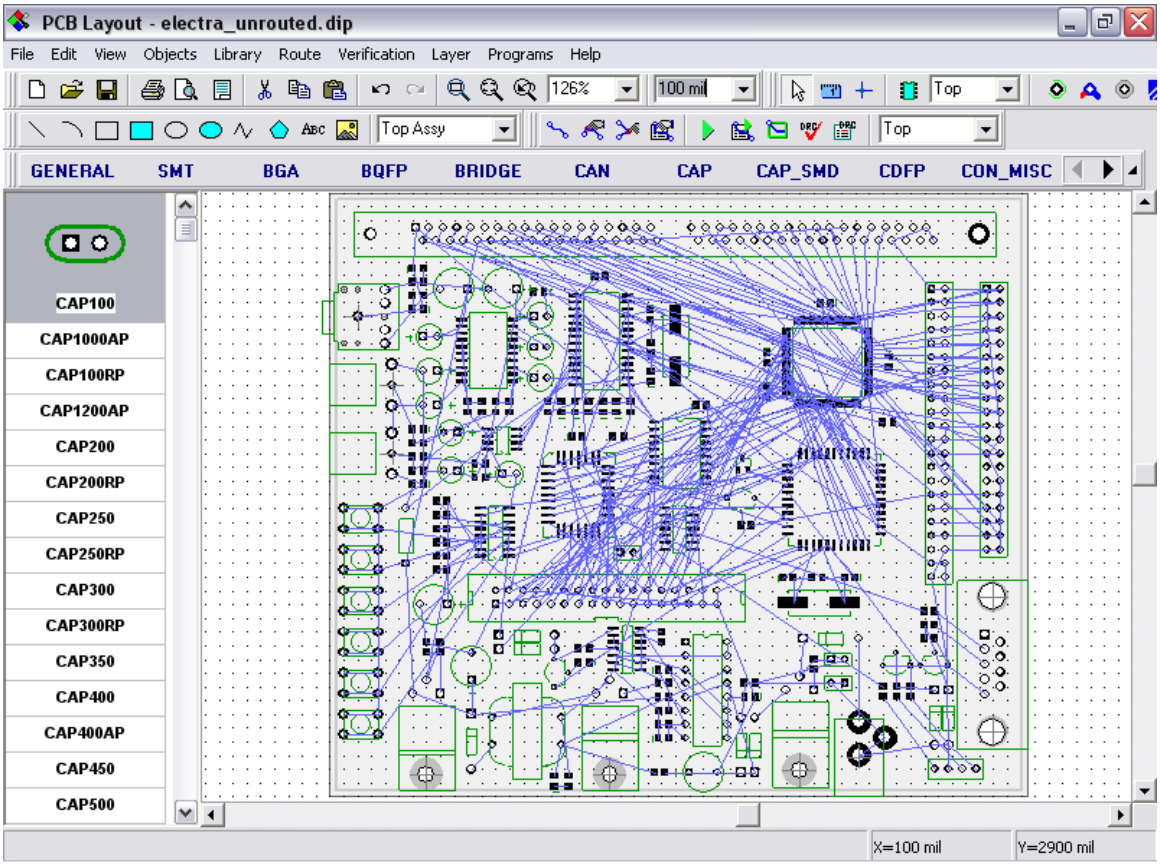
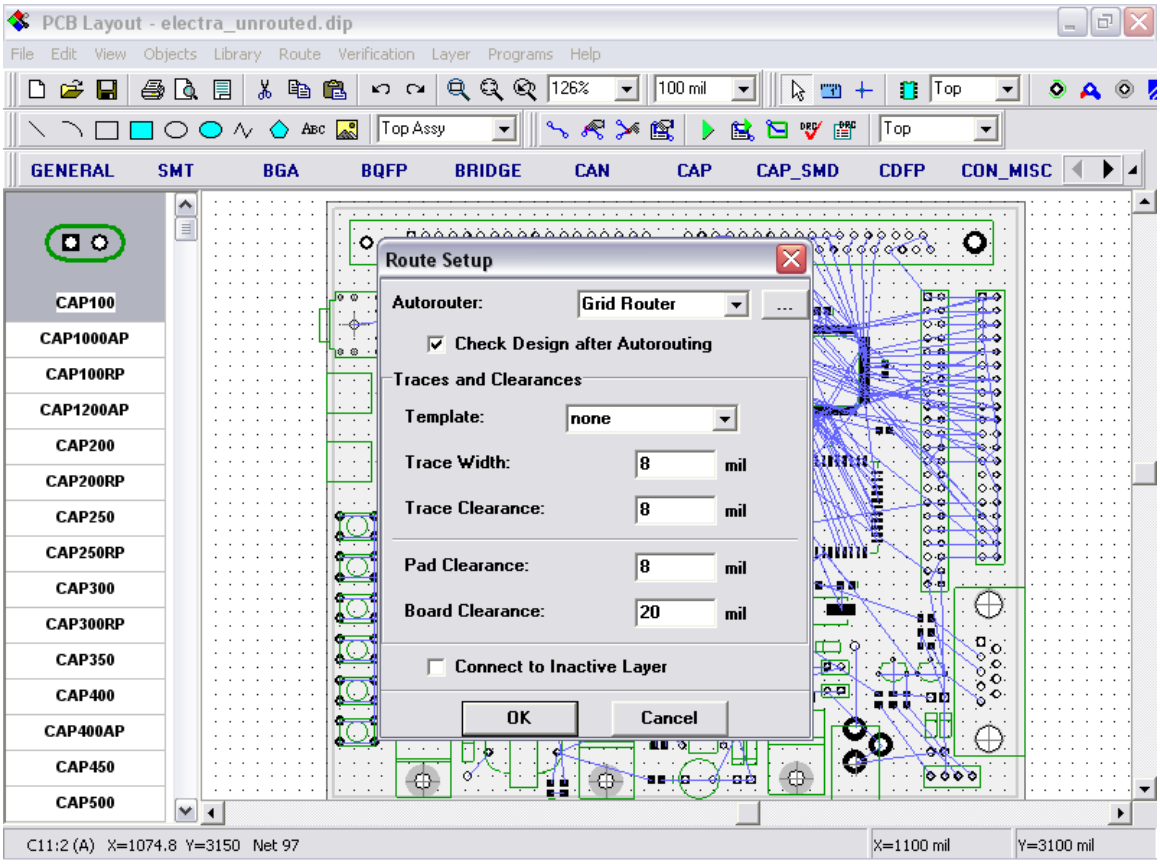
Using Electra auto-router

