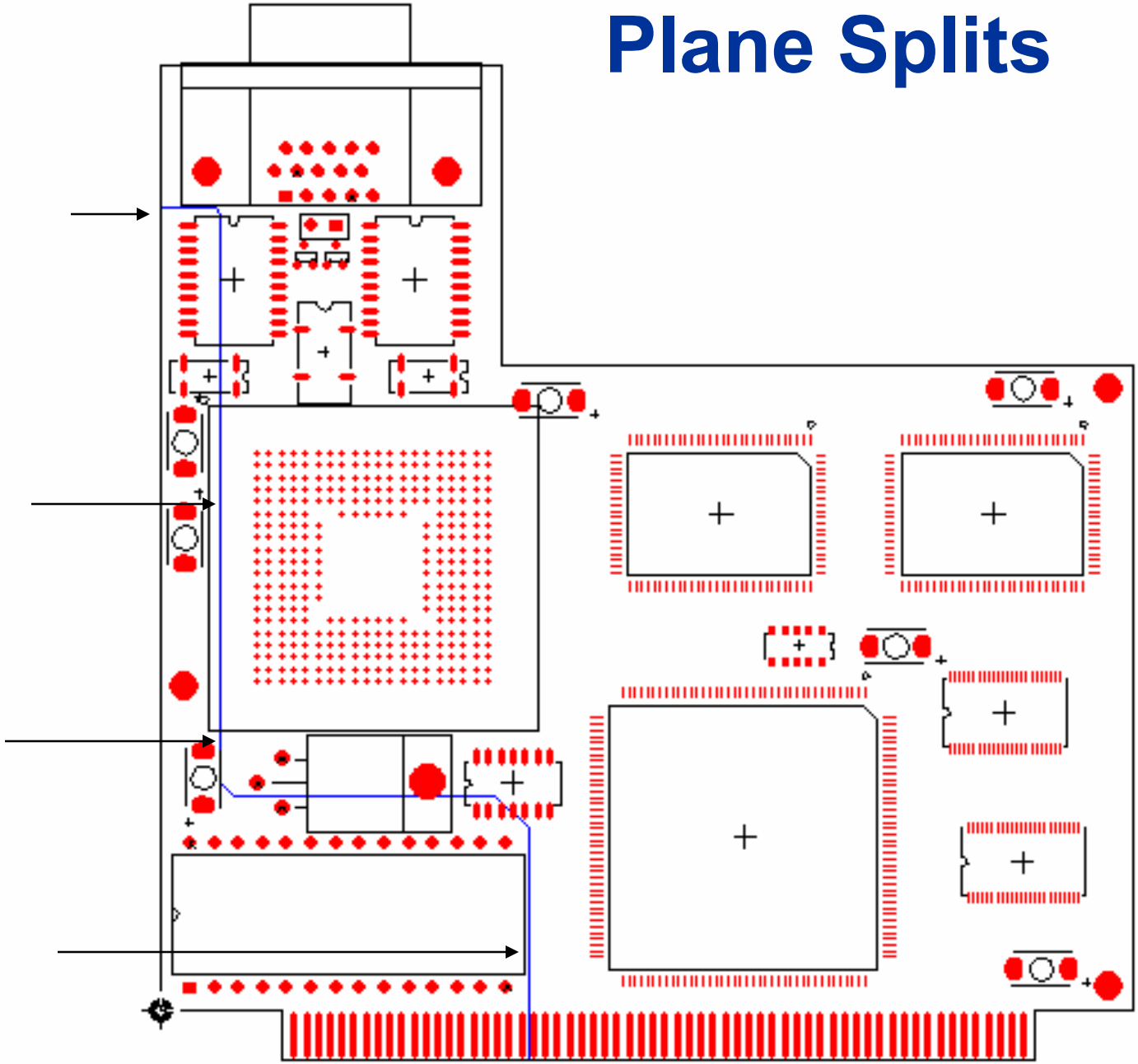
Exploded Placement 1



Exploded Placement 1



Plane Splits



Exploded Placement 1

- Double sided assembly
- 111 parts, 293 nets
- 6 layers 4S2P
- Layer count is driven by BGA
  - 8 mil trace, 7 mil space, 28 mil via
- All decoupling caps on back
- All resistor packs on back

