Tutorial 2 Automatic Placement & Routing

Please follow the instructions found under Setup on the CADTA main page before starting this tutorial.

1.1. Start Encounter

Log on to a VLSI server using your EE departmental username and password.
Go to your Encounter directory by typing:

cd ~/cad/se

cp your mapping verilog file without header and output lef file from previous tutorial and via fix pearl code:

cp ~/cad/cadence/ABSOutput.lef .

cp /home/cad/startup/EE6325/aux/addvias.pl .

Source the proper profile:

```
. /proj/cad/startup/profile.ic-5
```

Start the Encounter design environment by typing the following in your Encounter directory:

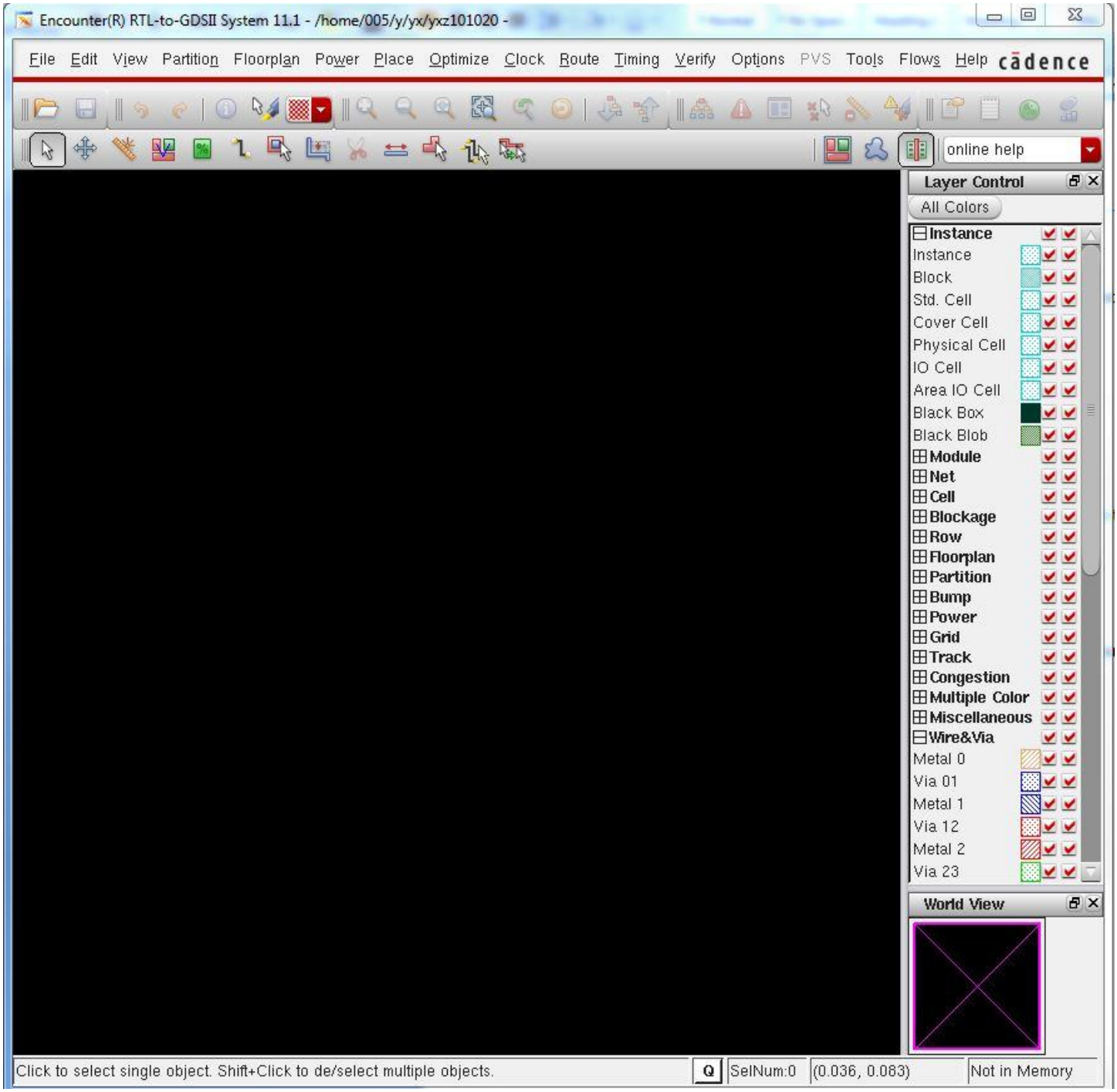
If you are on apache:

```
/proj/cad/cadence/edi_test/edi-11.10.000.sun4v/bin/encounter
```

If you are on txace:

```
/proj/cad/cadence/edi_test/edi-11.10.000.lnx86/bin/encounter
```

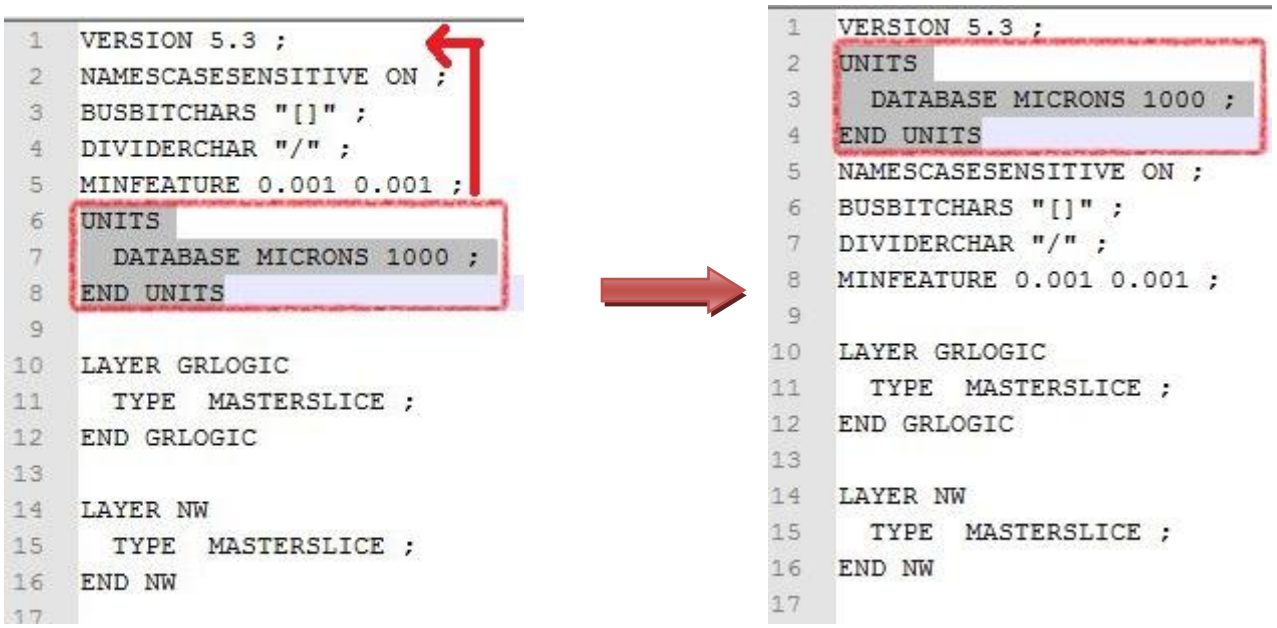
(MAKE SURE YOU DO NOT RUN THIS IN BACKGROUND!!!)


Encounter(R) RTL-to-GDSII System 11.1 - /home/005/y/yx/yxz101020 -
File Edit View Partition Floorplan Power Place Optimize Clock Route Timing Verify Options PVS Tools Flows Help cadence
online help
Layer Control
All Colors
Instance
Instance
Block
Std. Cell
Cover Cell
Physical Cell
IO Cell
Area IO Cell
Black Box
Black Blob
Module
Net
Cell
Blockage
Row
Floorplan
Partition
Bump
Power
Grid
Track
Congestion
Multiple Color
Miscellaneous
Wire&Via
Metal 0
Via 01
Metal 1
Via 12
Metal 2
Via 23
World View
Click to select single object. Shift+Click to de/select multiple objects.
Q
SelNum:0
(0.036, 0.083)
Not in Memory

1.2. Import files

Before you import the LEF file, you will need to modified your LEF file a little bit, so that the Encounter11.1 can read your LEF file correctly.

Open the ABSOuput.lef, find " DATABASE MICRONS 1000 " part, cut this whole part and paste it in the beginning of LEF file just like following. Save the file.