DipTrace allows you to use highly efficient Electra auto-router. PCB Layout has modules that export data into *.dsn format used by Electra as source data for trace creation, and a module that imports routing result from *.ses format.
If you don't have Electra auto-router yet, you can download trial version from DipTrace web-site to evaluate it:
http://www.diptrace.com/downloads/electra_trial.exe
Please install Electra trial before continuing.

Now please run PCB Layout program and open electra_unrouted.dip file from “C: \ Program files \ DipTrace \ Examples”. Notice that your file includes 2 signal layers and 2 plane layers (Pwr and Gnd). Also plane layers contain unpoured copper pours that are connected to appropriate nets. In future if you would like to create several planes in single physical layer, you have to place several copper pours with appropriate shapes and properties into single plane layer. Electra connects SMD pads to plane layers automatically while routing traces.

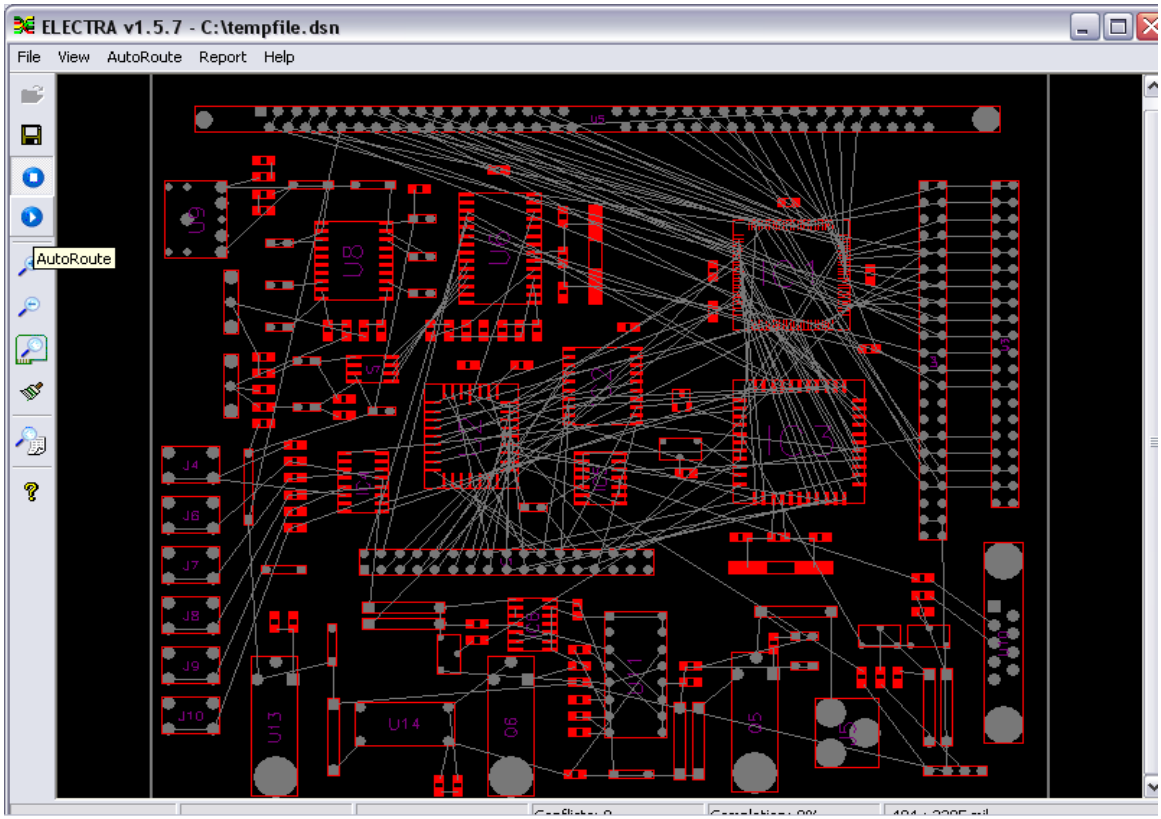


Notice that you have to define trace width and clearance before autorouting. Select Route/Route Setup and change trace width and clearance to 8 mil. Also you can define custom settings for some nets (for example power) via their properties (right click on the pad that belongs to net and select Properties).

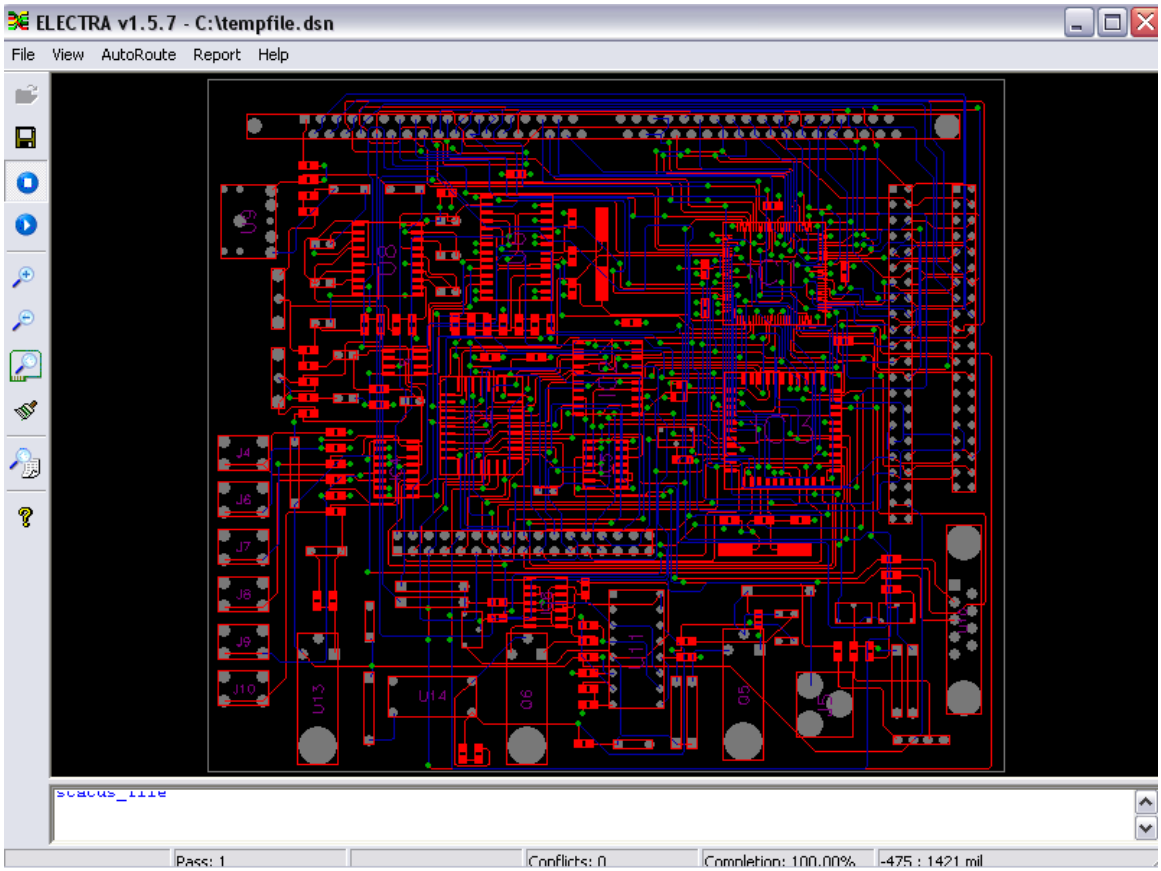


Trace width for the net should be not bigger than size of pads that such net connects. Otherwise in some cases auto-router will be unable to complete your design. Also check if via settings are correct (Layers/Via Properties). We use 40 mil vias with 20 mil hole for current design.