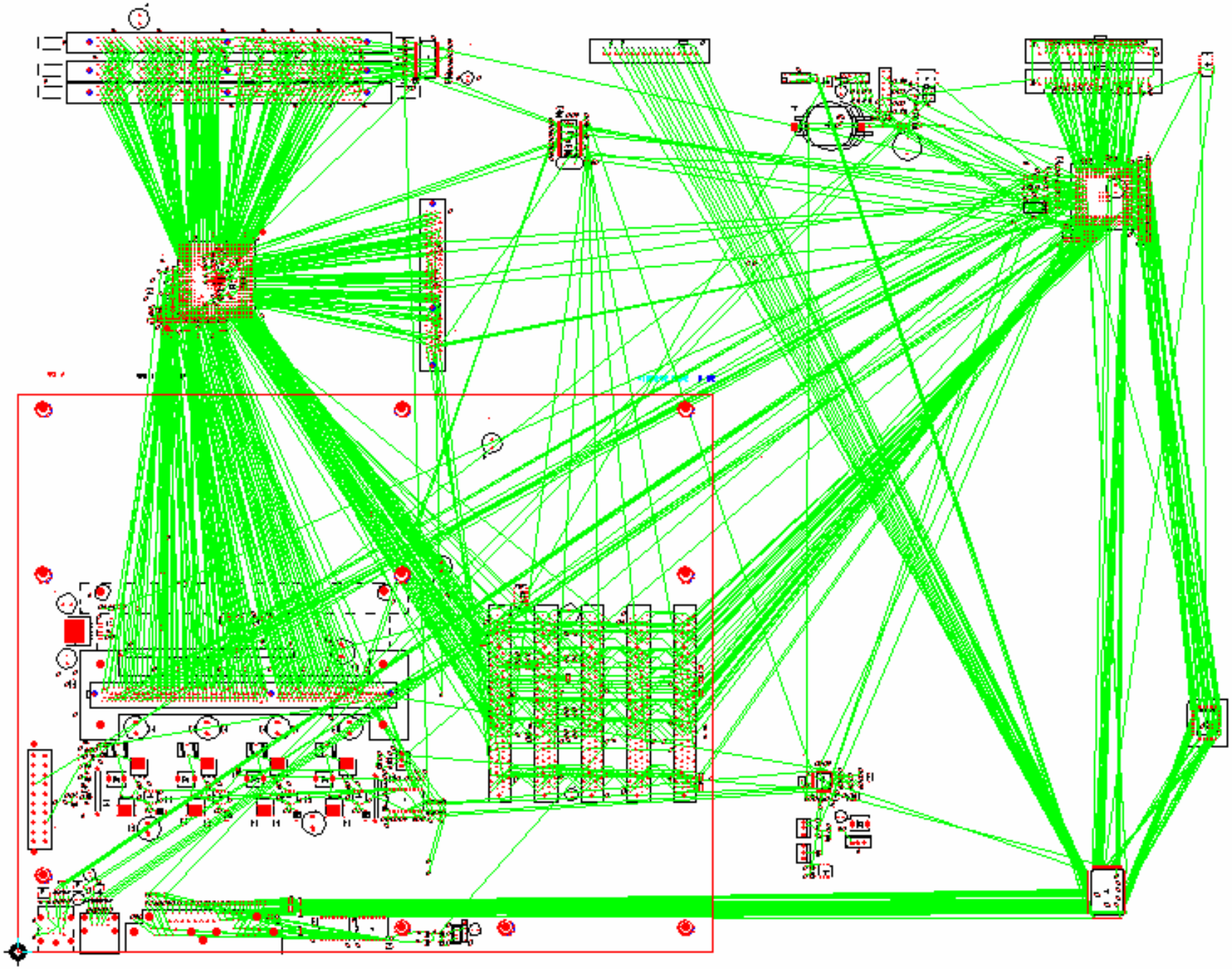
Exploded Placement 2

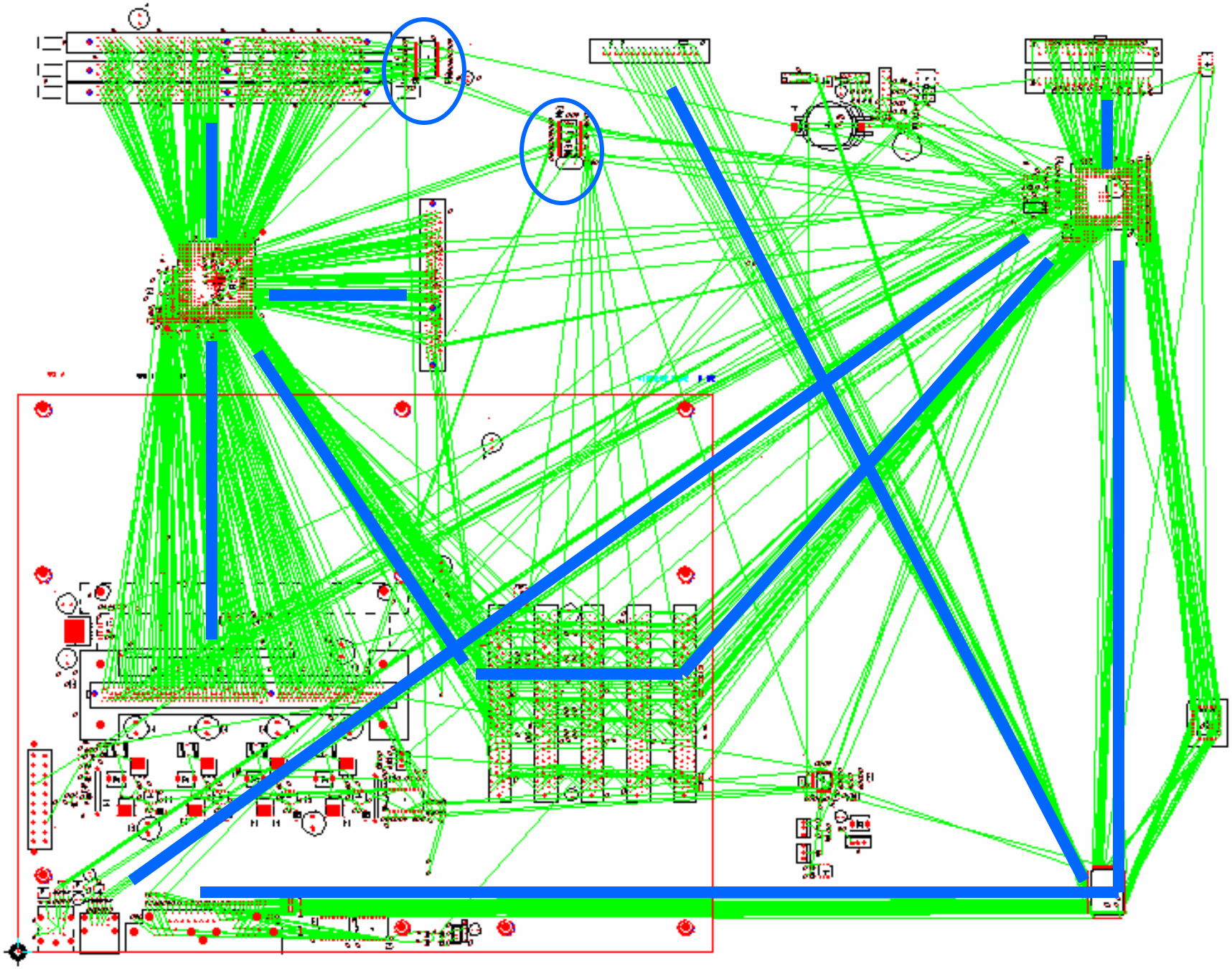
PC motherboard

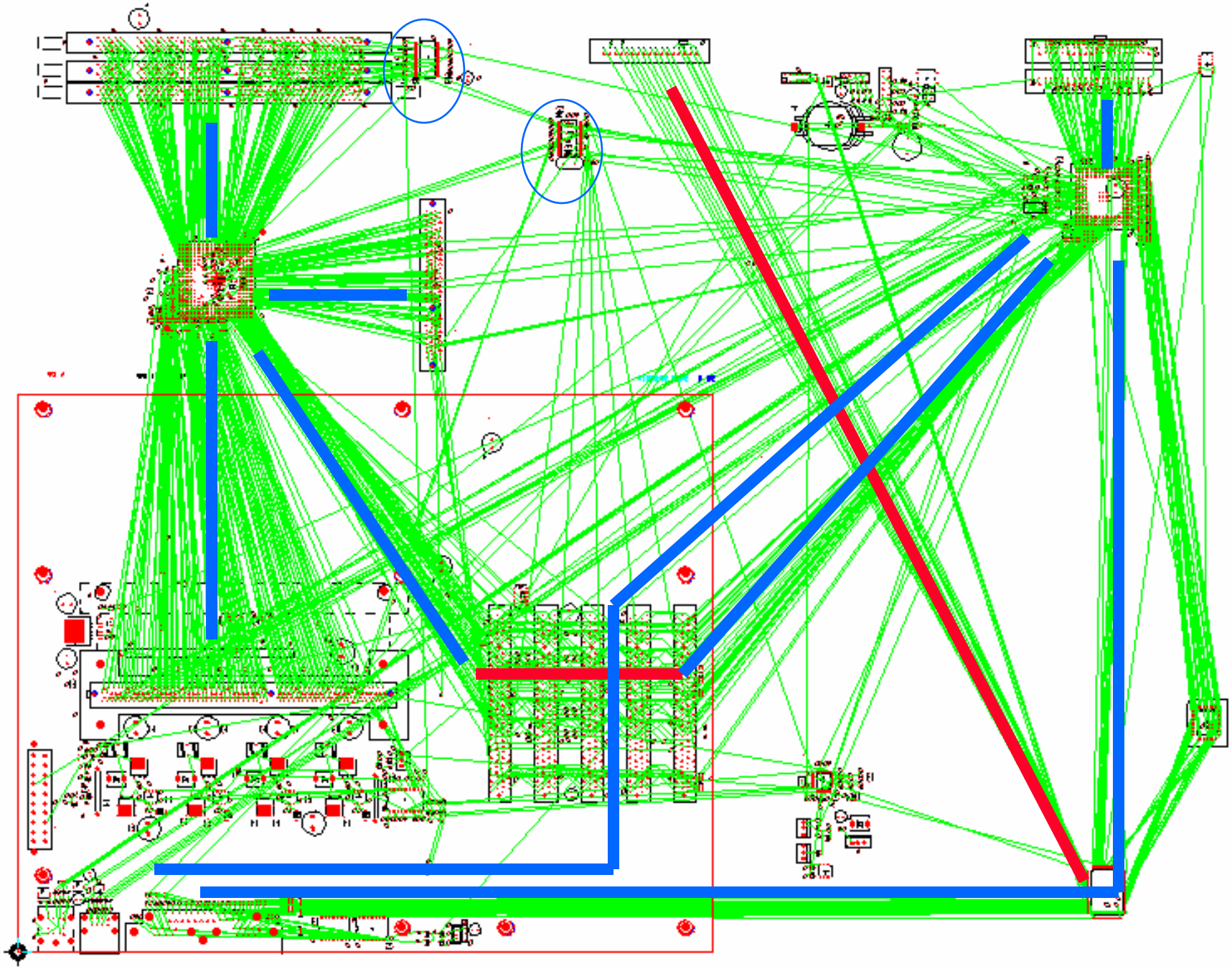
908 parts, 974 nets

4 layer, 2S2P

Single sided assembly







Exploded Placement 2

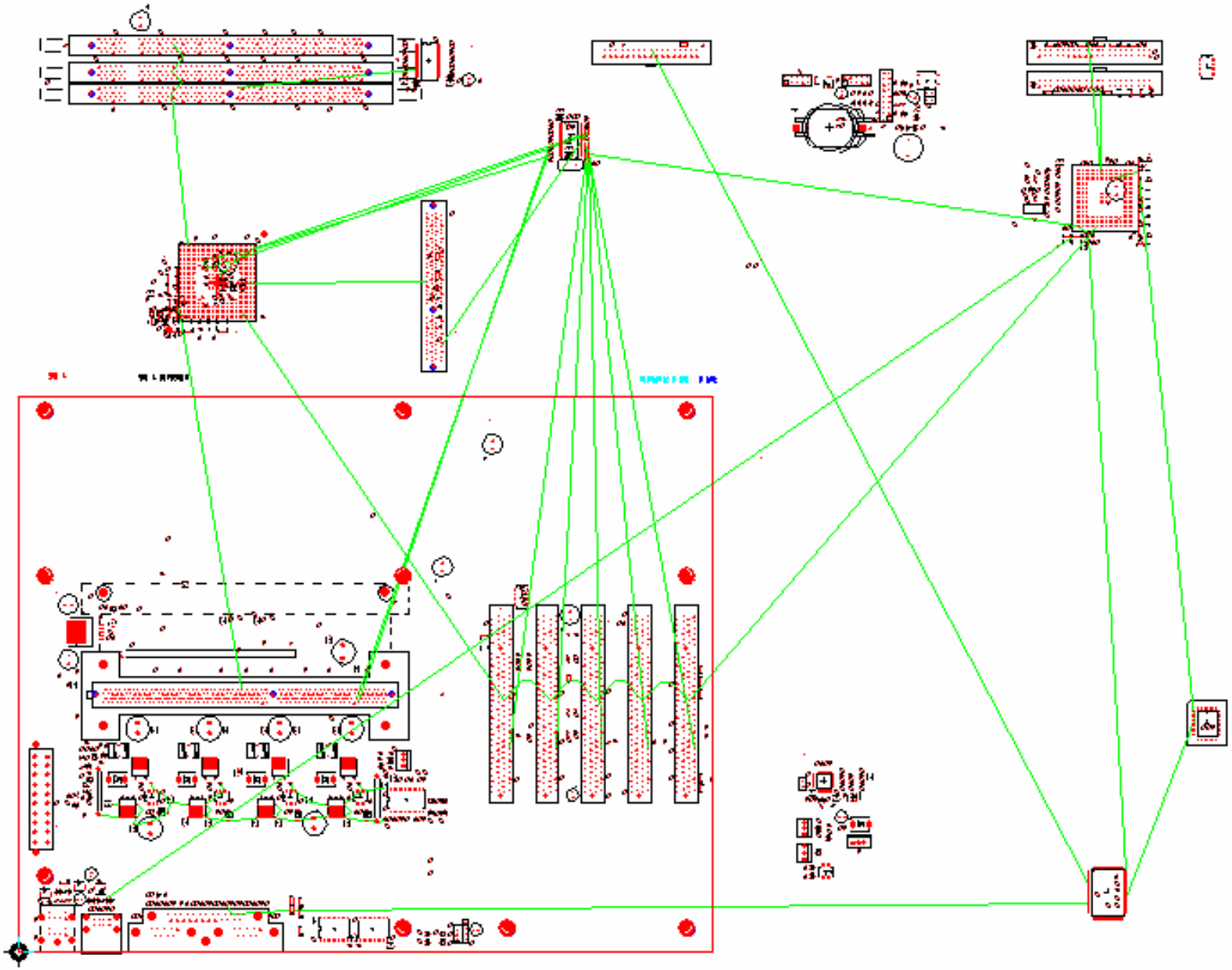
Single net for each bus

Clocks on

Differential pairs on

Power nets off

Miscellaneous nets off



Exploded Placement 2

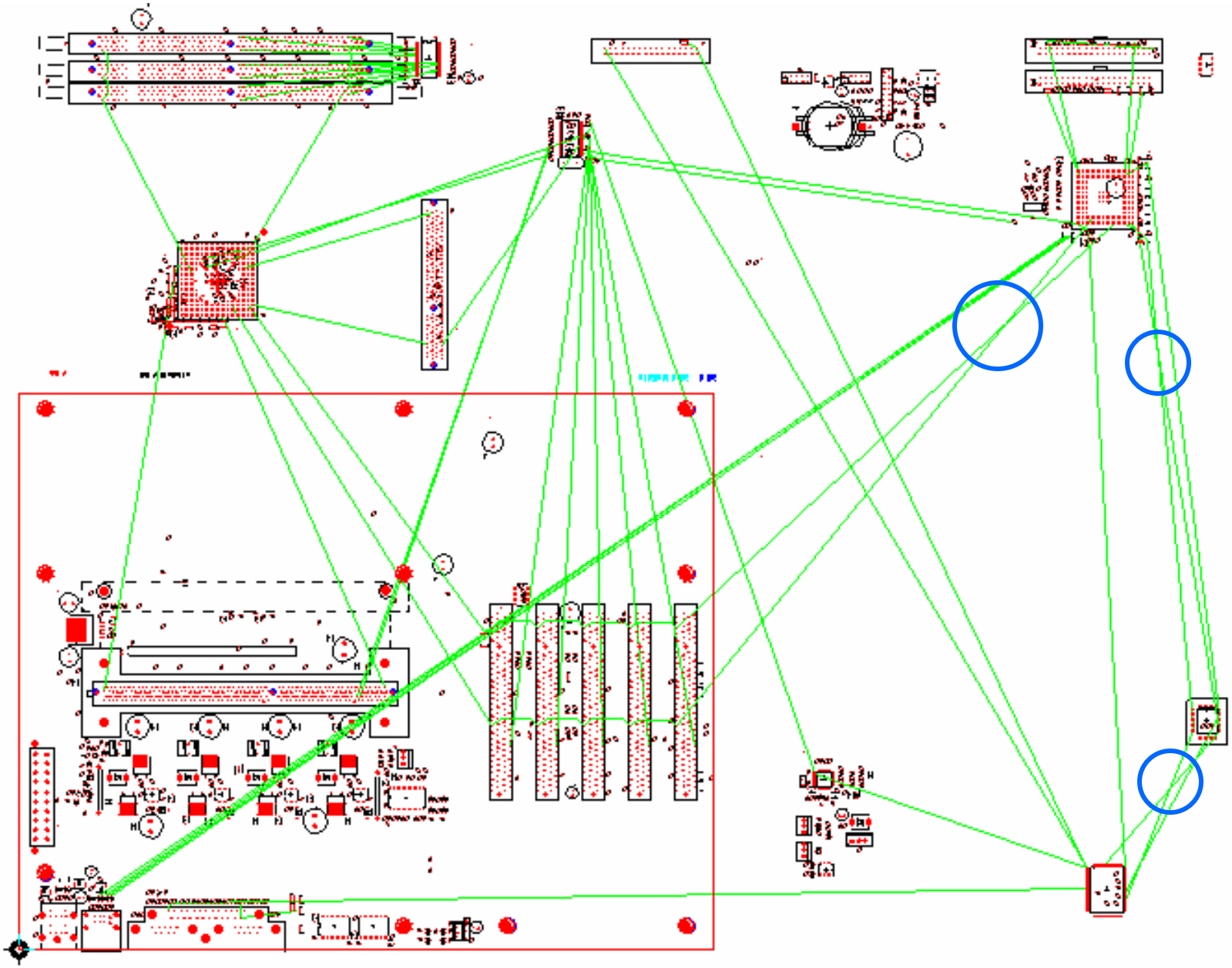
Dual nets for each bus

Clocks on

Differential pairs on

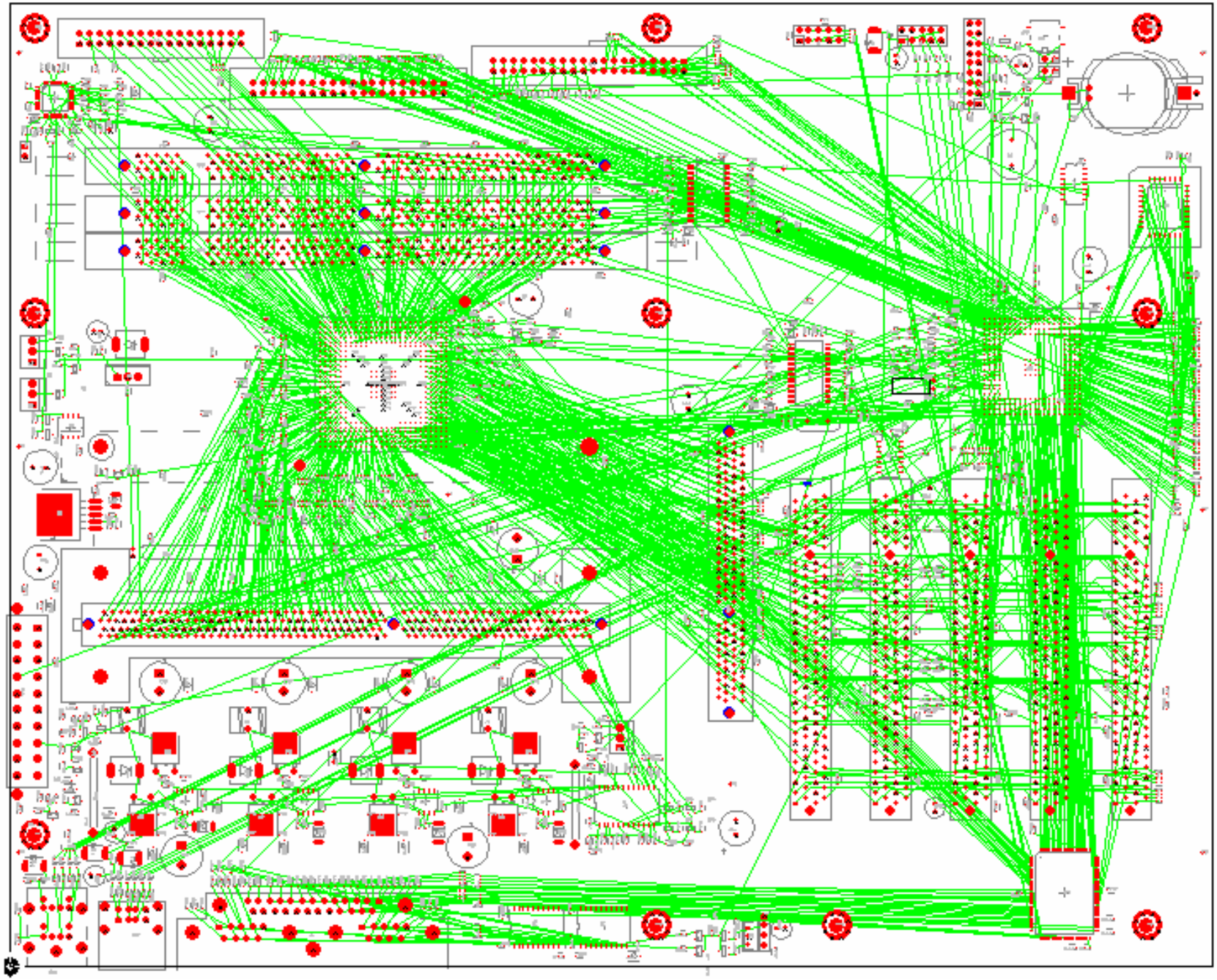
Power nets off

Miscellaneous nets off



Exploded Placement 2

All parts placed and detailed



Exploded Placement 3

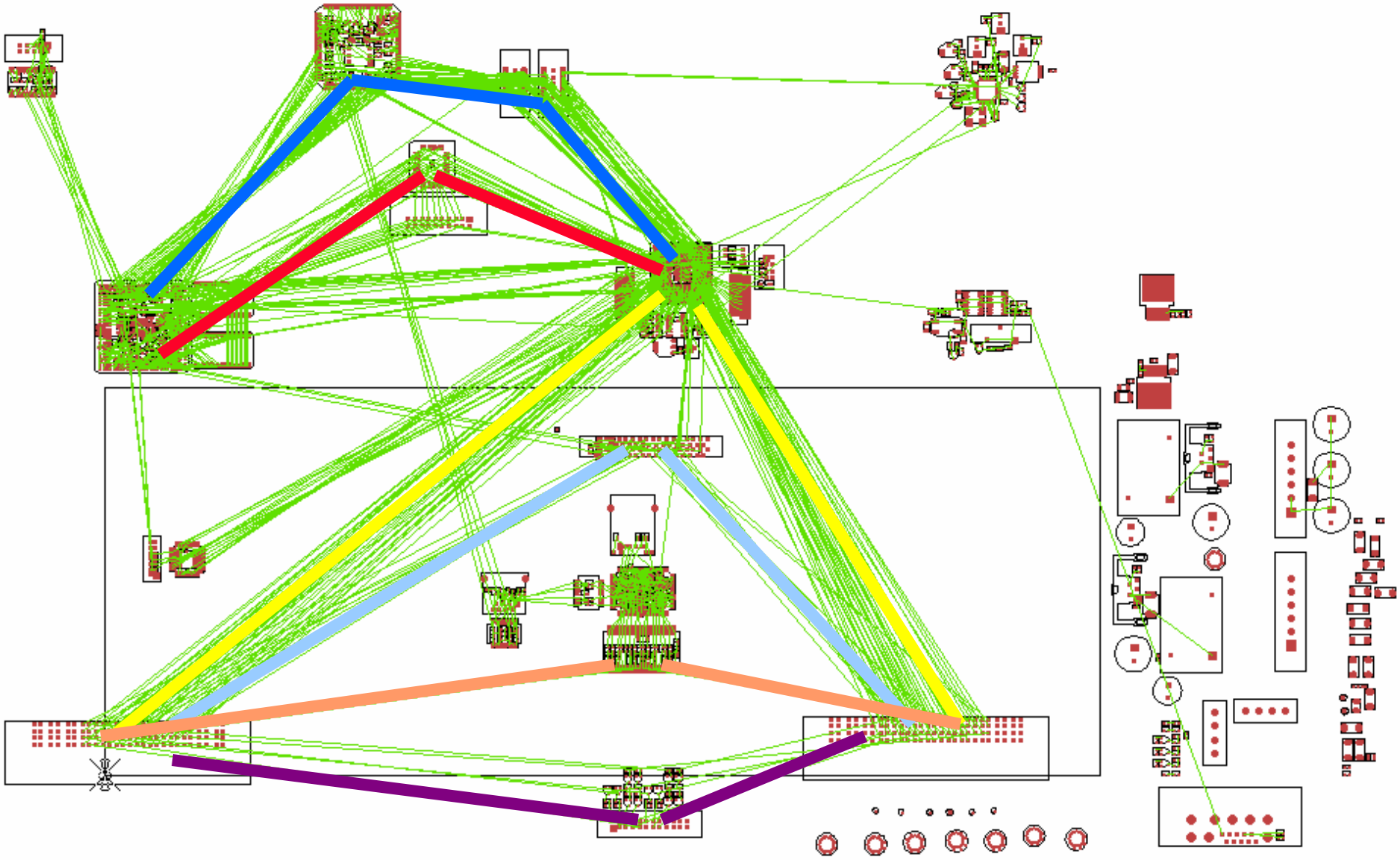
Test controller board

894 parts, 908 nets

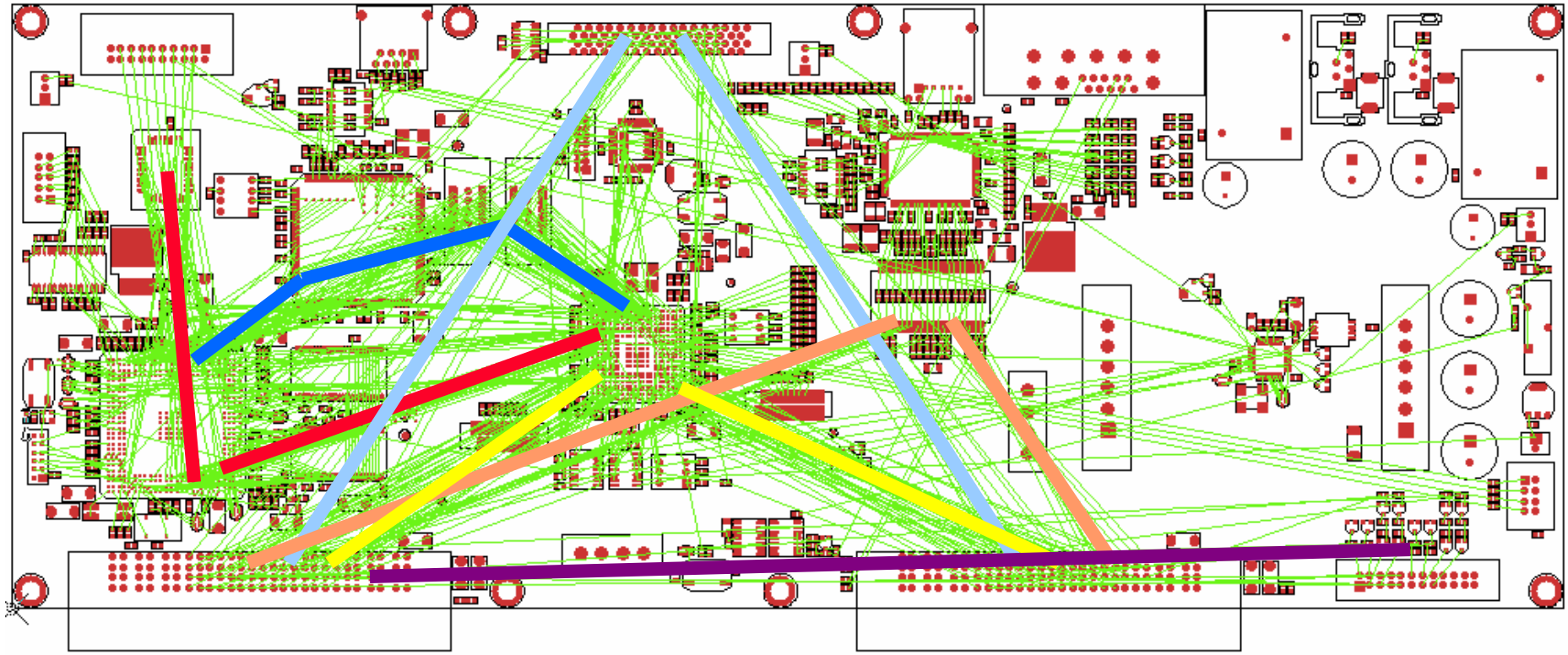
10 layer, 4S6P

Single sided assembly

Exploded Placement 3



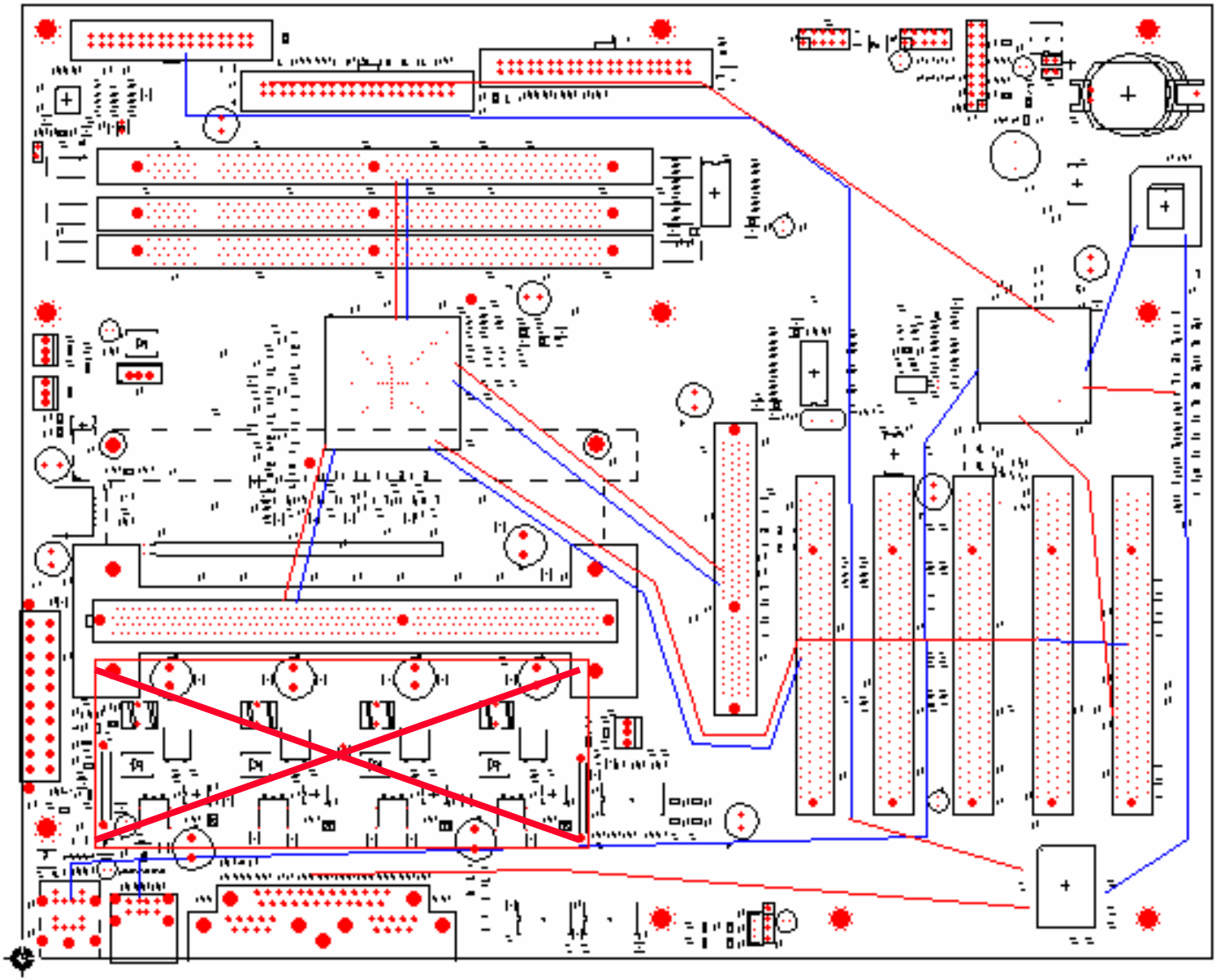
Exploded Placement 3



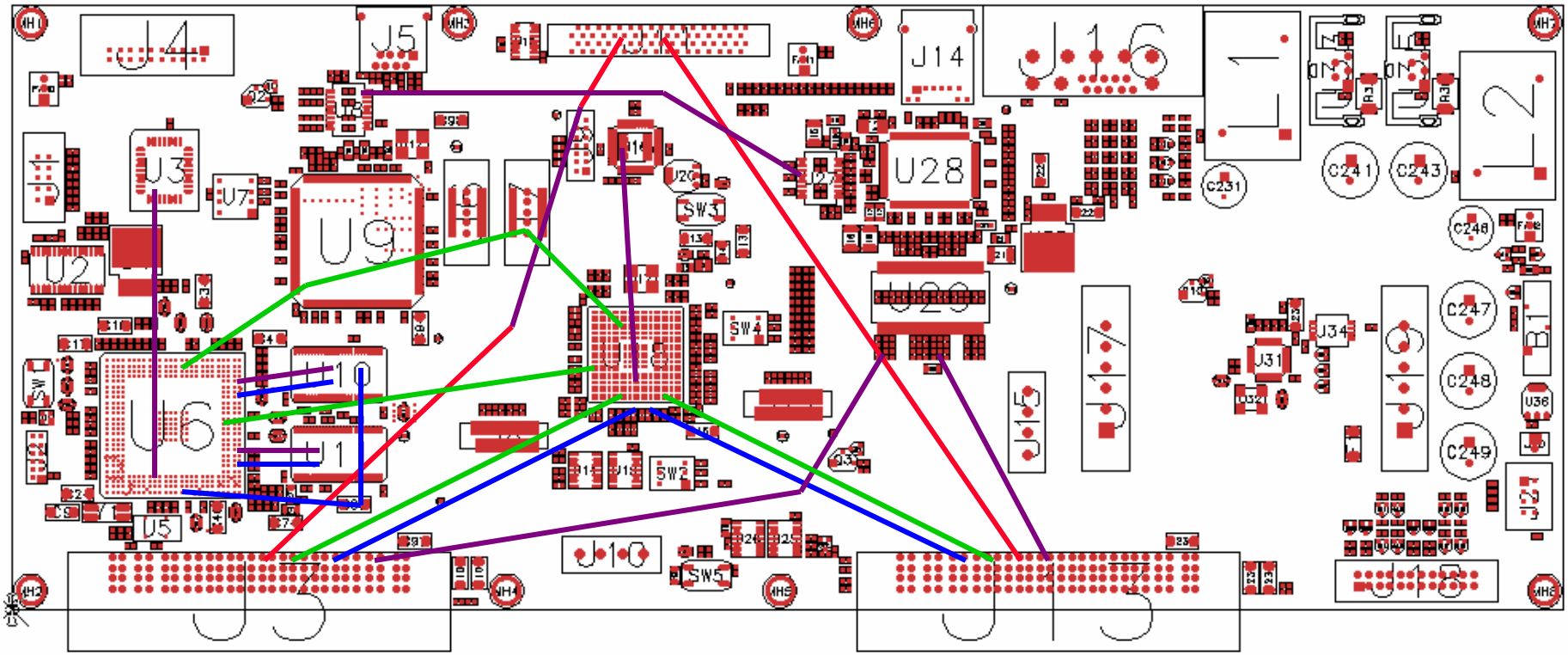
Routing Map

- Routing map
  - A map of the major busses and critical signal related to the layer usage
- Identify problem areas prior to routing
- Track your original routing plan on large designs
- Reference document for floor planning
- Coordinate multiple designer

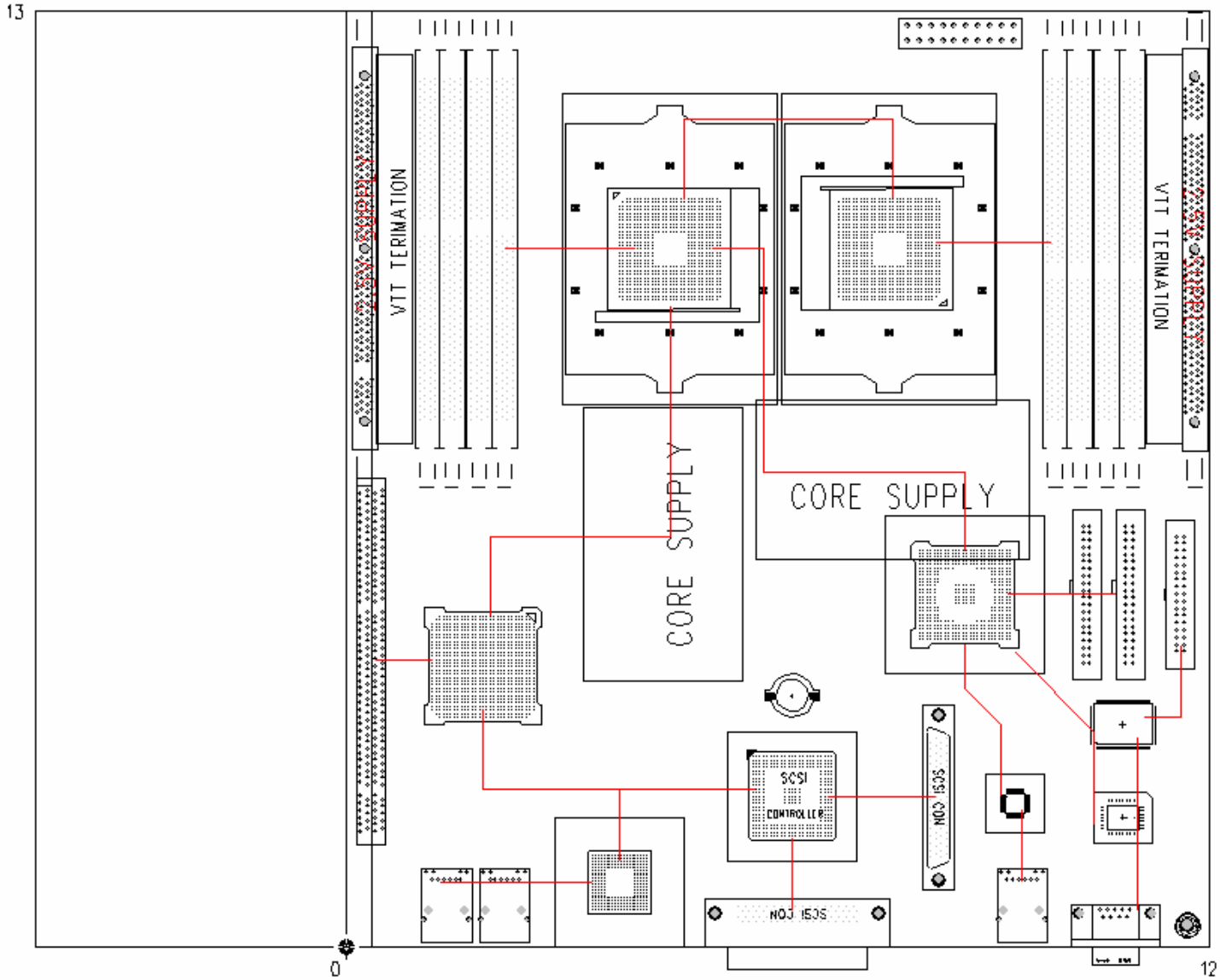
Routing Map



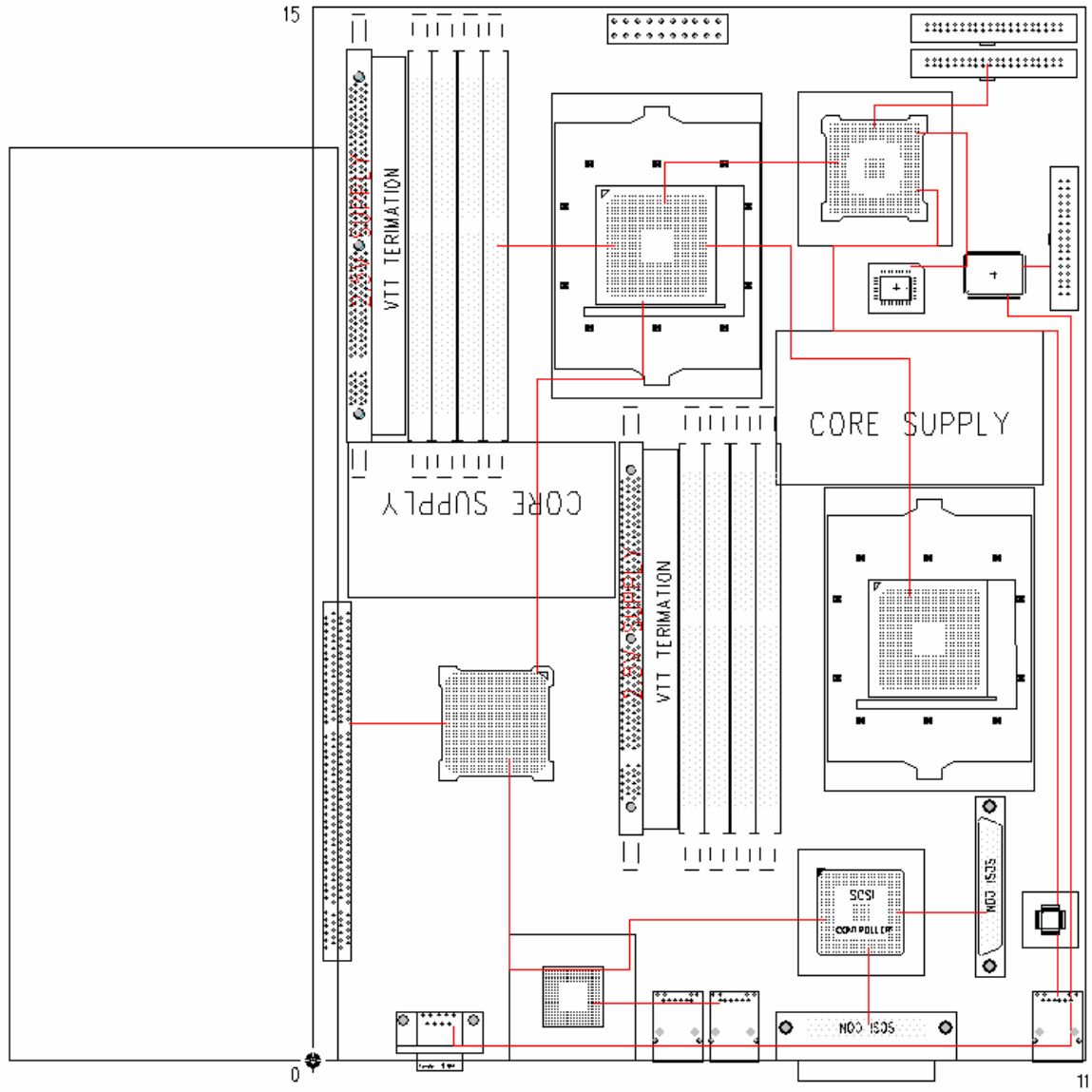
Routing Map


J4
J5
J14
J16
U3
U7
U9
U2
U6
U28
U29
J17
J15
J13
J31
U34
C231
C241
C243
C246
C247
C248
C249
SW1
SW2
SW3
SW4
SW5
J10
J3
J1
L1
L2

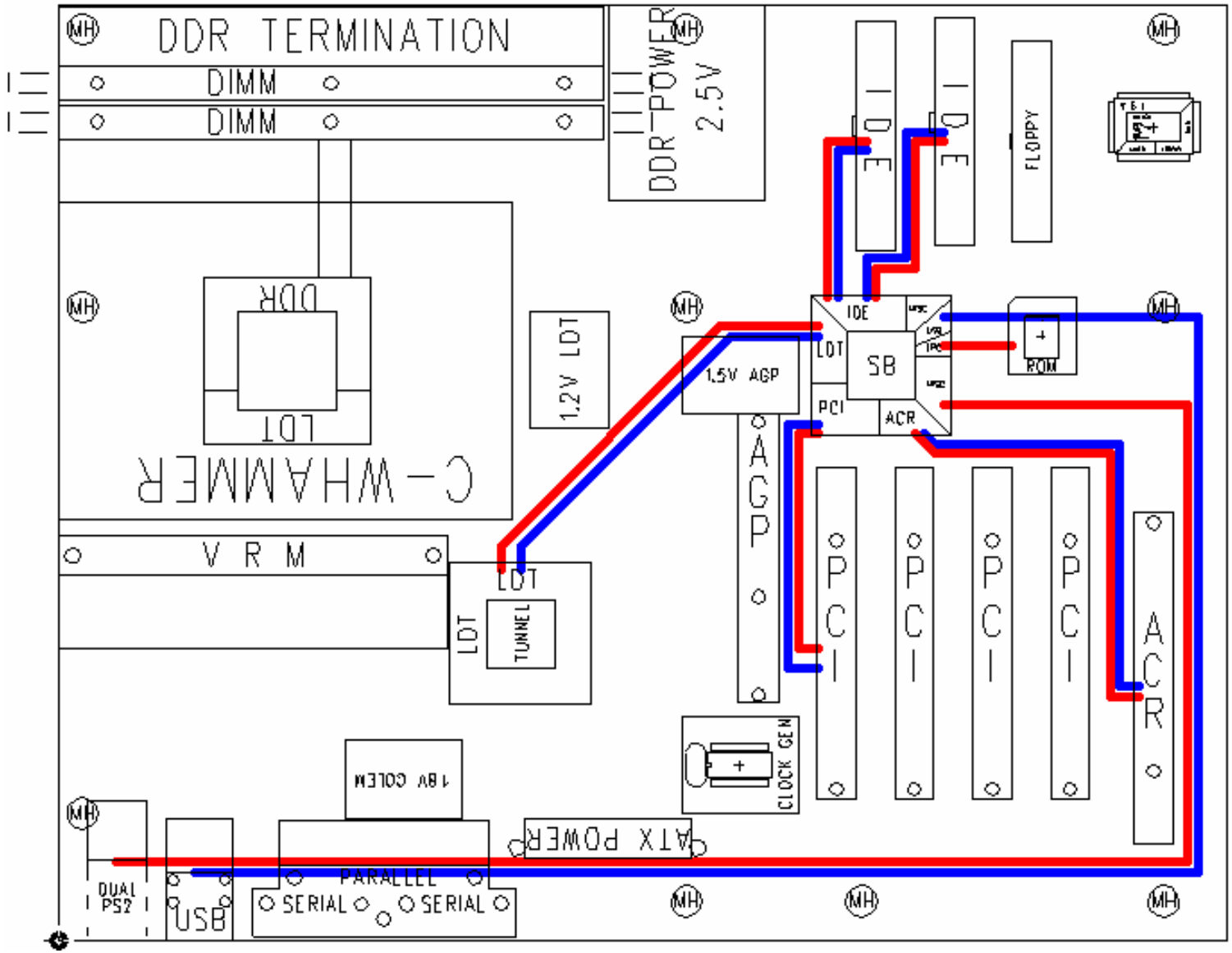
Routing Map


13
VTT TERMINATION
VTT TERMINATION
CORE SUPPLY
CORE SUPPLY
SCSI
CONTROLLER
0
12

Routing Map


15
VTT TERMINATION
CORE SUPPLY
CORE SUPPLY
VTT TERMINATION
SCSI
0
11

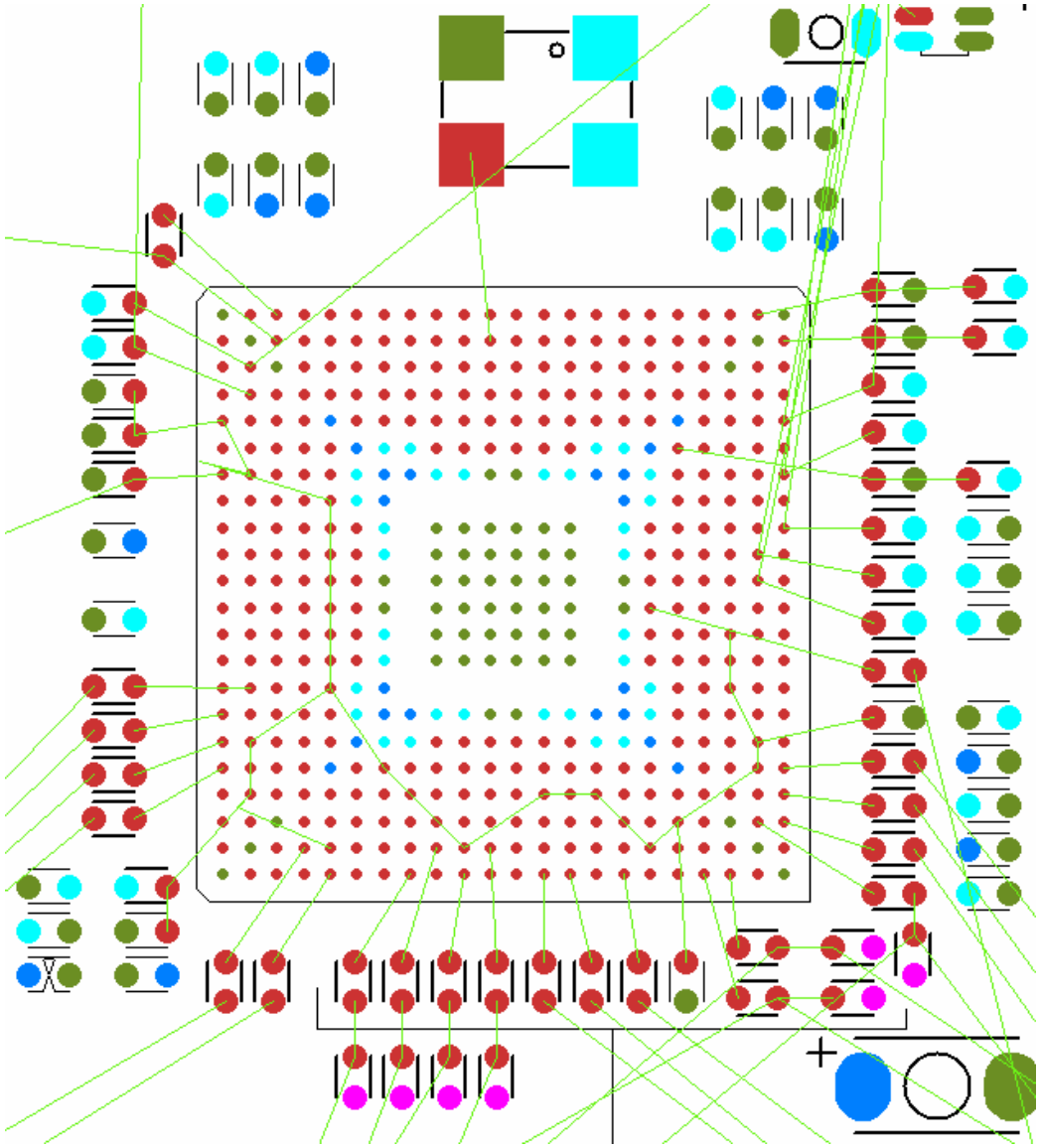
Routing Map


DDR TERMINATION
DIMM
DIMM
DDR POWER
2.5V
IDE
IDE
FLOPPY
DDR
LDT
C-WHAMMER
1.2V LDT
1.5V AGP
IDE
LDT
SB
PCI
ACR
ROM
V R M
LDT
LDT
TUNNEL
AGP
PCI
PCI
PCI
PCI
ACR
CLOCK GEN
1.8V COLEM
ATX POWER
DUAL PS2
USB
PARALLEL
SERIAL
SERIAL
MH

Detailed placement

- Clean up parts in groups
- Gate & pin swapping
- Alternate parts
- Test Points
- Special attention to termination
- Power supplies
- Bypass capacitors
- Bulk capacitors
- Adjust rough placement as needed to make room for supporting components
- Do not spend to much time tweaking

Detailed placement



Alternate Parts

Many electrical components come in different body styles and sizes.

Leaded or surface mount

Small outline or shrink small outline

Quad flat pack or ball grid array

Resistor packs or single resistors

Multi-gate logic or single gate logic

Several multi-gate logic parts or a single programmable logic part

Often the need for a change is not apparent until routing is taking place

If you believe a change will help the layout ask for it

Swapping

- Resistor packs
- Multi-gate logic devices
- Memory
- Clocks
- Asics
- Programmable logic devices
- Swapping of pins and gates often works better during routing

Test Points

- Test points are often forgotten until the end of the design
- Test points need to be considered during detailed placement
- If you don't leave room for TPs then you have to make room later when you thought the design was finished
- The best method I have found for TPs is to add them as components and place them during the fine placement.
- This will save you hours at the end of the design and will guarantee you 100% coverage before you lay a single trace

Placement Review

- Assembly shop review
  - Remind assembler that routing has just begun and you would like a quick response.
- Mechanical interferences
- Test fixture issues
- Blessing of electrical engineers
  - Everything is as expected
  - That's not what I meant
  - There is extra space, can we make the board smaller
- Sign off
  - Sign up

Mechanical Verification Board

- Top copper layer only
- Verifies land patterns
- Verifies form, fit and function
- Assembly review
- Prop for technical discussions
- Marks end of placement phase
- Good souvenir

PLACEMENT IS NEVER LOCKED

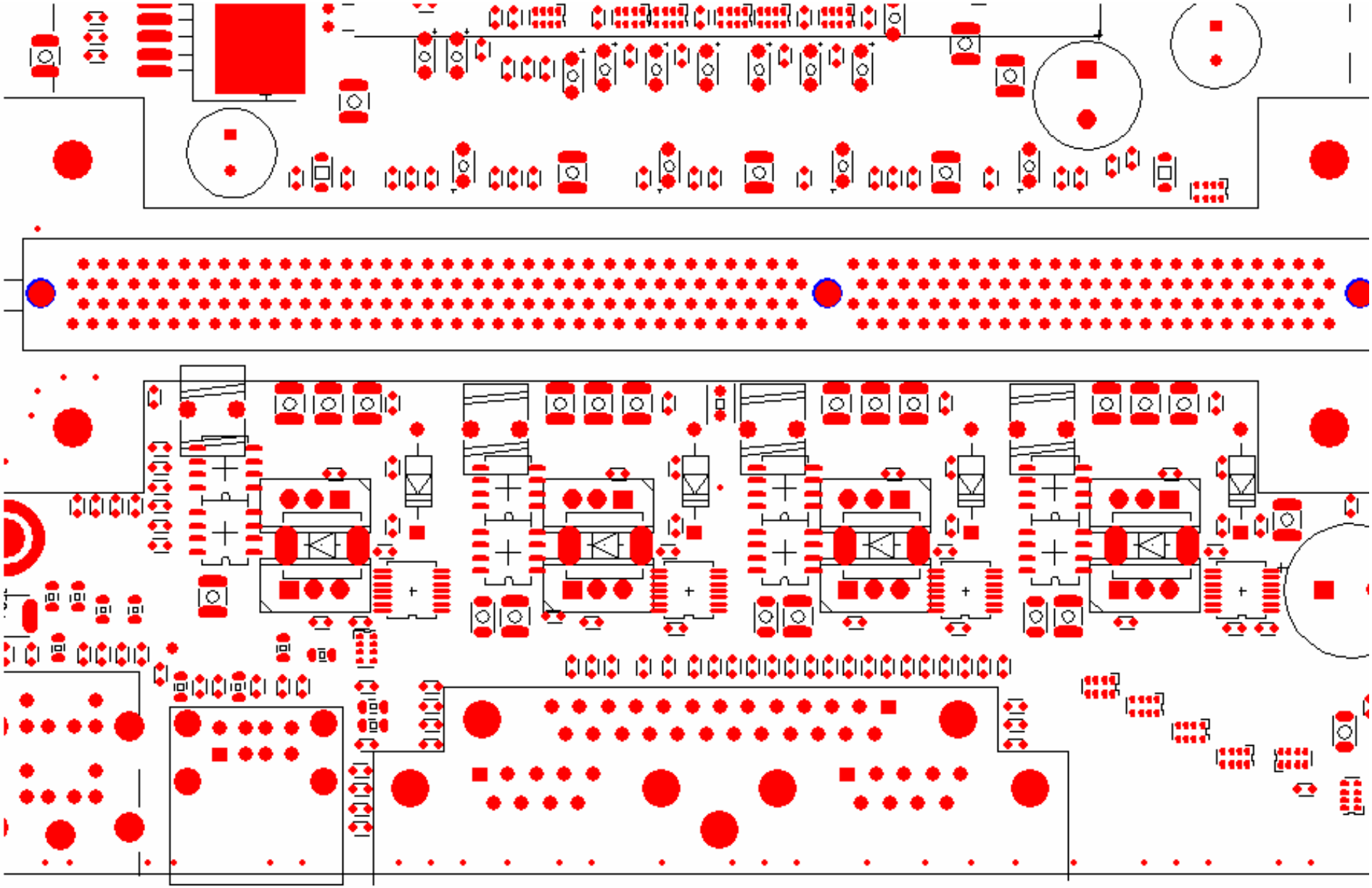
Moving of parts will continue during routing

This is an essential tool in completing high density designs

Placement Tips

- Build a pads only board to verify land patterns
- Place capacitors and resistors on same page as related components in schematic
- Routing study may be necessary to confirm the placement plan
- Block diagram
- Parts are 3 dimensional

3 Dimensional



Placement Tips

- High granularity placement grid
- Metric parts placed on a metric grid
- Parts do not have to face the same direction
- Parts can be placed at any angle
- Parts should only be aligned if it creates a routing advantage
- Smaller is not necessarily better
- Create a map if multiple designers will be routing the design

Double Sided Placement Tips

- Keep like parts on the same side of board
- Place all though hole devices on same side
- Identify how your assembler solders double sided boards
  - Glue and wave solder bottom side components
  - Double sided IR with selective wave solder
- Mirror image parts may be available
- Shadow placement
  - Not friendly to x-ray inspection

Exit placement ready to route

Meets all design requirements

Have a route plan

Have a power delivery plan

Have a test point plan

Have sign off

Design Flow Routing

- Review all requirements
- Visualize routing of major busses and critical signals
- Power and ground via fan out
- Escape high signal count parts
- Rough in routing with pin and gate swapping
- Via reduction
- Routing clean up
- Final pass signal tuning
- Verify all design requirements

Review Requirements

- The emphasis here is on rules that affect routing
- Traces that require length matching for timing reasons
- Differential pairs
- Sensitive signals requiring additional space for greater cross talk rejection
- Signals with layer limitations
- Signals sensitive to voltage drop, inductance or carry high current

See Before You Start

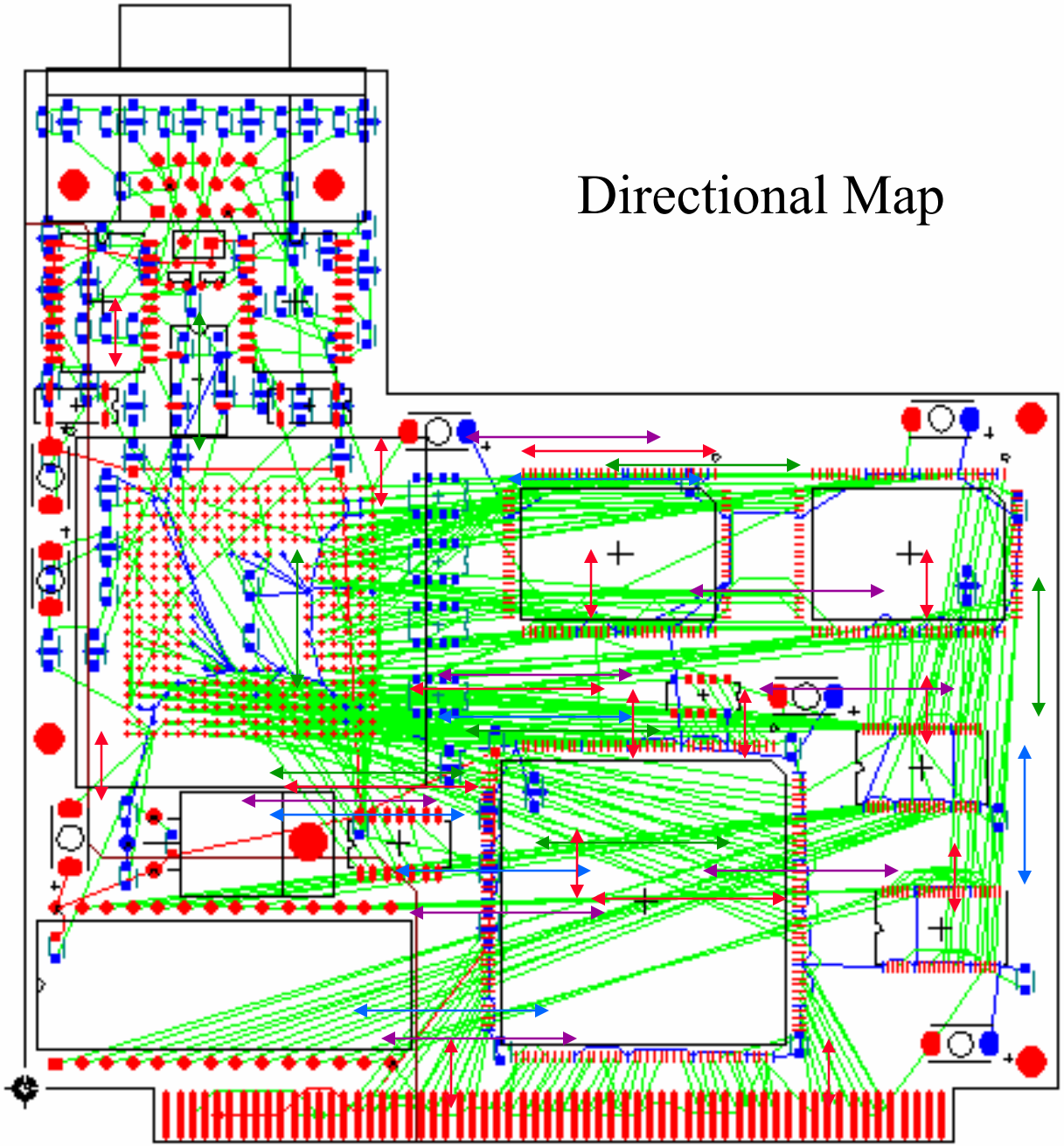
Visualizing the routing is the same as having a set of blue print to build with

