

# TopoR Tutorial

This document allows you to get started with ease by routing simple projects using TopoR autorouter. The tutorial includes step-by-step design guide that allow you to discover basic program features. If you have any questions while learning the tutorial, contact our support staff: [support@toporouter.com](mailto:support@toporouter.com). We will be happy to be of assistance and gladly answer all your questions. We use TopoR Lite and DipTrace CAD system, all projects are included into DipTrace examples. You can use another CAD system that can create DSN files and other layouts (there is no big difference). This version of tutorial was created with TopoR ver. 4.3.89.14 and DipTrace ver. 2.0.07

## I. Downloading and installing

First we should download and install TopoR Lite and DipTrace.

Notice that if you already have TopoR Demo, you should download TopoR Lite to be able to save files and export them back to CAD system. TopoR Demo allows to route larger projects to evaluate TopoR capabilities, but doesn't allow to save files.

Open <http://www.toporouter.com/download.php> and download TopoR Lite, then install it to your PC. Notice that during installation is also adds Visual C++ 2005 Redistributable package, please do not stop installation of that package.

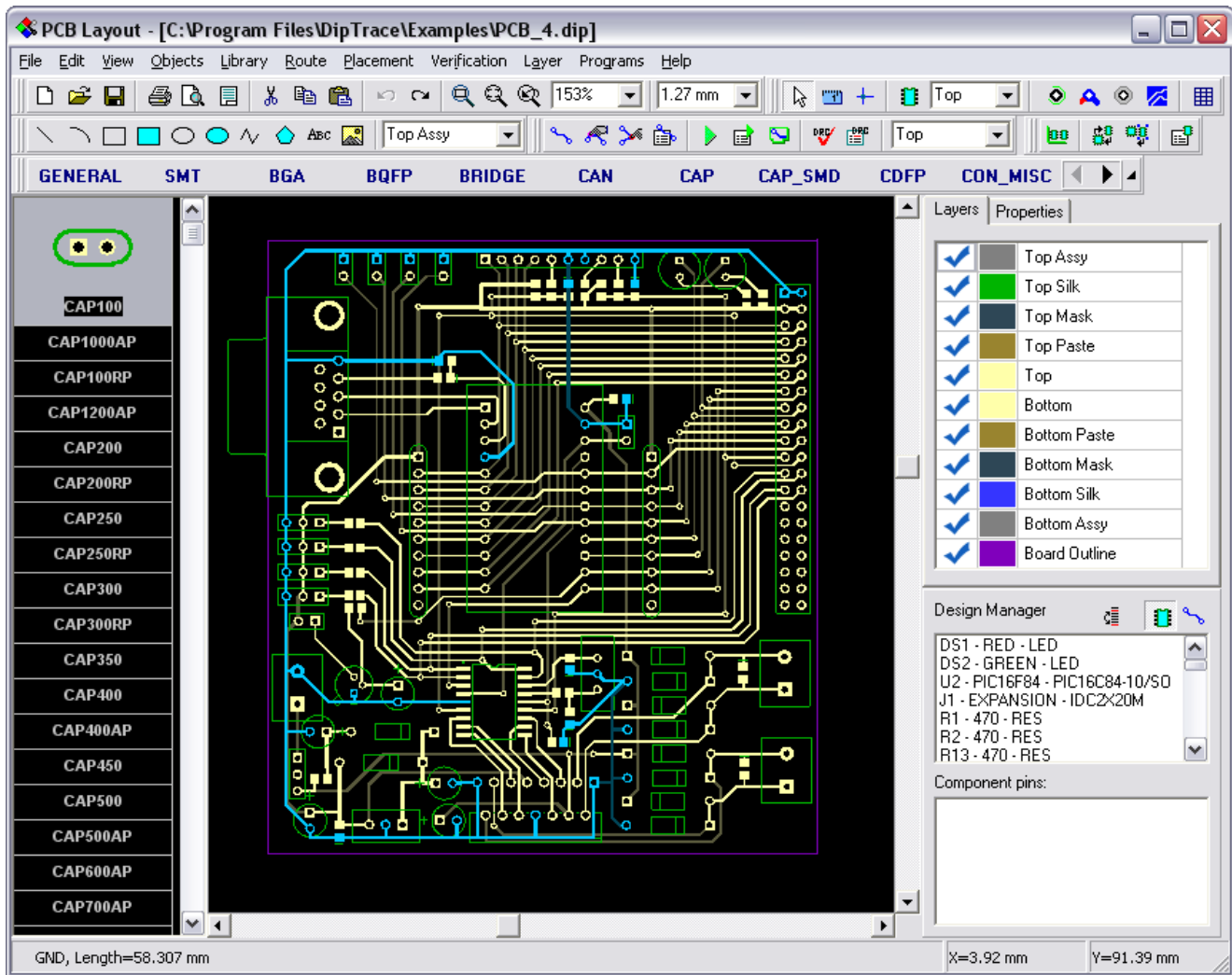
Open <http://www.diptrace.com/download.php> and download DipTrace Freeware, install it to your system. You can also upgrade it to Non-Profit Lite version (500 pins) without charge by sending request about that to [support@diptrace.com](mailto:support@diptrace.com) (say you got information about that upgrade from TopoR tutorial), however DipTrace Freeware can open files of unlimited size, so we can use it too.

## II. First project

### 2.1 Exporting DSN from DipTrace CAD system

Our first project will be simple PCB from DipTrace examples.

Run DipTrace PCB Layout from Start/All Programs/DipTrace and open C:/Program files/DipTrace/Examples/PCB\_4.dip. You can see simple PCB that was routed manually.

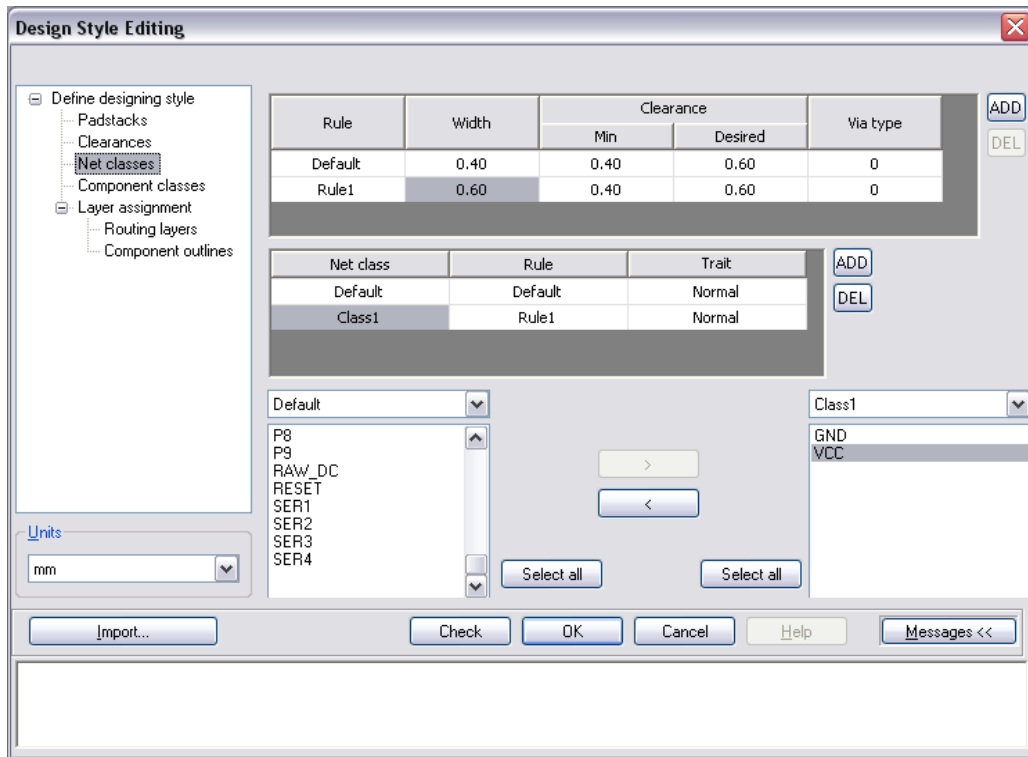


We can configure net classes, etc. in DipTrace (those classes will be transferred to TopoR via DSN format), however we will do that in TopoR. In DipTrace select «View/Units/mm», then «Route/Unroute All» to unroute all traces and «Route/Route Setup» to set default trace properties. Set 0.4 mm for trace width and clearance, OK to apply settings. Select «Route / Electra/Spectra Intergration / Export Autorouter DSN» and save pcb4.dsn file. Minimize DipTrace.

## 2.2 Importing DSN and configuring net classes

Run TopoR Lite from «Start/All Programs/TopoR Lite». Select «File/Import/Import PCB File», find your pcb4.dsn file and open it. Then OK in «Import» window and «Coach Message».