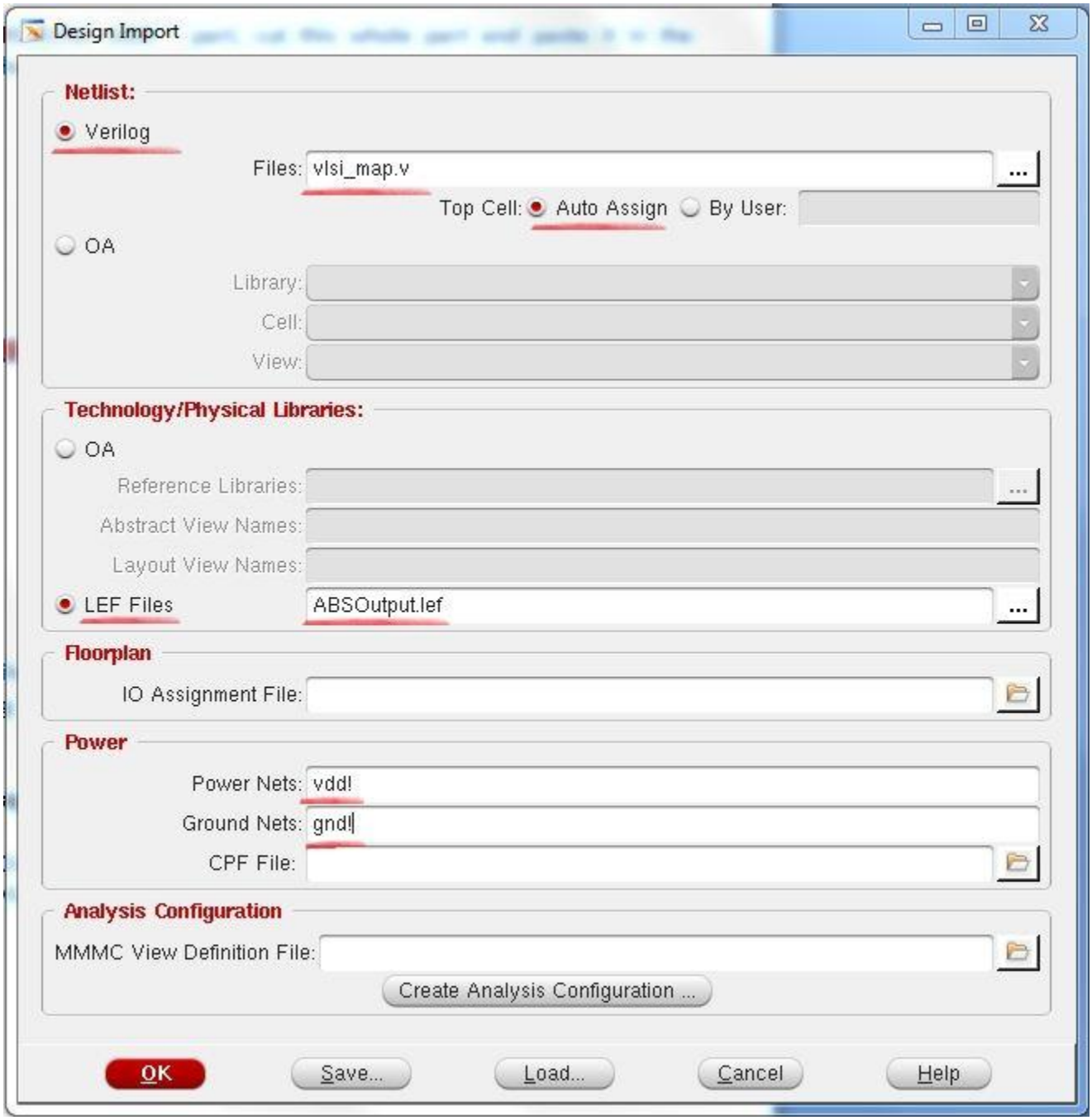
Now you can import both your Cadence LEF file (which contains information that Encounter needs regarding your cell library) and your synthesized Verilog netlist (which contains an electrical description of the circuit) into the Silicon Ensemble environment.

In the Encounter GUI, Click ***File -> Import Design***, the Design Import window will pop up.

In the Netlist part, check ***Verilog*** option, select your verilog file for, and check ***Auto Assign*** option for Top Cell.

In the Technology/Physical Libraries part, check ***LEF Files*** option, select your ABSOuput.lef file.

In the Power part, put **vdd!** and **gnd!** for Power/Ground Nets . and then Click **OK**

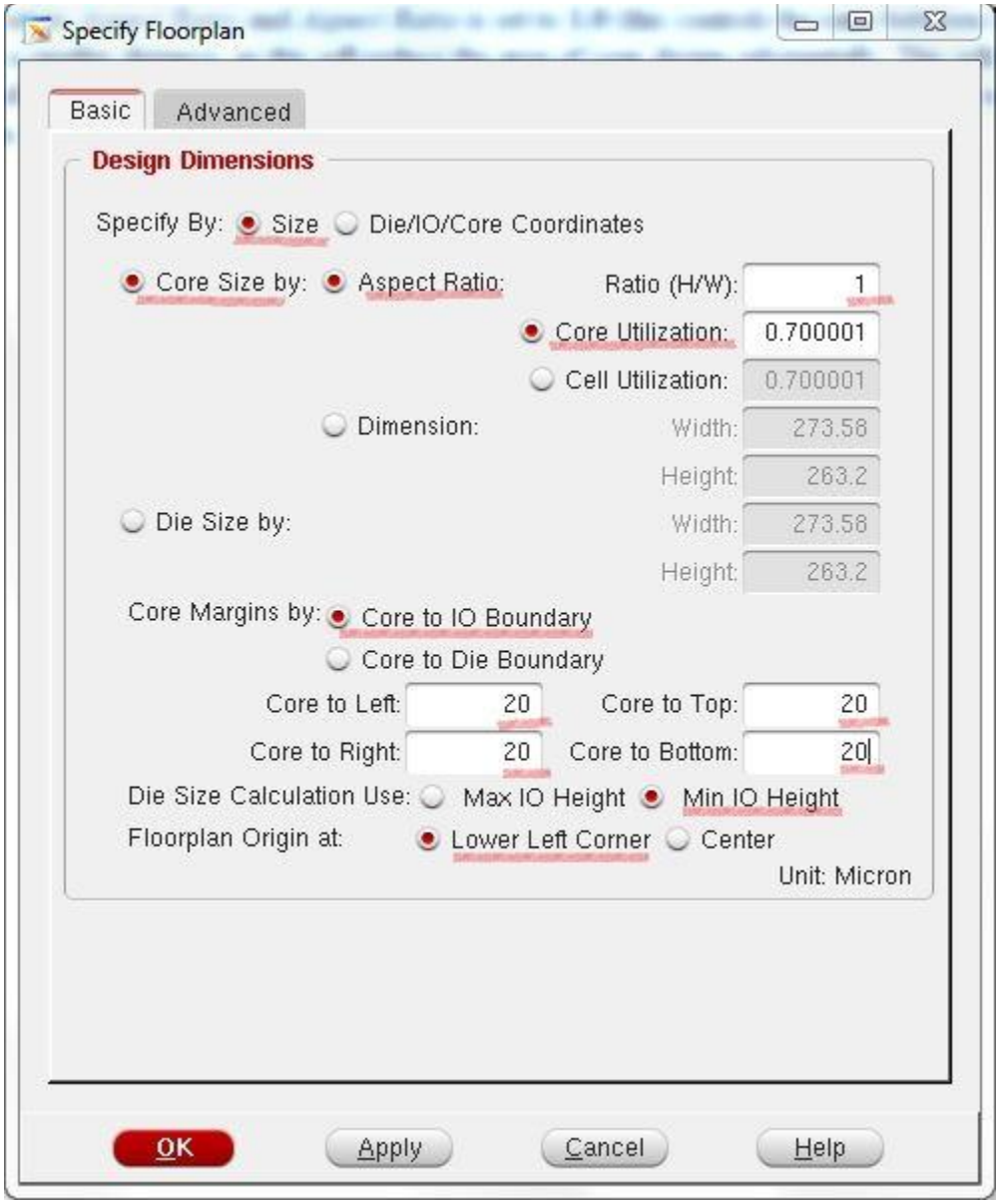


1.3. Floorplanning

Floorplanning is the stage of the design where the area of the design is defined. Two regions exist within the design: the core, which contains the cells arranged in rows, and the I/O area, which contains power and ground rings and I/O pins.