If you have difficulty keeping this information in your head use a routing map

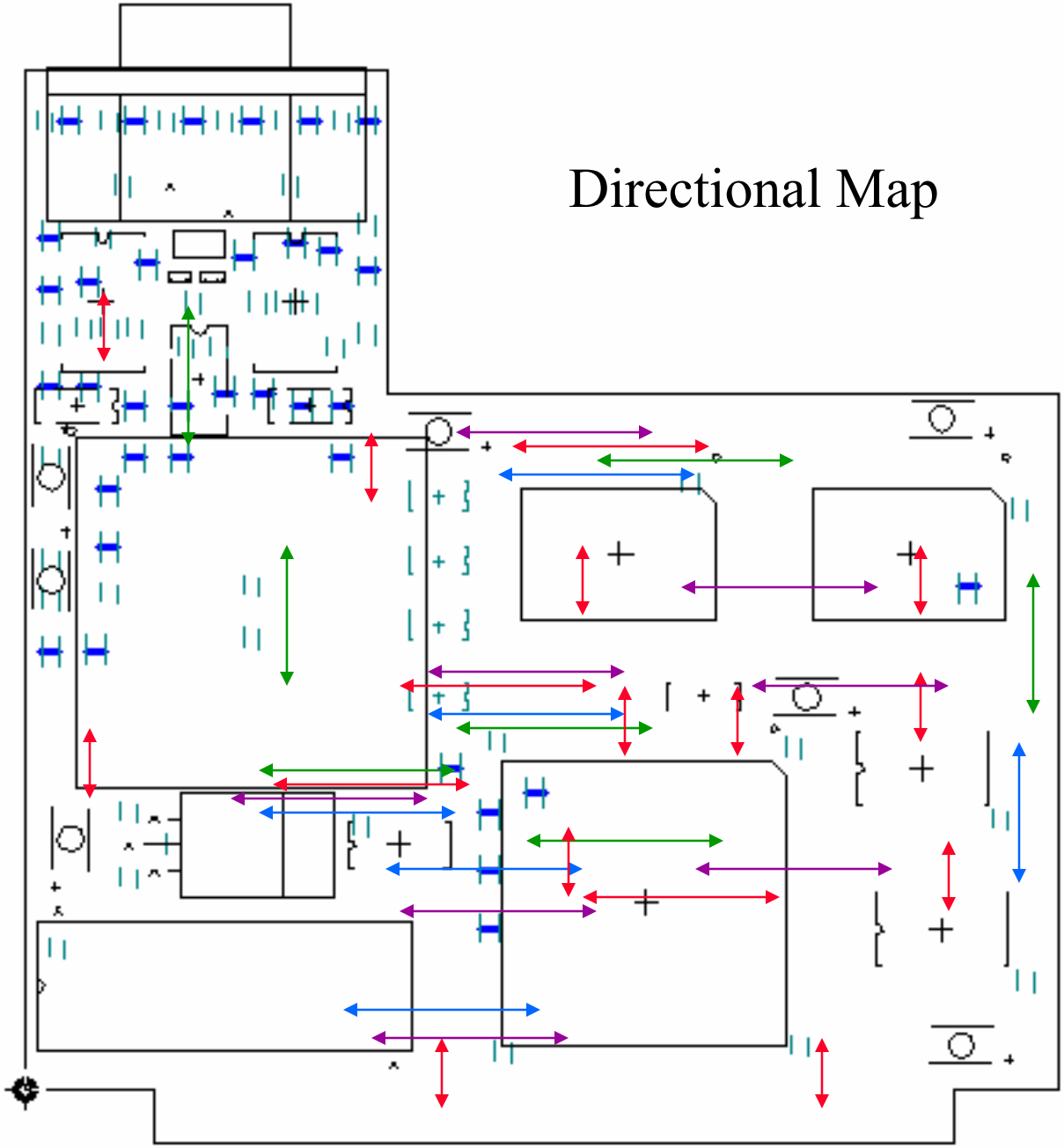
The amount of detail is up to the individual

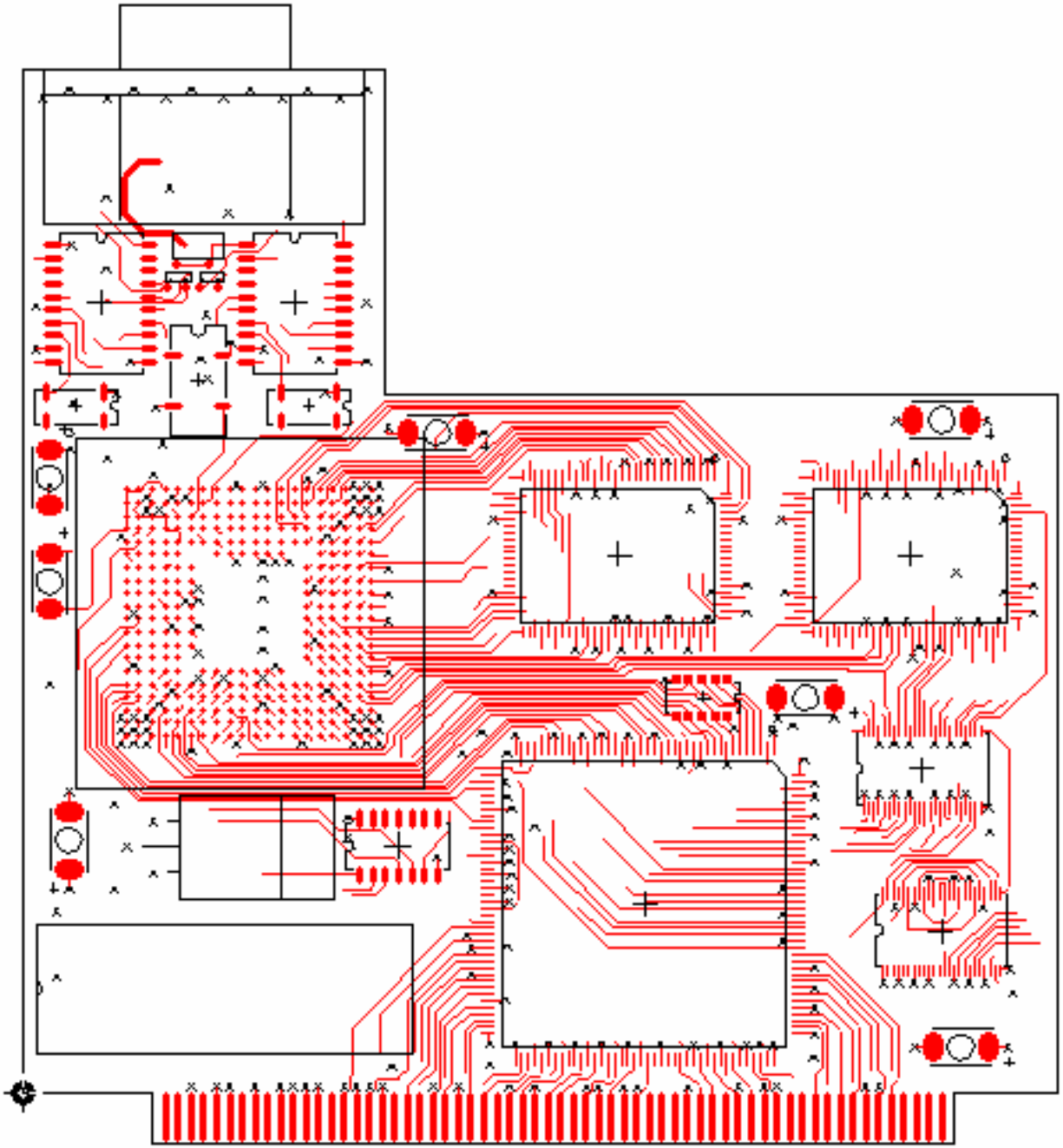
If you have done this well during placement then all that is left is building the board