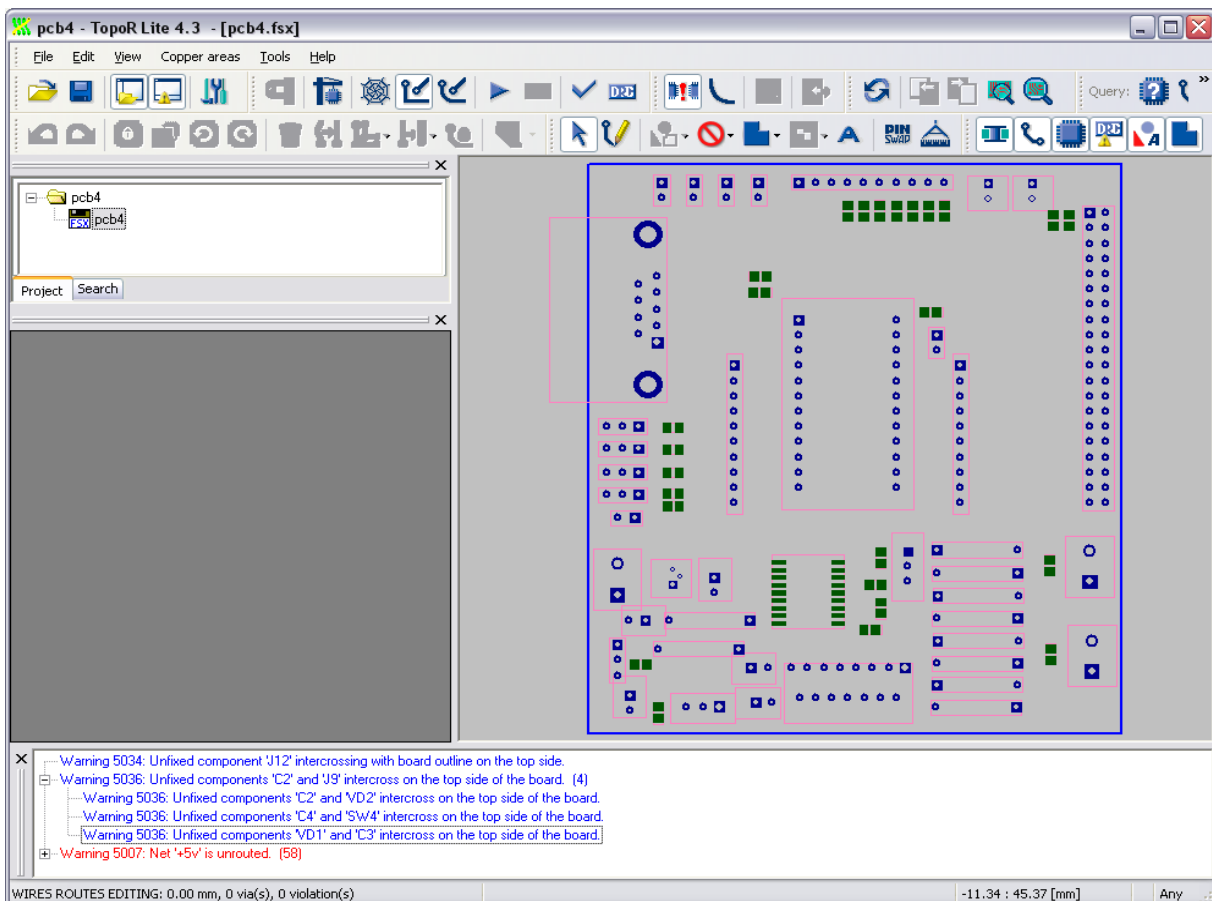
In Design Style Editing dialog box select «Net classes» in the left side and «Units/mm» in lower-right. As you can see there are 2 net classes: Default and Class 1 for GND net. These classes are imported from DipTrace. Change width for Class 1 to 0.6 mm — in the Top grid click on the width field for Rule 1 and change its content to 0.6. Also move VCC net to Class 1, move it from the left list of nets (Default class) to the right list (Class 1) — scroll left list to the bottom, find VCC, select it and press «>>» button.




Press OK to import design with these settings. Also OK it Catch message, you can also read catch messages to learn program features more in-depth and check «Do not show this message again».

## 2.3 Preparing to route

After import we can see components with outlines.






DipTrace makes rectangular outline based on component graphics for each component. For our design this caused intercrossing of some components (see warnings below). To use TopoR auto-placement and placement correction features we should fix such warnings. If you will not use automatic placement correction skip this step. Double click on the warning to highlight intercrossing components and then move them (to move component left click on it to select and then drag using left mouse button). Do not correct intercrossing with board outline warning as J12 is connector that has fixed position.

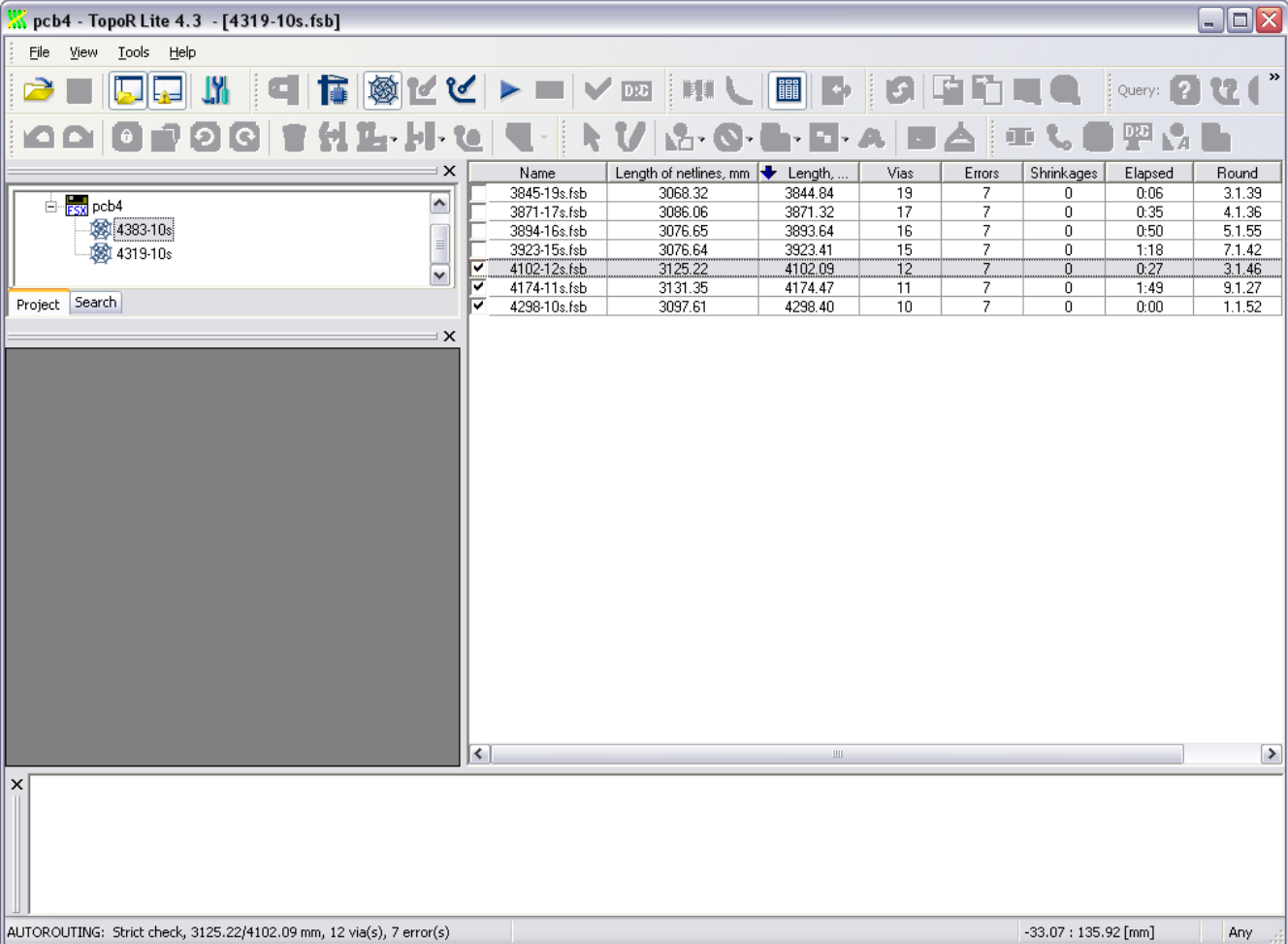
Next step is locking position of some components. If you don't lock components TopoR thinks that it is possible to correct their position after making traces. That may cause making more traces than possible in some areas. So select connectors and other components with fixed position (use Ctrl for multiple select) and press «F» key or  button to lock them. The outline color for locked components will be different.

## 2.4 Autorouting

Now we are ready to auto-route our PCB.

Press  button to start autorouting. Ng auto-routing TopoR optimizes several auto-routing variants simultaneously, selects the bests and places them into the table. Please wait few minutes while it makes normal routing results and press  button to stop auto-routing.

You can also preview topology results — right click on it in the table and select «Show», to return to the results table press  button. TopoR doesn't make traces when auto-route PCB, but only finds optimal design topology.





The screenshot shows the TopoR Lite 4.3 software interface. The main window displays a table of routing results. The table has the following columns: Name, Length of netlines, mm, Length, Vias, Errors, Shrinkages, Elapsed, and Round. The data rows are as follows:

Name	Length of netlines, mm	Length, ...	Vias	Errors	Shrinkages	Elapsed	Round
3845-19s.fsb	3068.32	3844.84	19	7	0	0:06	3.1.39
3871-17s.fsb	3086.06	3871.32	17	7	0	0:35	4.1.36
3894-16s.fsb	3076.65	3893.64	16	7	0	0:50	5.1.55
3923-15s.fsb	3076.64	3923.41	15	7	0	1:18	7.1.42
<input checked="" type="checkbox"/> 4102-12s.fsb	3125.22	4102.09	12	7	0	0:27	3.1.46
<input checked="" type="checkbox"/> 4174-11s.fsb	3131.35	4174.47	11	7	0	1:49	9.1.27
<input checked="" type="checkbox"/> 4298-10s.fsb	3097.61	4298.40	10	7	0	0:00	1.1.52


The status bar at the bottom of the window displays the following information: "AUTOROUTING: Strict check, 3125.22/4102.09 mm, 12 via(s), 7 error(s)" and the coordinates "-33.07 : 135.92 [mm]".