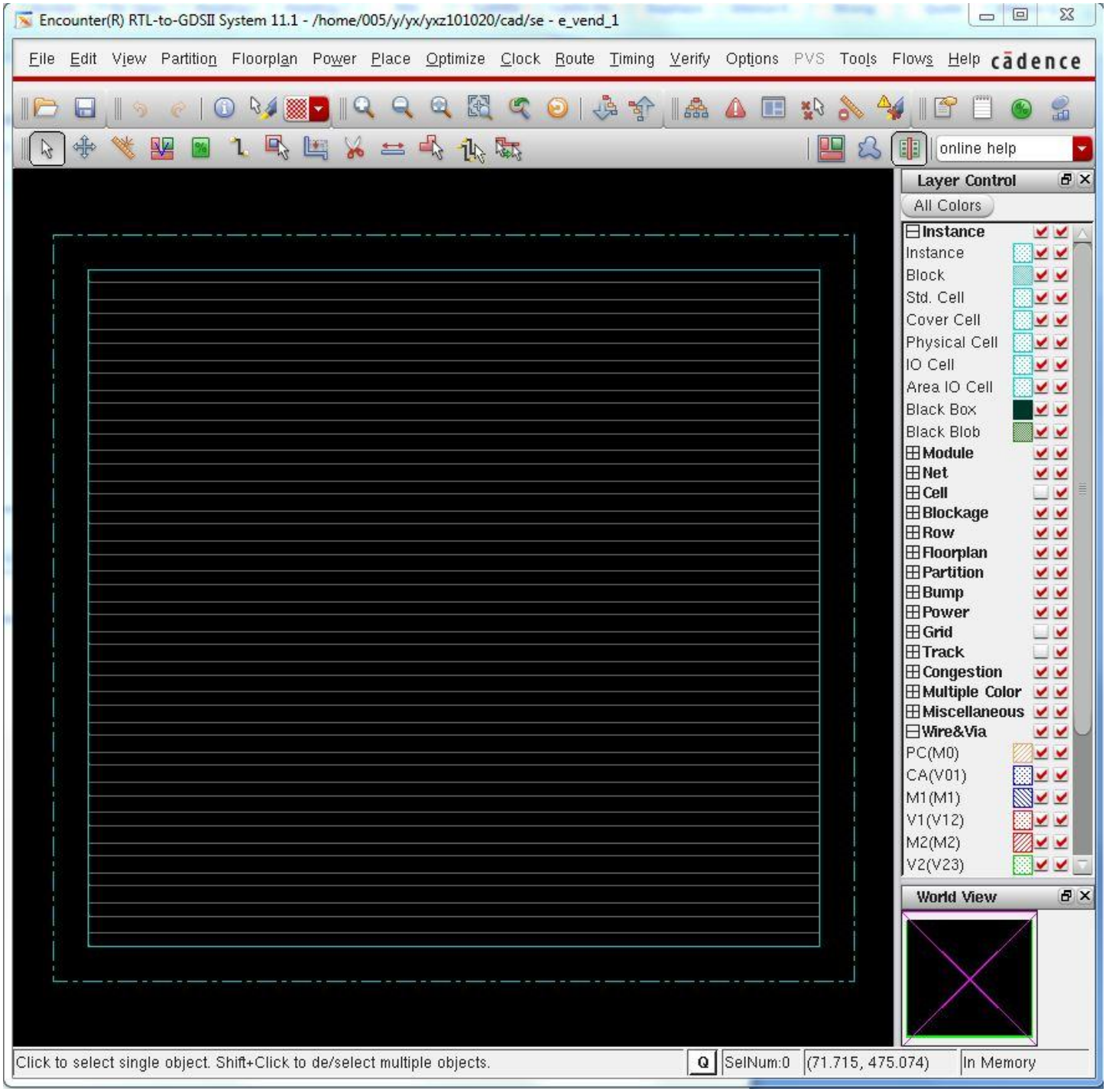
Select **Floorplan -> Specify Floorplan** to open the *floorplanning* window. In the ***Basic*** tab. Make sure ***Core Size*** is set to ***Aspect Ratio*** and *Aspect Ratio* is set to **1.0** (this controls the ratio between height and width of the design). Set the ***Core to IO Boundary*** *Distance* to 20 microns all directions. (If the design has room after routing, you may want to go back and re-run floorplanning with a smaller distance, as this will reduce the area of your design substantially. This will mean however that you will need to re-run all steps after floorplanning as well).

Do **NOT** click OK now.



Then, go to the ***Advanced*** tab. Change your ***Double-back rows*** as picture. The ***Row Spacing*** setting controls the distance between rows of cells (space where routing can occur on the metal1 layer) and should be initially set to several microns. It may be possible to later reduce this distance to shrink the design area, but it is necessary to view the routed design to determine this first. Make sure the ***Bottom IO Pad Orientation*** as picture. Click **OK** to finalize the floorplan.

And after floorplanning, your Encounter will looks like this. (You can use key “F” to zoom fit)

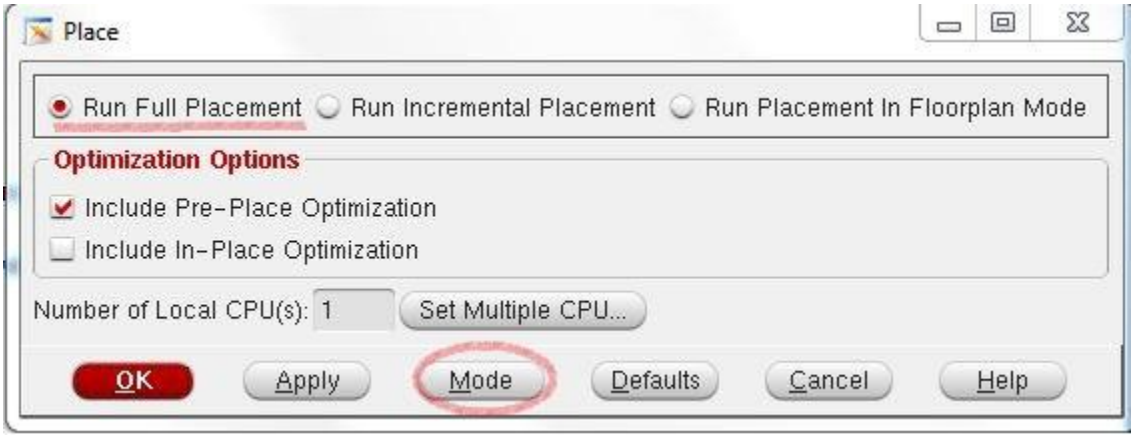


1.4. Placement

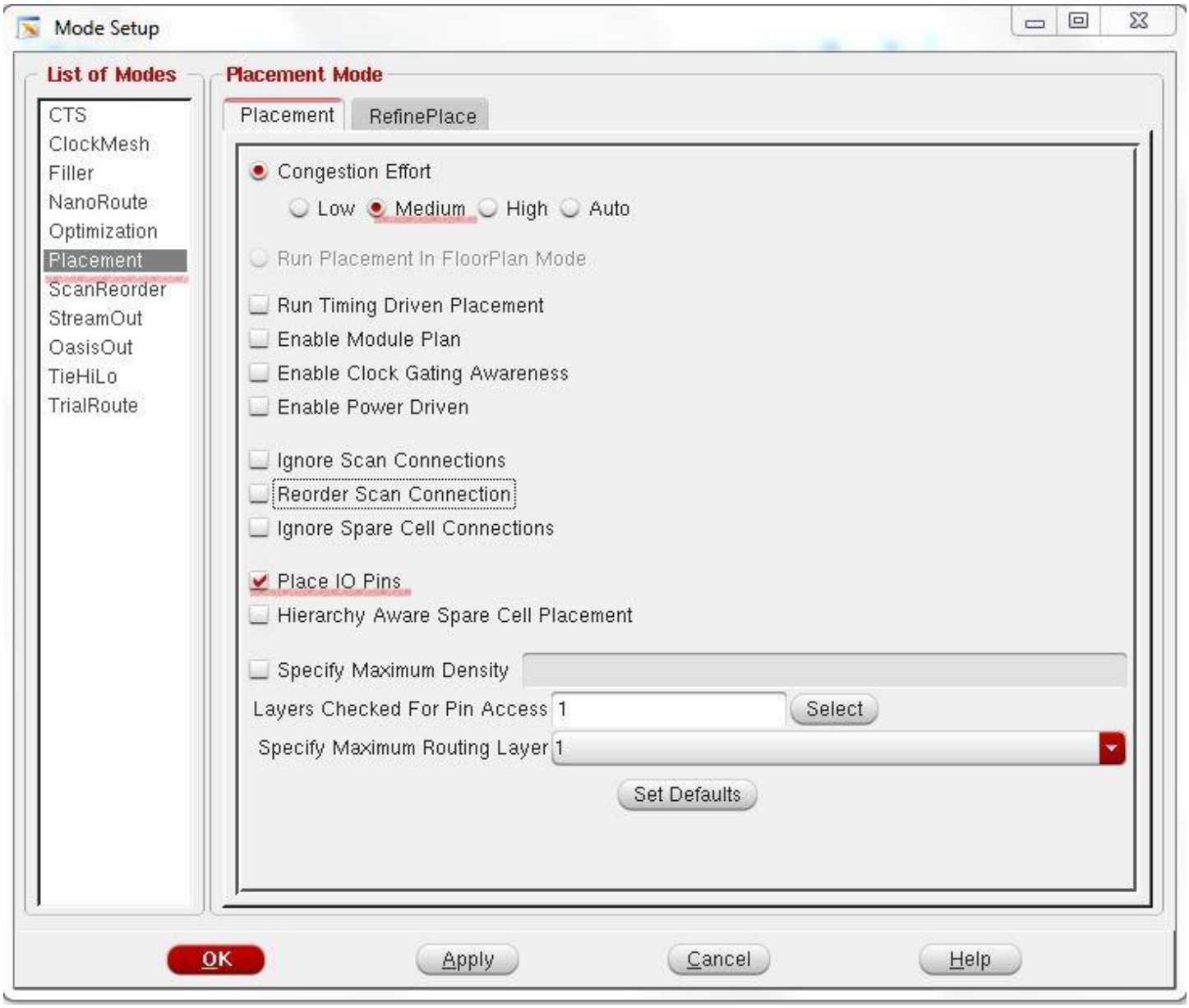
1.4.1. Place Standard Cells

Once the floorplanning is complete, IO ports and cells need to be placed in the alotted rows.