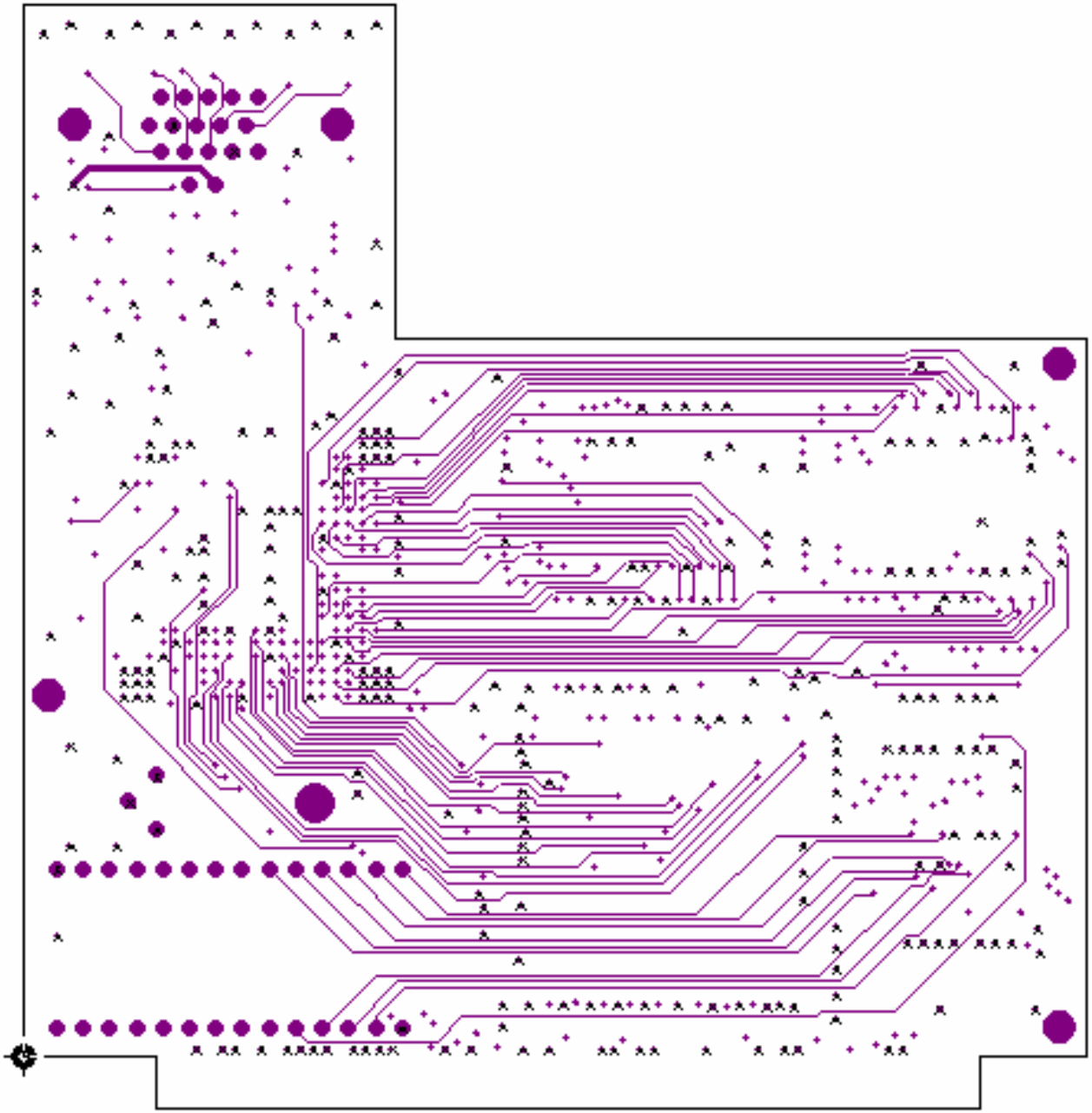
Directional Map

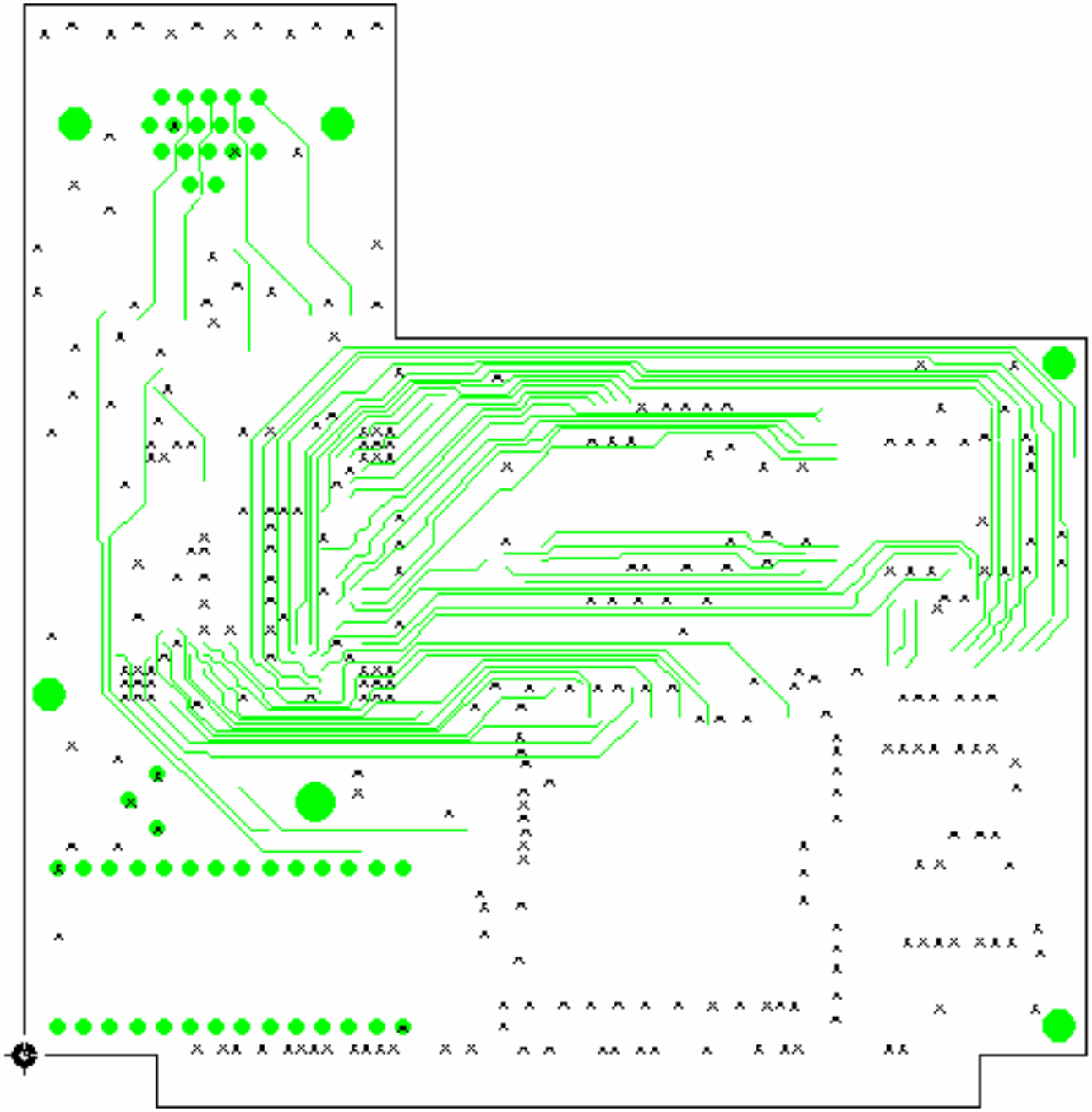


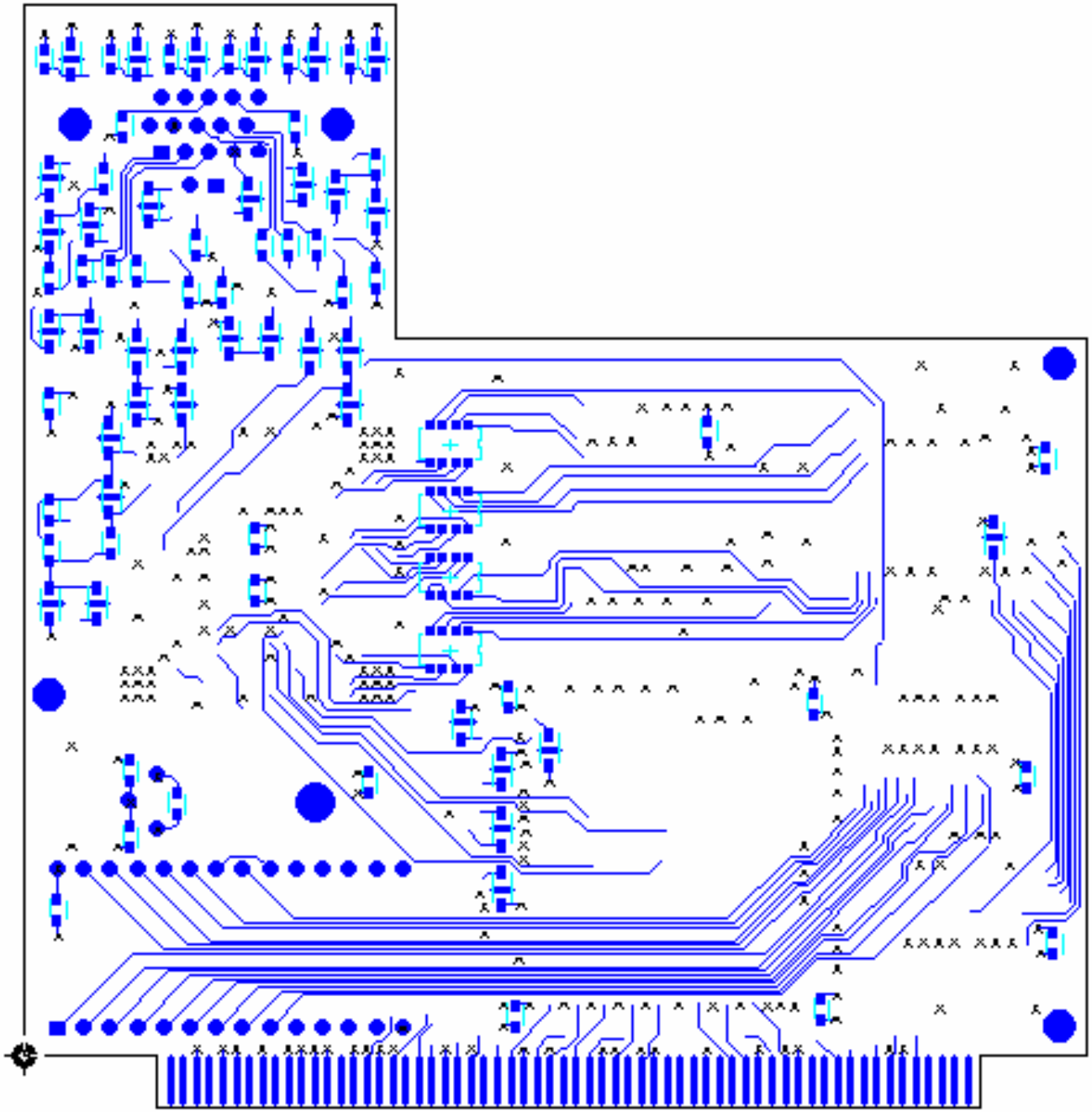
Directional Map

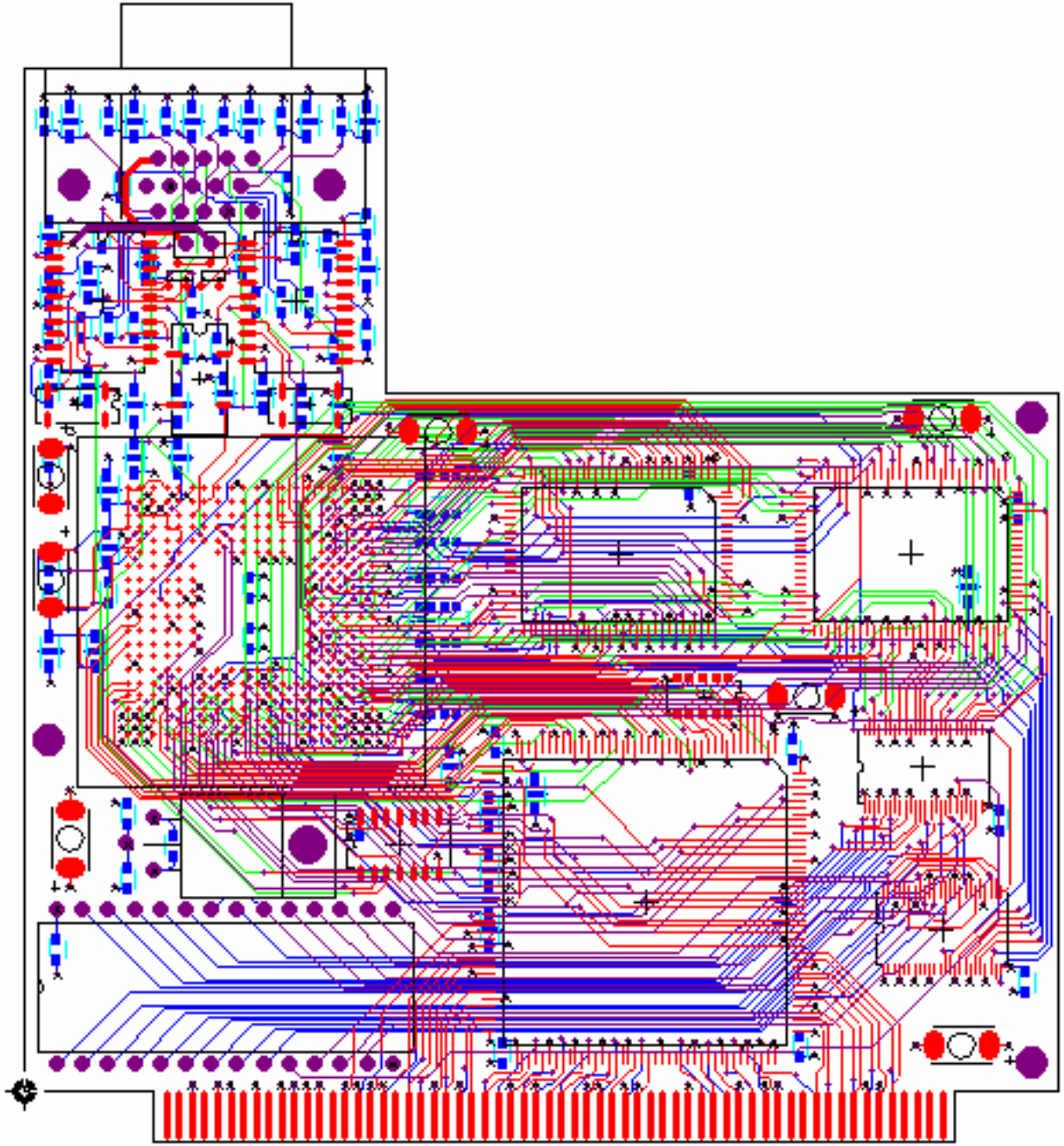


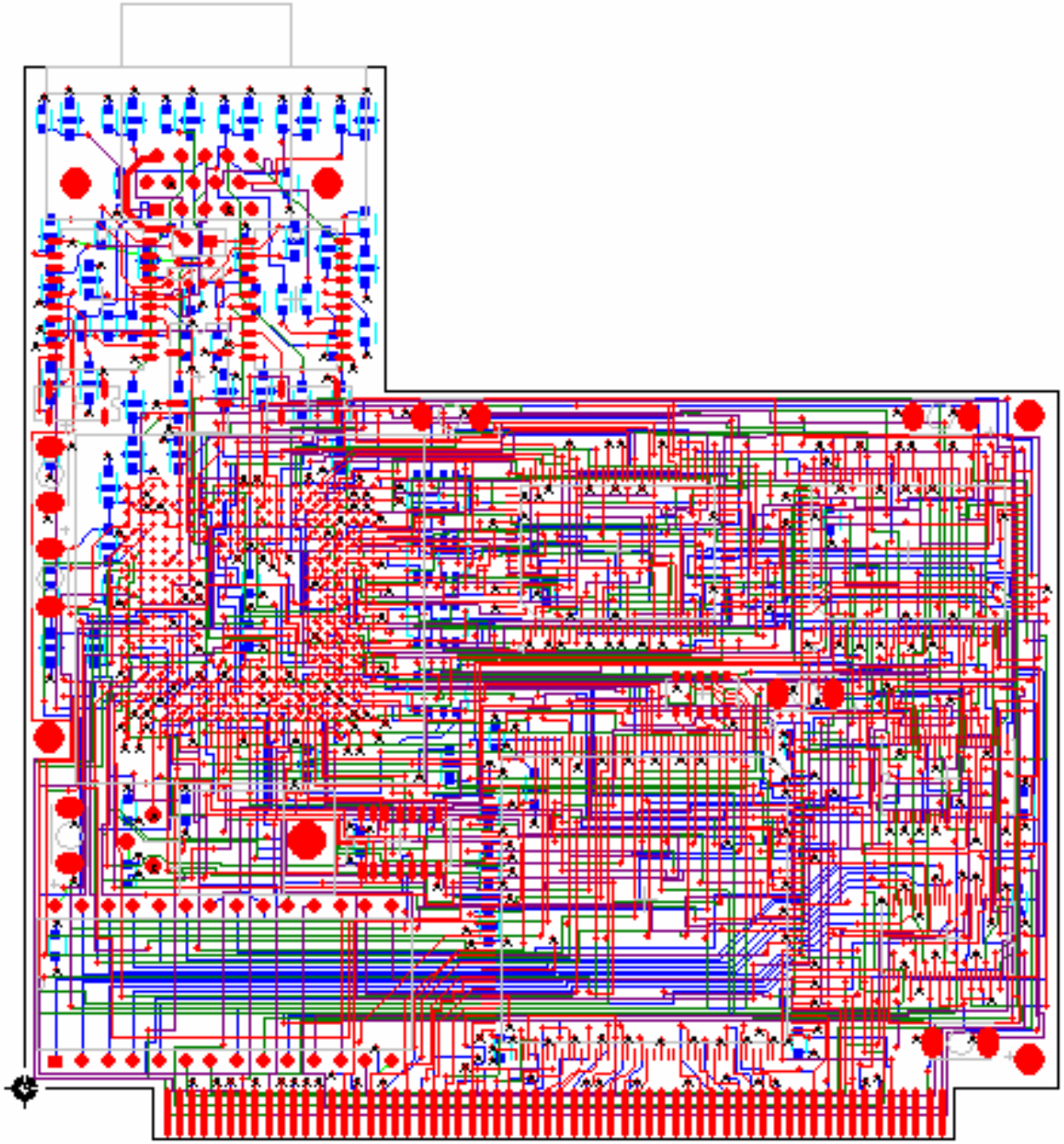


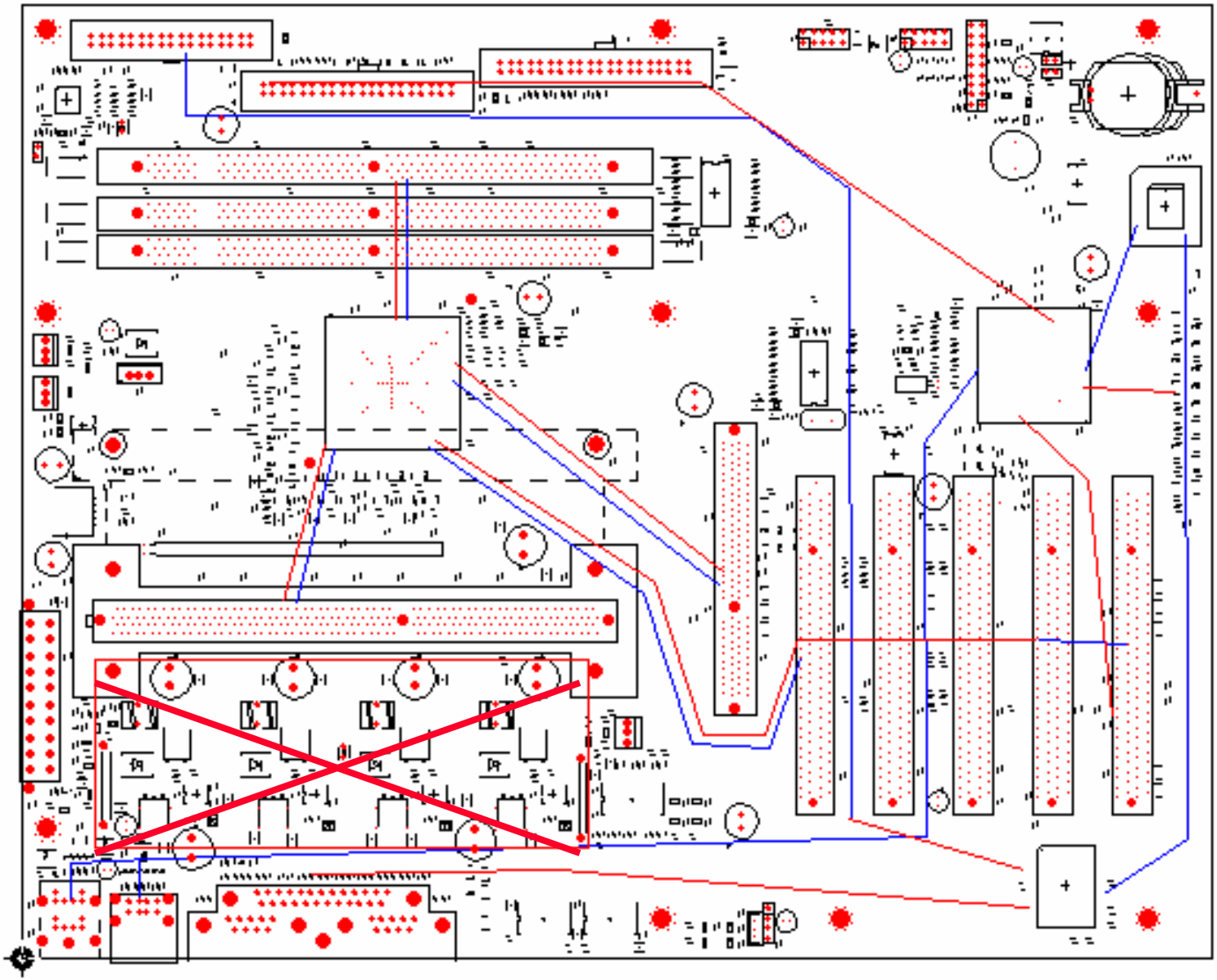


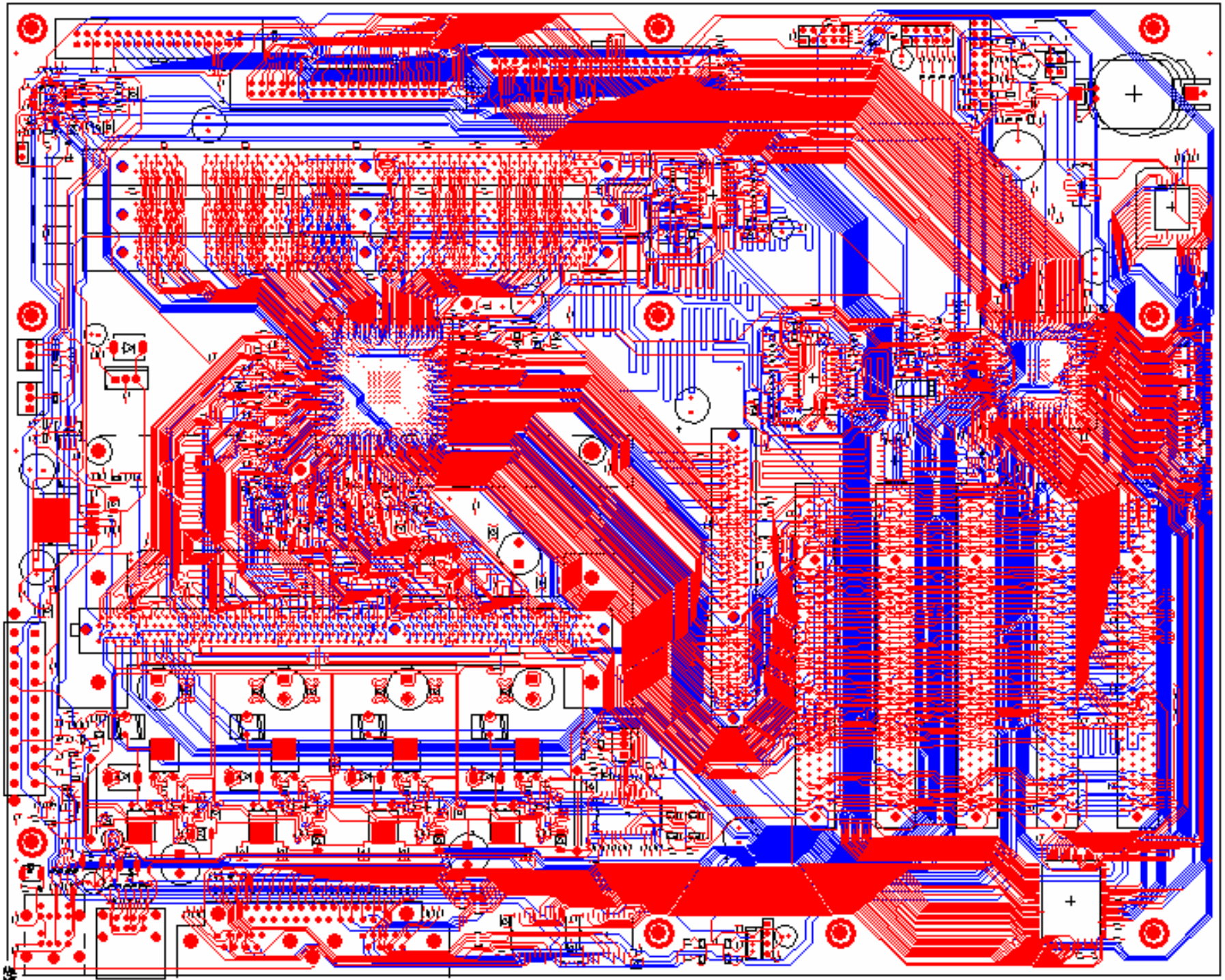


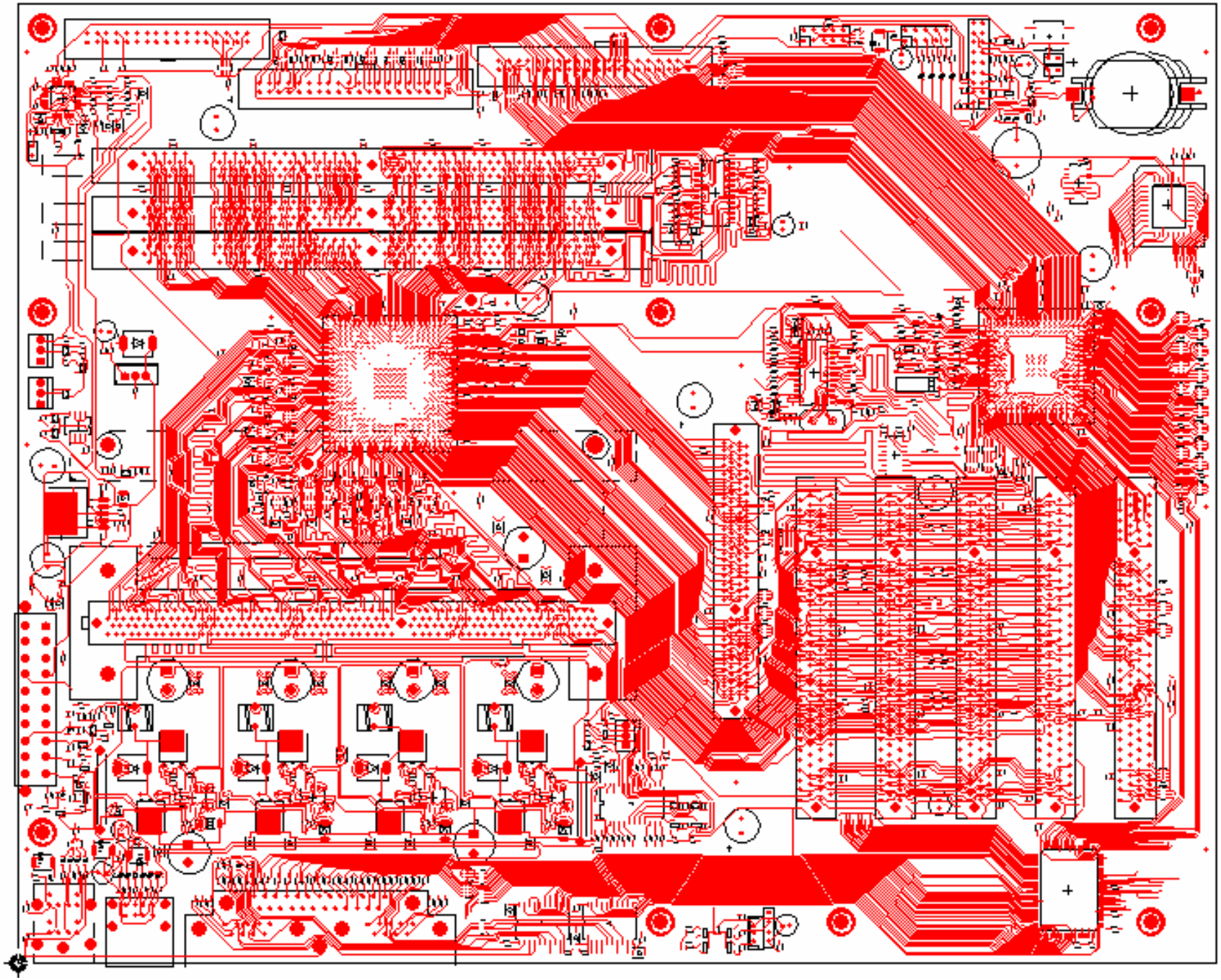


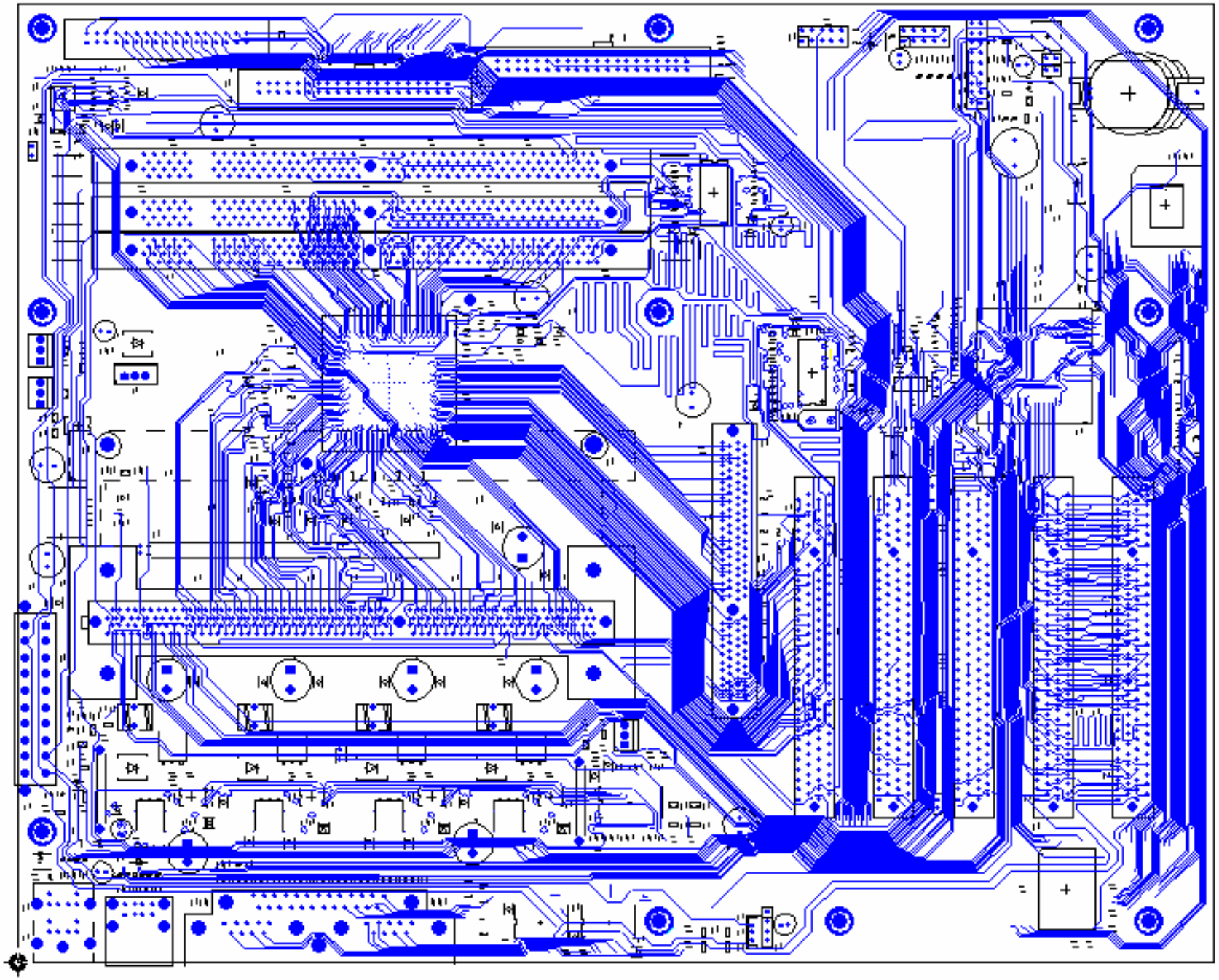










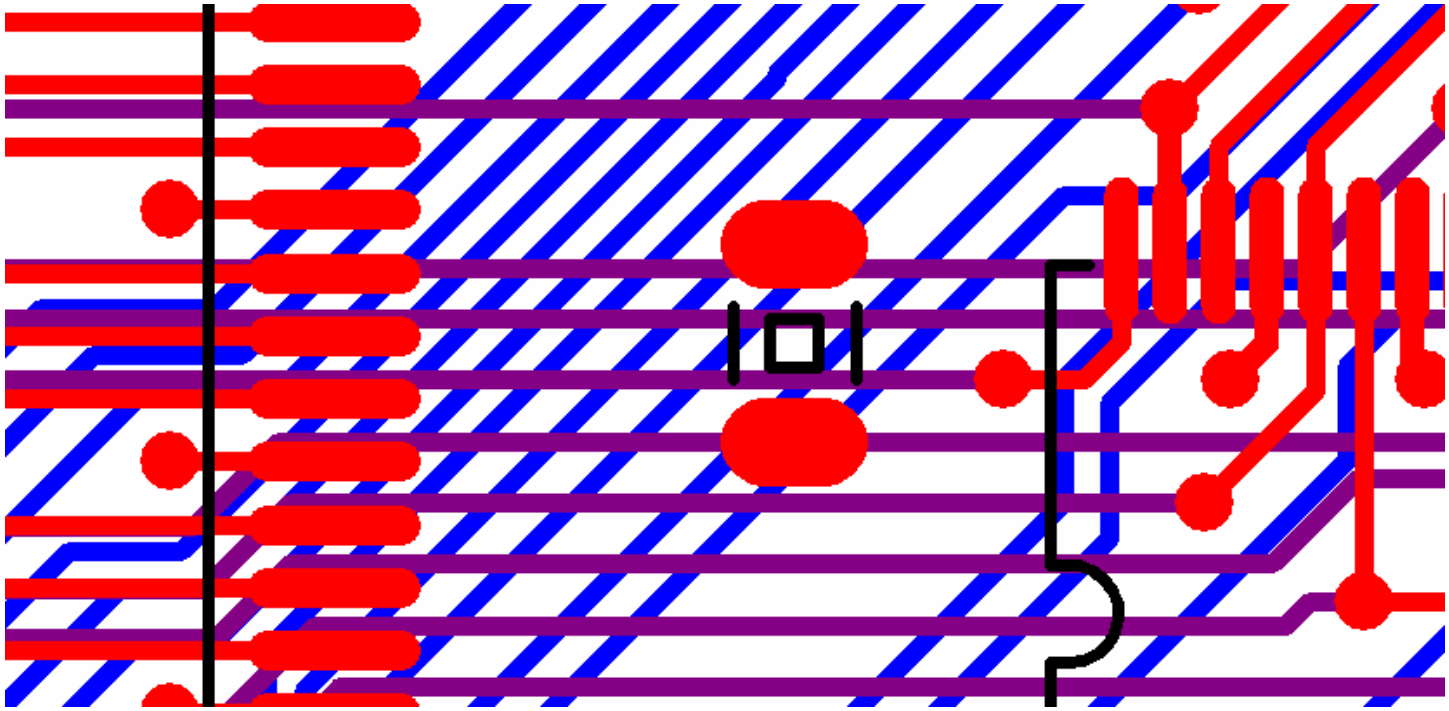


Power fan out

Make all planer power connections first

Confirms power plane division

Eliminates the possibility of power connection difficulty later

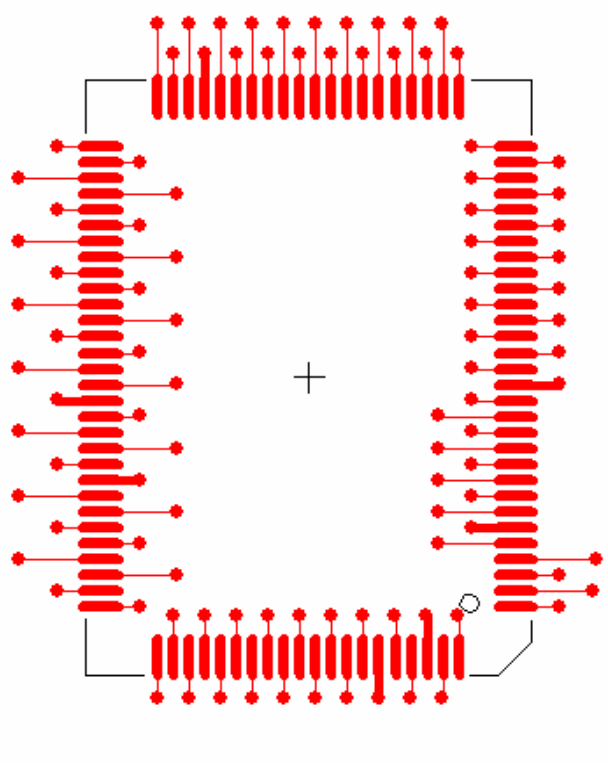


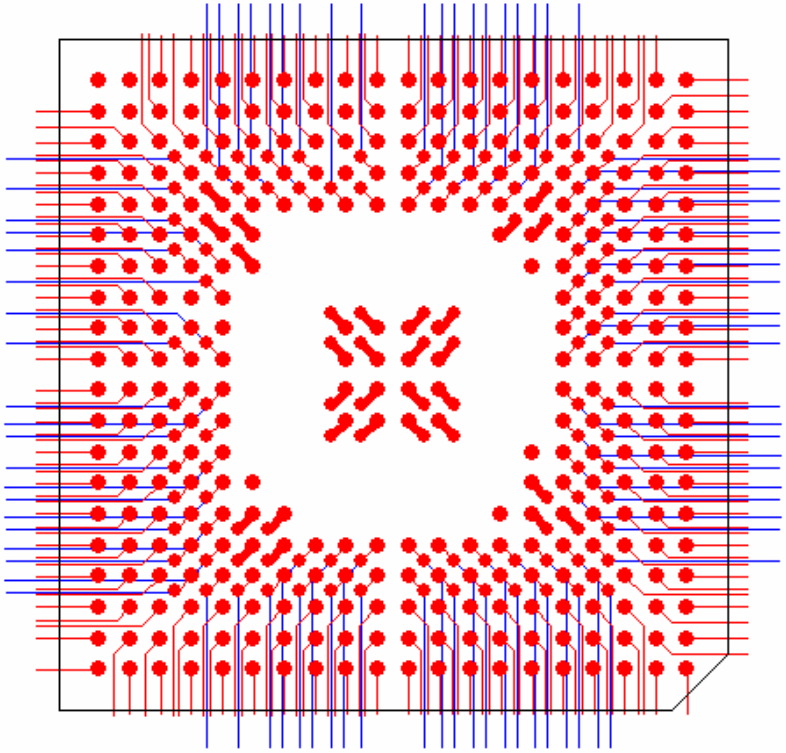
Escape

Establish breakout pattern for high density parts

Repeating patterns are easily copied

Prevents you from having difficultly later

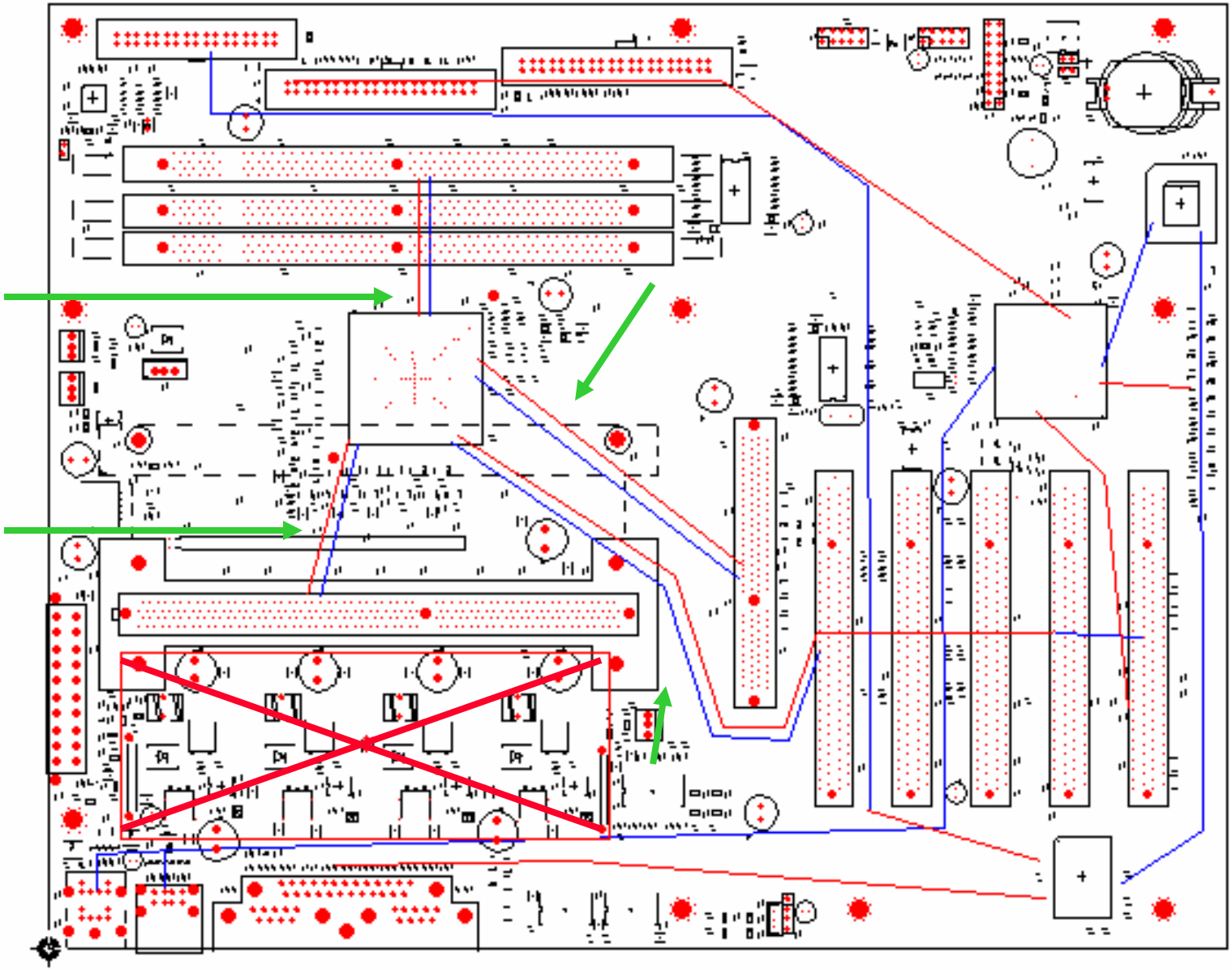




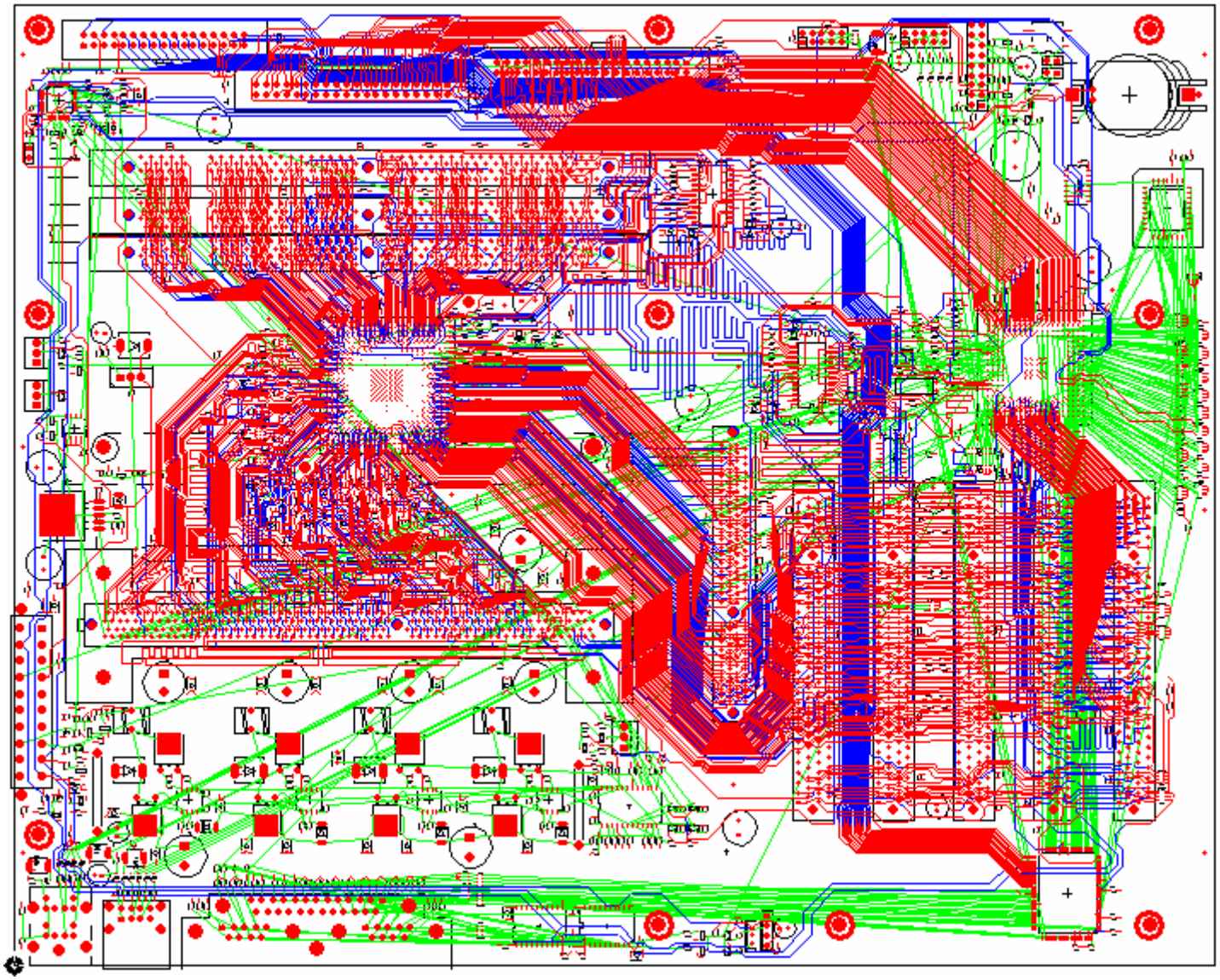
Rough In

- Bus first routing method
- One pass routing method
- Route problem areas first
  - Start at high density parts and work out
  - As you route adjust parts as necessary
  - Pin swap and gate swap as necessary
- Save clean up for later