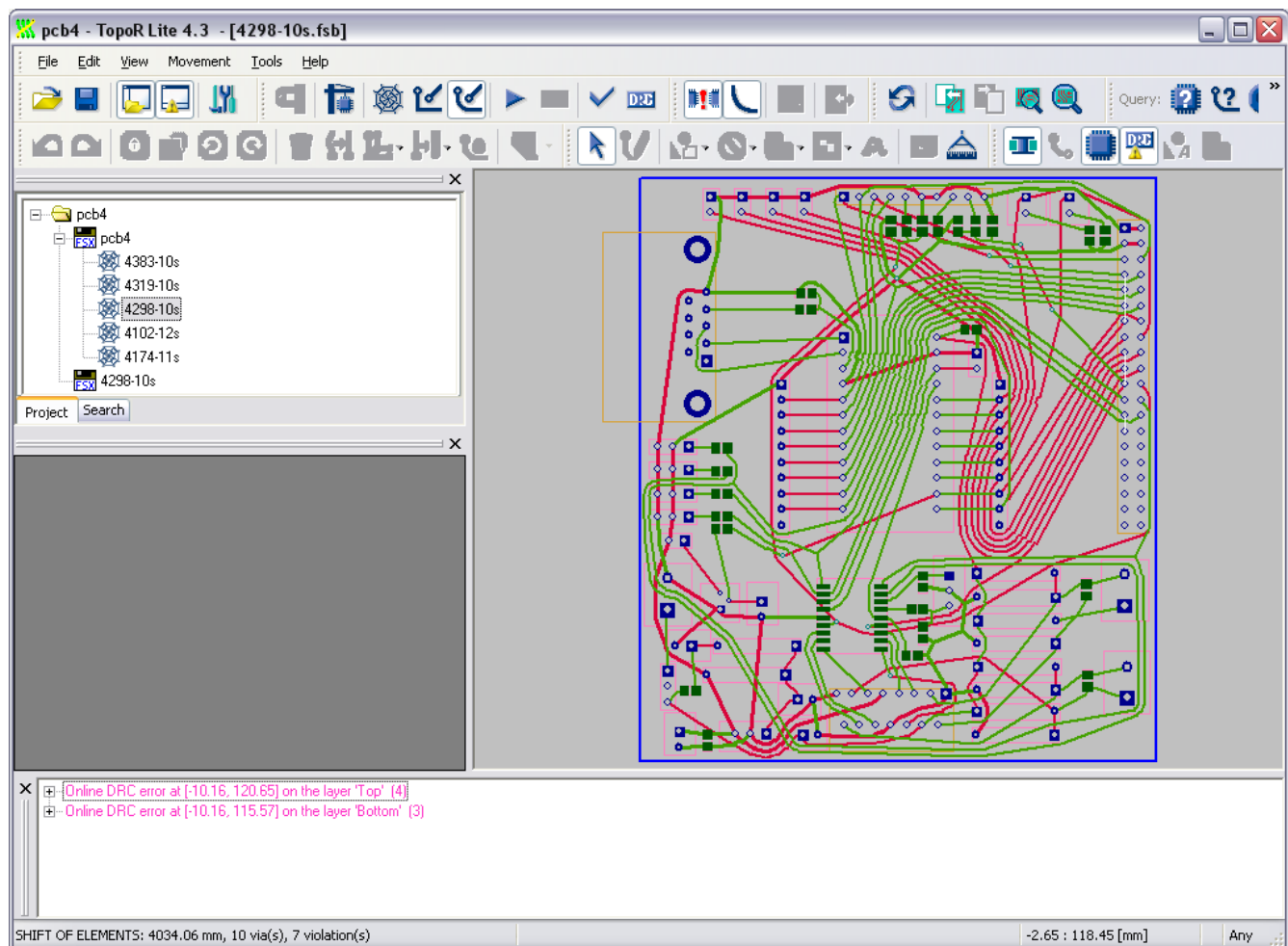
Place checkboxes for the results you like, then right click on the best one and choose «Open for Editing». On this step TopoR makes traces and places vias, for large designs it may take some time.

Traces may include arcs and lines or lines only (arcs are approximated). DipTrace supports arcs, so we can use them, press «  » button to enable arc-like traces. To see another result select it in the project bar in the left side and press «  » button. If you already corrected existing topology, you can save it as fsx file.

## 2.5 Correcting results

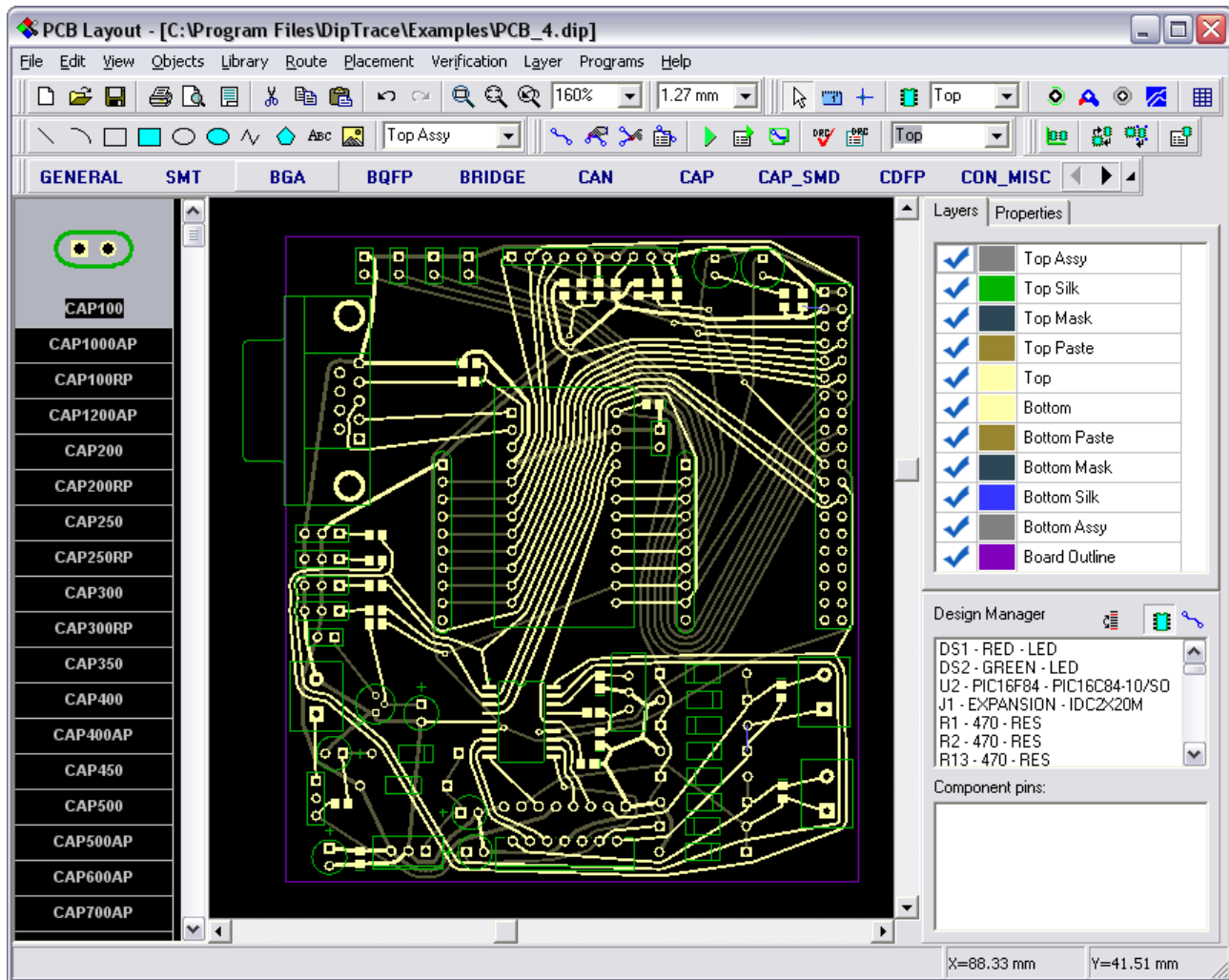
Our routed design have some DRC errors, that we should fix or keep without changes if clearances are ok. It is possible to do that in two ways: allow TopoR to correct component positions or correct positions manually. It is also possible to route/correct traces manually (TopoR has specific tools to work with arc-like traces), however for our simple layout it is not necessary.

For manual correction zoom-in places where you see white lines, see if the clearance is really non-acceptable and move components (left click to select and drag), traces will be recounted automatically. For automatic correction press «  » button and select «Start components and vias automatic movement with wires refining». TopoR will try to remove all DRC errors and minimize trace lengths by moving all unlocked components and vias. After this procedure we can also move components if necessary. We will move components manually as our layout has only few errors.