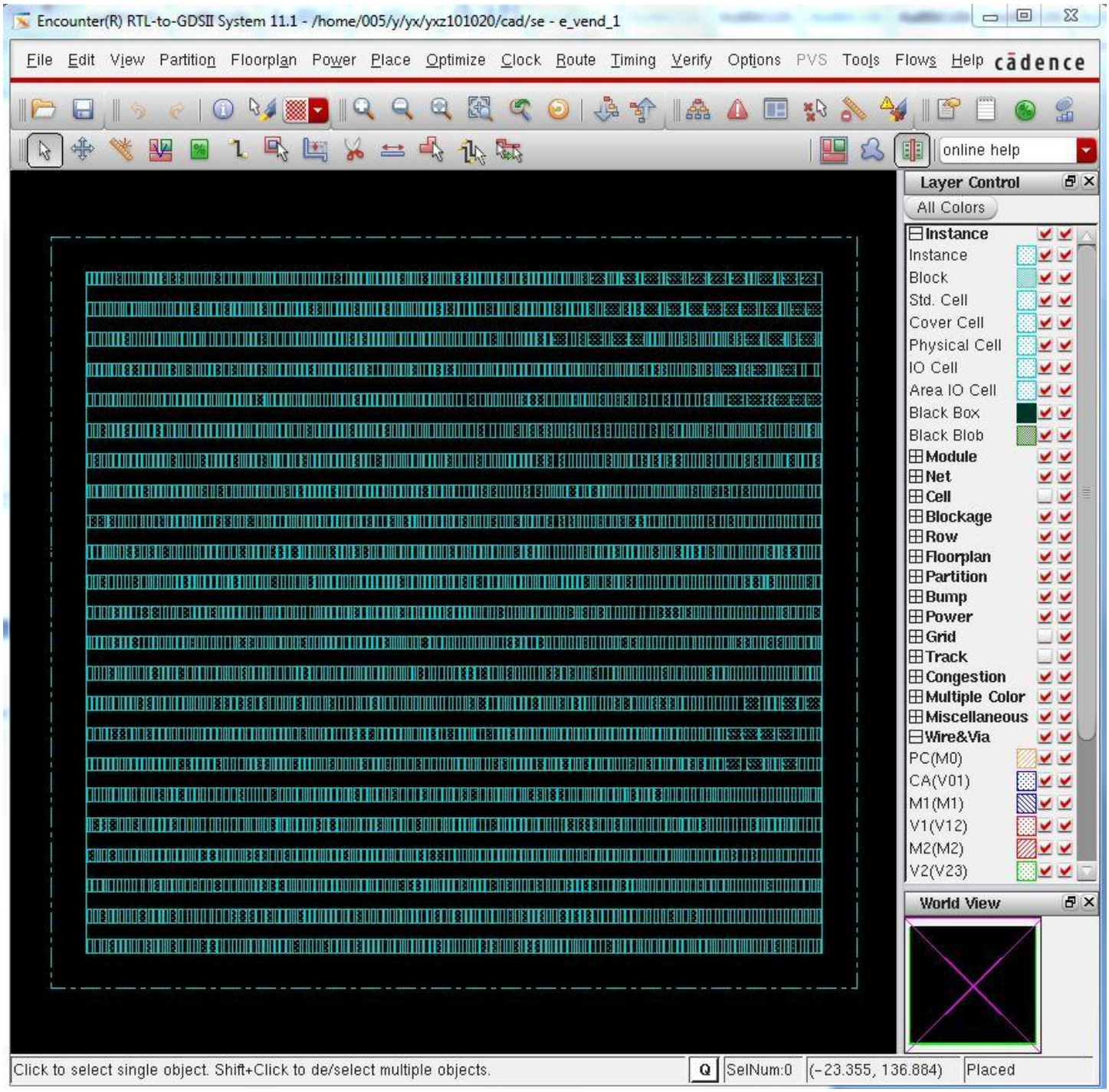
Select ***Place -> Place Standard Cells***. In the Place window, make sure ***Run Full Placement*** is selected. Do **NOT** click OK now. Click ***Mode.***



And the Mode Setup window will show like following. Select ***Placement*** in the List of Modes, use ***Medium*** for Congestion Effort. Since we do not do the timing analysis and power analysis, we do not need all these options. Just make sure ***Place IO Pins*** is selected. Click ***OK***.

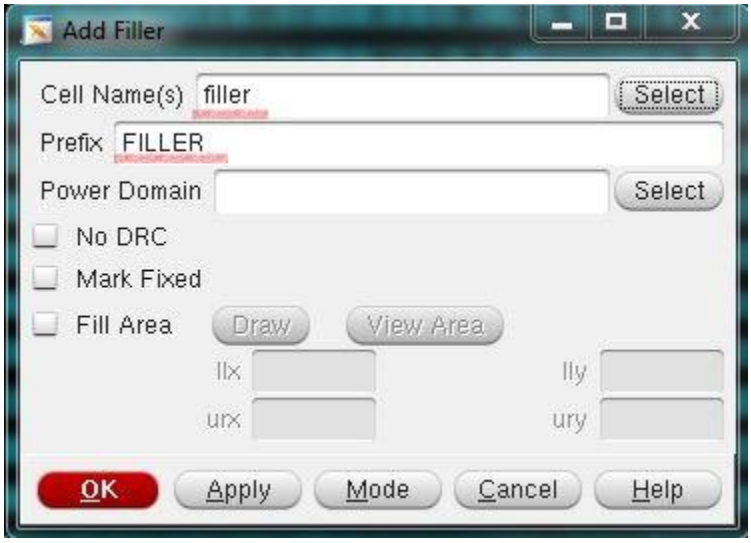

Mode Setup
List of Modes
CTS
ClockMesh
Filler
NanoRoute
Optimization
Placement
ScanReorder
StreamOut
OasisOut
TieHiLo
TrialRoute
Placement Mode
Placement
RefinePlace
Congestion Effort
Low
Medium
High
Auto
Run Placement In FloorPlan Mode
Run Timing Driven Placement
Enable Module Plan
Enable Clock Gating Awareness
Enable Power Driven
Ignore Scan Connections
Reorder Scan Connection
Ignore Spare Cell Connections
Place IO Pins
Hierarchy Aware Spare Cell Placement
Specify Maximum Density
Layers Checked For Pin Access
1
Select
Specify Maximum Routing Layer
1
Set Defaults
OK
Apply
Cancel
Help

Now, you can click ***OK*** in your Place window. The Encounter will show like following. (This process may take a while)

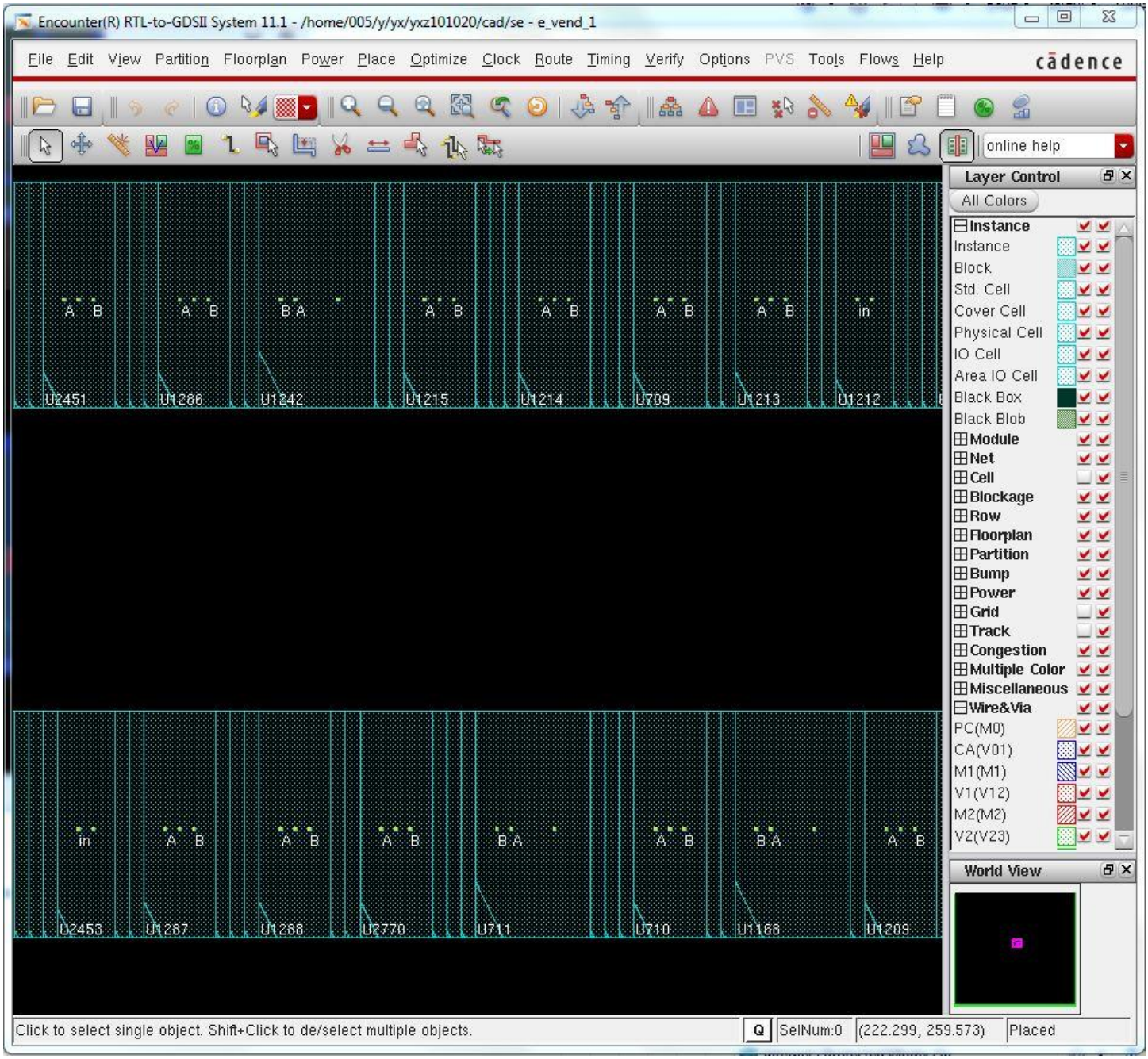


1.4.2. Place Fillers between cells

If you have filler in your library, place the filler right now. Select ***Place -> Physical Cells -> Add Filler.*** The Add Filler window will pop up. Select your filler's name in the ***Cell Name(s)***. If you used different name for filler, put the right name there. Click ***OK*.**



Then try to change your zoom or move the view, you will see the fillers.