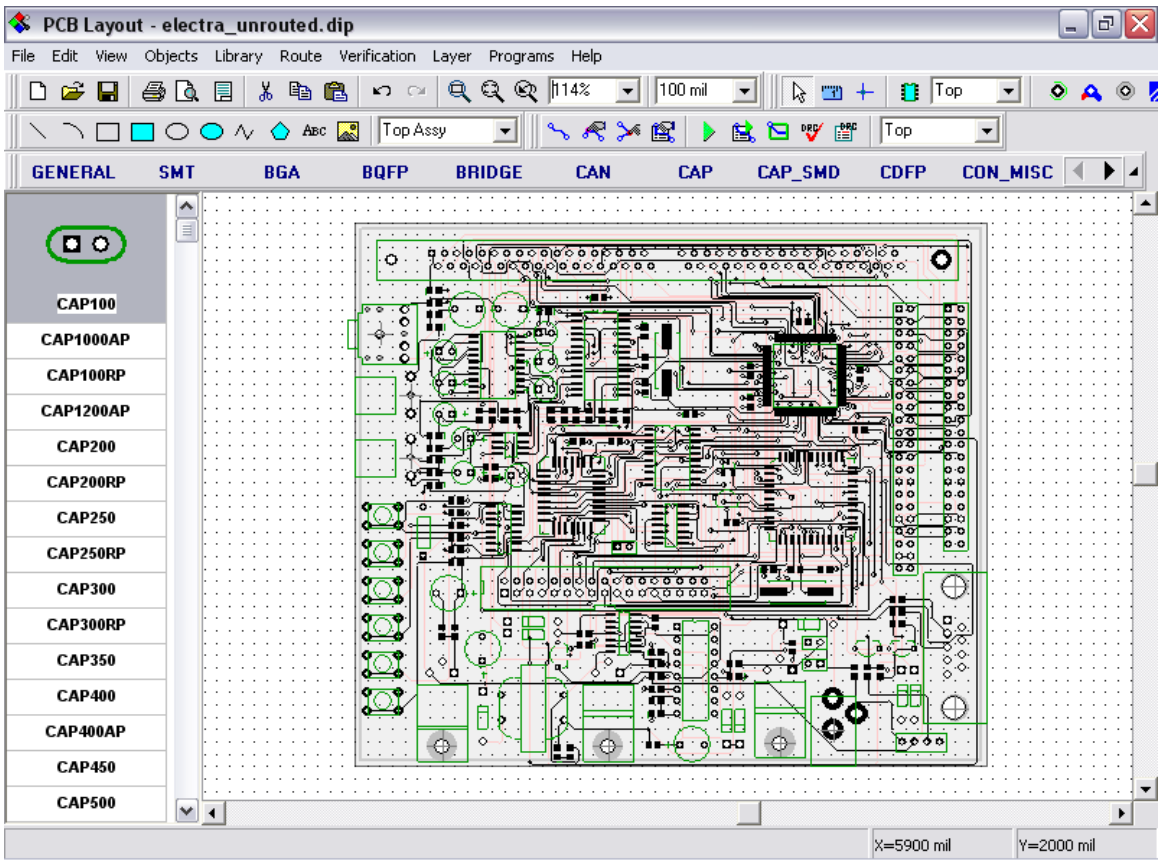
Now select “Route / Electra/Specctra Integration / Export Autorouter DSN” from main menu and save your file. Notice that you should keep existing project without changes. Run Electra auto-router. If you have trial version click Continue in the first window, then uncheck “Autoload Demo Design” box in the next window and click OK. Select File/Open from main menu and open your DSN file.