## 2.6 Exporting SES-file back to CAD system

After completing correction we can export results as SES-file back to DipTrace or export gerber/drill files directly from TopoR. To export SES-file select File/Export/Specetra SES and save pcb4.ses Minimize TopoR and choose DipTrace PCB Layout on the Windows taskbar to show it. Select «Route / Electra/Specetra Integration / Import Autorouter SES» from the main menu and specify pcb4.ses. Now you can edit this file in DipTrace, export gerber/drill files etc.



### III. Second project

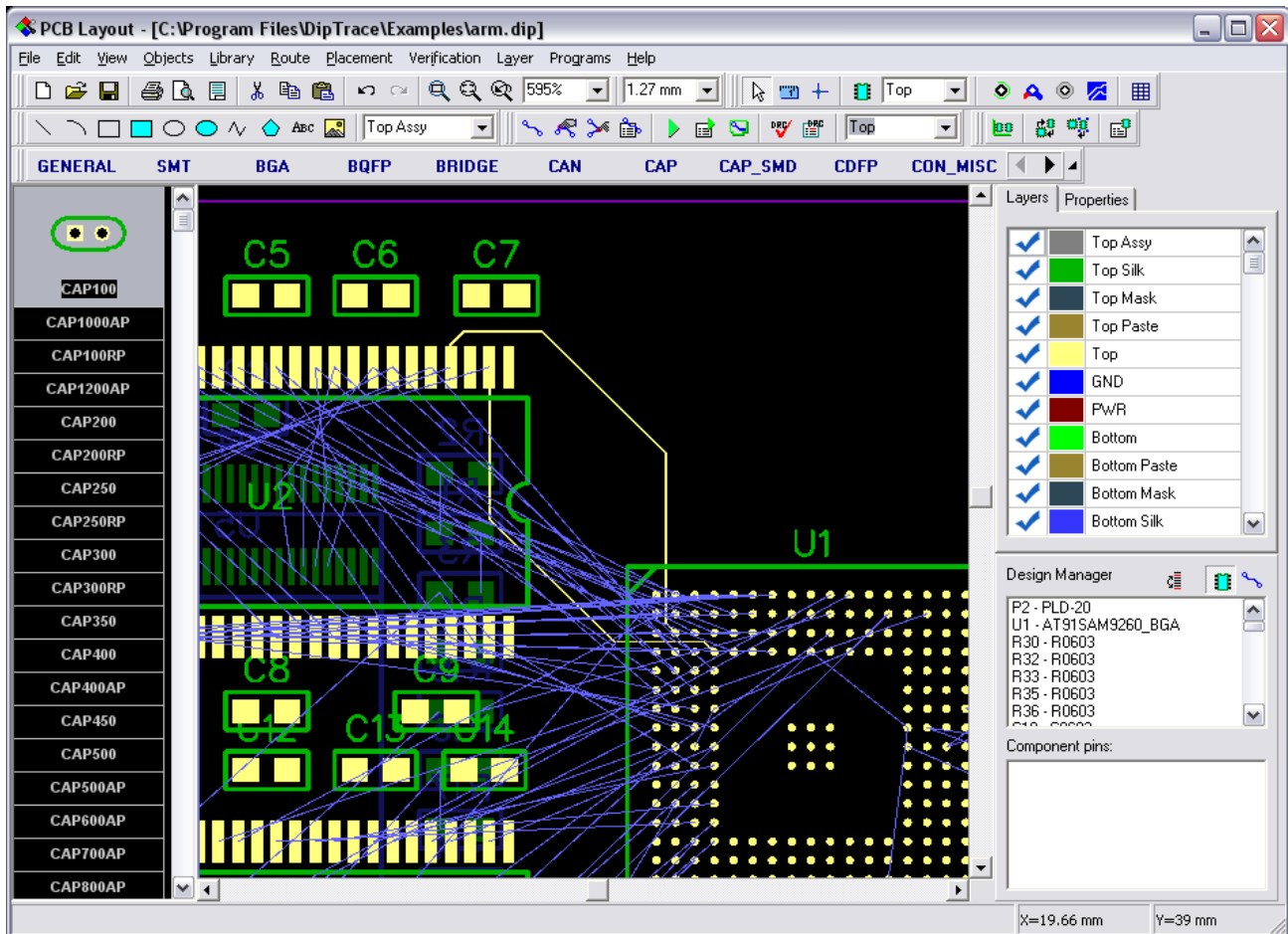
#### 3.1 Preparing PCB in DipTrace CAD System

Our second project will be more complex board with power/gnd planes and BGA component. We will also try to edit it manually after auto-routing in TopoR.

Run DipTrace PCB Layout and open C:/Program files/DipTrace/Examples/arm.dip

As you can see the file is already routed with TopoR, so please unroute using «Route/Unroute All» command from main menu. Then select Route/Route Setup and set trace width and clearance to 0.1 mm (View/Units if units are not mm). Small trace width and clearance are necessary to be able route BGA-pattern. Also change default via size through Layer/Via Properties, it should be outer diameter — 0.4 mm and hole diameter — 0.2 mm. Notice that our PCB has 2 signal (routing) layers — Top and Bottom and 2 plane layers — GND and PWR used for ground and power nets. In DipTrace you can add/change layers from the Layers menu if necessary.

Also we will add a few traces manually (let us assume that we need to route critical traces before auto-routing). First please select Layer/Layer Setup from main menu and change color of the Top layer, currently it is red and the pads of the routed net will not be highlighted as pad highlight color (View/Colors) is the same. Then select Route/Tools/Route Manual from the main menu or Route Manual button on the route panel and make several traces for existing nets in DipTrace (left click on the first pad/trace, then define trace by points and left click on the second pad/trace).

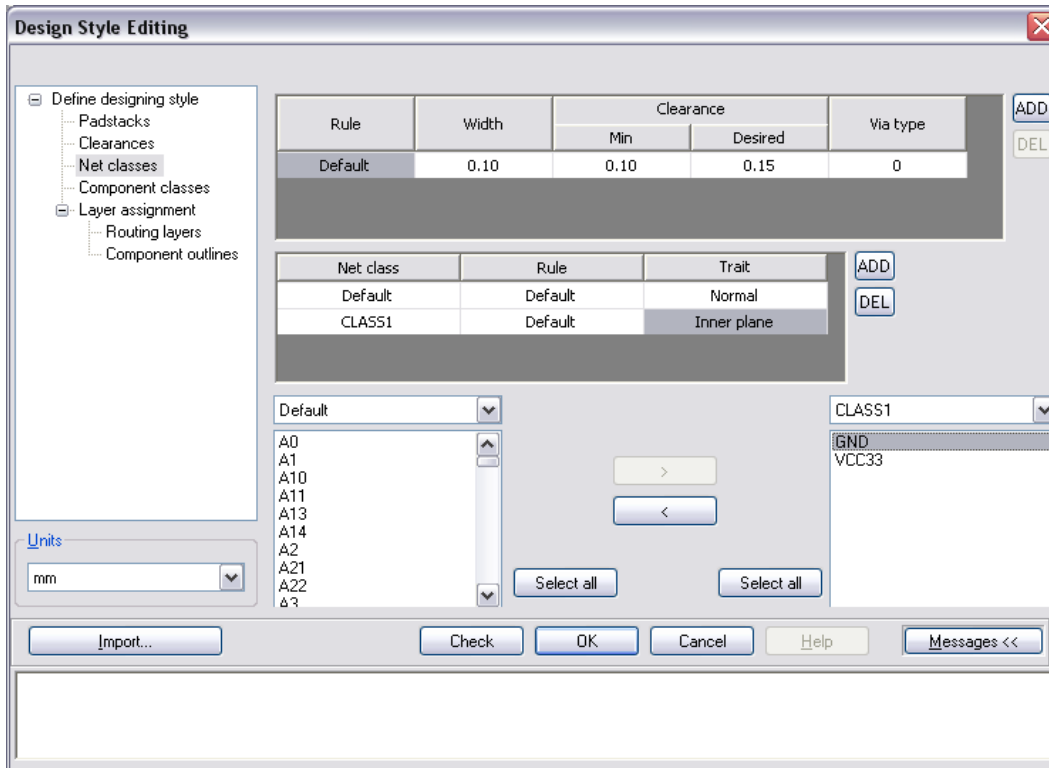




Select «Route / Electra/Spectra Integration / Export Autorouter DSN» from main menu and save arm.dsn file, minimize DipTrace.