As you see there is no space between cells.

1.5. Power/Ground

1.5.1. Tie-hi and Tie-lo nets

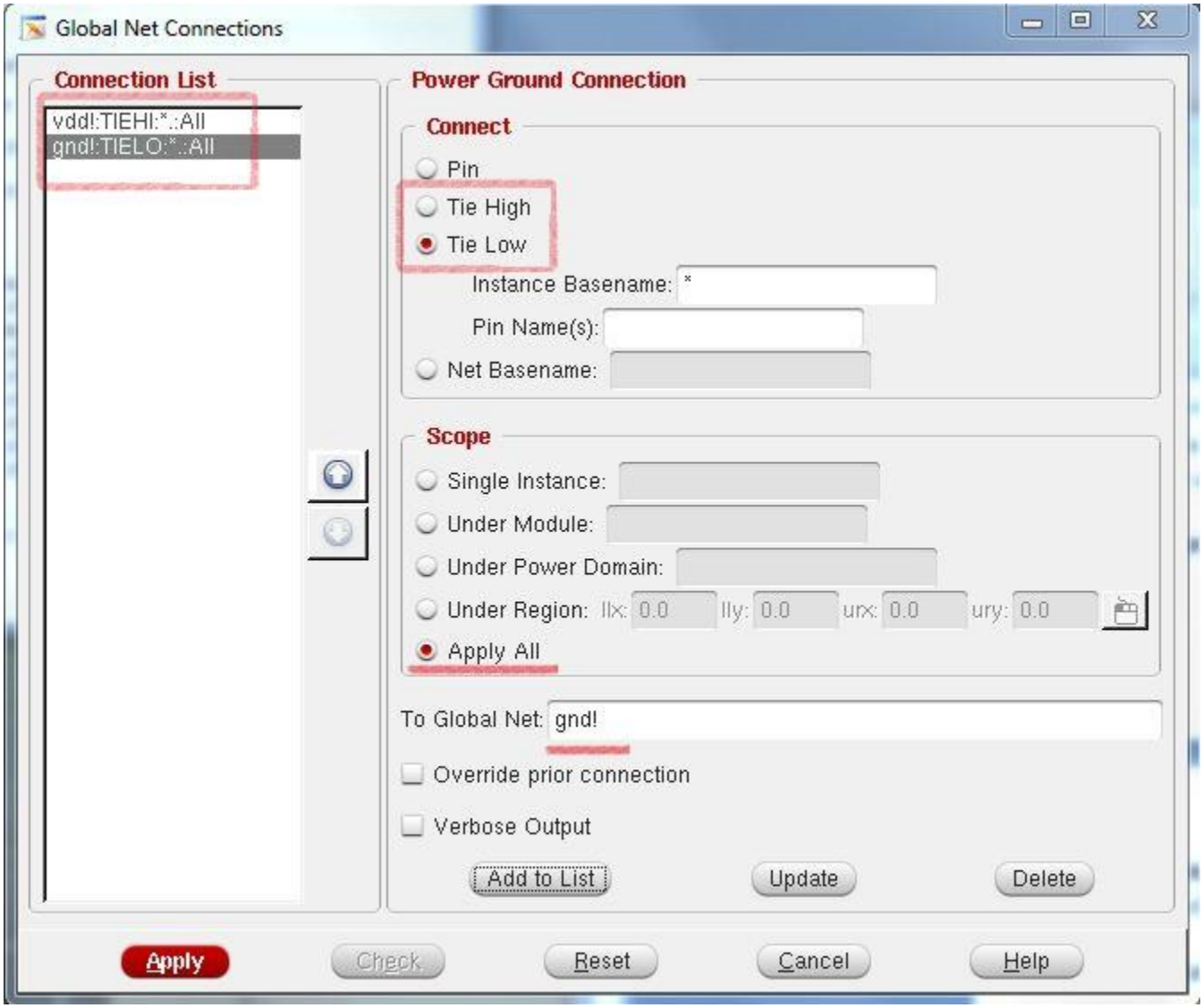
Since there are no Tie-hi/Tie-lo cells in the library, you need to connect between the Tie-hi and Tie-lo nets to the appropriate power and ground net. These are keywords in the Verilog netlist, such as 1'b0, 1'b1.

Select ***Power -> Connect Global Nets,*** the *Global Net Connections* window will pop up.

Select ***Tie High*** in the Connect part, Select ***Apply All*** in Scope part, put ***vdd!*** In To Global Nets, click ***Add to List***. Then you will see “vdd!:TIEHI:*.:ALL” in the left Connection List.

Select ***Tie Low*** in the Connect part, Select ***Apply All*** in Scope part, put ***gnd!*** In To Global Nets, click ***Add to List***.

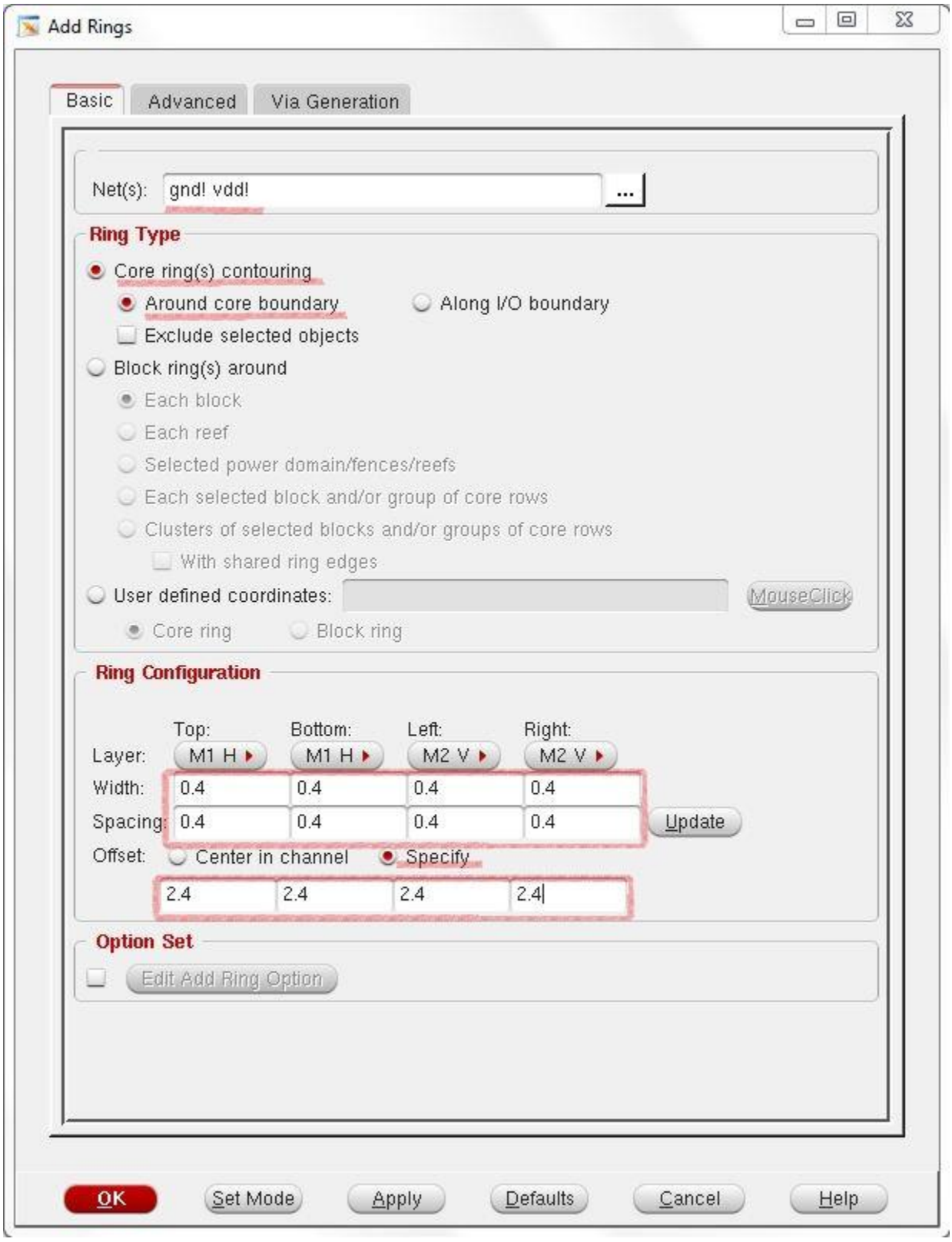
The window should looks like this:



Click ***Apply,*** then Click ***Check,*** there should be no error in your terminal. Close the window.