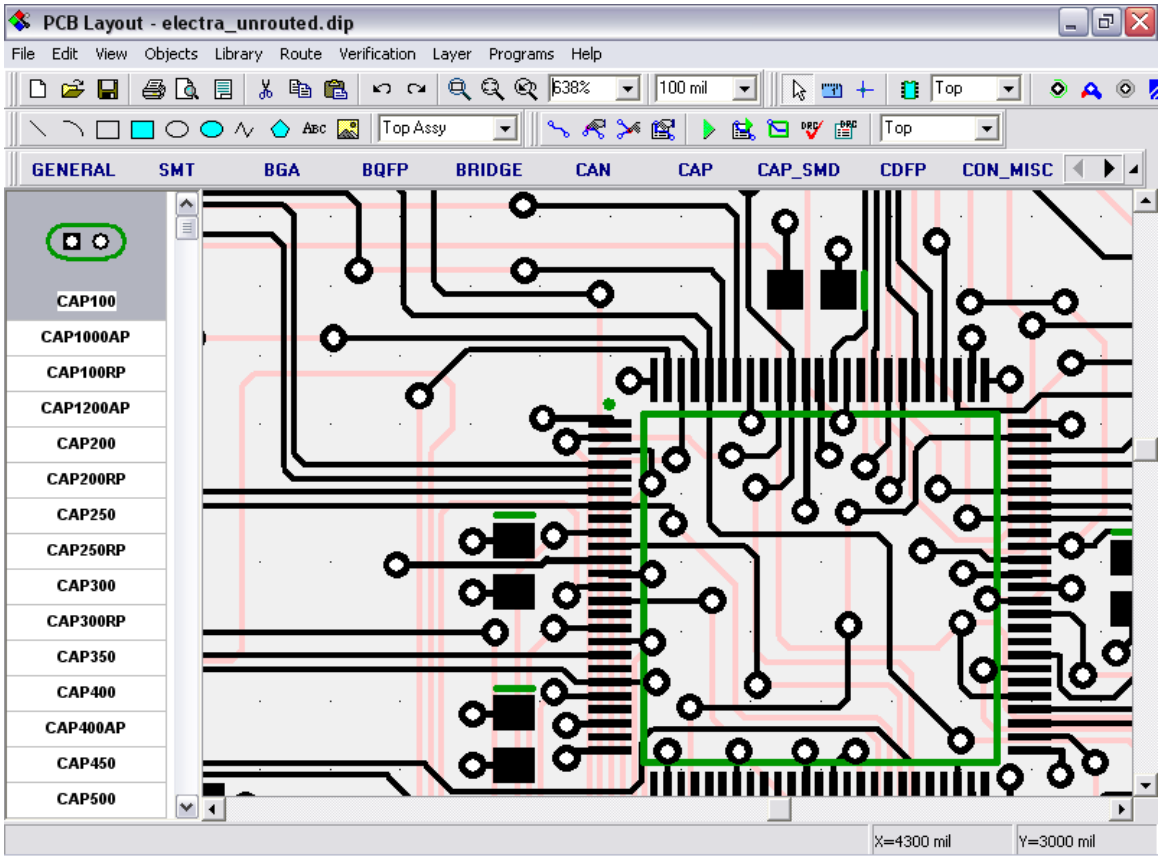
Click AutoRoute button on the left panel to route your PCB and wait a few minutes while Electra routing and optimizing your board. Notice that such design is not very complex, so you can easily route it using Electra 2L (2 signal layers, unlimited pins). When the routing process is completed, press Save button and save routing results as SES file, that you will be able to import into DipTrace.



Now switch to DipTrace PCB Layout program. Notice that component positions of your design should be kept without changes. Select “Route / Electra/Specctra Integration / Import Autorouter SES” from main menu and open your file.



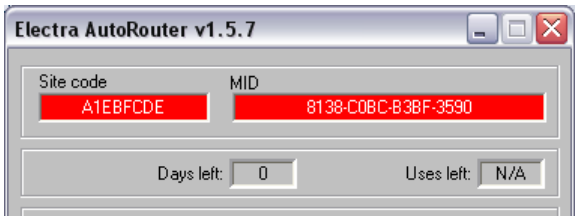
You can also Zoom In your design to check routing accuracy.



Also to complete your design please select all, right-click on one of the copper pours in plane layers and choose Update to pour them. Then run Check Net Connectivity to verify if all nets of your design are properly connected.

Notice that discounts on Electra auto-router are available for registered DipTrace customers. You can find prices and discounts at http://www.diptrace.com/electra.php

To register Electra auto-router you should know Computer Site code and Machine ID.
You can find them in Electra trial nag-screen (red fields in the top of the window).



Now go to http://www.diptrace.com/electra.php, select your edition and click Order via Plimus.
Computer Site code and Machine ID should be entered into additional fields in the bottom of order page.

If you have any questions regarding Electra Autorouter or ordering process, please contact ConnectEDA (http://www.connecteda.com/contact.htm) or DipTrace Team (http://www.diptrace.com/contacts.php)