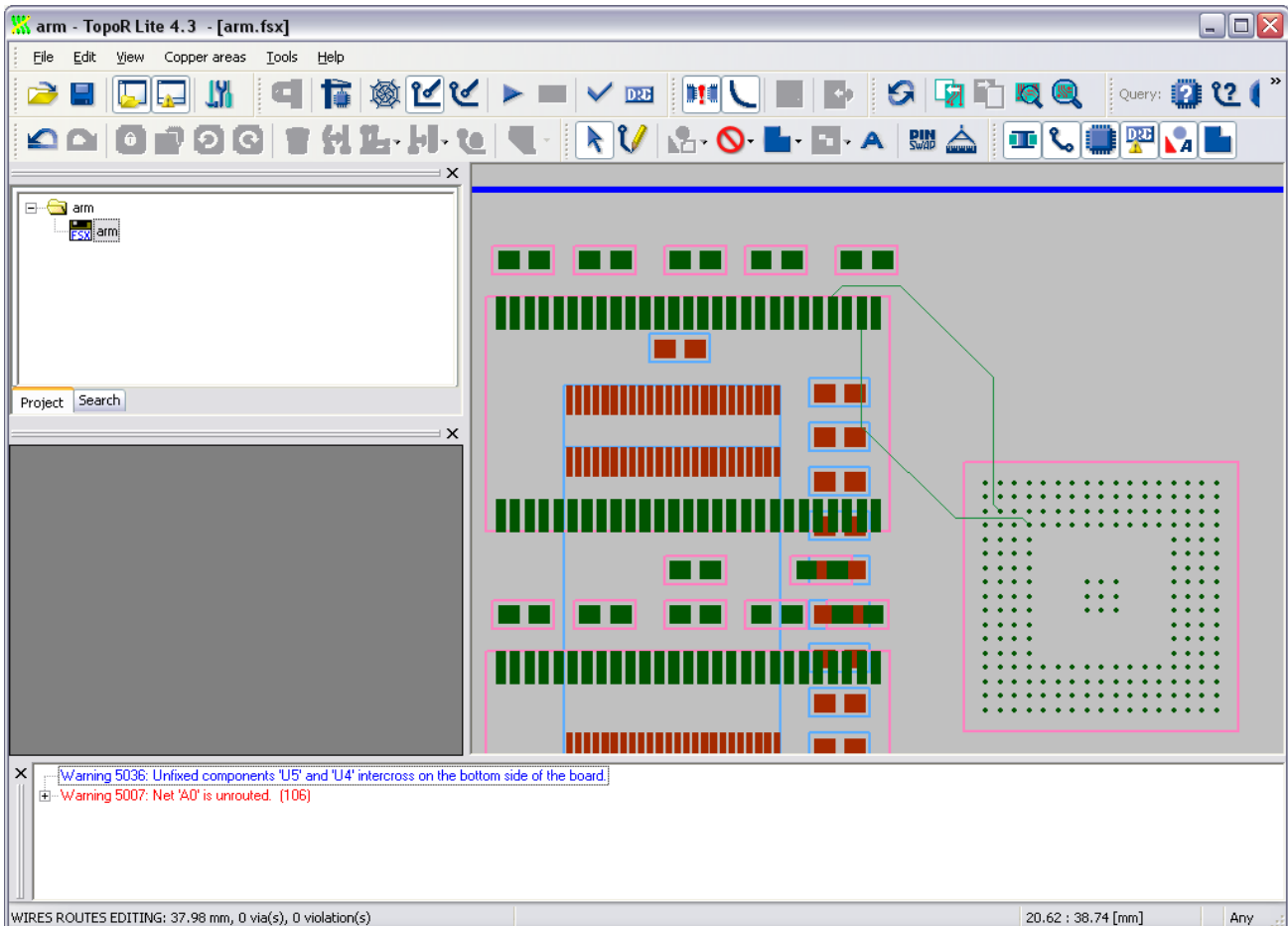
### 3.2 Importing DSN and preparing to route.

Now run TopoR, select «File/Import/Import PCB file» from main menu and then your arm.dsn. In the «Design Style Editing» dialog box we should specify that our VCC33 and GND nets are connected to inner plane layers to let auto-router only make vias for them.

Select «Net classes» in the left side, add class (Add button near the grid with «Net class», «Rule», «Trait» field), select new class and set change «Trait» to «Inner plane». Move GND and VCC33 from Default to CLASS1. Press OK to import DSN file.



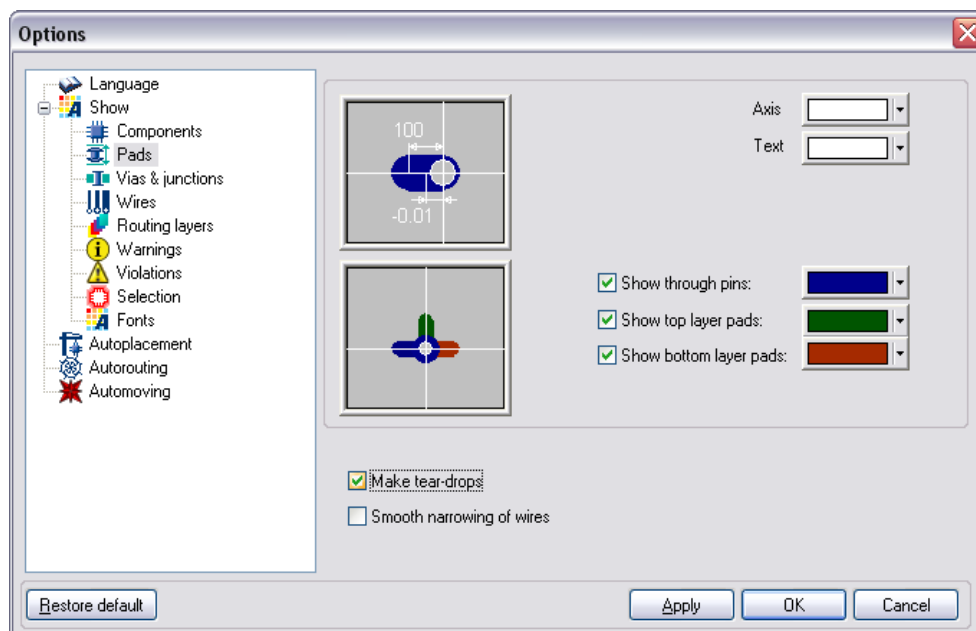
Now select routed traces using box selection and press «  » button to lock them. Then select connectors and other components with fixed positions and press «F» or «  » button to lock them.







The layout has two intercrossed components — U4 and U5. You should not fix this as only one of these components will be placed to the PCB.

We will also enable tear-drops for this layout, select «Tools/Options» from main menu, then Show/Pads in the left listbox and check «Make tear-drops» box, press Apply.

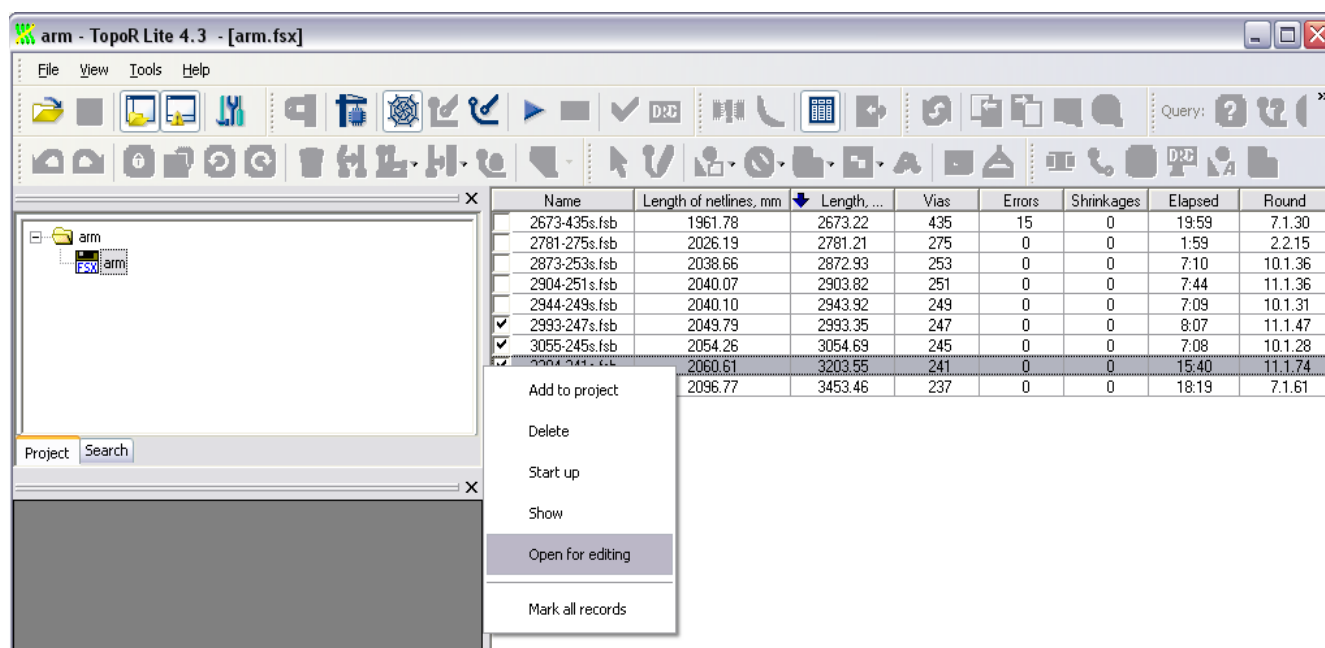


You can also disable or enable special BGA-component routing in this dialog-box, See «Autorouting» options. Choose OK to close the dialog box.

