

Features summary for ELECTRA version 2

- Extending rules hierarchy

PCB < Layer < Class < Class_layer < Net < Net_layer < Area

- Rules-by-Area (width and clearance by region)
 - Especially useful around BGAs and fine SMDs
 - Interactive region editor to define area, width and clearance

- Router Strategy Dialog

- New Graphical User Interface

- Layer/Object panel visibility control
- Design stats panel
- Mouse wheel for easy zoom in/out
- Interactive net selection
- Distance measuring ruler

- Report of routing via styles

- Improved layout display

- Polygon outline and blind/buried via display

- Reset and restart

- Try different routing strategies with a reset command that deletes all routing

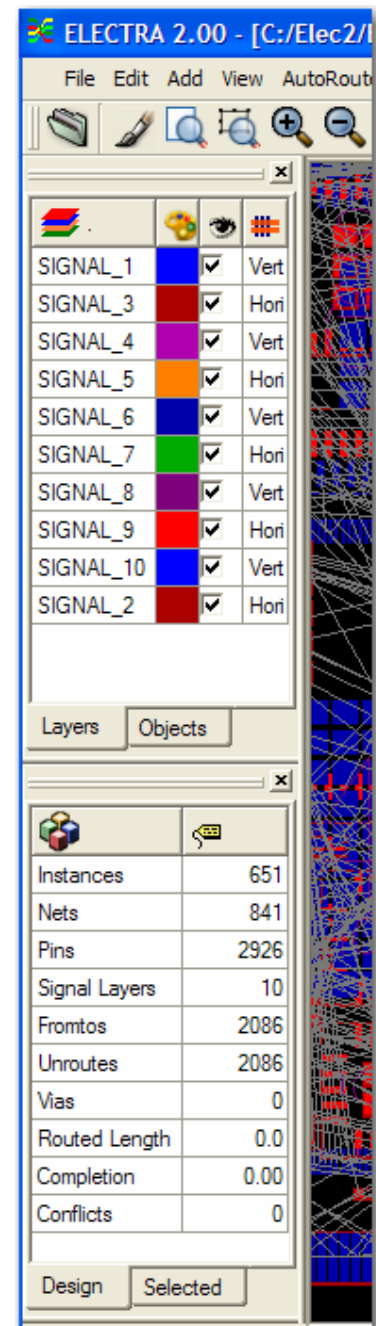
- Simplified and robust licensing

- For node-locked licenses, Site and MID codes are now unaffected by changes to BIOS, MB and OS
- One step configuration upgrade
- Hassle-free floating license setup

- Improved User Guide with Tech Tips section

Electra v1.xx customers who initially purchased maintenance are entitled to get Electra version 2 free-of-charges

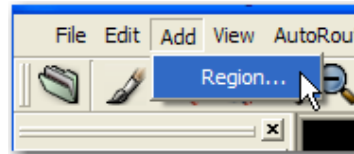
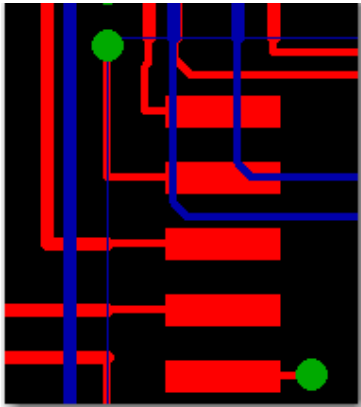
Electra v1.xx customers who are not under maintenance are offered a version 2 upgrade at 35% of the license cost



Extending rules support

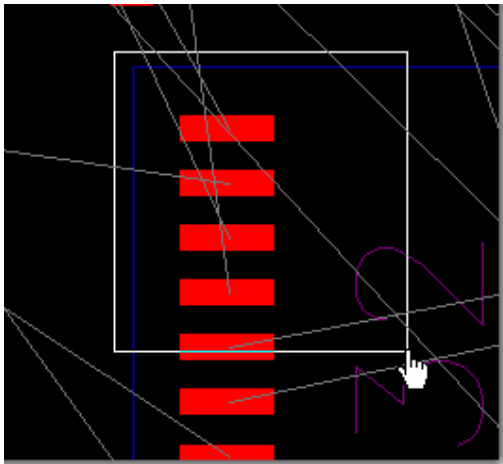
Rules-by-Area

This is especially useful around small complex components such as BGAs and SMDs. Also called region rule, it is a user defined rectangular area specified on specific layer(s) and a user defined track width and clearance within the area. The automatic router will adhere to the defined area rules during autorouting of nets that are crossing the region.