1.5.2. Add Rings and Stripes

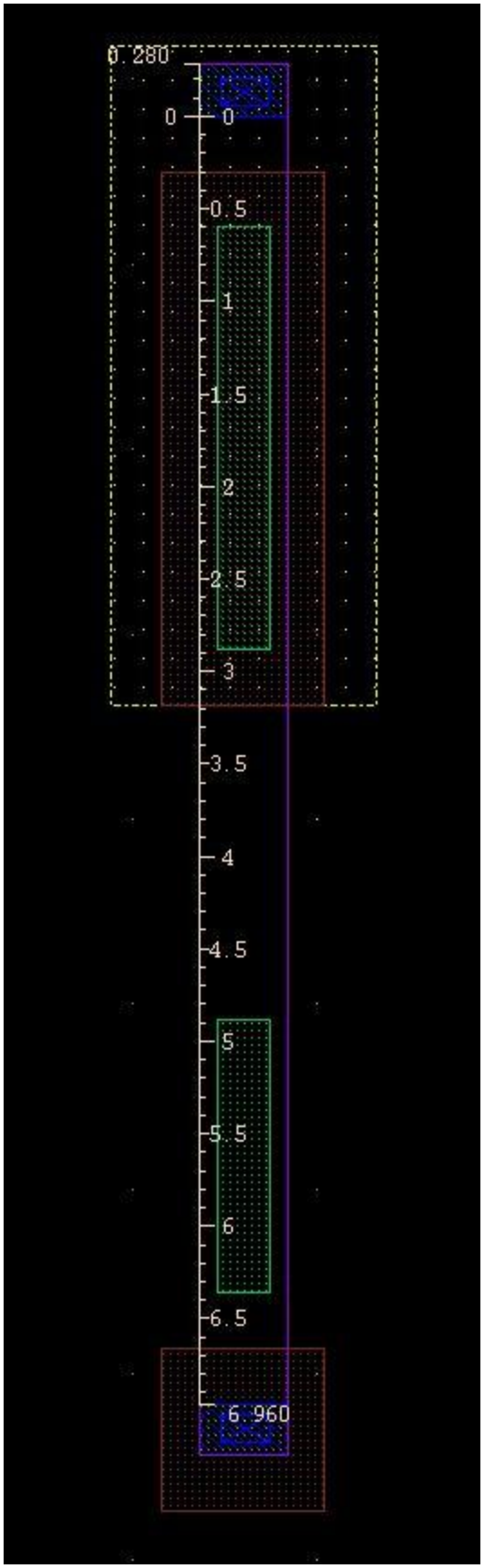
Select ***Power -> Power Planning -> Add Ring***, the *Add Rings* window will pop up.



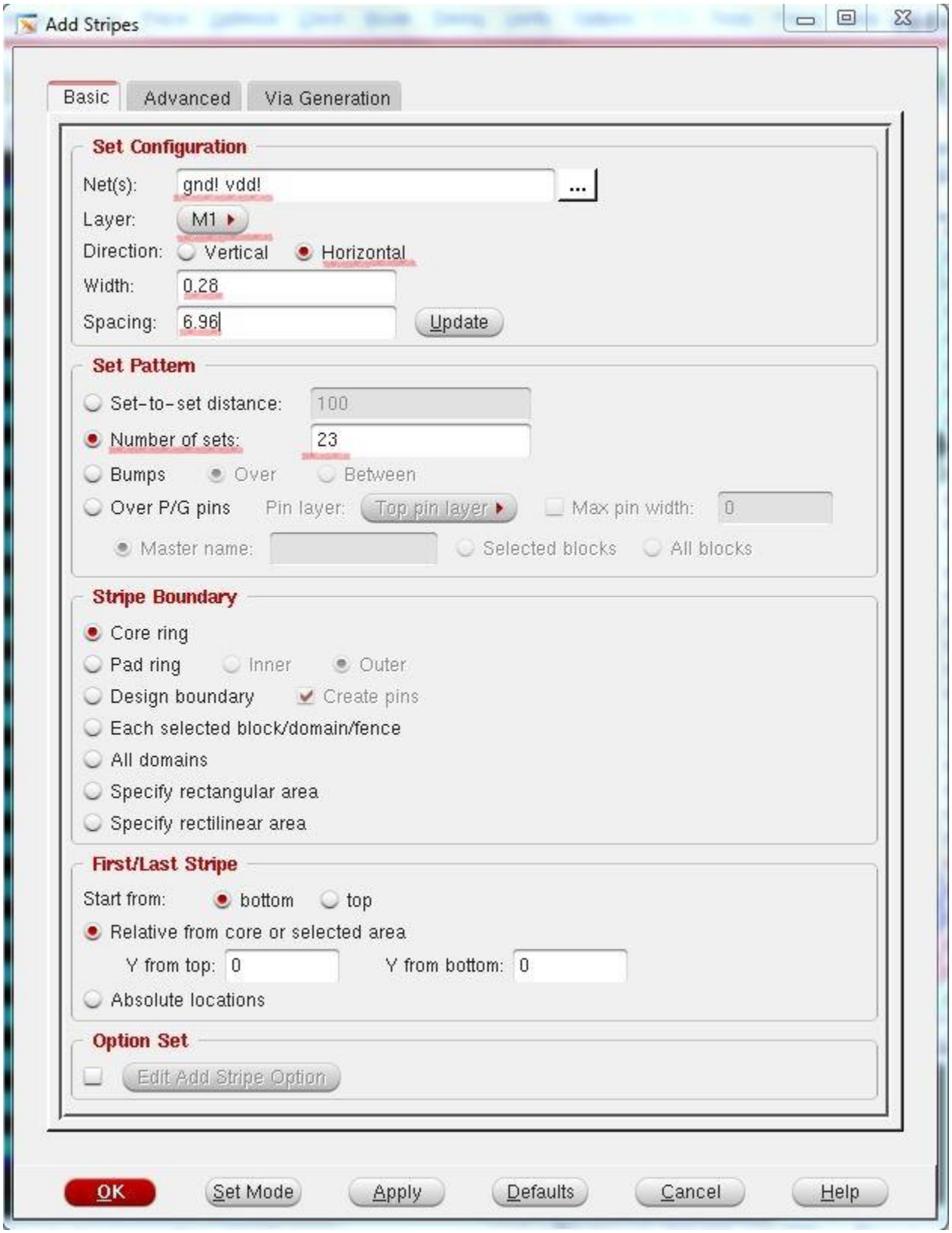
Make sure your power and ground net names (**vdd! gnd!**) appear in the *Nets* window (you can select the net names with the button beside), set the *Width* to **0.4** microns, the *Spacing* to **0.4** microns, and make sure the *Offset* parameters are set to **Specify**. For offset, PUT some number with on grid. If your grid is 0.48, put 2.4 or 4.8 or some number (n* grid). Click **OK** to apply the rings and close the *Plan Power* window.

Selcet ***Power -> Power Planning -> Add Stripe,*** the *Add Stripe* window will show.

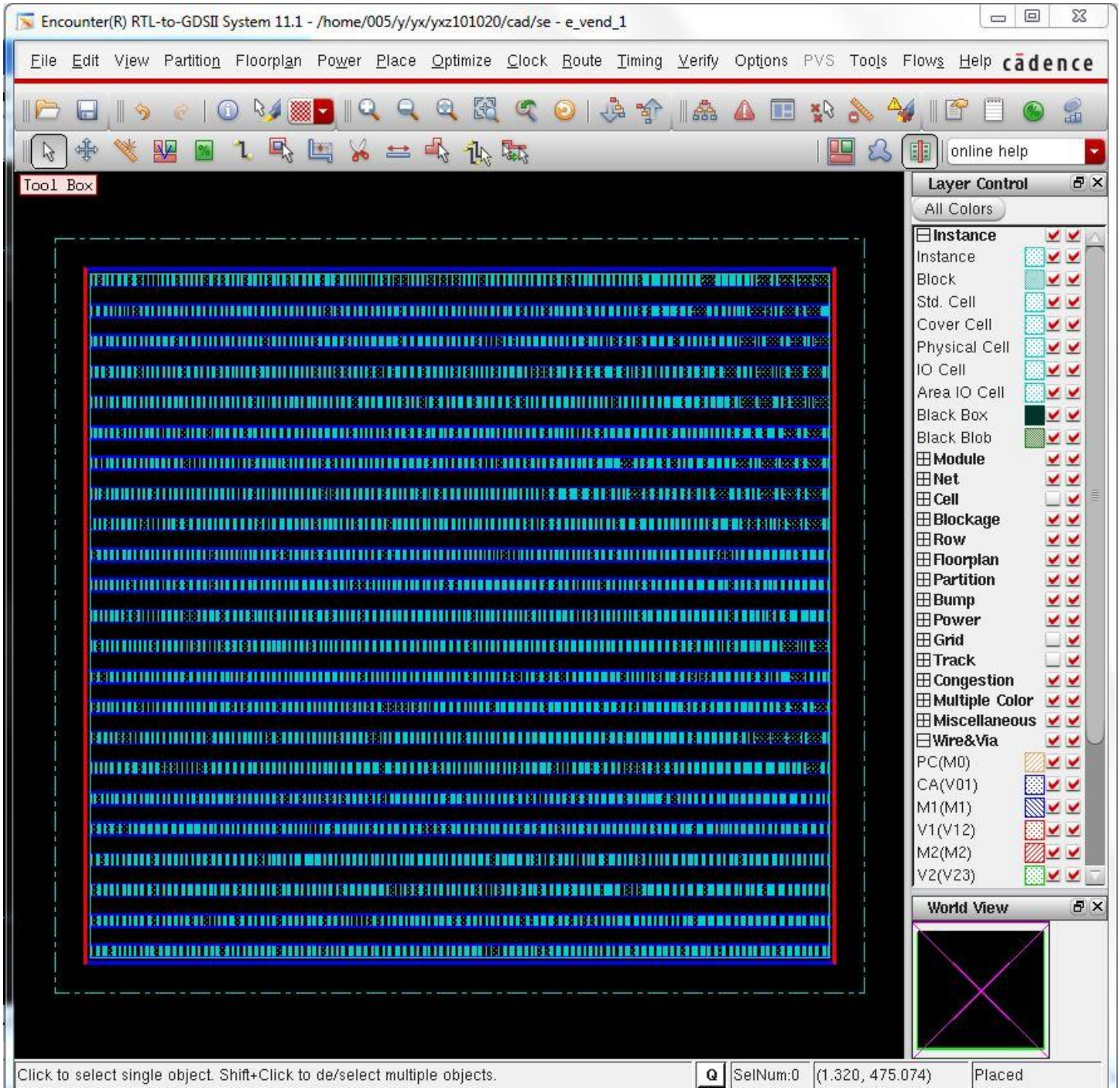
Make sure (**vdd! gnd!**) appear in the *Nets* window again. Change Layer to **M1**. Change Direction to **Horizontal**. Width is supposed to be equal to your **vdd and gnd M1 width** in your cells. Spacing is equal to the space between vdd! and gnd! Trail.