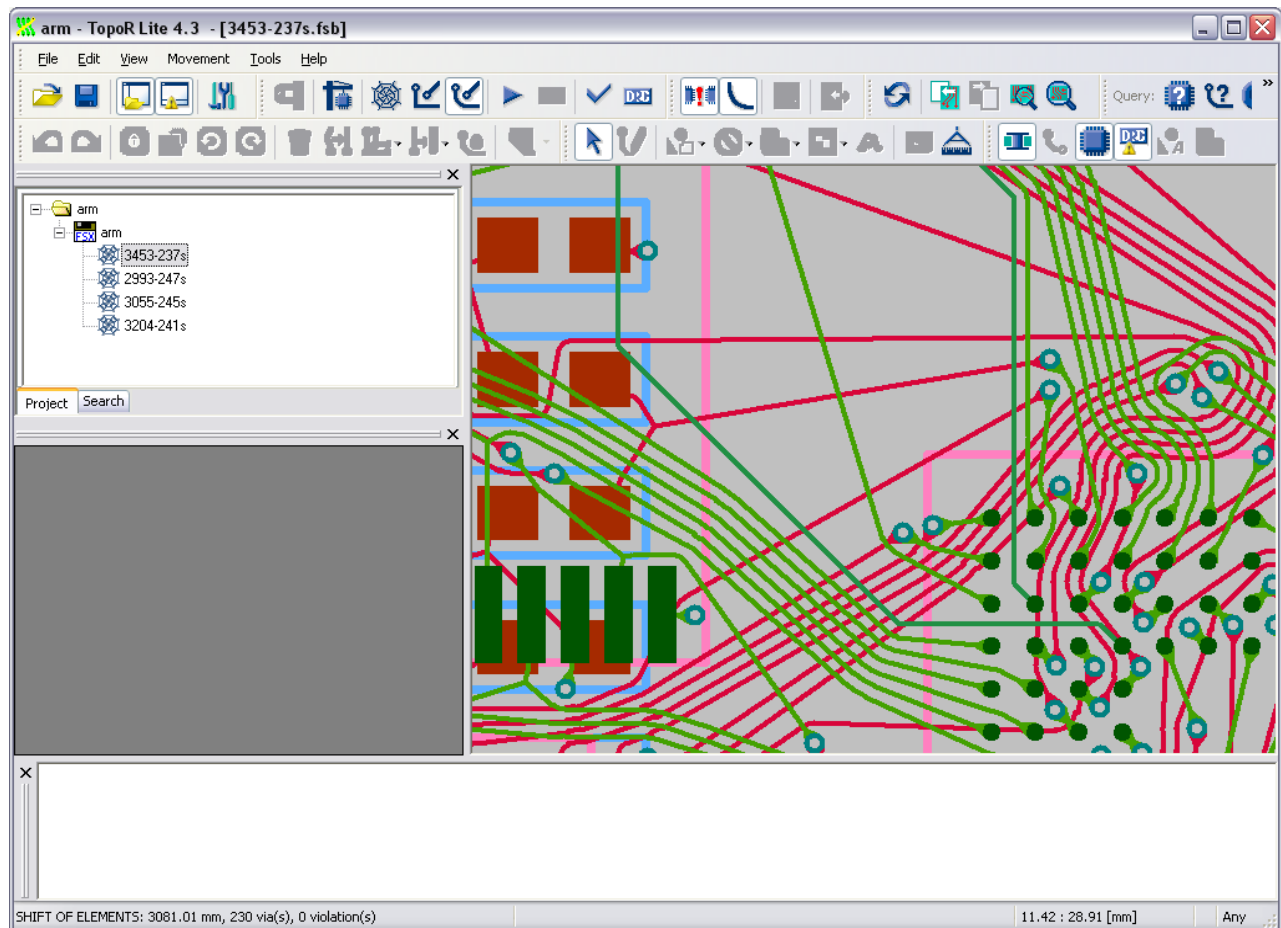
### 3.3 Auto-routing

Press «» button to run auto-router, wait 5-6 to get optimal results with minimum 4-5 depth (first digit in «Round» field) and stop auto-router using «» button.

Check the results you like, right click on one of them and select «Open for Editing».




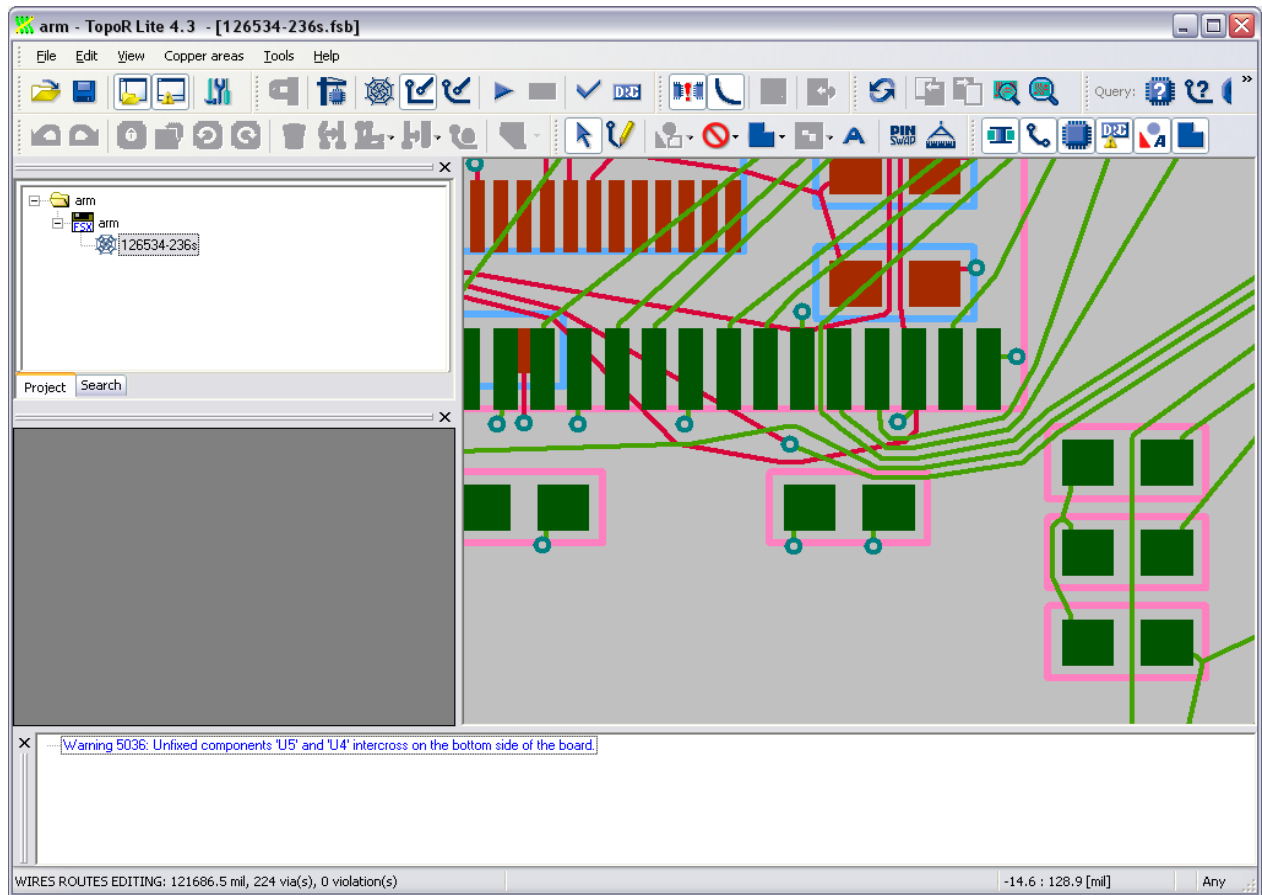
In our case routing comes without DRC errors, so we don't need to move components, etc. However if your variant have errors you can move components/vias to fix them.