Bus First

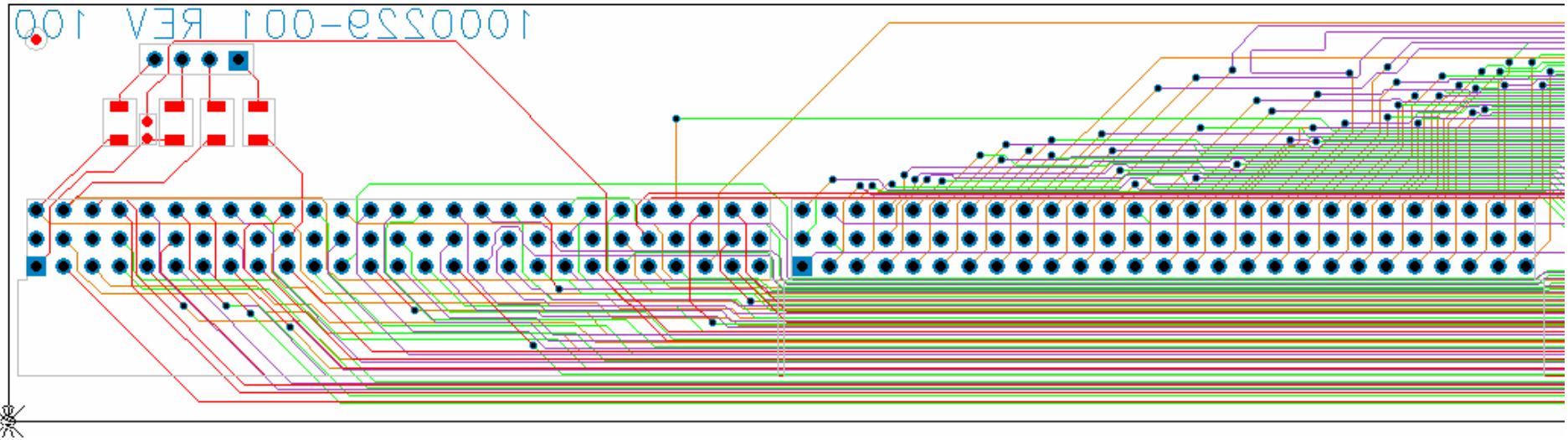


Bus First

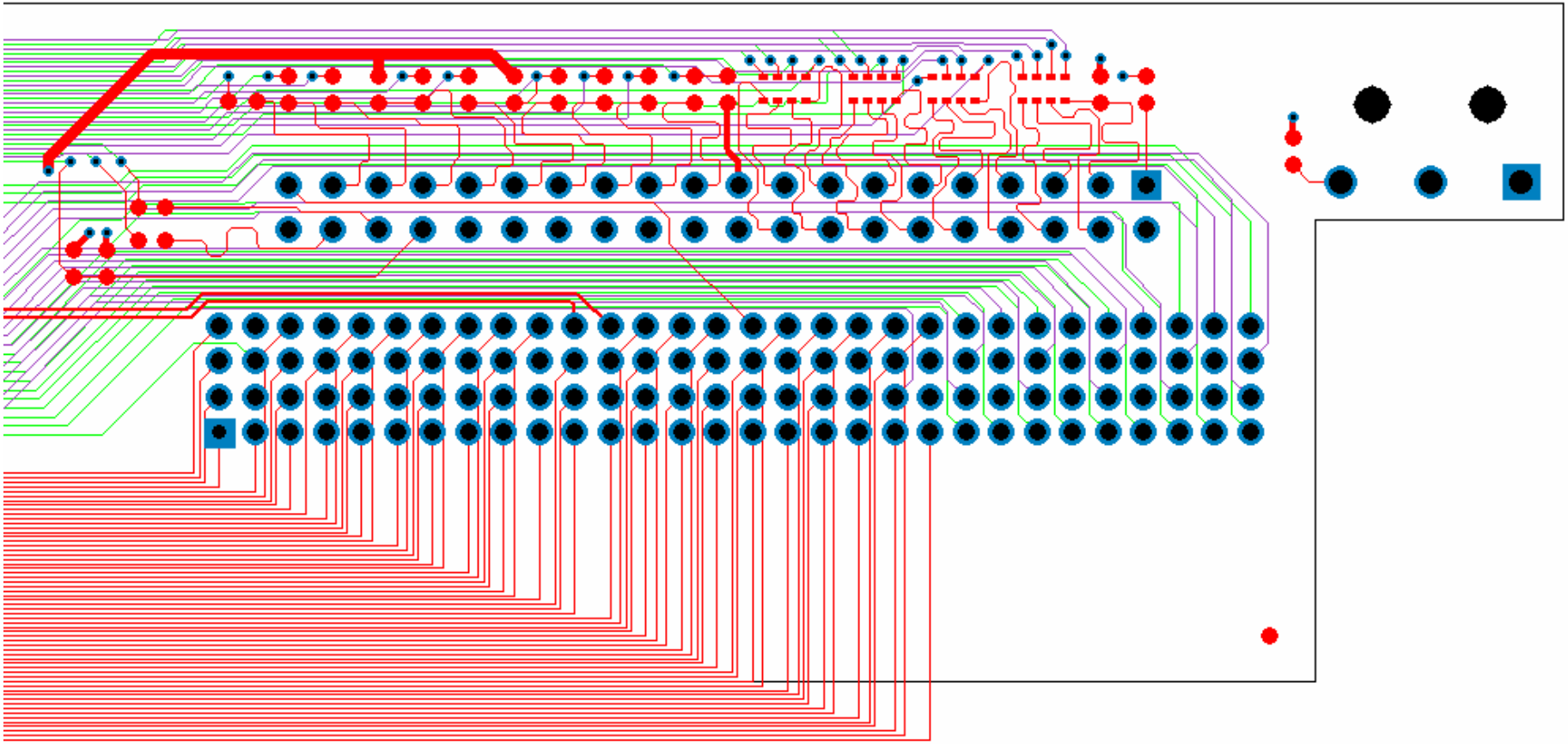


One Pass Routing

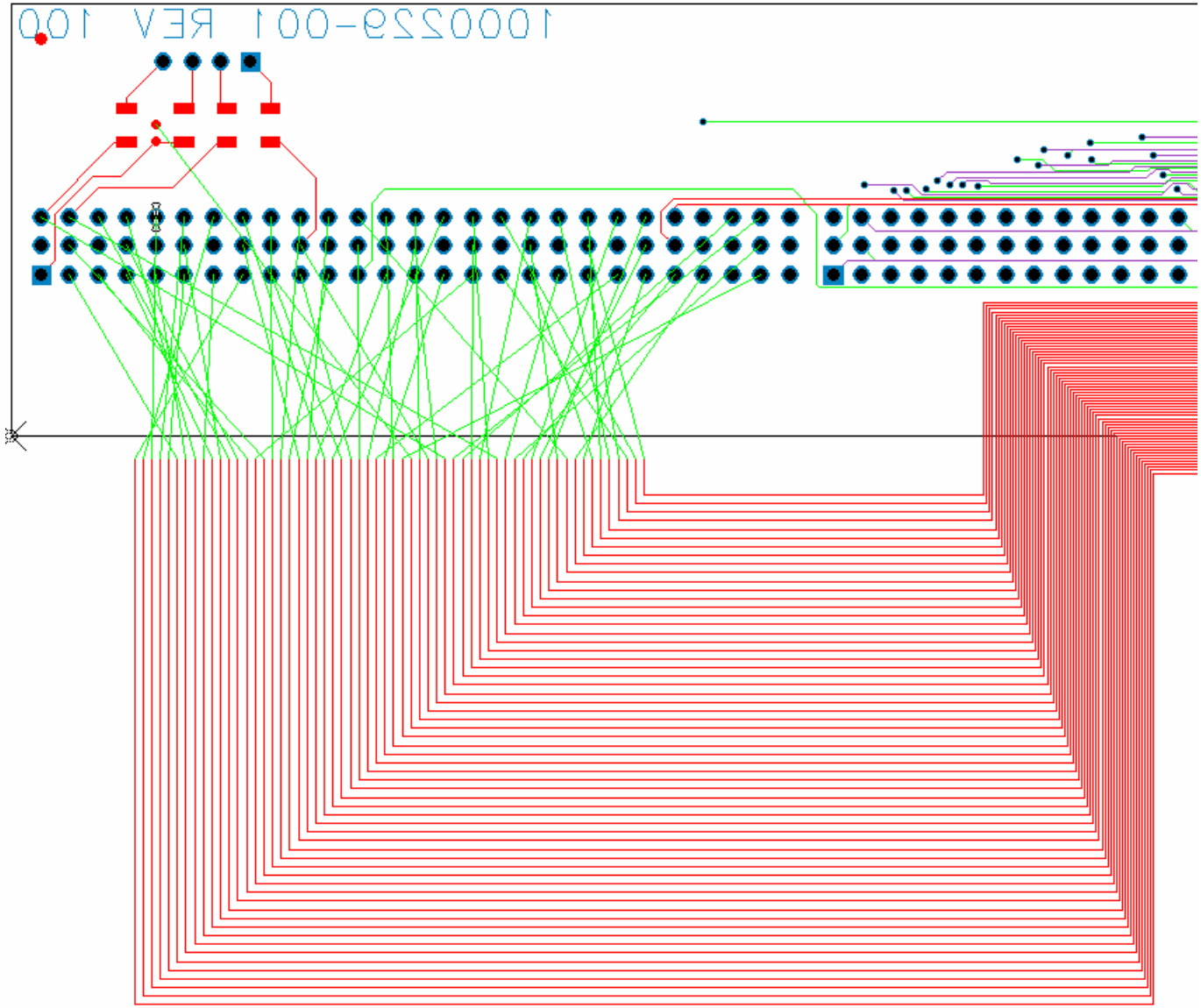
One Pass Routing



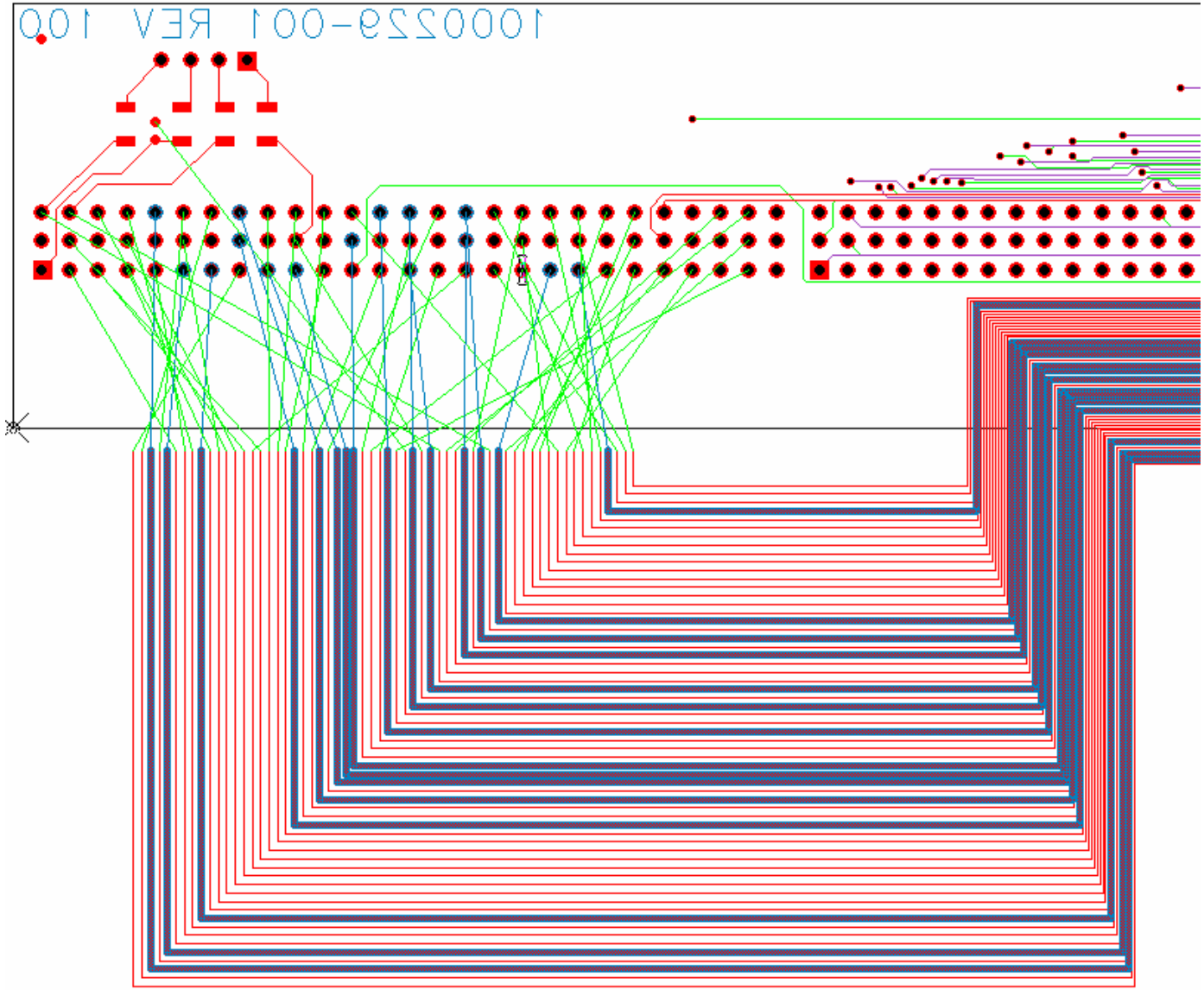
One Pass Routing



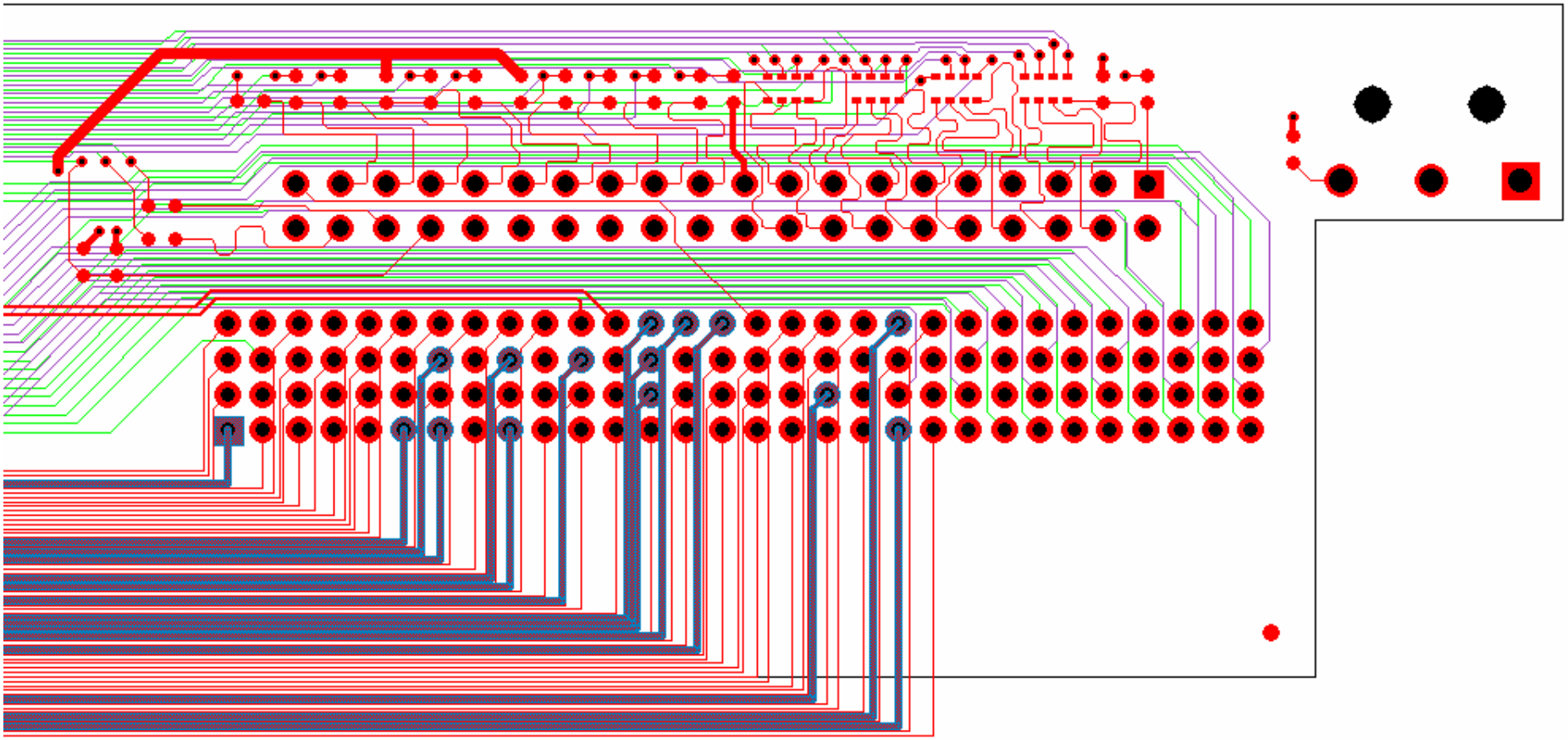
One Pass Routing



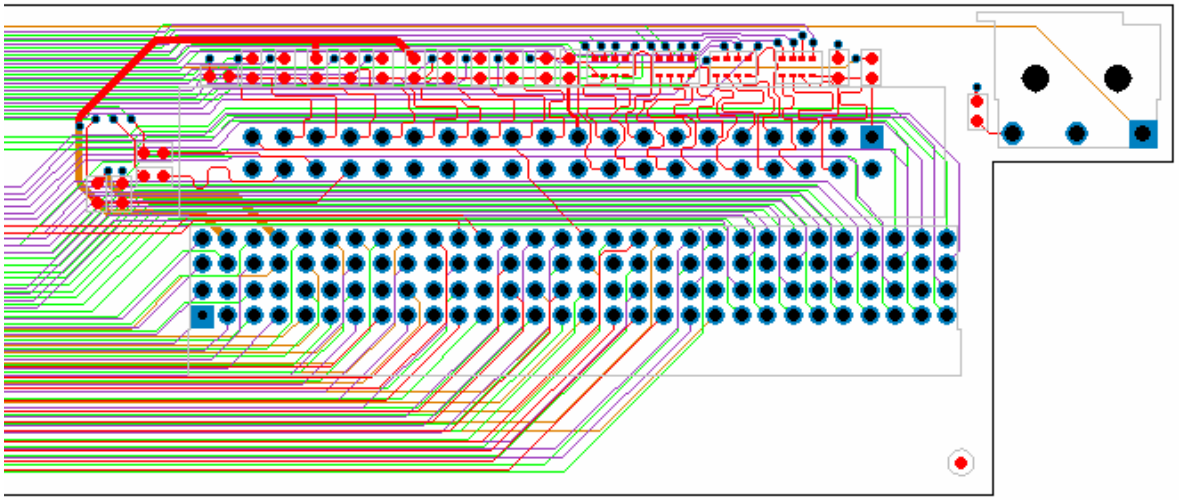
One Pass Routing

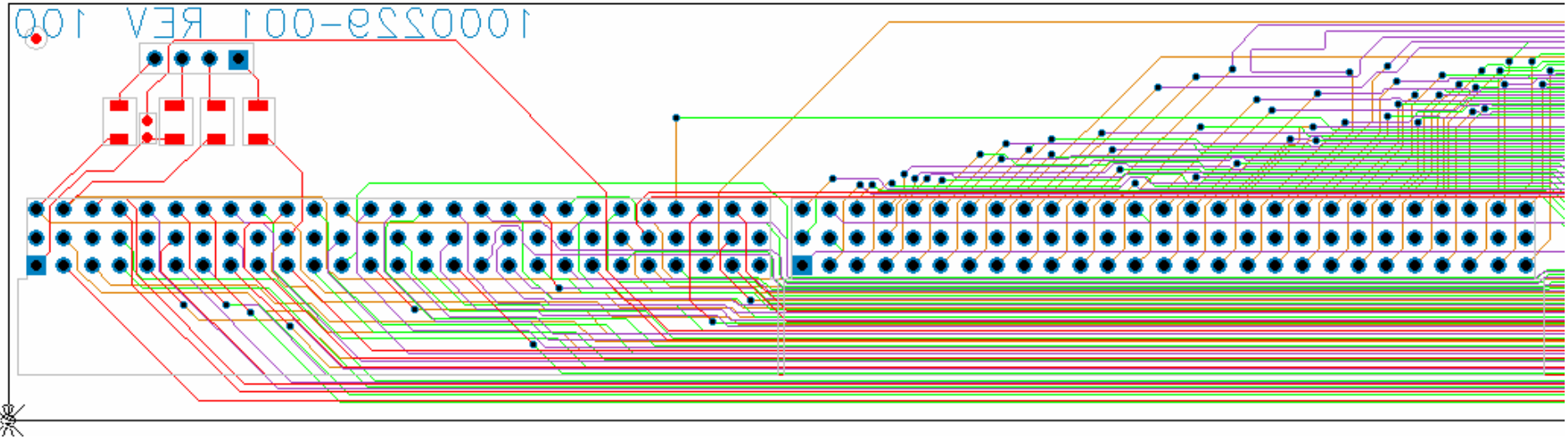


One Pass Routing

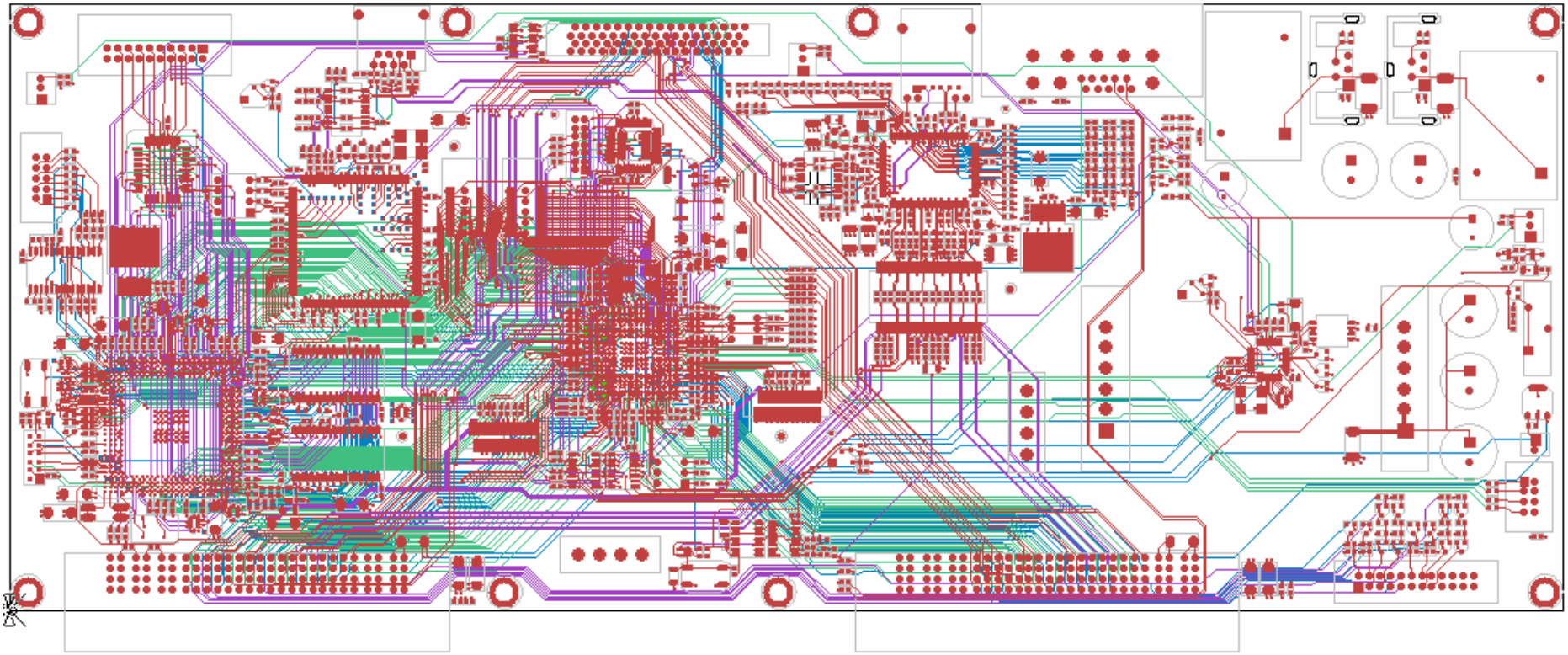


One Pass Routing

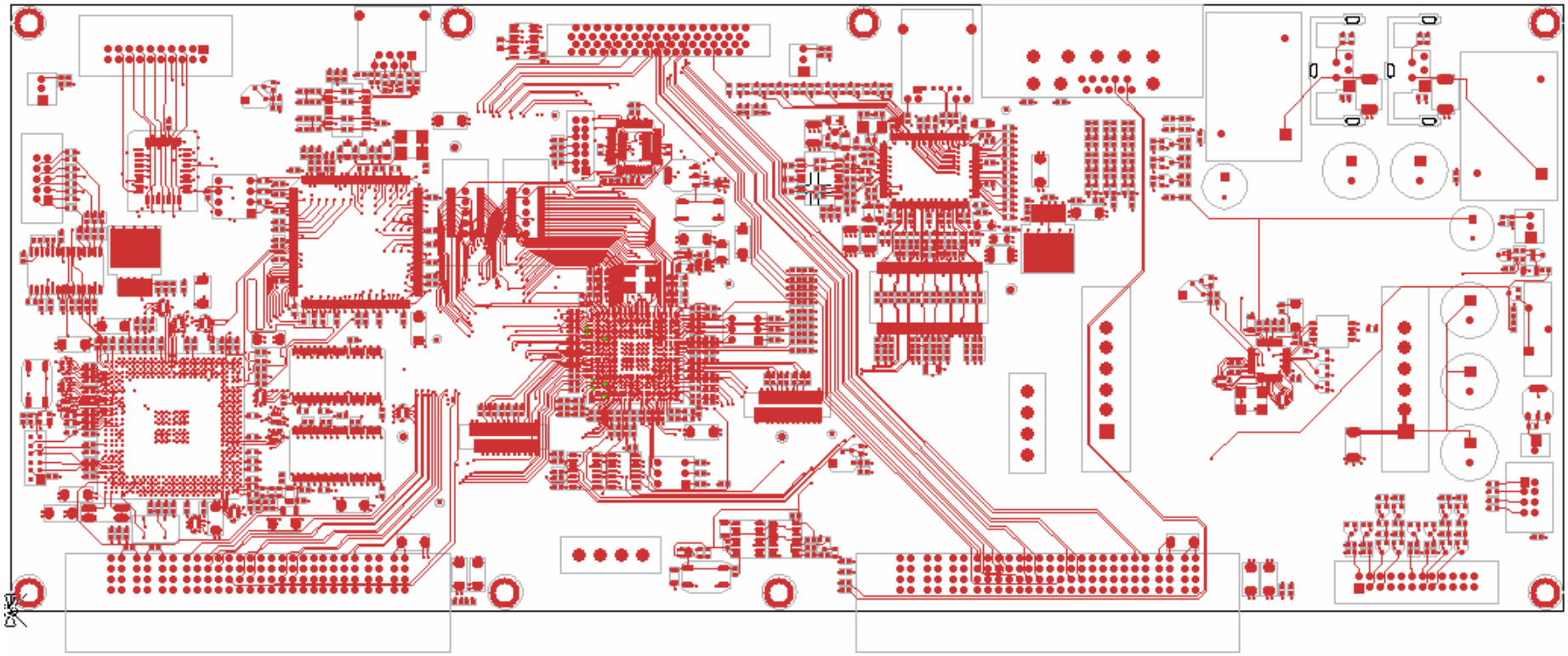




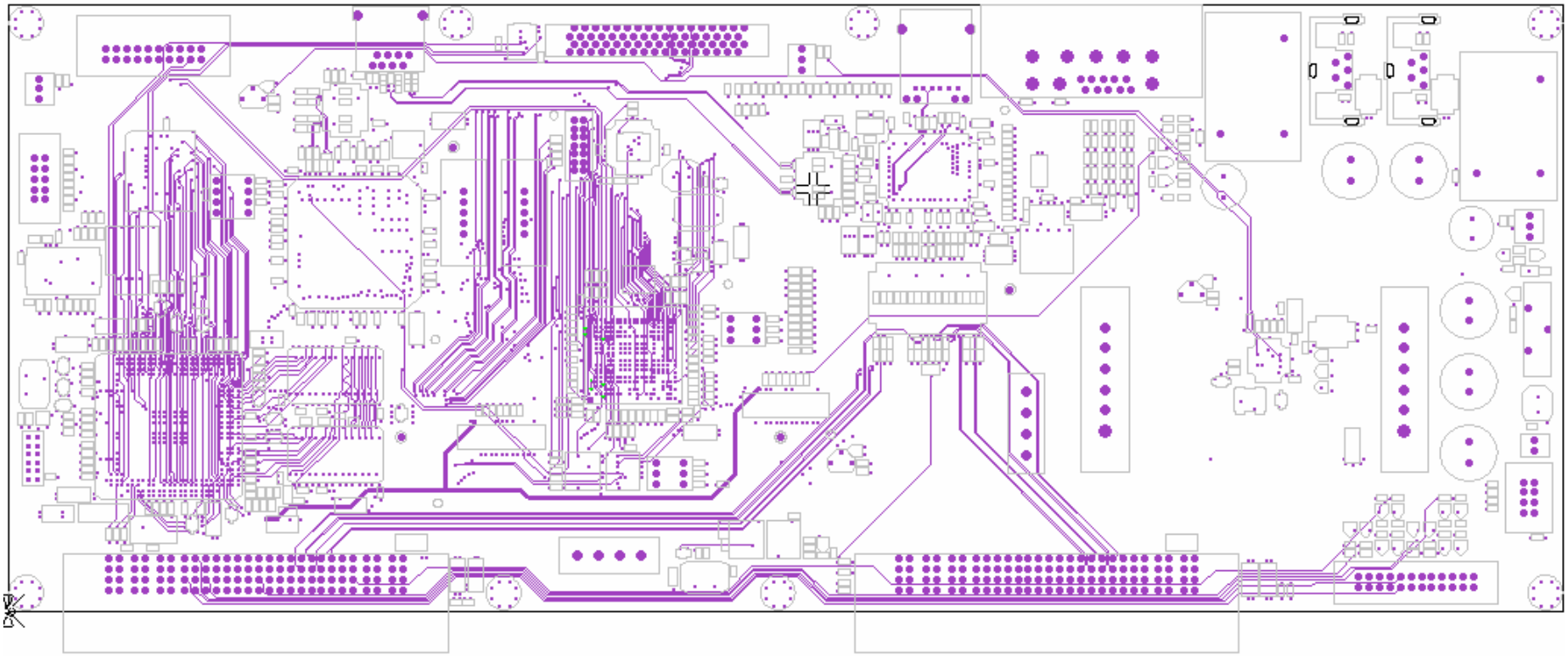
Swapping



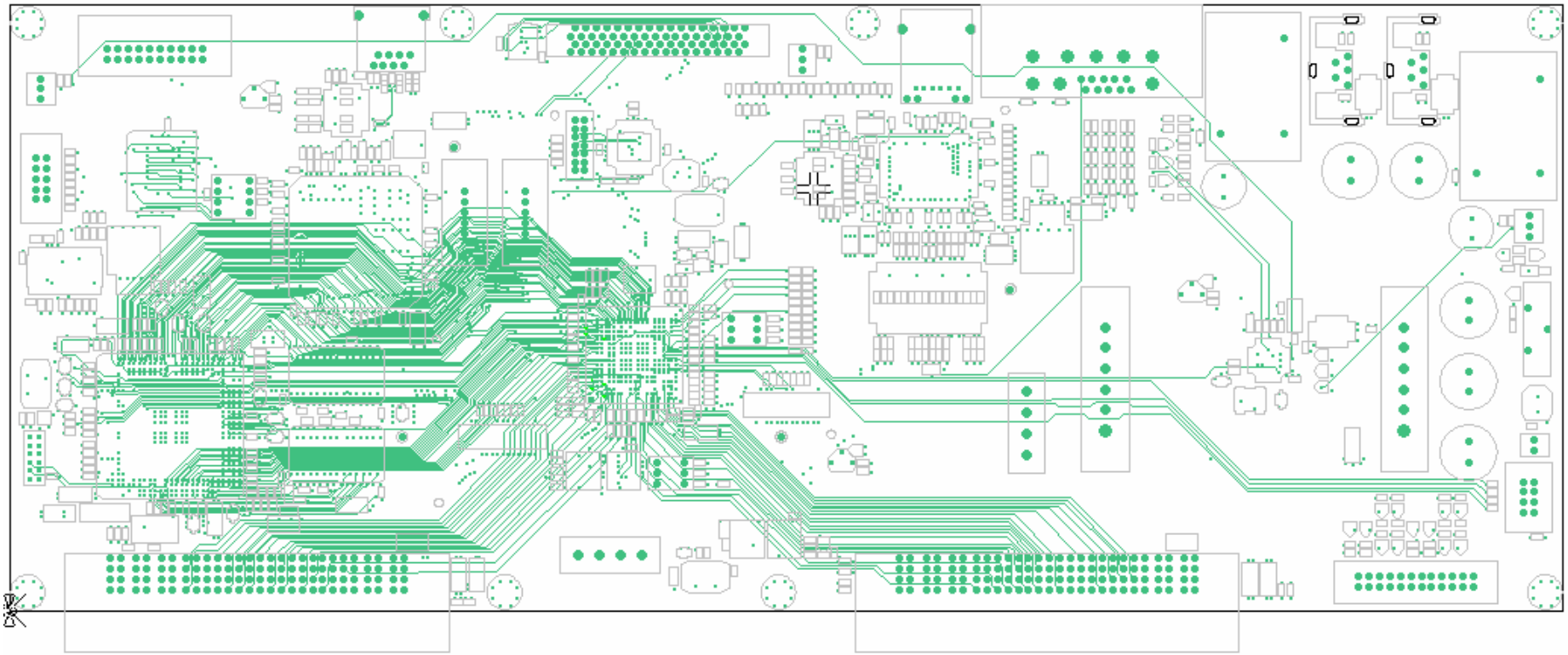
Swapping



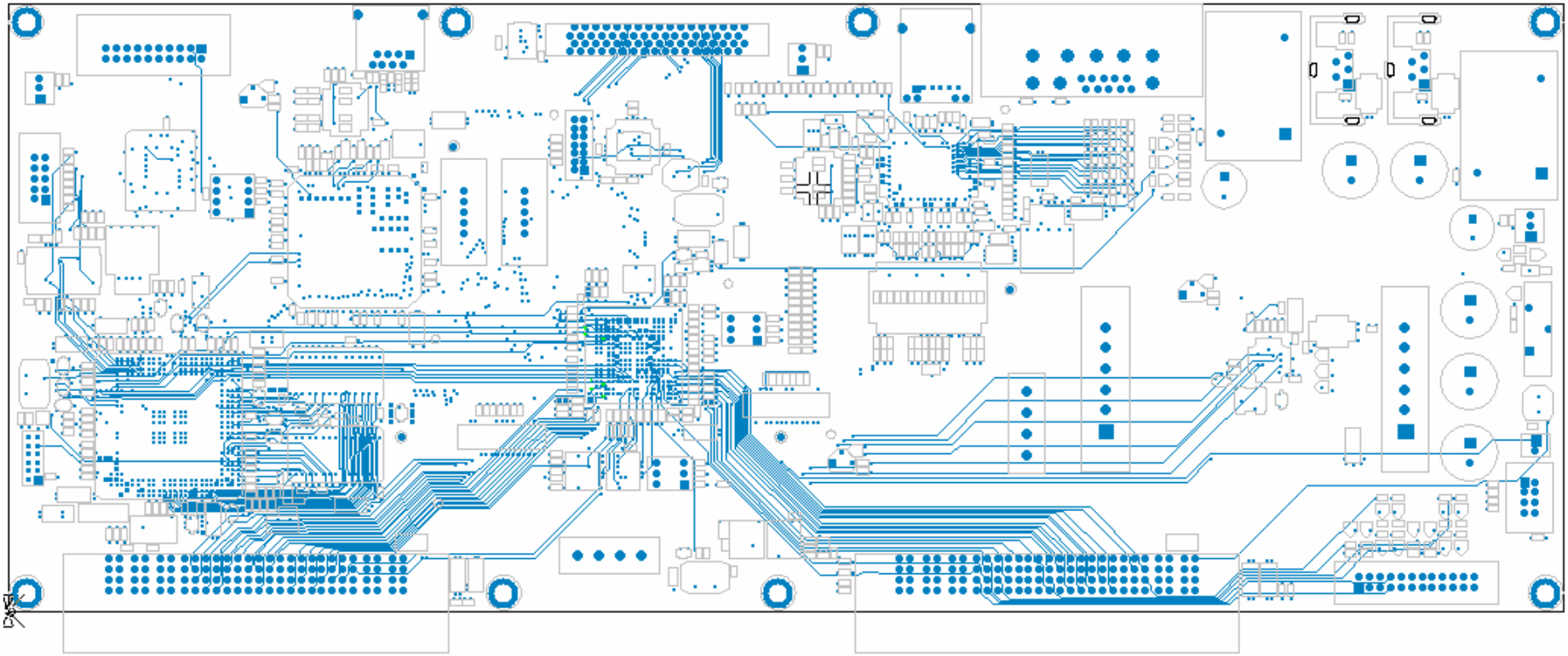
Swapping



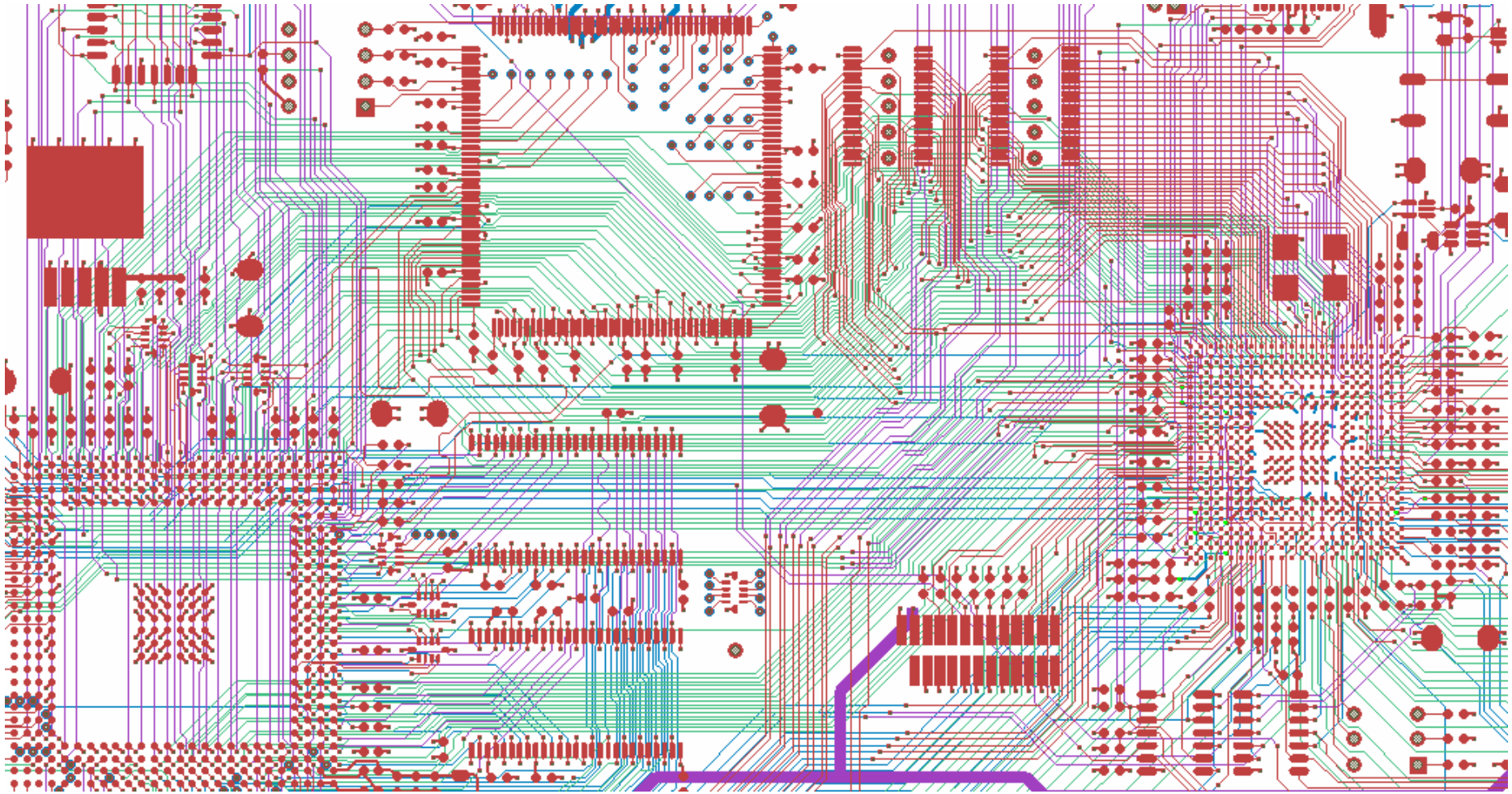