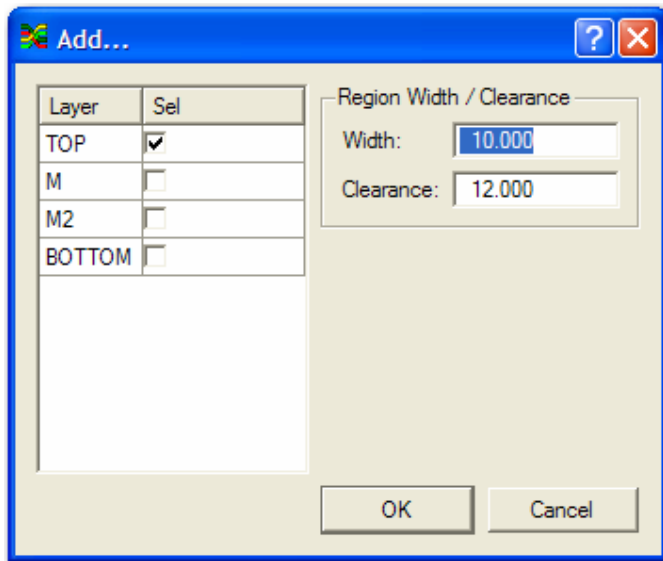


How to use it:

1. From the top menu bar, select Add/Region...
2. A "hand" cursor appears and you are prompted to define the rectangular area by dragging from one corner to the other corner of the rectangle.



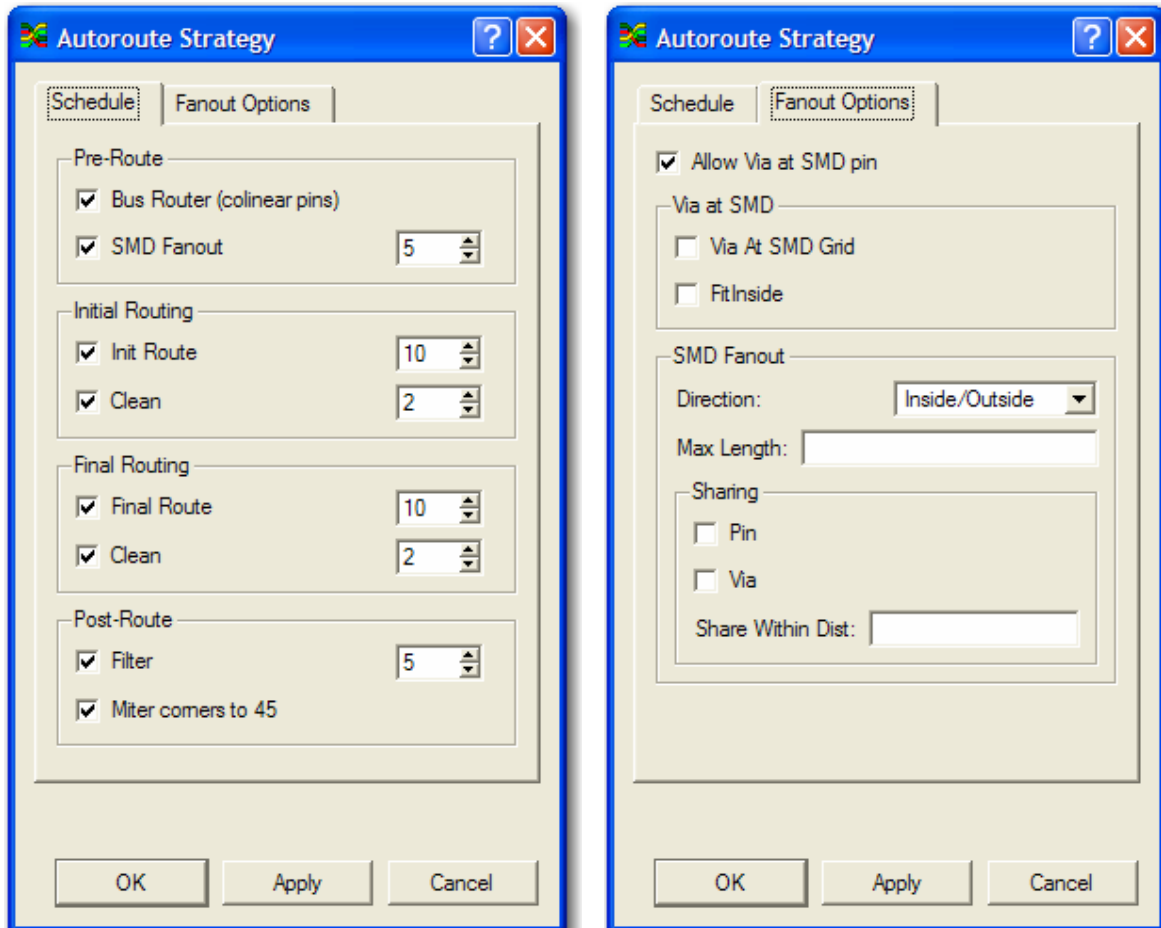
3. Once the area is defined, then a region rule dialog will show up to allow selection of the layers and region width/clearance:



Note that the regions are automatically saved and restored the next time you reload the design. This is accomplished by saving the user defined region definitions in the design directory. The region file is named after the design name with the .region extension appended. You can freely edit the file. See the following example:

```
demo.region
#
# Region file created by ELECTRA V2.00
#
# Syntax: add region <layer_name> width clearance x1 y1 x2 y2
#
unit mil
add region BOTTOM    6.000   12.000 9549.583 8638.587 10083.456 9306.394
add region TOP     6.000   12.000 9549.583 8638.587 10083.456 9306.394
|
```

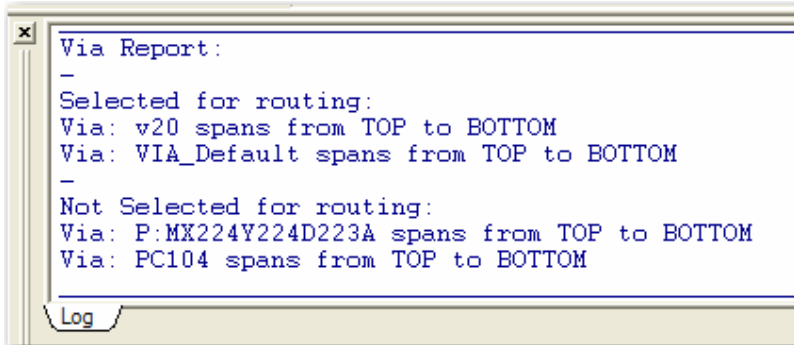
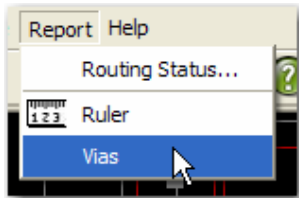
Strategy Dialog Editor



A basic routing strategy can now be defined and reused without “text editing” a DO file. The strategy settings are automatically saved and restored the next time you reload the design. This is accomplished by saving the strategy settings in the design directory. The strategy file is named after the design name with the .rtr extension appended. For more advanced requirements, the use of a DO file is of course still available.

Reporting of routing via styles

Reports the vias that are selected for routing. You can verify that the via defined in the DSN file are as expected.



You can also select/unselect a via style as needed. For example here is the effect of typing the following command:

unselected via v20