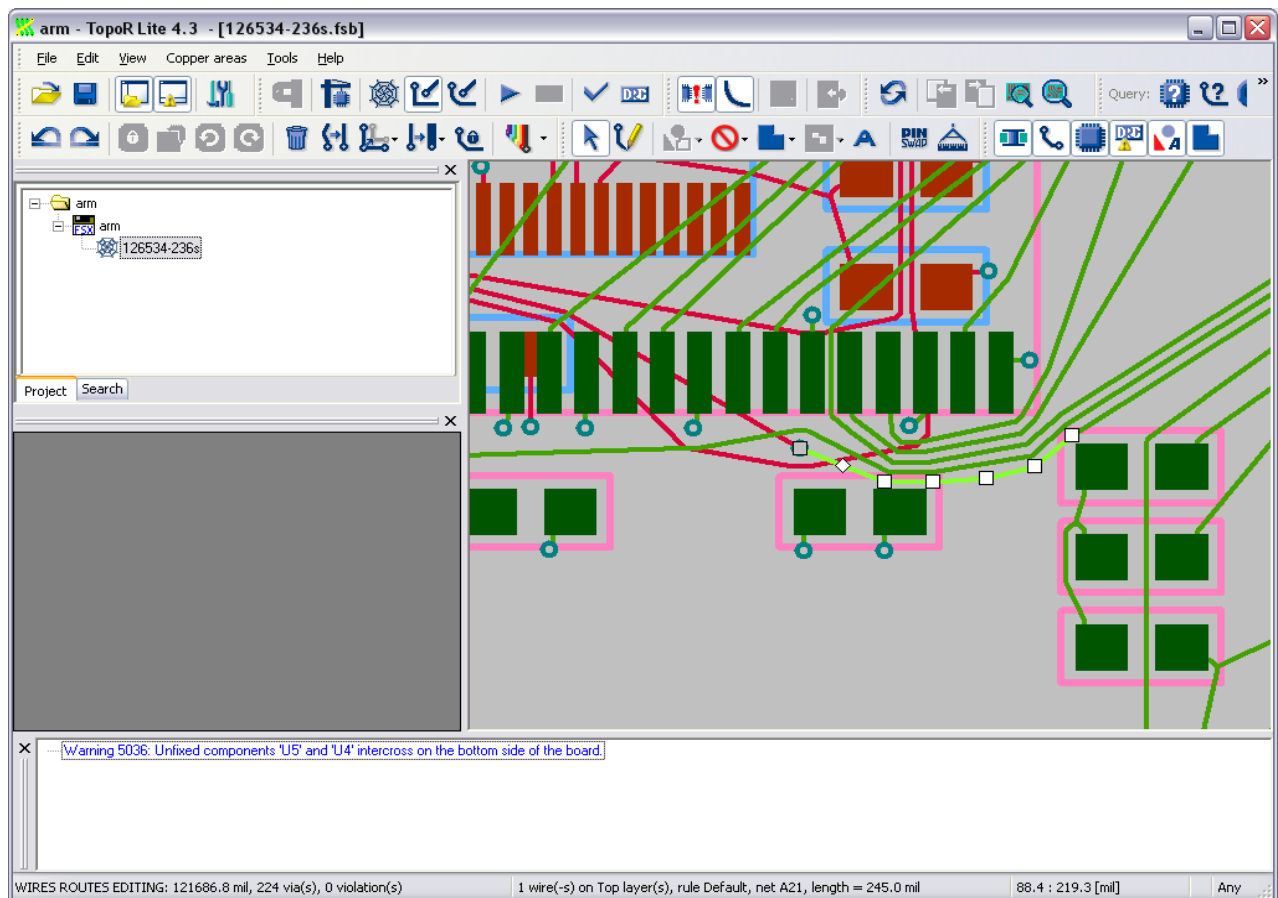
### 3.4 Manual editing traces



TopoR has specific features to edit arc-like traces manually. Editing such traces in DipTrace or other CAD program without specific tools of topology editing and automatic trace making based on topology feature is complex task. For simple layouts manual editing is not necessary, however we will learn how to do it as you may need it for more complex layouts.

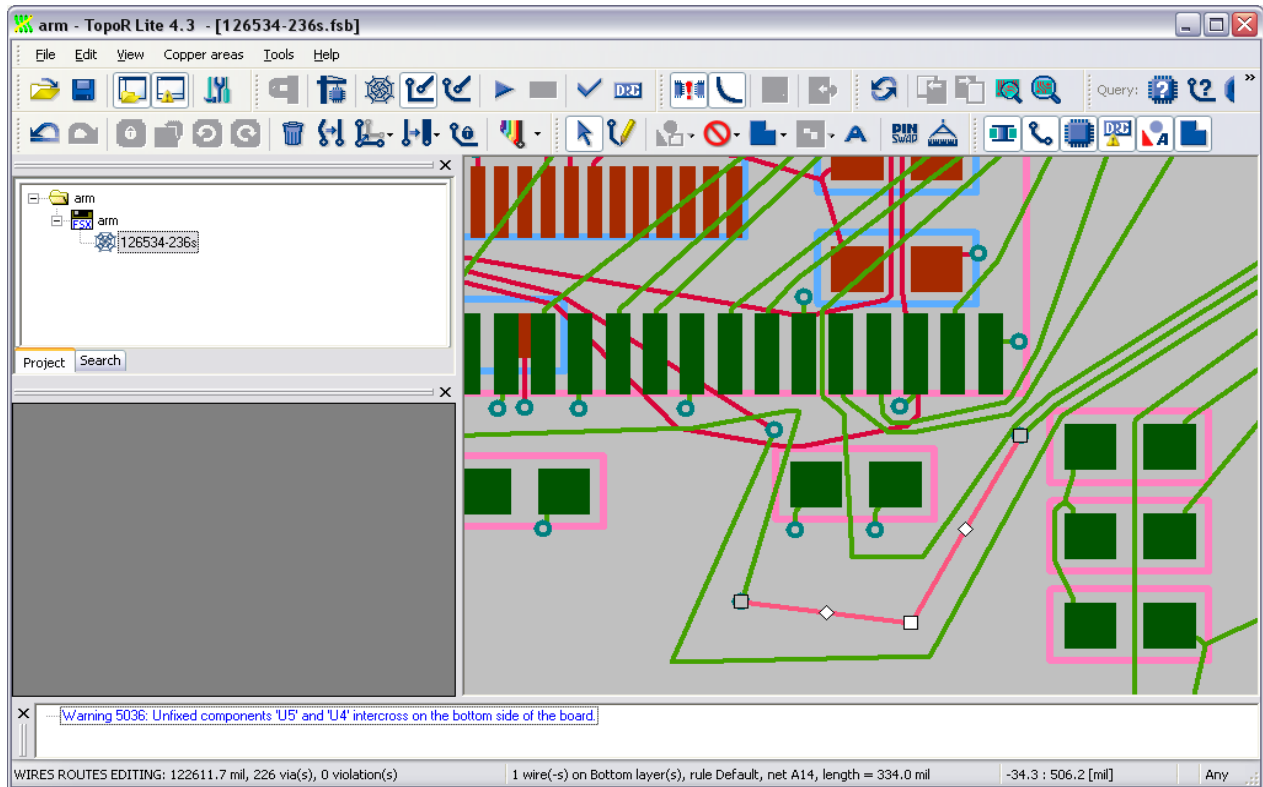
Scroll your design to some place where you want to edit traces. BTW use mouse scroll for zooming and right mouse button for panning. We will scroll our layout to the bottom-left area (C16 component). Press «  » button to switch to manual editing mode, arc traces will be converted to polylines without arcs.




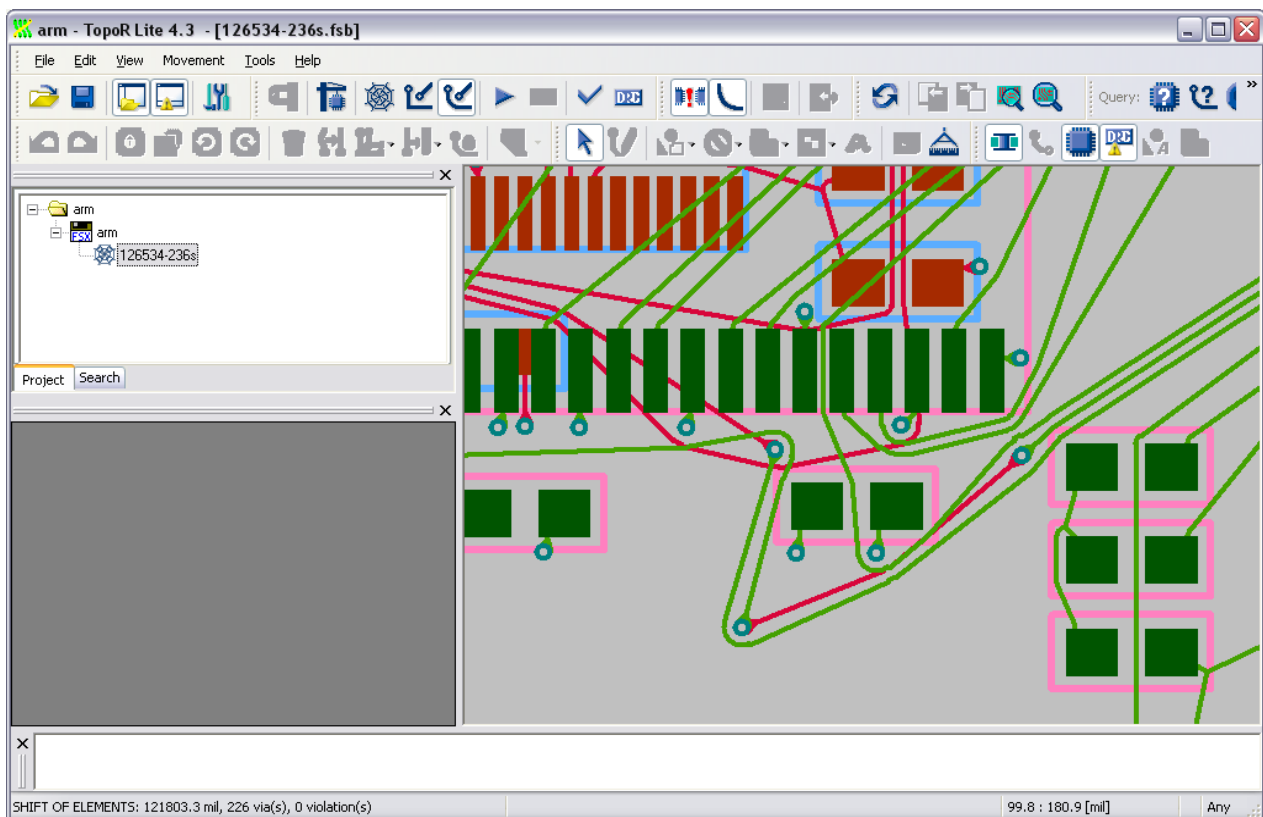
Now select the part of trace — hold down Ctrl key and left click on the parts of trace, you can also box selection to select several parts of the trace.