Swapping



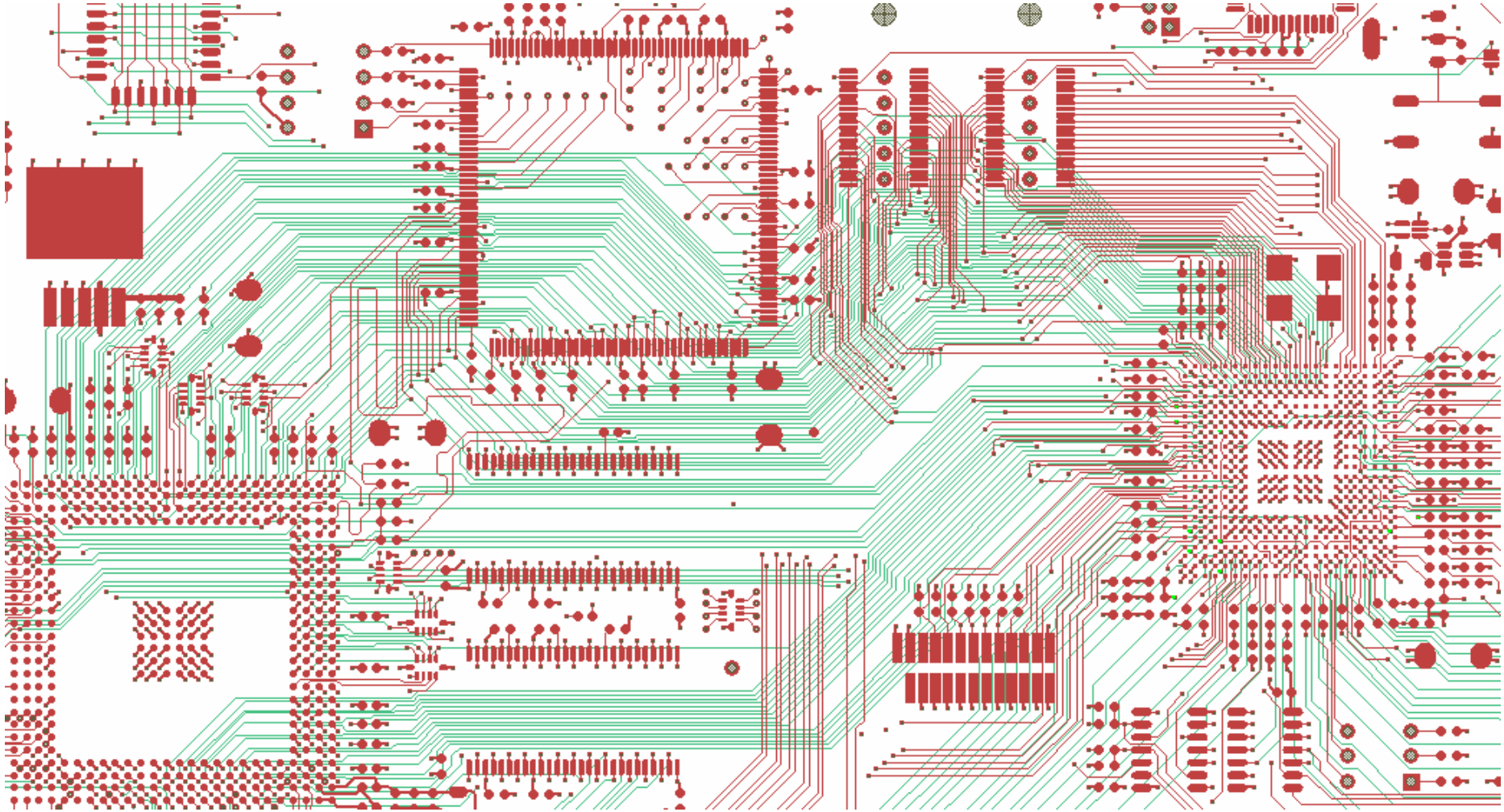
Swapping



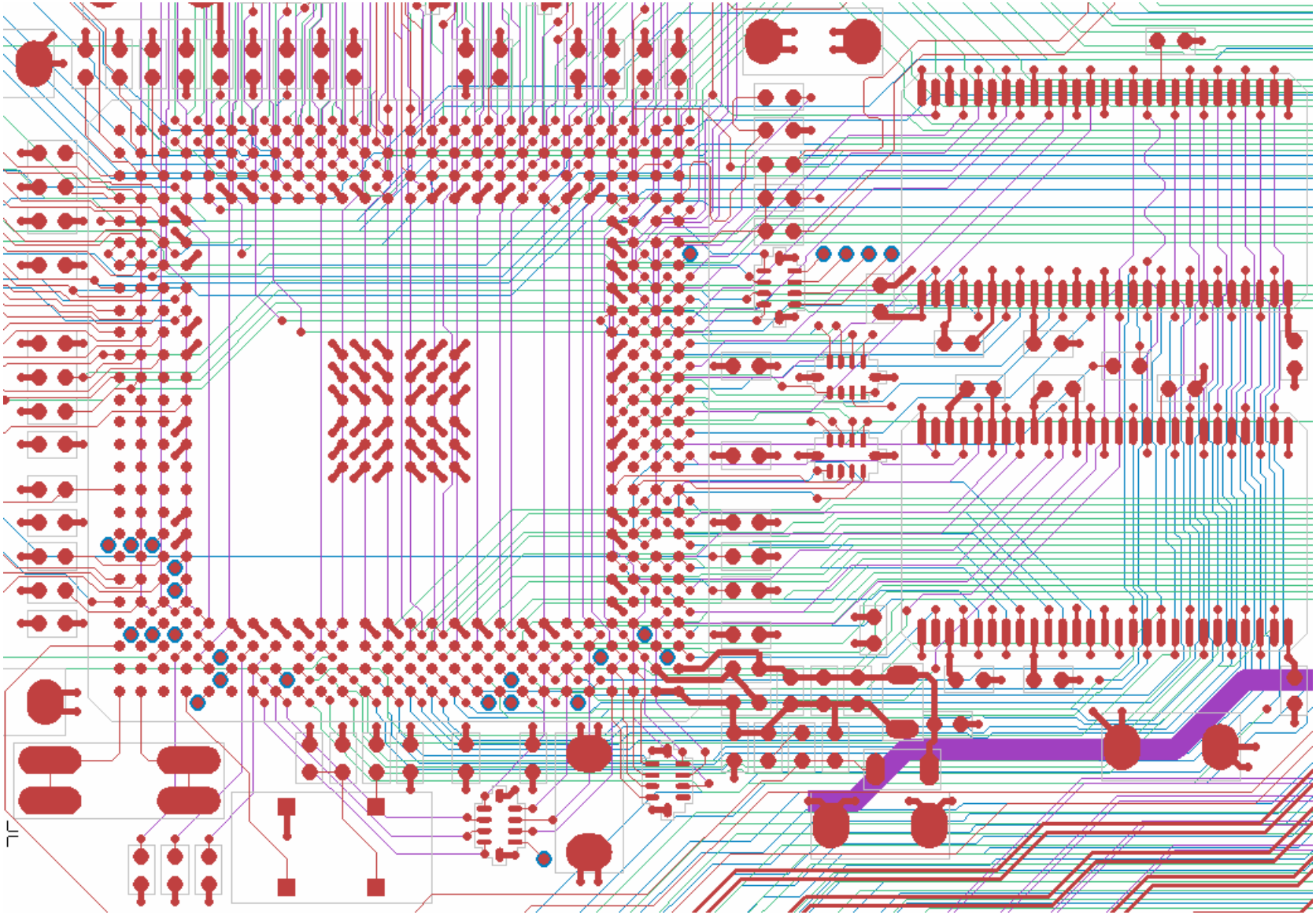
Swapping



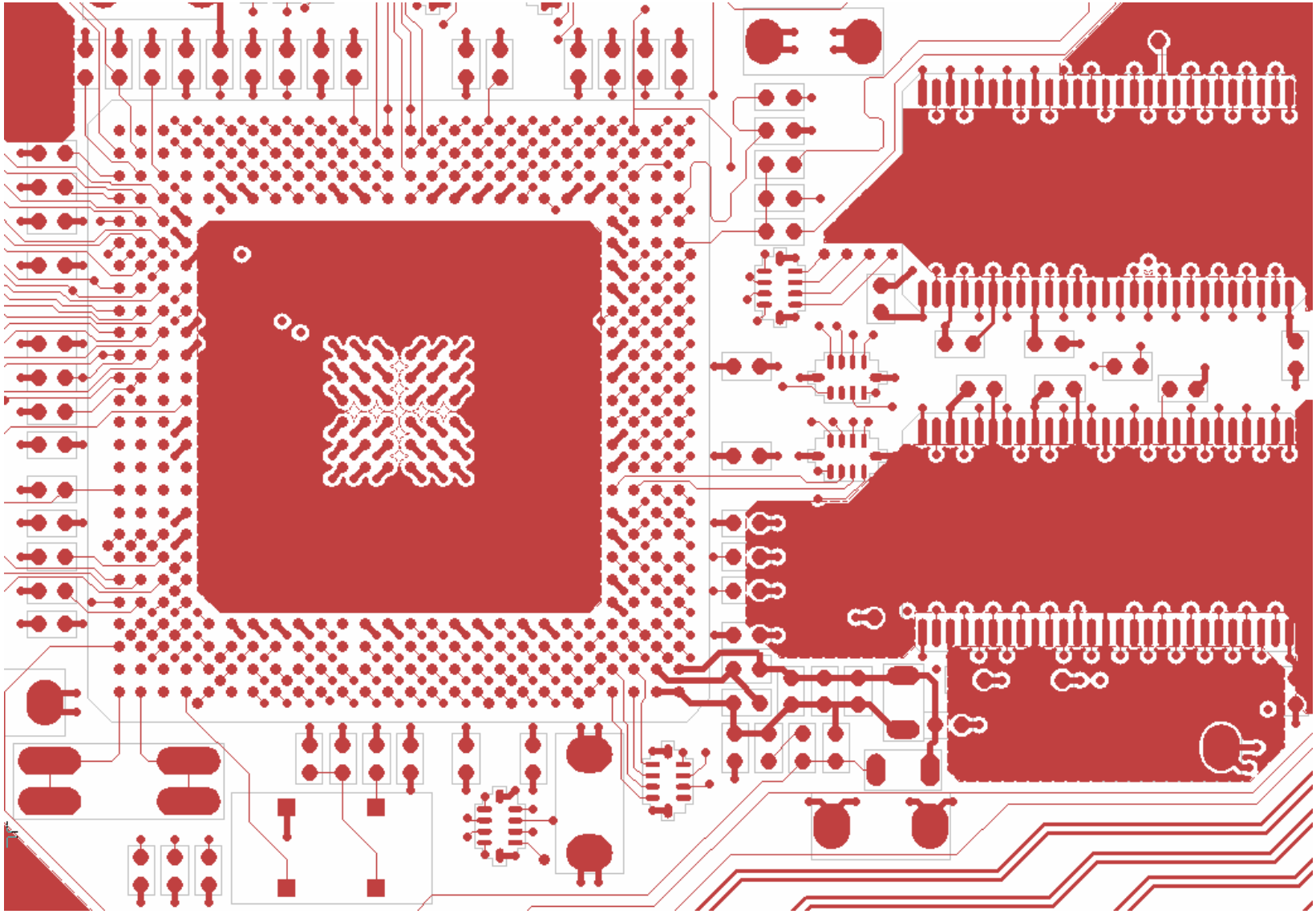
Swapping



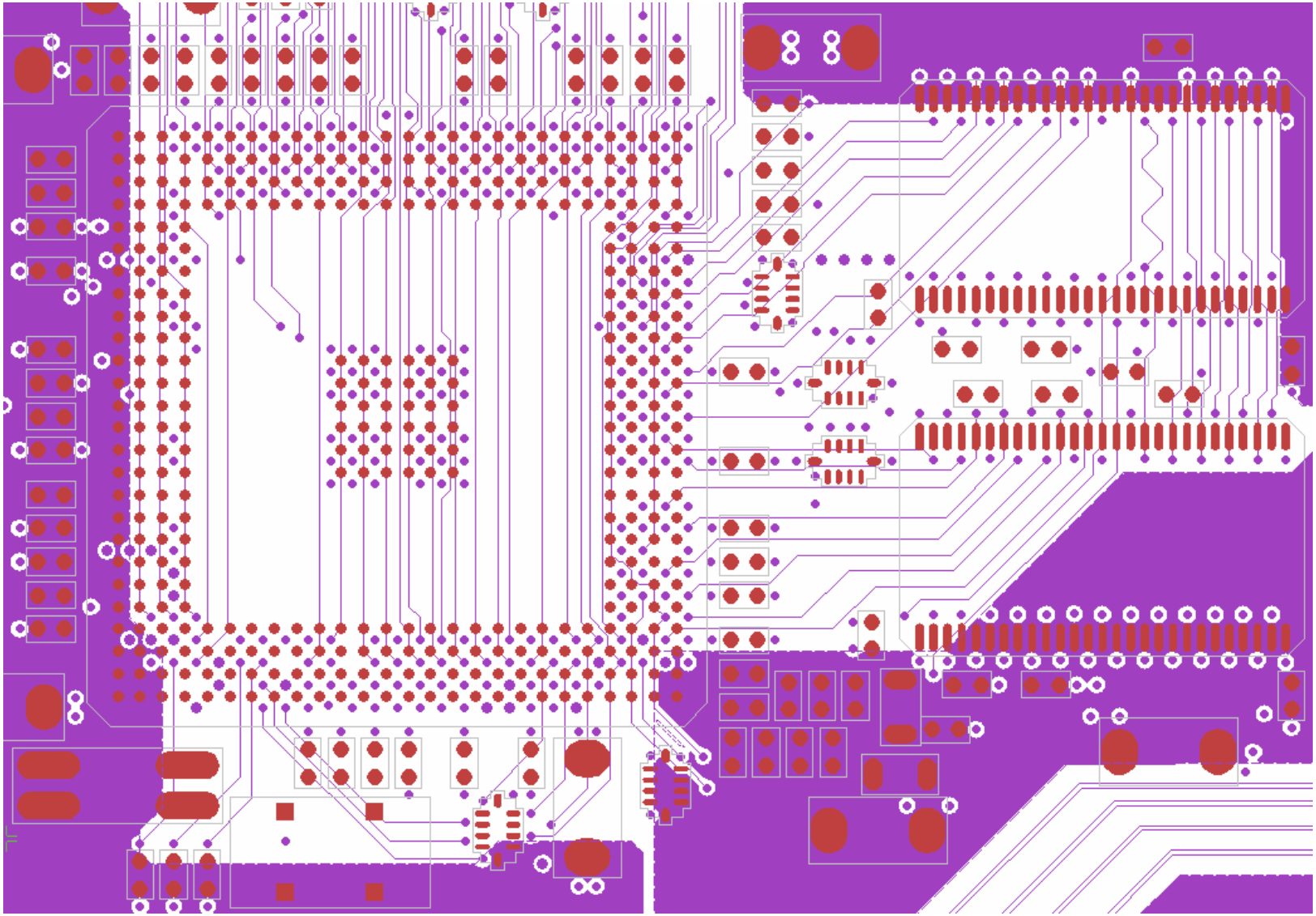
Swapping



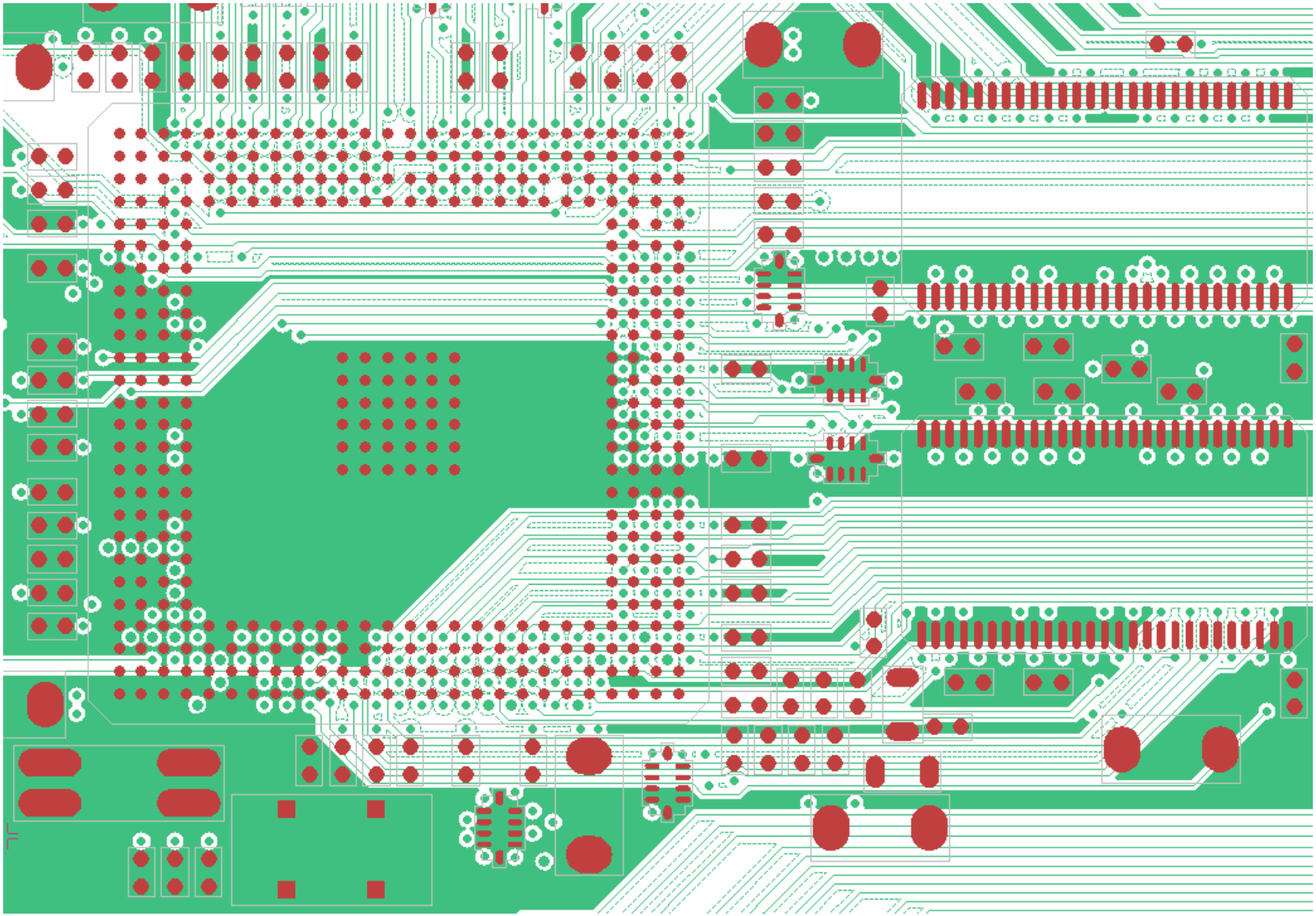
Swapping



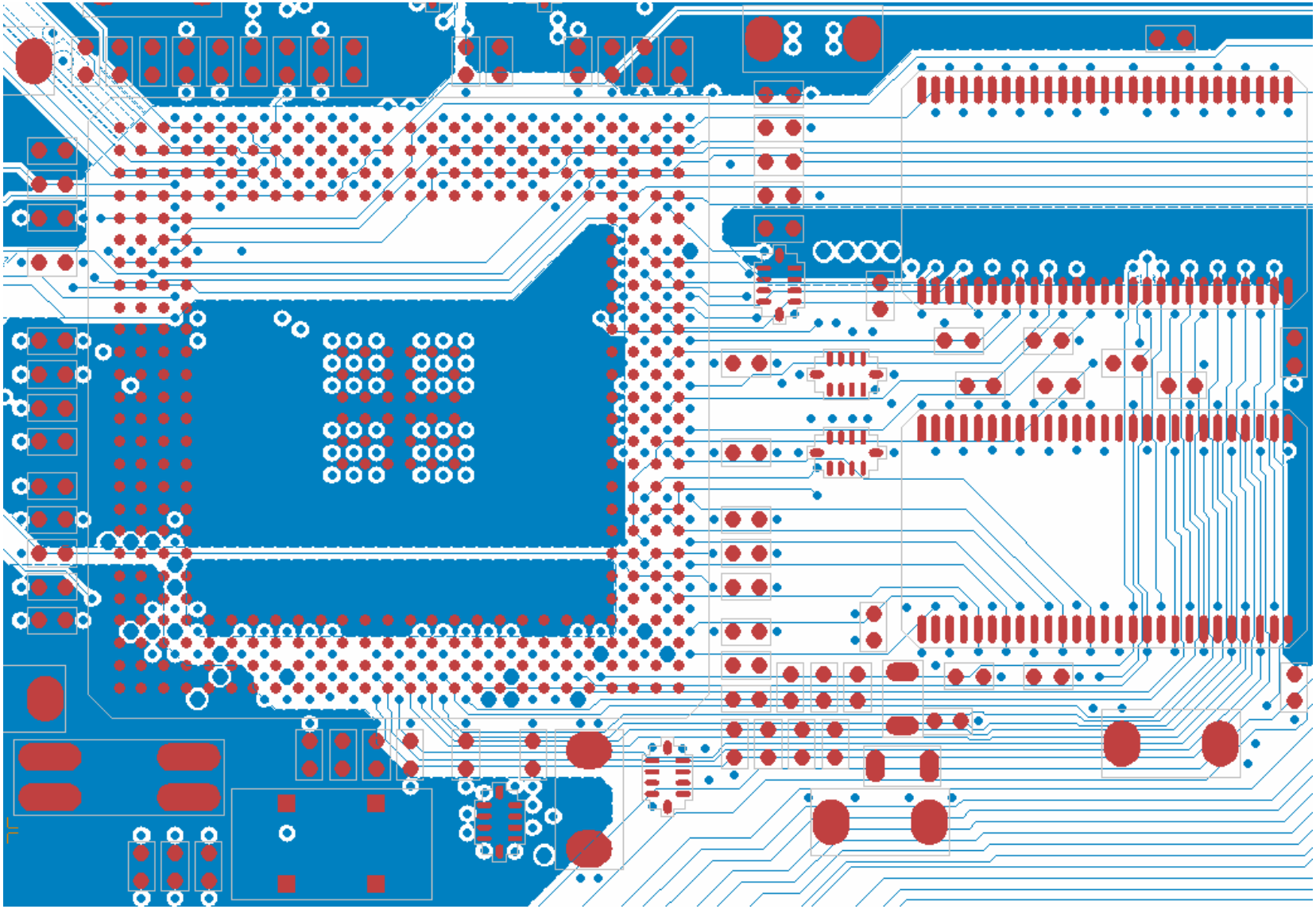
Swapping



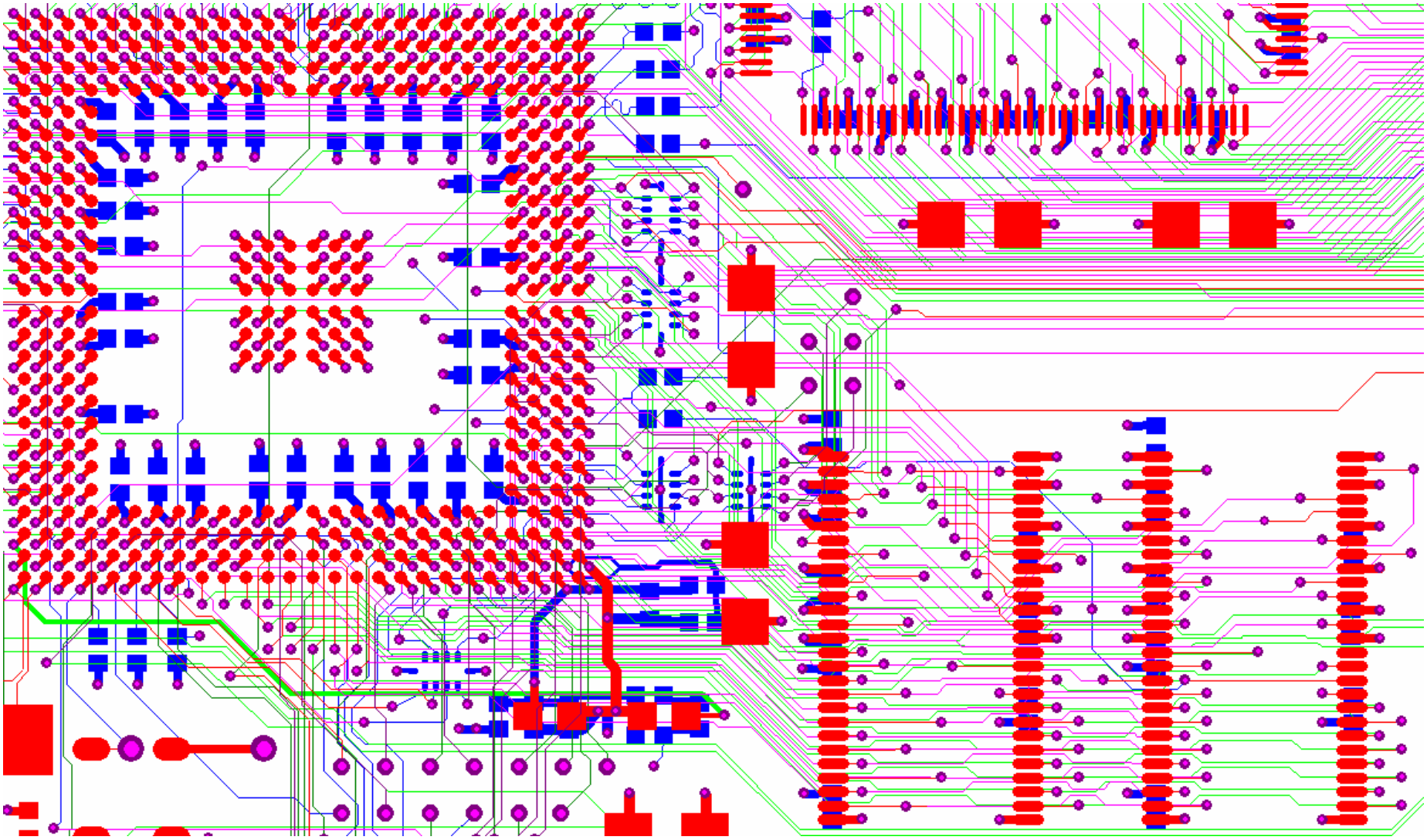
Swapping



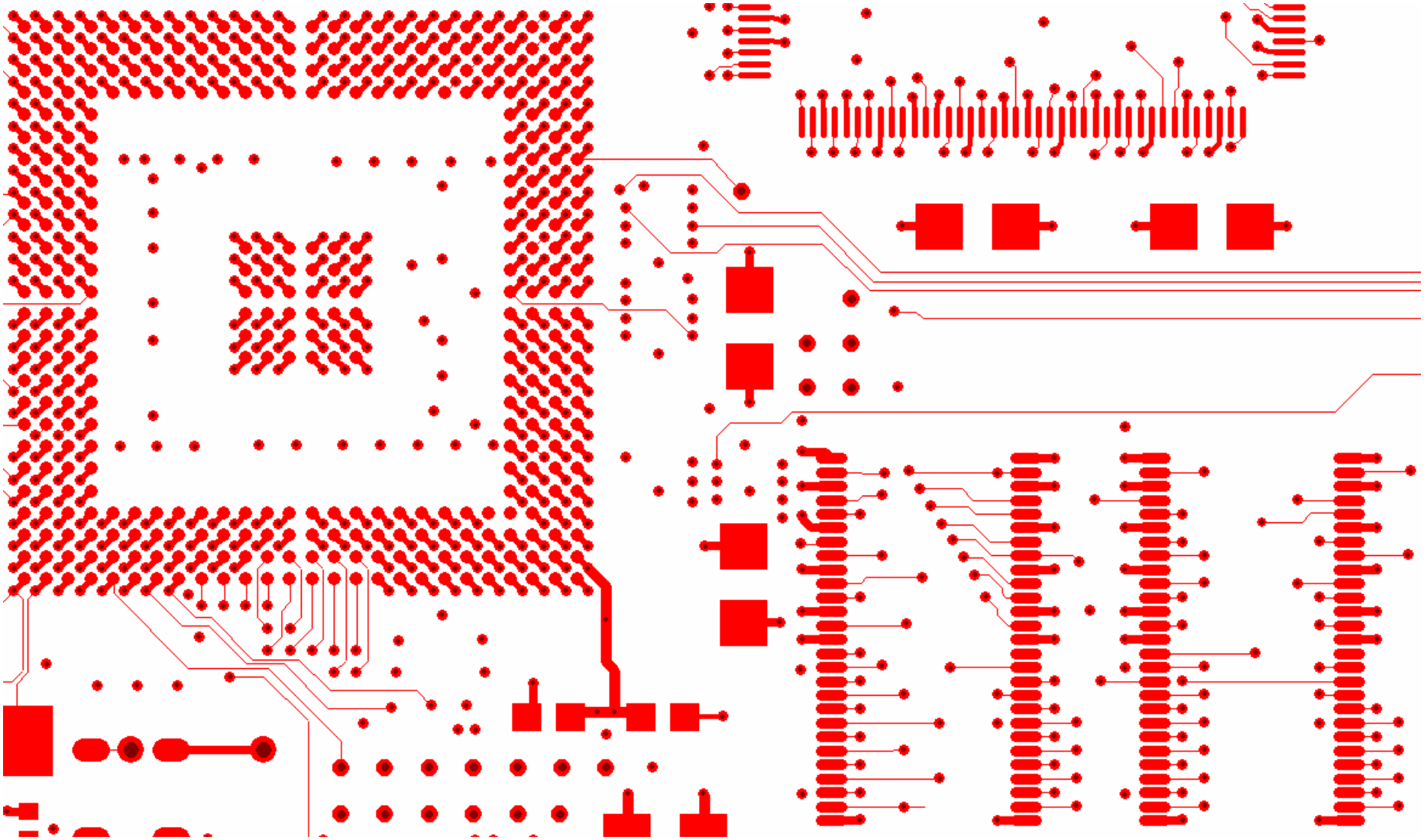
Swapping



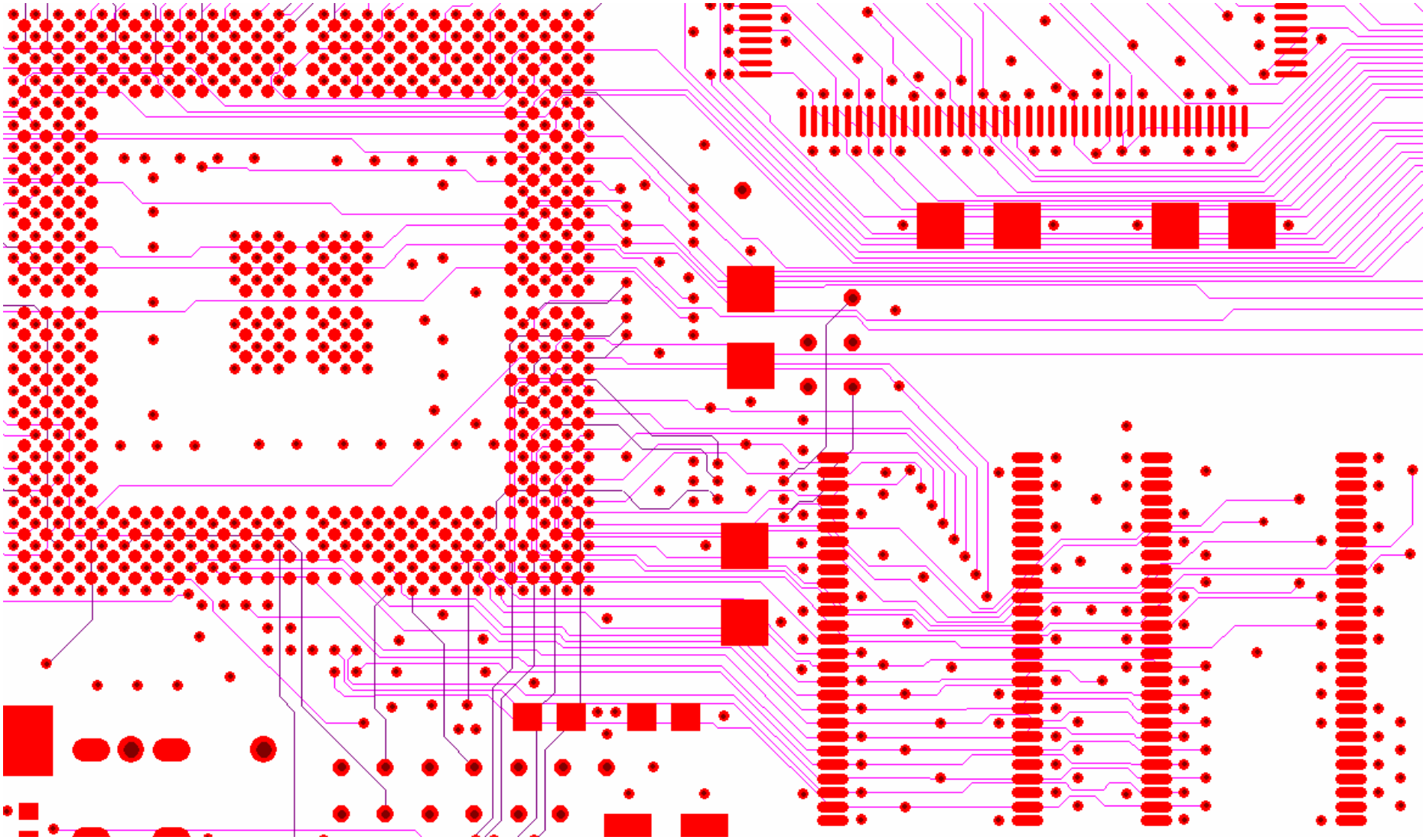
Swapping



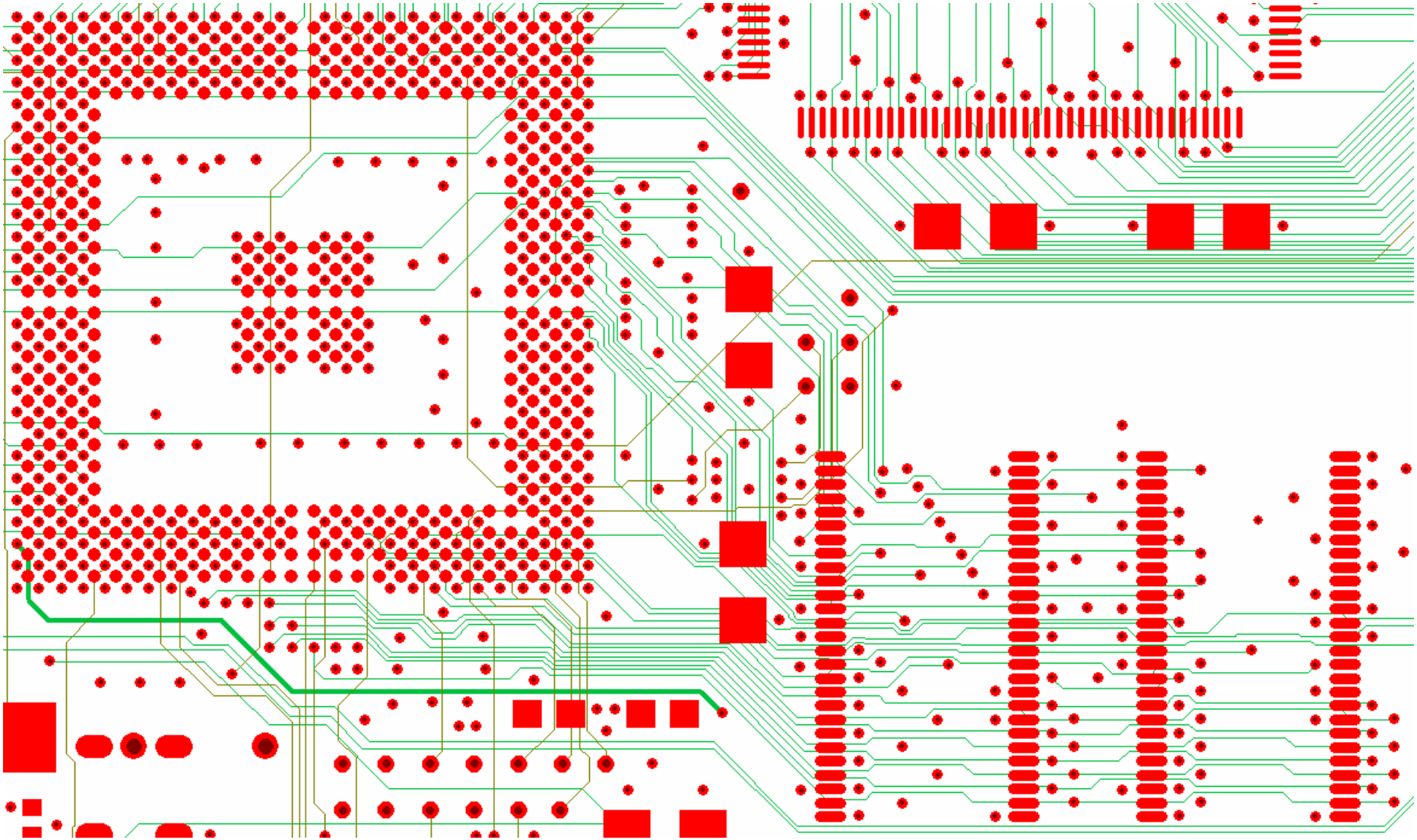
Swapping



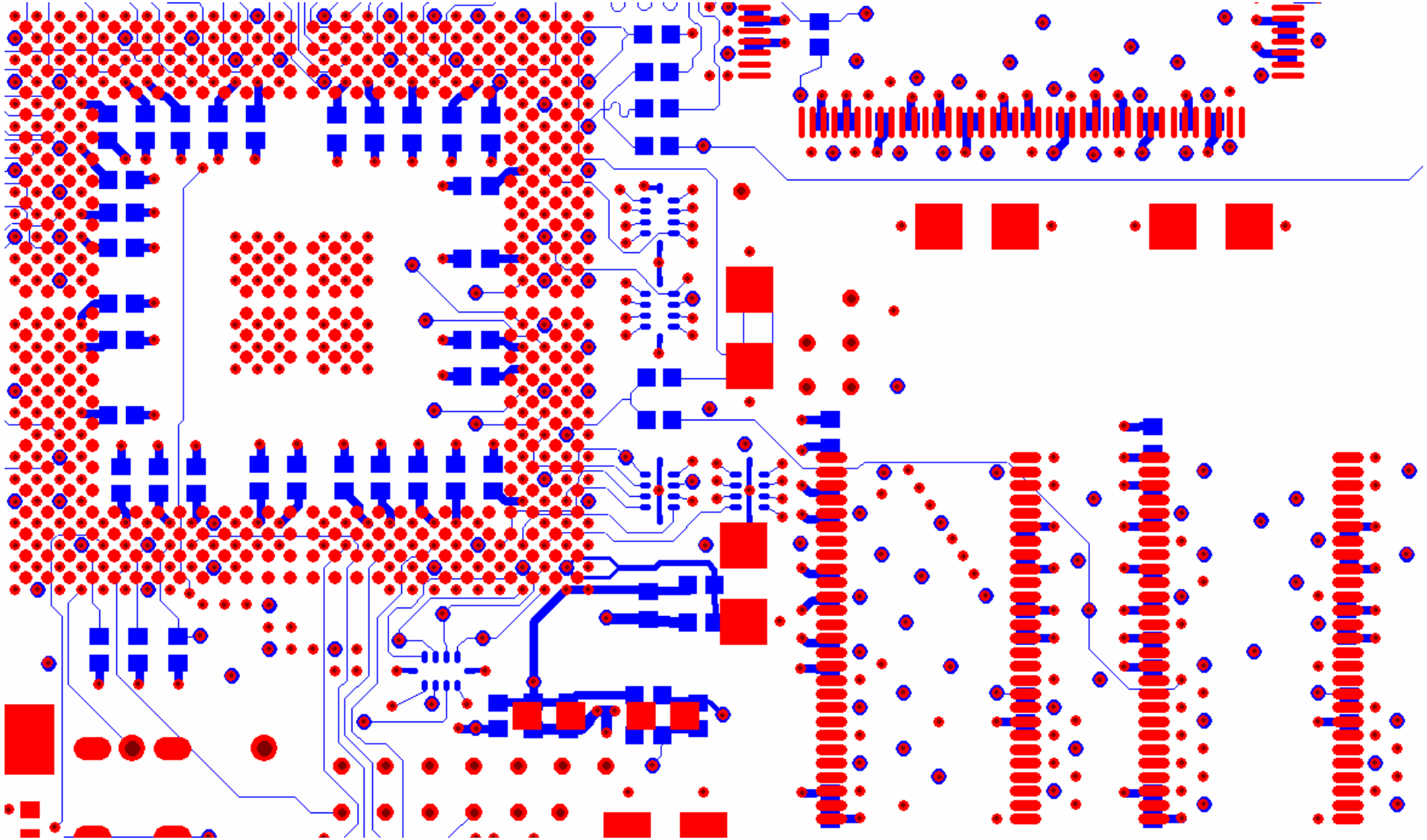
Swapping



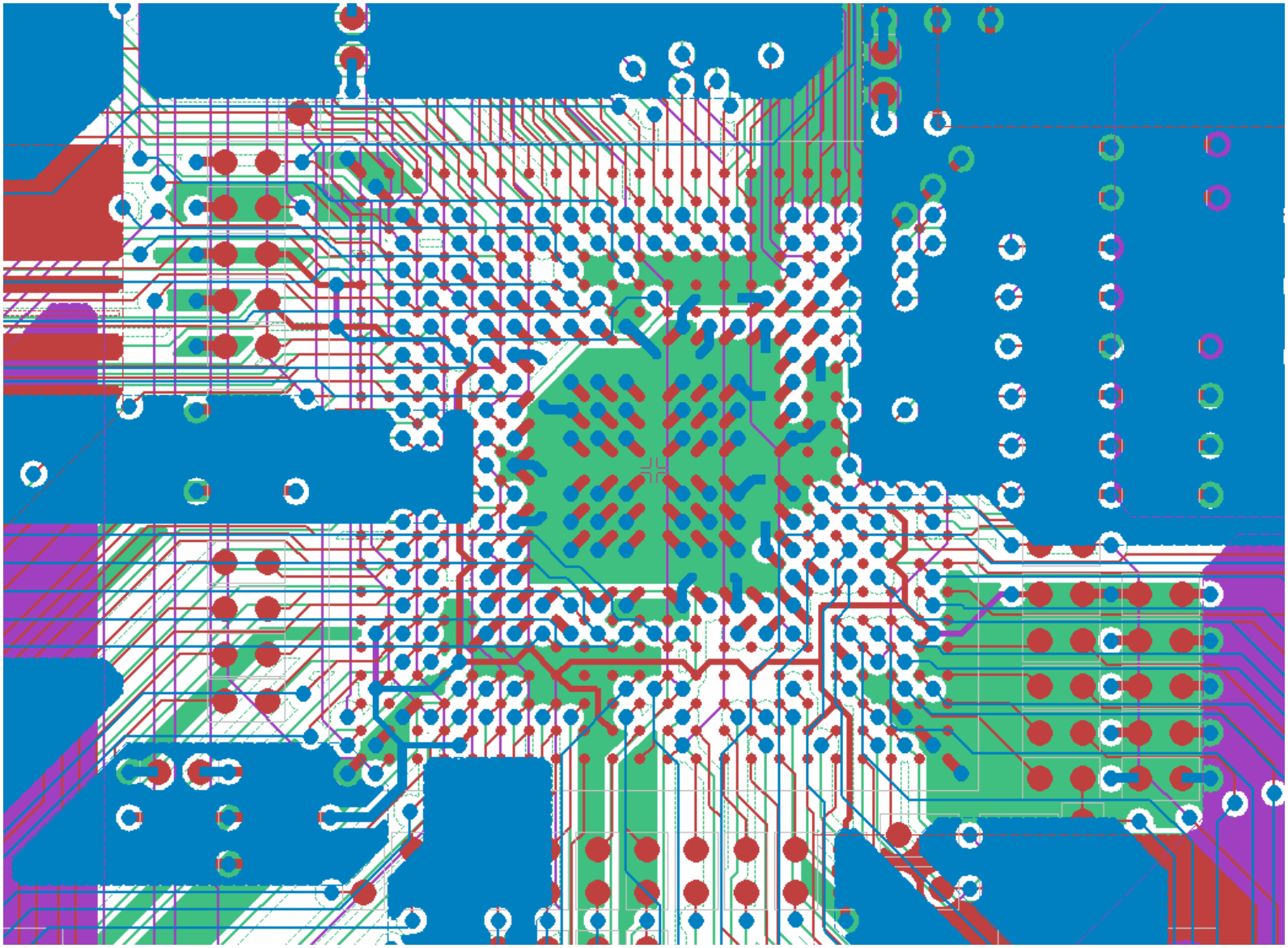
Swapping



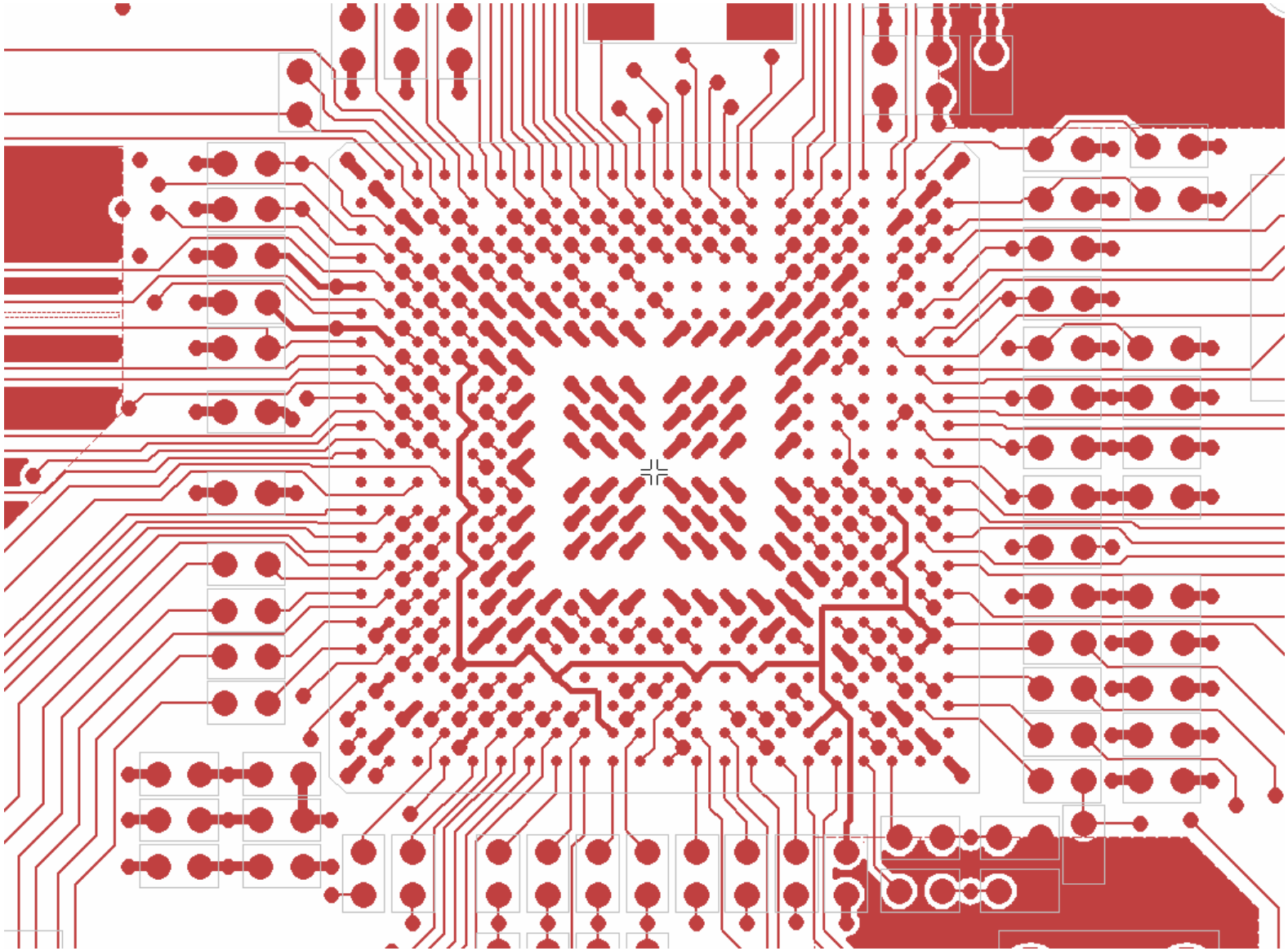
Swapping



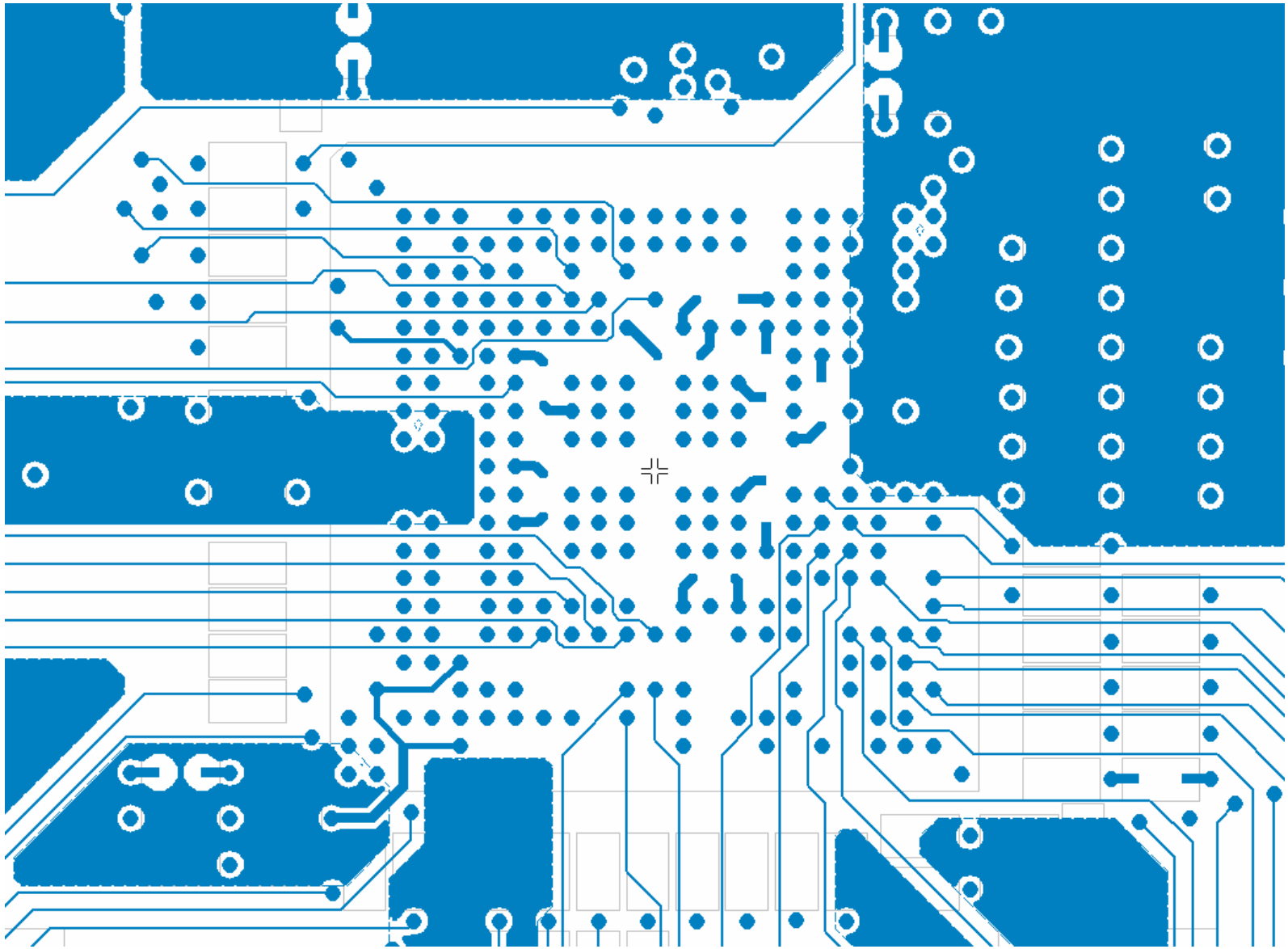
Swapping



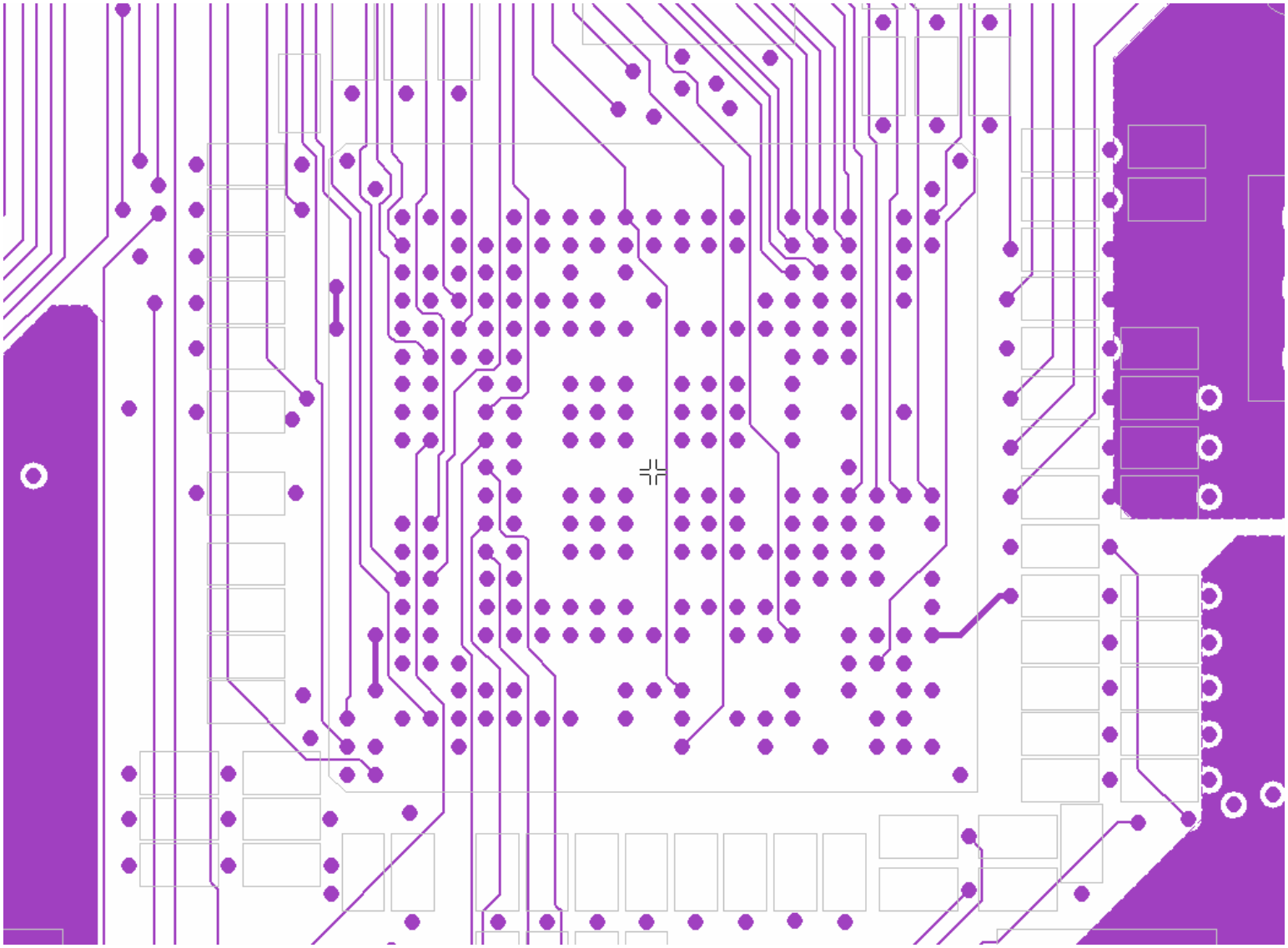
Swapping



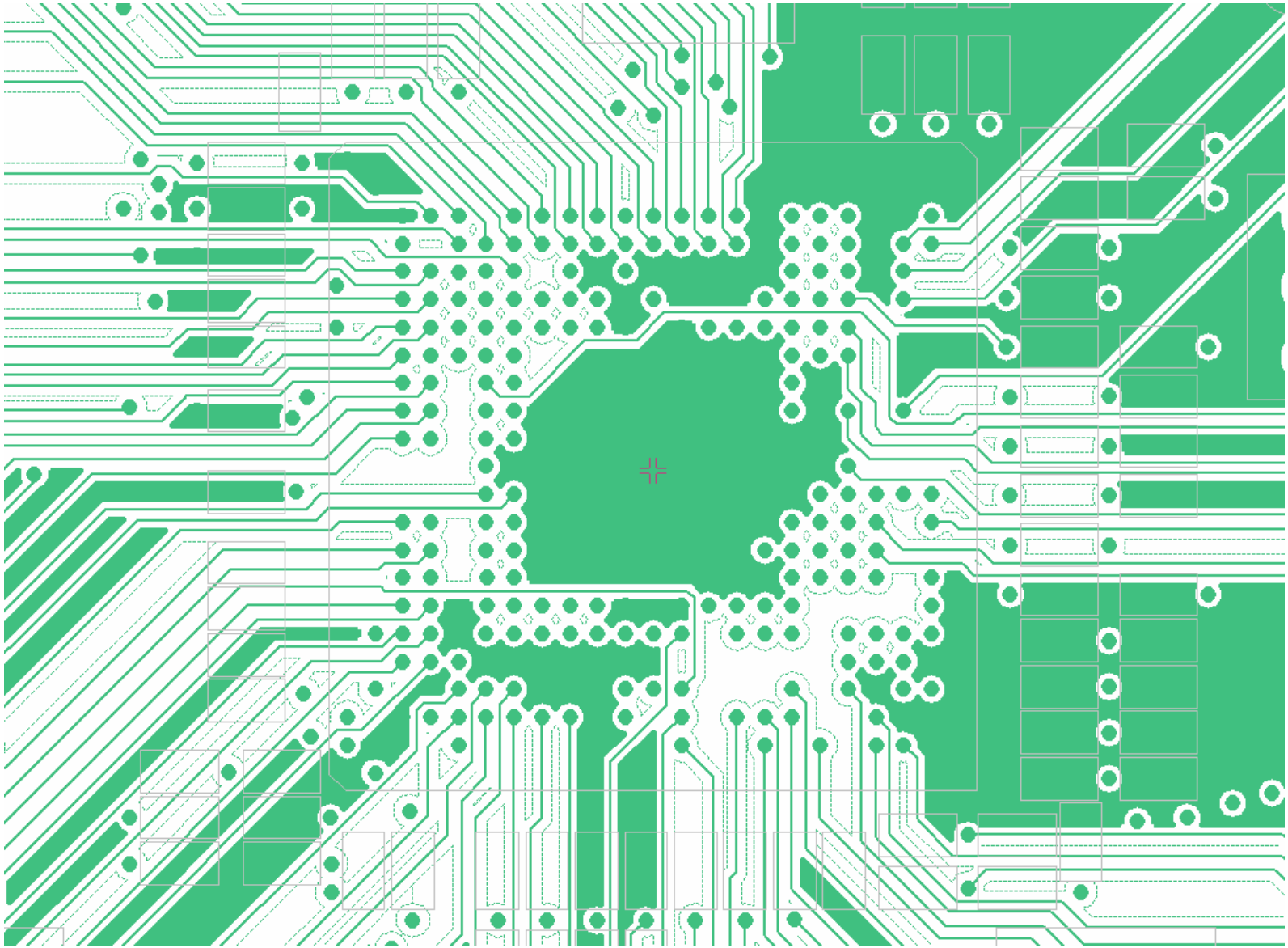
Swapping



Swapping



Swapping



Clean Up

Fix spacing errors

Tune signal lengths

Spread traces

Miter corners

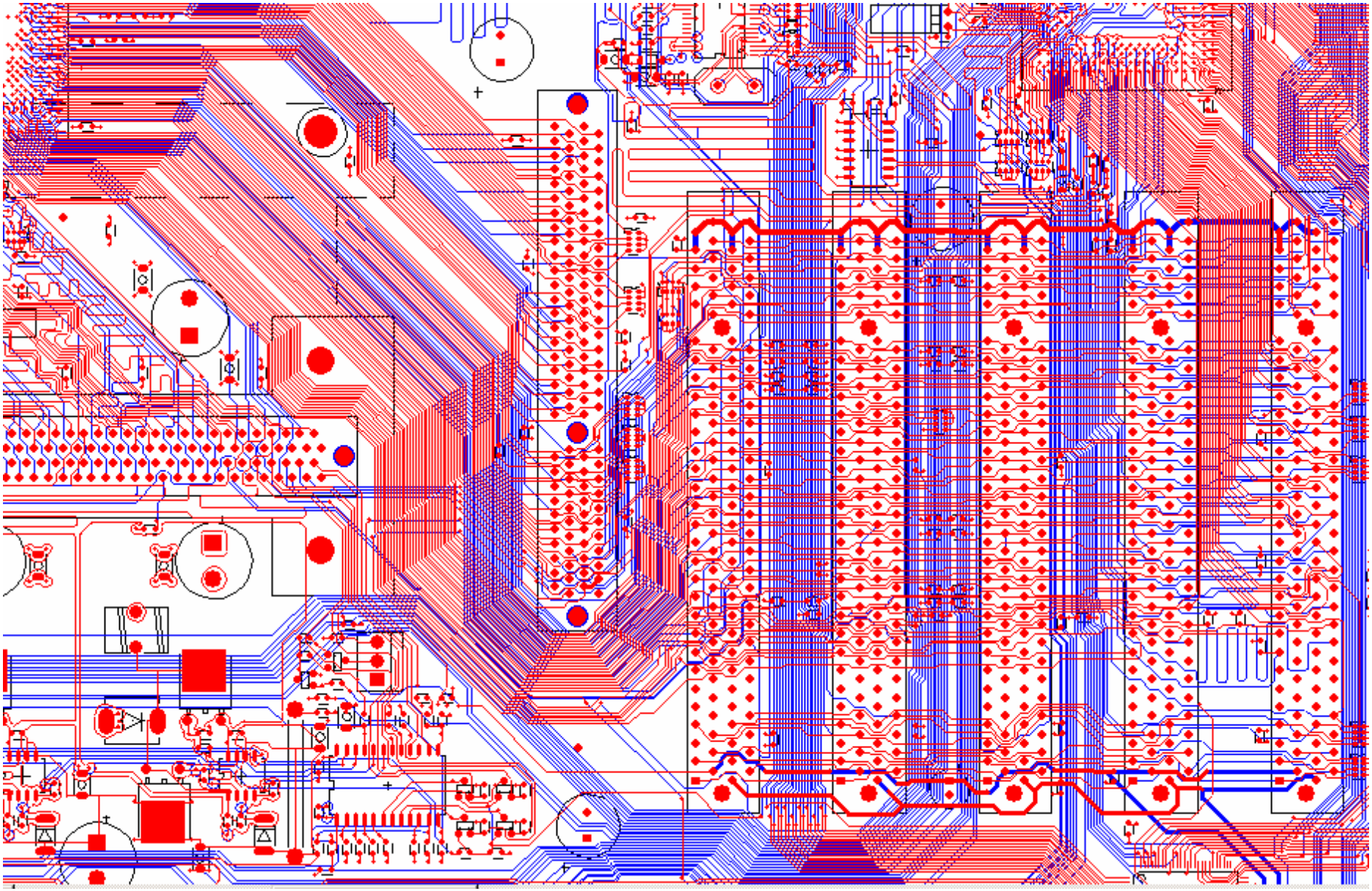
Remove unnecessary vias

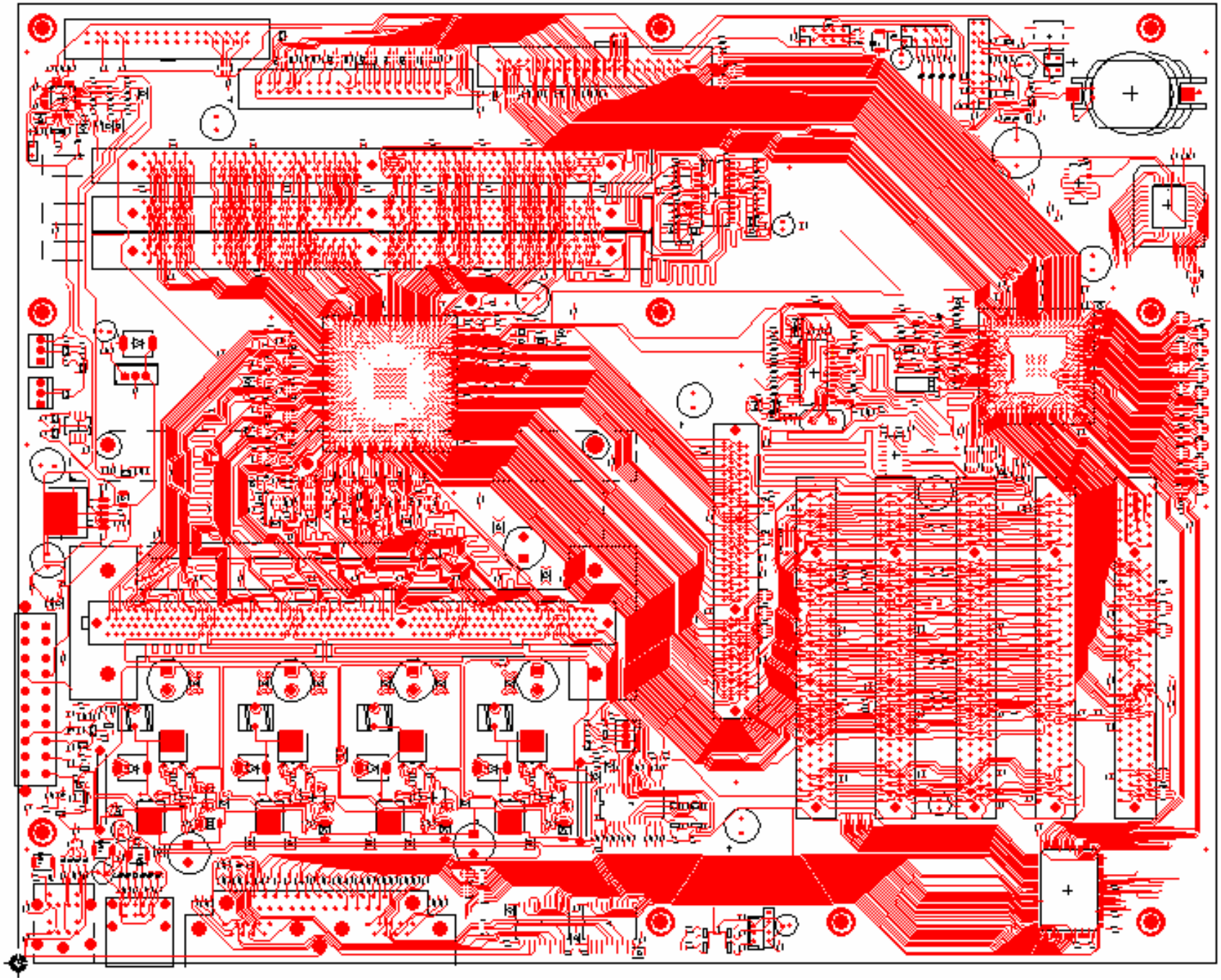
Write your name with one of the traces

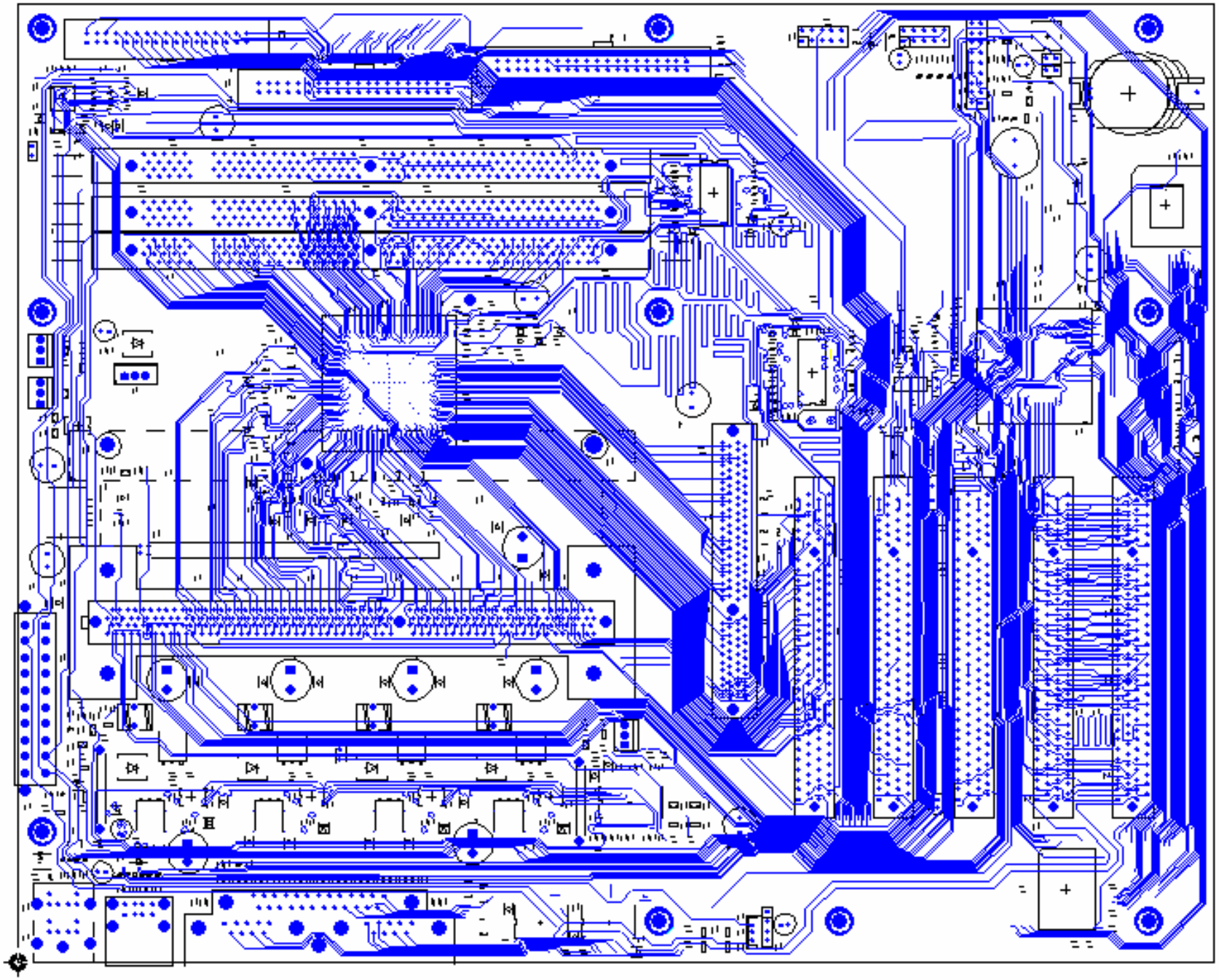
Routing Tips

- Work in metric if most of your parts are metric
- Granularity
- 100% connection before clean-up
- Route tight or trouble areas first
- Plan your areas for uncrossing
- Work with all routing layers on
- Use directional rules only when they work for you