Press cut-off button (  ) to convert all selected segment into one — usually this is necessary to simplify editing. Move center point of this segment to another location to change topology. Now do the same with few other traces. Use «  » button to change layers of selected segments. Also you can move vias and components in this mode. In our case we only play with this feature, however usually it is used to fix errors. Our final picture after moving several traces will look as follows:



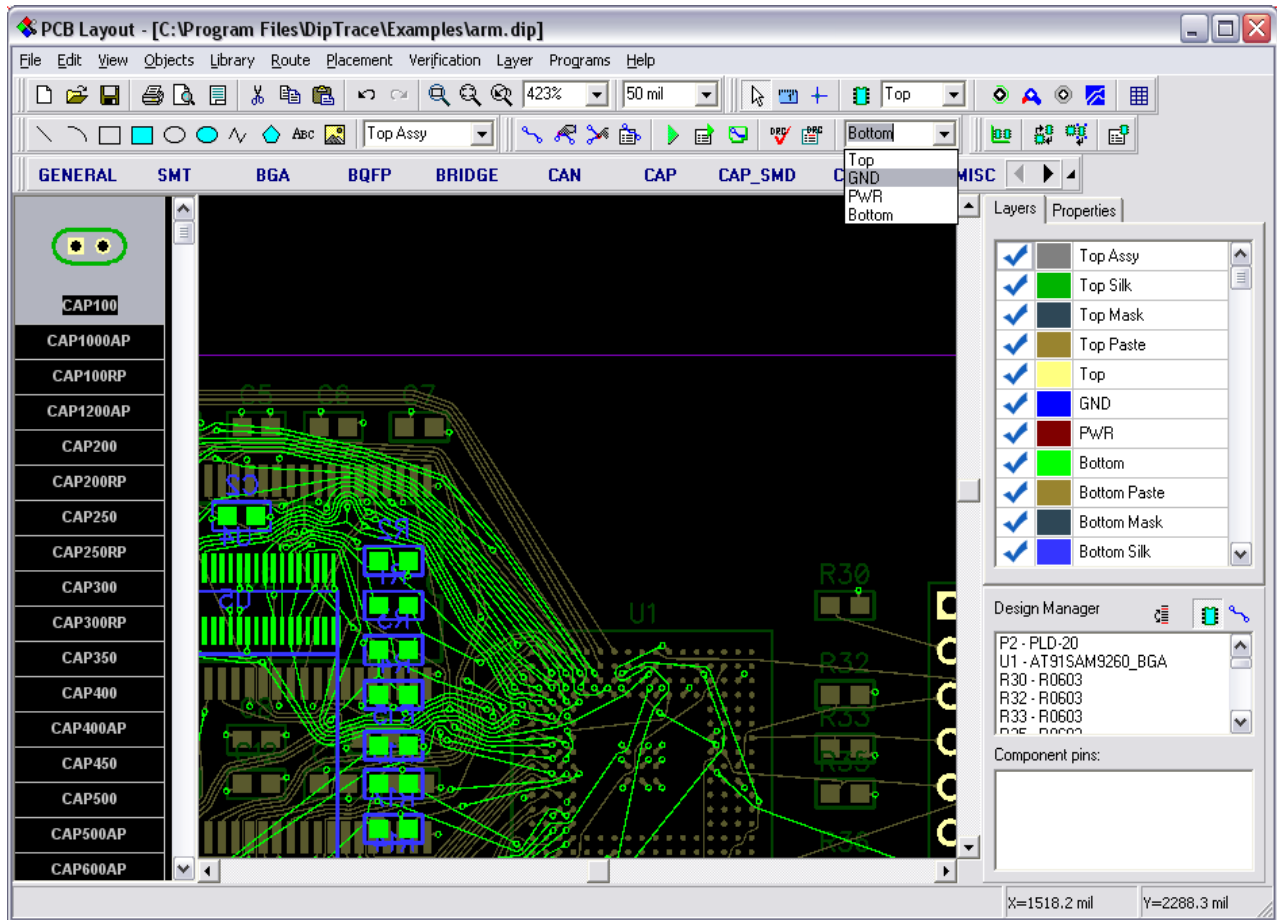
Press «  » button to return to freestyle editing mode and make arc-like traces based on our modified topology.



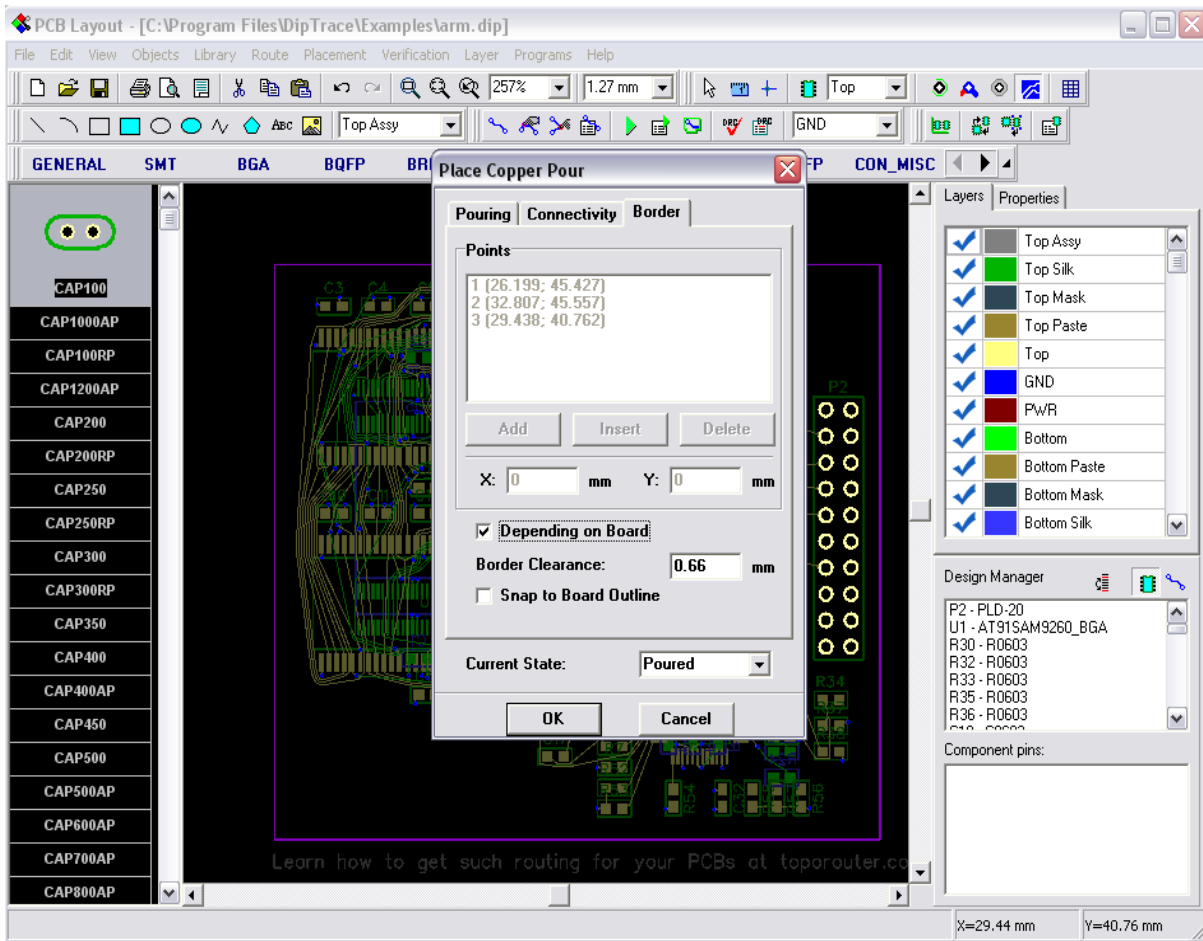
### 3.5 Exporting back to DipTrace and adding planes.

Select File/Export/Specetra SES from main menu and save arm.ses file, then minimize TopoR and select DipTrace PCB Layout on the task bar. Choose Route / Electra/Specetra Integration / Import Autorouter SES and then your arm.ses file.

Next step will be placing copper pours to plane layers and connecting them to ground and vcc vias. Select «GND» layer in the «Current Signal/Plane Layer» box.



Then «Objects/Place Copper Pour» from main menu of button on the objects panel, place few points, right click and Enter. In the copper pour placement dialog box select «Connectivity» tab and «Connect to Net : GND» - it should be already set as we place copper pour to the plane layer with gnd net in properties. Define thermal connections, it is usually «Direct» for vias. Then select «Border» tab and check «Depending on Board» to make copper pour border based on board outline, define clearance between copper pour and board outline. Press OK.



The same for VCC layer. We can also define different thermal settings for pads of P2 connector, right click on one of the pads, select «Thermal Settings», then uncheck «Use copper pour settings» and select different thermal type. Update copper pours (right click on the copper pour border and Update). To ensure all connections are ok use Verification/Check Net Connectivity feature.

That's all for now.

Thank you for learning this quick tutorial. If you like TopoR and want to learn it more in-depth please use user manual that is installed together with TopoR.

Join our group at Yahoo! to communicate with other TopoR users, suggest new features and get support from the developer: <http://groups.yahoo.com/group/topor>

Order TopoR online at <http://www.toporouter.com/order.php>

Links:

Other TopoR tutorials: <http://www.toporouter.com/tutorials.php>

DipTrace web-site: <http://www.diptrace.com>

DipTrace tutorial: <http://www.diptrace.com/books/tutorial.pdf>