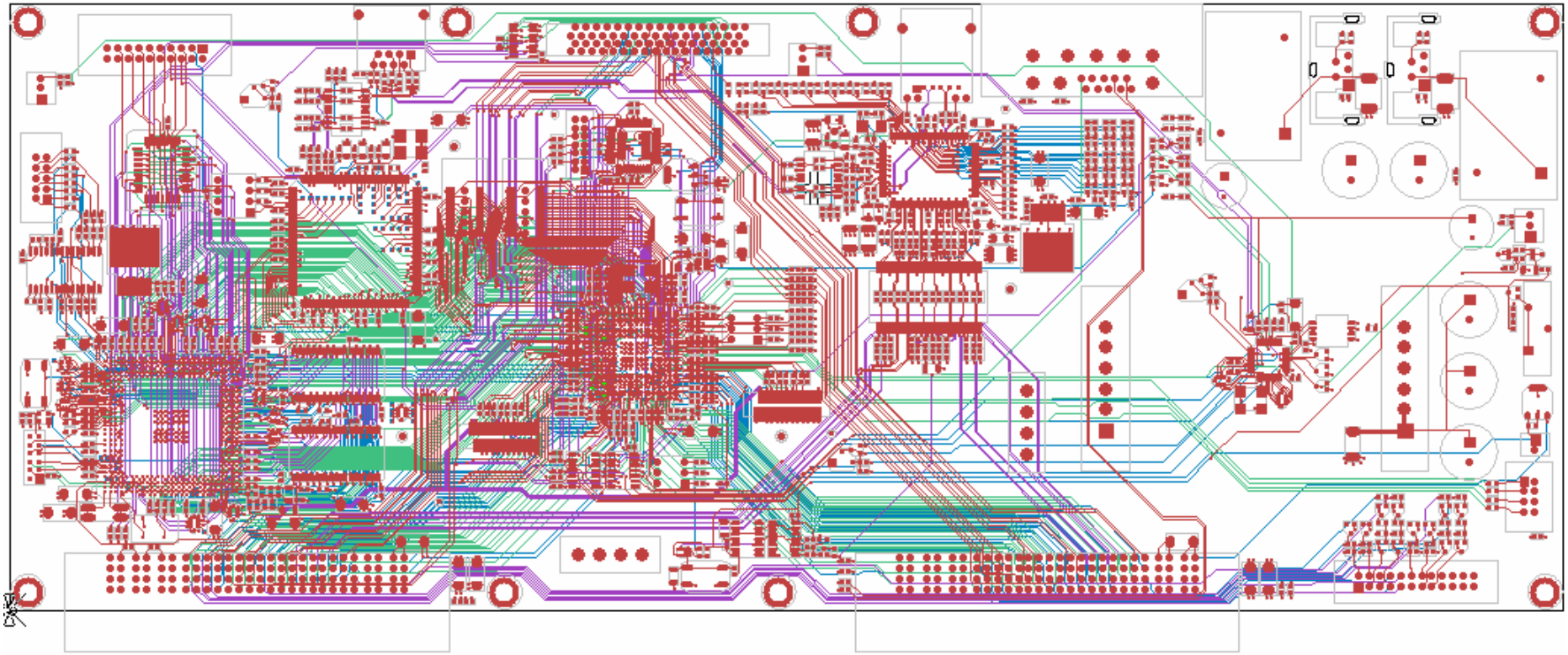
Directional Rules



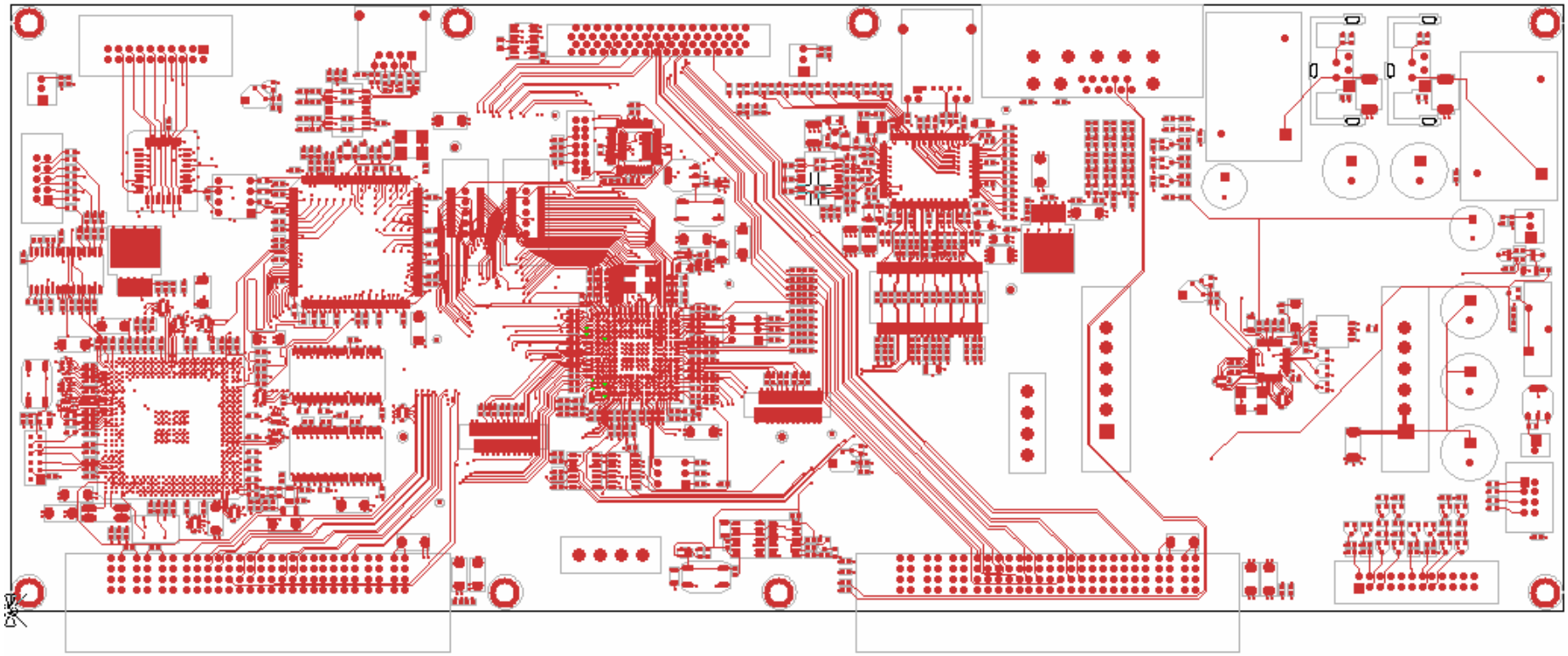




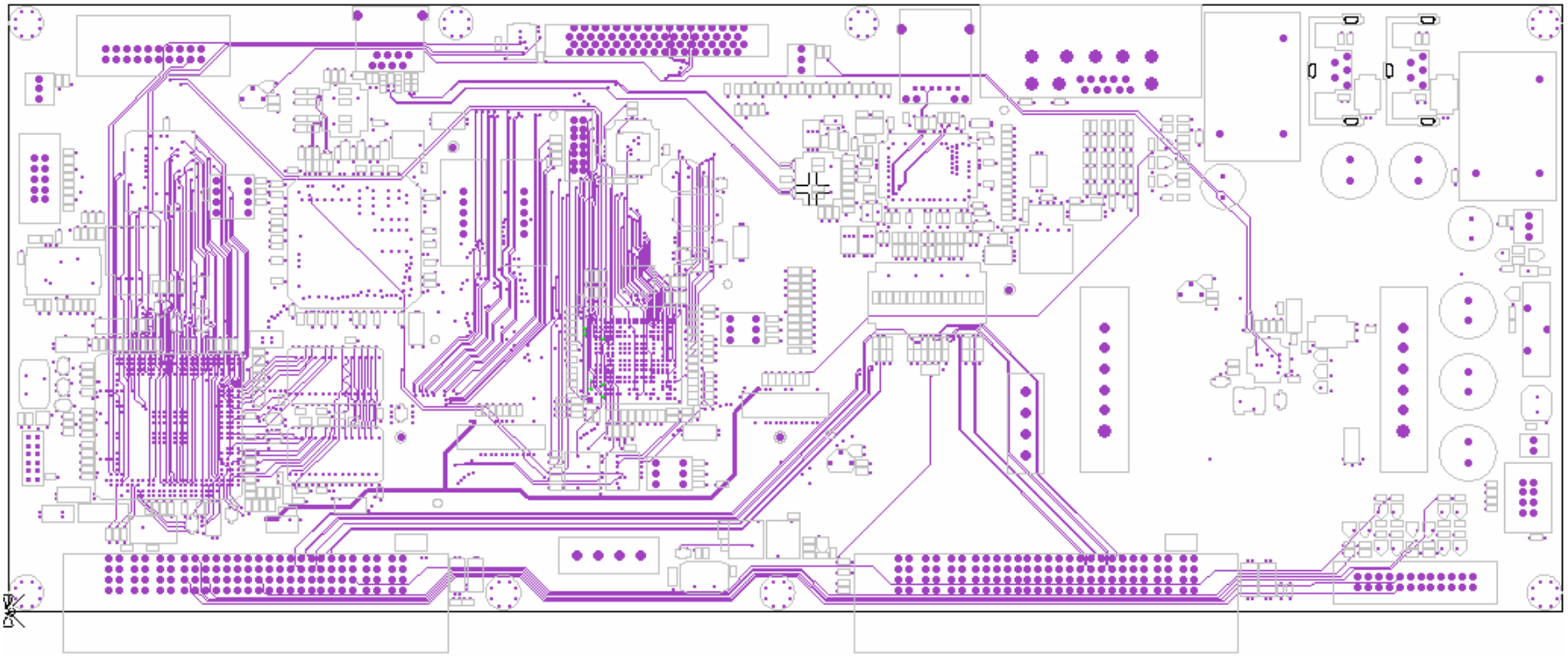
Directional Rules



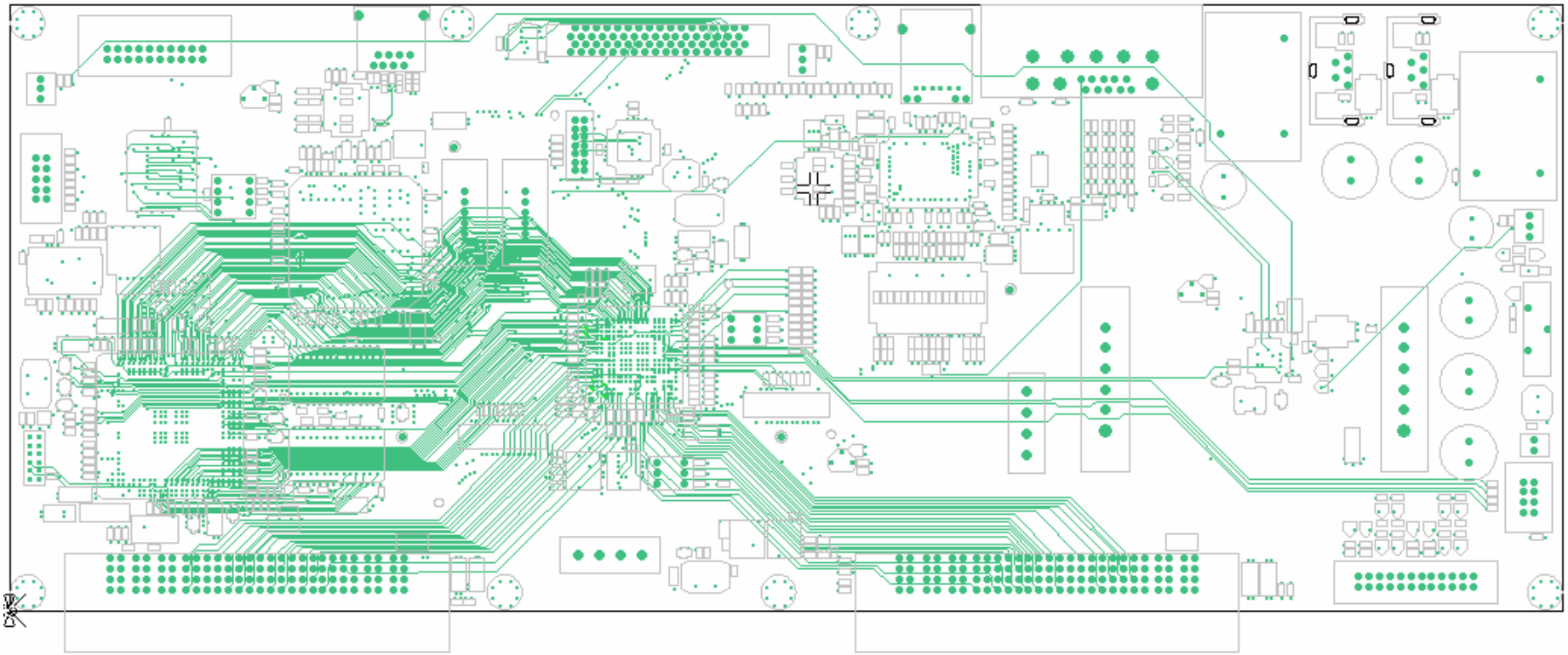
Directional Rules



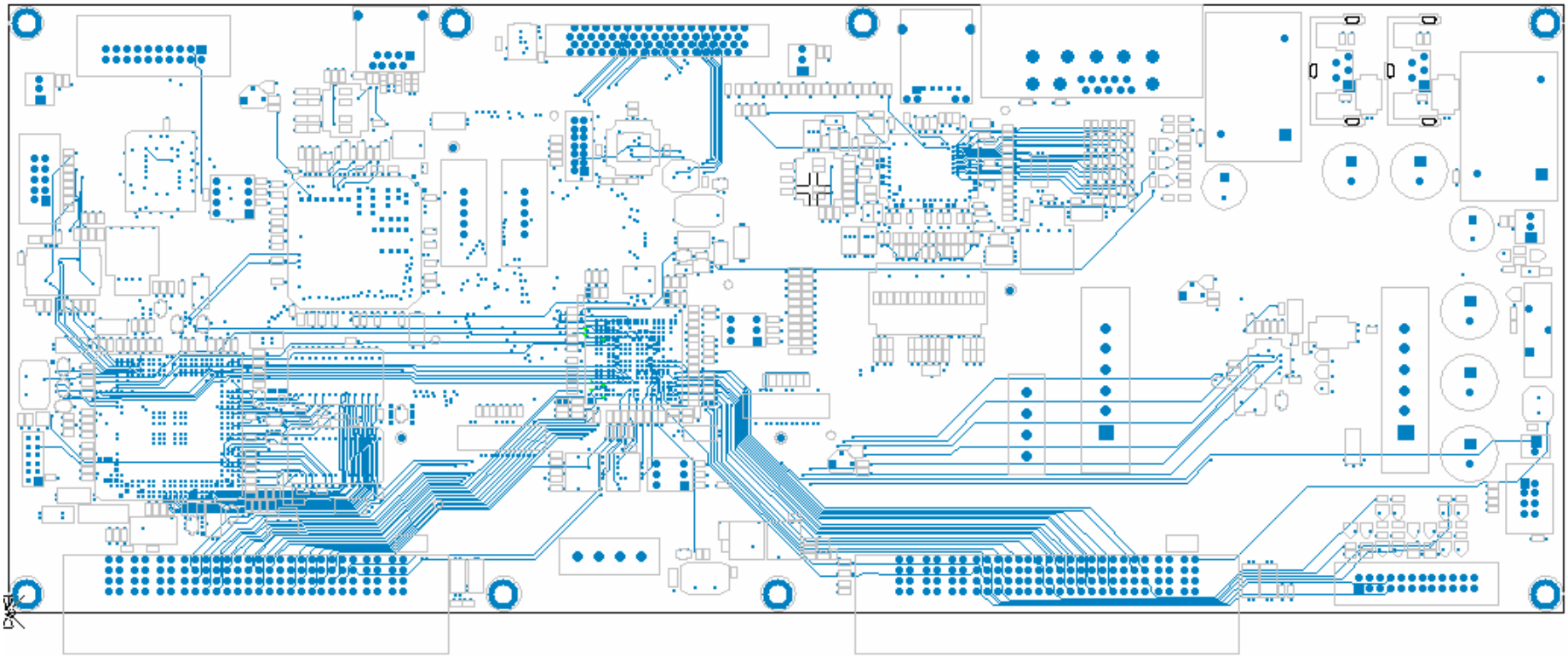
Directional Rules



Directional Rules



Directional Rules



Multi-layer Routing Tips

- Assign 2 layers to be diagonal routing
- Vias spacing equal to 2 spaces + 1 trace
- Push for replacing any thru-hole parts with surface mount parts
- Spread copper density evenly between layers if possible

Closing

I hope this presentation has given new ideas

For a copy of this presentation send email to wayne.pulliam@amd.com

Please put PCBeast in the subject and name the presentation you would like

Remember, The harder you work the luckier you get!

"Hand Routing"

Insights to reduce board design time and layer count.

This is a description of the board layout techniques I have developed and learned over the last 14 years. Used together these techniques are a design process that reduces wasted time, effort and space. The majority of this paper will focus on single sided assembly and 4-layer board construction. An abbreviated overall design flow will be followed by detailed placement and routing flow. General placement and routing techniques will be included during the design flow and listed at the end of the paper for easy review.

Understanding the elements
What the designer receives is a collection of data. When properly interpreted it can be used to create a physical electrical circuit. The elements of the PCB are:
Parts (land patterns)
Mounting holes
Vias
Traces
All of these elements take up space and must fit within the physical dimensions of the board. The physical designer can influence all the elements but is in total control of only the traces. The majority of space saved on the PCB by the designer will be in reduced trace length. Therefore placement is the most affective way of saving space on a board.

Key to reducing time
Do tasks only once, the major key to this is do tasks in a logical order. If tasks are not in a logical order then procedures begin to repeat. Identify problems that need to be resolved early in the design process. This is closely related to the first reason, as most solutions to problems require a task to be done again. The ideal board goes from start to finish without a single task being done twice. This is an unrealistic goal, but working toward it will reduce your design cycle time.

Overall design flow
Receive, review and understand design requirements
Build land patterns, set up design rules and create board outline
Place parts
Create power plane splits
Route signals
Review design for accurate adherence to design requirements
Create documentation for board fabrication.

PLACEMENT FLOW
Understand all design requirements
Some physical limitation may includes, board size, fixed part locations, mounting hole locations height restrictions and any other considerations for correction interaction with the enclosure. Some of the electrical requirements might be, power delivery, high-speed signal timing requirements, analog isolation or any other electrically derived requirements. Assembly and fabrication must also be accounted for. Understand how the priorities of all of the requirements are related. Occasionally requirements will be in conflict with each other. For example, there is a 1mm BGA that requires 4 routing layers to escape but the design is calling for only 2 routing layers.

Group parts
Using a block diagram of the electrical design, if one is available, find the primary component of the each block and group it with the supporting components. They need

not be placed neatly; in fact the fastest method is to stack all of the small parts inside the larger component's outline. This is done outside the board outline, as much space will be needed in the next step. The objective here is to see how the primary components connect to each other. I call this an exploded placement. If there is no block diagram and/or you choose not to use the schematic this task is still easily accomplished on small designs using the netlines. Simple take the large and or high pin count components and spread them around in the database. Then following the netlines place each of the smaller components inside the larger components. Remember to turn off all plane nets when doing this to avoid grouping components by voltage instead of logical connection. Group bulk capacitors and other power delivery components together and leave them off the board until last phase of rough placement.

Fixed location components

Connectors are the most common fixed location part, but nearly any component can have a fixed location. Mounting holes almost always fall in to this category. Place these in their designated location and take note of their relative locations. These can be moved around in the exploded placement as long as you keep then relative to each other.

Exploded placement

Now it is time to uncross the netlines. If the board is supplied by more than 1 voltage then you will need a way to keep like voltages together. Try highlighting the power nets without the netlines on, this will reduce the confusion. Moving the component groups around, try to uncross all of the major busses. If you are unable to uncross all of the busses try for an arrangement that has the least amount of crossings. The voltages should be treated only in relation to each other. Try to keep them separate with clear divisions. The shape of the divisions doesn't matter. Usually the resulting pattern will not fit inside the shape of the board outline. It will be to long, to wide or the shape will be wrong for the board. Center the entire circuit over the board. If you maintained the relative location of the fixed components use them as a guide. Replace the fixed components to there fixed location.

Visualize the signal flow that you decided on and place the component groups back on the board as best you can while retaining the least amount of crossings. Keep in mind the number of parts in each group and adjust the placement to make room. This is a very hard concept to communicate. It is totally visual way of deciding on the best possible placement. If the fixed location parts cause poor signal flow now it the time to request a change. Don't wait until the board is partially routed to see if you concerns are founded. Poor placement is a major contributor to increased layer count.

There are many variations to this type of placement. If the density of the board is high, you may want to reduce down to 1 or 2 netlines per bus and turn off the miscellaneous nets. This reduces the clutter and allows for easier viewing the flow of the major busses. If your board size is based on the circuit and not on some predetermined enclosure. Then after the exploded placement is finished, pull the parts close together and your ready for detailed placement. If you board has only 1 major component then skip the exploded placement. Always tune you process to the board you are working on.

Detailed placement

Clean up placement of parts in each group. Groups may need to be shifted to make room for this process. Many things should be check during detailed placement. First and foremost is a review of all electrical design requirements. Review the packages of the components. If a change in package style will improve the routing ask for it. This includes dividing resistor packs or consolidating resistors. Many logic gates are available in single gate packages. If a large

multi-gate package is causing problems ask for a change. The physical designer is the only person truly qualified to see how part packages affect the layout. Some gate and pin swapping can be done now. But most of the swaps will not be affective until routing is underway. Do not spend time tweaking part locations; this will be done during routing. Placement should never be "locked". The ability to shift parts while routing is important in high-density designs.

Test points

These are a subset of the detailed placement. Test points can be a real detriment to design time. Left till the end of the design they will cause many parts and traces to be moved. The end result is usually a compromised test field with less than 100% coverage. The best method for test point control is to add them to the design as parts. Place 1 single-pin test-point part on each net. Nets that connect to through-hole parts do not require test-points. This will require a modified net list and/or changes to the schematic. If you plan to go this route it is best to ask for these test-point parts to be added as soon as you receive the design requirement. Place these during the detailed placement phase adhering to all test point requirements. One of the benefits of having the test points in the design this early is the ability to lock the test point locations and generate a test fixture map before the design is finished. Test fixtures are traditionally a long lead-time item. If you chose not to use this method than try to take into account the space needed to have complete testability of the PCB while doing detailed placement.

Assembly review

When all parts are on the PCB near their final locations send placement data to you assembler for review. Ask for a quick turnaround and remind them that you have not started routing yet. Having assembly feedback before you start routing will save time.

Pads only board

Another tool for early detection of errors is have a board built with the parts only data. This board can be built with top copper only. (Bottom copper required for double-sided assemblies) Plated holes are optional depending on what you expect from the sample. If you want to solder through-hole components then plated holes will be needed. It can be used to confirm land patterns are correct. The parts only PCB can be used to verify form fit and function. It is also a useful as a prop for design discussion and makes a good souvenir.

General placement tips

Place all discreet components that are related to a device on the same sheet in the schematic. This is a tip to pass on to the electrical engineer. If he will place the bypass caps etc. on the sheet with the component that they are for then you don't have to ask later.

Placement granularity can have an affect on design time. Use a broad placement grid during early stages of the design. Using a fine grid will tend to lead you into tweak mode. (Gee if I squeeze these together I can get two more in the row) You don't want to go there the time is better spent on something else in the design. Place your BGAs with a grid equal to there pin pitch. This will make break out a snap. If your BGA is a true metric (1mm for example) part try changing your database units to metric and place the part on a grid equal to the pin pitch. When you get to the part for breakout use the metric units with a metric grid. This will save you time and frustration.

Parts do not have to face the same direction anymore. Check with your assembler they should confirm this. The only remaining advantage to like component facing is manual visual inspection.

Parts do not have to be placed orthogonal. Modern placement equipment has the ability place at any angle in 1-degree increments.

Parts need not be aligned in neat rows. Again the need for this has faded into the past. Only spend time aligning parts if it improves routing.

Smaller is not necessarily better. Fine pitch devices do not work well in 4 layer (2S2P) designs. They block routing on half of the available layers.

Make a rough map of your routing plan. The one you visualized. This will be valuable as a reminder of what your plan was especially if it is a large design. It will also be useful as a guide if several designers are working on the PCB.

Tips for double sided assembly
Shadow placement of parts is very useful. If you have like component bodies on both sides of the board place them directly on top of each other. This only works well if the parts share connection in the same location. Shadowing also works for bypass capacitors and resistors. These can be placed under the leads of larger components saving valuable via fan out space.

If you are working with memory devices in a double-sided environment keep in mind that mirrored image pin outs may be available. If you are dealing with large quantities of these parts you can request mirror image parts. With gull wing leaded parts this is a simple as flipping the part over before lead-form.

Keep like parts on the same side of the PCB.
Balance the number of parts on the front and backside of the board.
Keep tall and large components on the same side of the board.
Keep through-hole components on the same side of the board.

These are all things your assembler may ask you for when they review you data. If you want to save time do them before they ask.

Know your assembler's method for soldering double sided assemblies is very use full as well. If the assembler plans to wave solder components on the backside of the board special land patterns may be required. If the assembler uses 2 sided IR soldering then there may be a keep out requirement around leaded components for a wave solder fixture.

As you exit the placement phase you should be able to check off the following.
Meets all design requirements
Assembly review completed
Test point plan completed
Power plane plan completed
Routing plan completed

ROUTING FLOW
Review your routing plan
If you have a hard copy or have stored this information in the design now is a good time to review it. If you only have the plan in your head it is even more important to spend some time examining the placement to refresh you memory. The intent is to connect all of the nets one time only with no false starts.

Power plane fan out
Connect all power nets that connect to power planes. This reserves the space for the vias and verifies the power plane splits before any signal net traces are laid. If you find an error in the power plane strategy it can be corrected without losing any routing.

Escape
Ensure you have an escape path for all parts before routing through them with other signals. BGAs are the difficult to route without a set escape pattern. Establishing a repeating pattern for escape of devices is a time saver. The pattern can be copied in most tools. When you complete the escape

of a component simple copy it to all parts with the same land pattern. All of the components may not make good use of the pattern but the amount of time spent is very small and the gain in efficiency is usually large. Again the main concept here is to avoid moving traces later to enable the connection of a device.

Rough in routing

Connect the nets without clean up or tuning. Routing net lines without cleaning them up is one of the hardest things for established designer to change. What you want to accomplish is 100% connection as quickly as possible. This will identify any problems with you routing strategy. What you don't want is time spent on making routing pretty only to rip it up when you find it doesn't work.

There are a couple of techniques I use to assist me in routing traces once and avoid having to move them later. First is to route all major busses. Second, route critical signals. Third, connect remaining miscellaneous traces. Digital busses make up the bulk of the routing. Connect them first to identify any routing bottlenecks. Generally, individual signals can find a path through a design even if they have special spacing or length requirements. Route critical signal first and you will most certainly have move them when you route the major busses. Again, why do things twice

Another method that works well is one-pass routing. Taking all of the nets in an area and routing them together in one direction. This works very well on small boards with single functions. When the grouped traces arrive at an impasse you know exactly how much space is required to solve the problem. This method may be difficult to visualize but is easy to understand when you see an example.

Rough routing is the time to do any gate and pin swapping. Route the bus or signals to the devices where you intend to do the swapping. With the netlines coming off the traces in close proximity to the parts it is easy to see the correct swaps. Remember to verify all swaps with the electrical designer. Some of the devices that can be swapped are resistor packs, capacitor packs, multiple gate logic devices, programmable logic devices, ASICS, and memory devices.

Clean up

With 100% of the PCB connected it is time to clean up the routing. Clear any spacing error you have created. Tune any traces requiring special timing. Spread traces when possible for lower cross talk. Reduce via count and miter corners if you have the time. Verify all design requirements.

Routing tips

Metric parts, especially BGAs can cause painfully slow routing. One way to deal with the occasional metric BGA is to change database units to metric while escaping the part. One of the earlier placement tips was placing your BGA on a metric grid. If the majority of the board is metric parts you will save yourself a great deal of time by keeping the database metric and learning the metric equivalents.

Plan areas for uncrossing busses. If a bus needs to change layers in mid-route additional space will be required for the vias. This is the same for busses that are twisted. The easiest way to untwist a bus is to have a planned layer transition. It also helps if the transition includes a 90% turn.

Directional rules are for auto routers. Never limit routing direction of an entire layer in 4-layer (2S2P) design. Doing so can increase the via count as much as 2X. Vias take up more space than traces and space is at a premium in a dense design. Instead use direction rules wisely by planning crossing areas. Set directional rules by area, and let the need for route crossings determine the direction.

Work with all routing layers visible. This is another tough one if you are in the habit of routing with only the active layer on. The harder this is to do the more important it is. If you only have 2 route layers and a single

sided assembly it is easy to focus on the active layer. There is less to see and very little interference to worry about. So why have it on. In a design with 6 route layers and double sided assembly it close to impossible to remember what is on a layer in certain area. When the layers are all on it is difficult to focus on the active layer. But if you learn to do this it will enable you to quickly insert traces into a design that is mostly routed. Instead of cycling through the layers looking for the clear routing layer and then looking for an area were a via can be dropped you can simply see the path the trace must take.

Multi-layer routing tips

Converse to low layer count boards. If you are designing a board with many layers be sure to assign directional layers. Include 2 diagonal routing layers one sloping one / and the other \. These will usually end up being the densest layers. It is a rare day to open a design and see most of the netlines running vertical and horizontal.

Via spacing is more important in a design with multiple routing layers. Vias that are to close to each other become a block to routing. Make your via to via spacing equal to 1 trace width plus 2 spaces. This gives you just enough room to route 1 wire between any 2 vias.

Closing

It is not the intention of the paper to provide an A to Z design process. That would take a book. It is not intended to apply every technique to every design. That would only work if all designs were the same. It is intended to give insight were time can be saved in the design process and give some ideas to use when tackling low layer count high-density design.

Everything I have described in this paper is from actual experience. I hope that some of my experience will help you reduce you design cycle time and route PCB with fewer layers.

Placement tips

Ask for changes
Make a routing map
Parts only board
Placement is never locked
Have a block diagram
Metric parts placed on a metric grid
Parts can face any direction
Parts can be placed at any angle
Align parts only if it improves routing
Understand you assemblers solder process
Place like components on the same side of the board
Place all through-hole parts on the same side of the board

Routing tips

Connect plane power first
Route traces only once
Breakout patterns for high pin count parts
Clean up after 100% connection
Metric parts routed on a metric grid
Use directional rules wisely
Work with all routing layers on

Wayne Pulliam
PCB Physical Design Lead
Advanced Micro Devices
For a PDF of the presentation slides email wayne.pulliam@amd.com