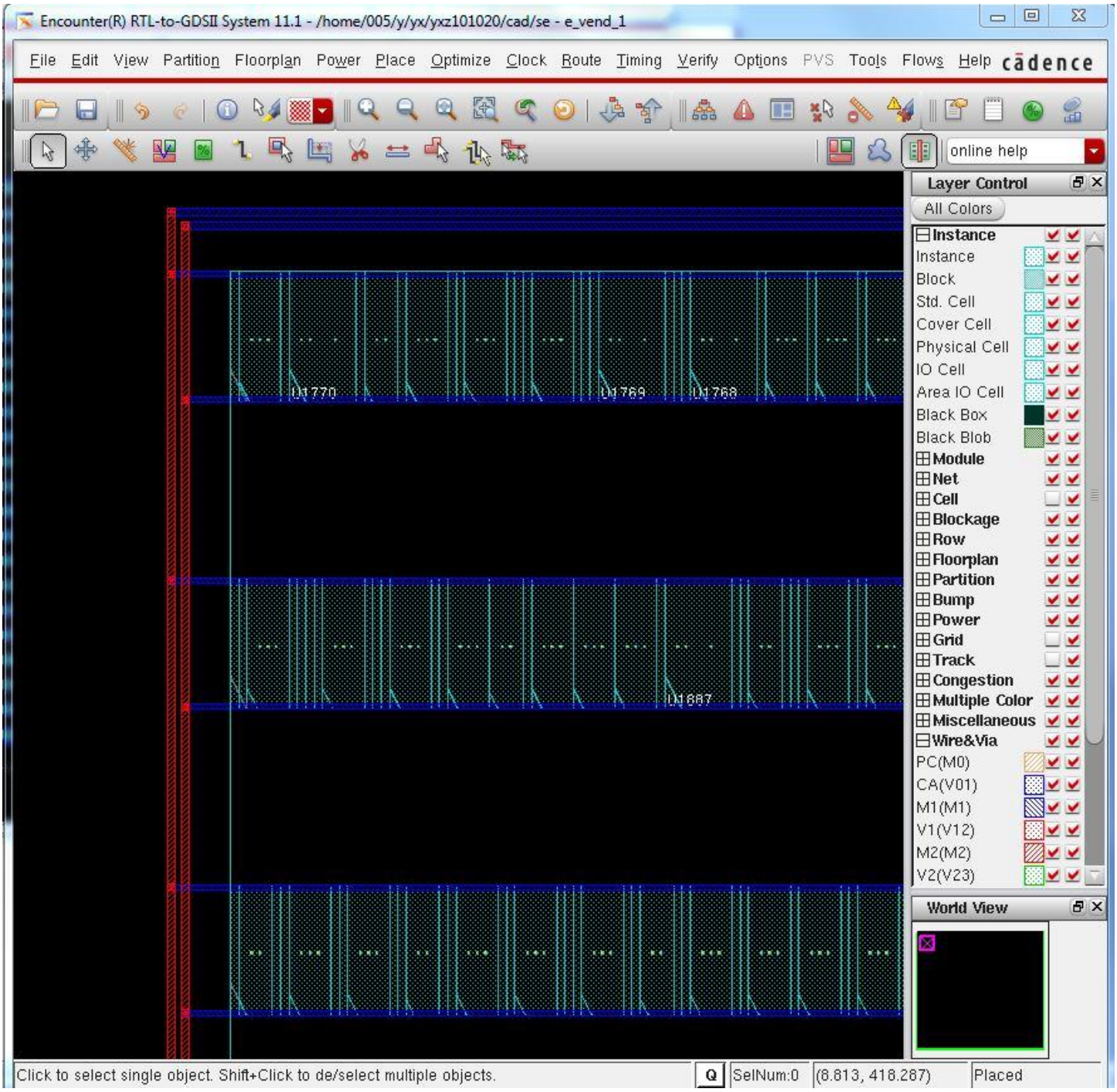

0.280
0
0
0.5
1
1.5
2
2.5
3
3.5
4
4.5
5
5.5
6
6.5
6.960

Set Pattern needs to be checked at **Number of sets** and put the **number of rows you have in your design,** then Click ***OK***.



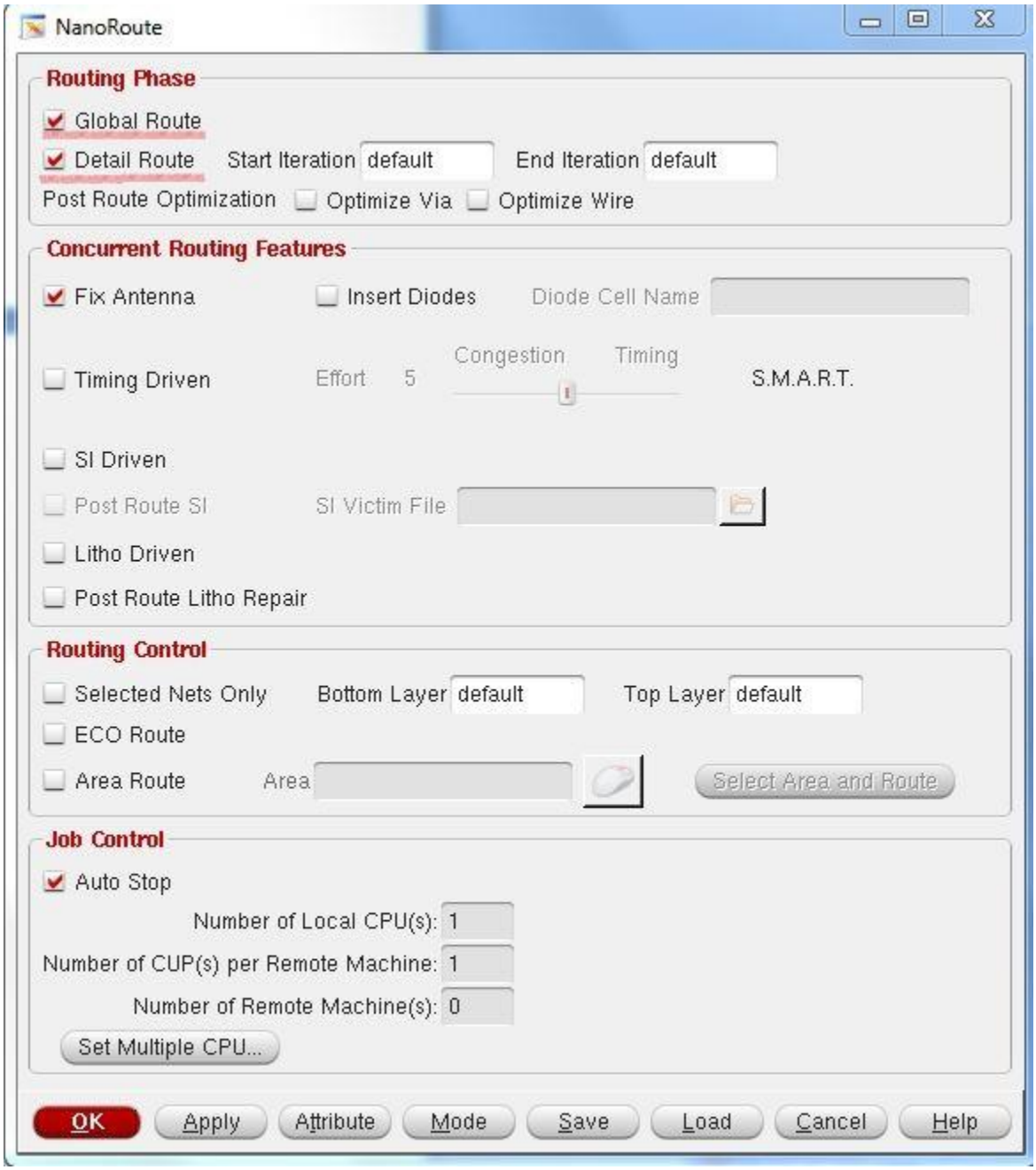
Now you encounter should looks like this:



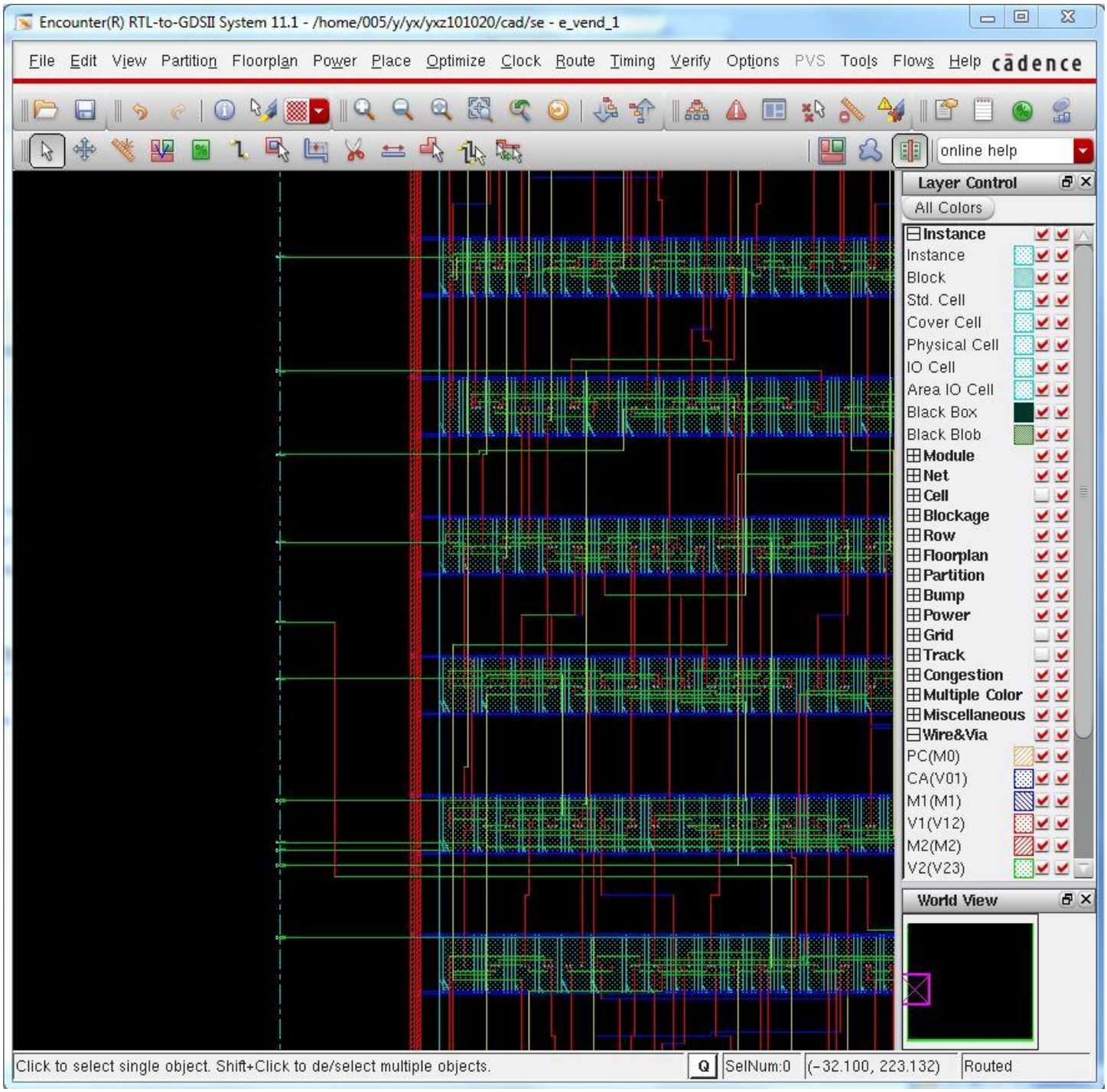

Encounter(R) RTL-to-GDSII System 11.1 - /home/005/y/yx/yxz101020/cad/se - e_vend_1
File Edit View Partition Floorplan Power Place Optimize Clock Route Timing Verify Options PVS Tools Flows Help cadence
online help
Layer Control
All Colors
Instance
Instance
Block
Std. Cell
Cover Cell
Physical Cell
IO Cell
Area IO Cell
Black Box
Black Blob
Module
Net
Cell
Blockage
Row
Floorplan
Partition
Bump
Power
Grid
Track
Congestion
Multiple Color
Miscellaneous
Wire&Via
PC(M0)
CA(V01)
M1(M1)
V1(V12)
M2(M2)
V2(V23)
World View
Click to select single object. Shift+Click to de/select multiple objects.
Q
SelNum:0
(8.813, 418.287)
Placed

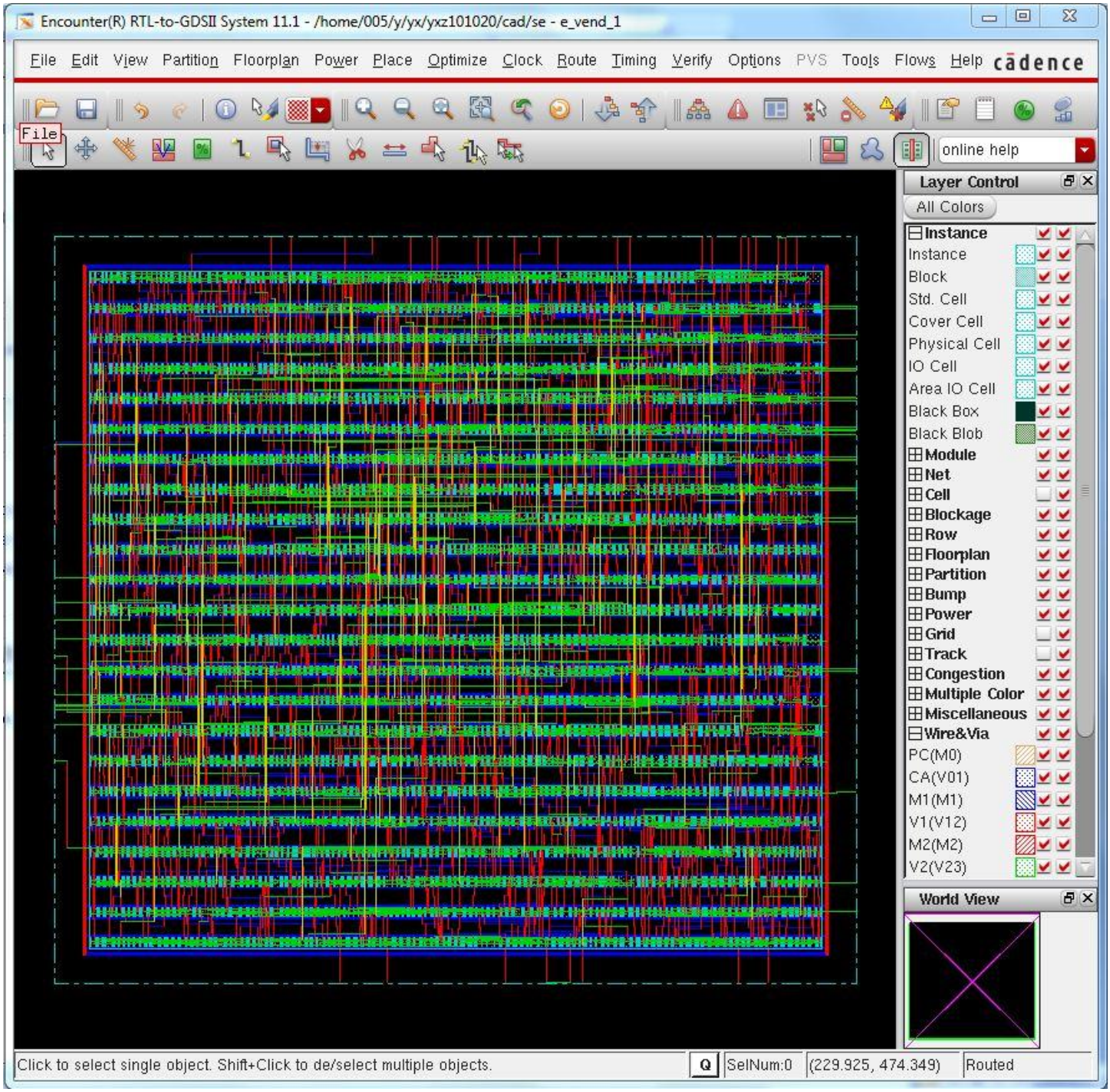
1.6. Routing

Now we will connect the cells together and to the I/O pins by routing the circuit nets. Select ***Route -> NanoRoute -> Route***, make sure ***Global Route*** *and* ***Detail Route*** is selected, and click **OK**.



This process may take a while. When it is complete you should have wires connecting your pins together:


Encounter(R) RTL-to-GDSII System 11.1 - /home/005/y/yx/yxz101020/cad/se - e_vend_1
File Edit View Partition Floorplan Power Place Optimize Clock Route Timing Verify Options PVS Tools Flows Help cadence
online help
Layer Control
All Colors
Instance
Instance
Block
Std. Cell
Cover Cell
Physical Cell
IO Cell
Area IO Cell
Black Box
Black Blob
Module
Net
Cell
Blockage
Row
Floorplan
Partition
Bump
Power
Grid
Track
Congestion
Multiple Color
Miscellaneous
Wire&Via
PC(M0)
CA(V01)
M1(M1)
V1(V12)
M2(M2)
V2(V23)
World View
Click to select single object. Shift+Click to de/select multiple objects.
Q
SelNum:0
(-32.100, 223.132)
Routed


Encounter(R) RTL-to-GDSII System 11.1 - /home/005/y/yx/yxz101020/cad/se - e_vend_1
File Edit View Partition Floorplan Power Place Optimize Clock Route Timing Verify Options PVS Tools Flows Help cadence
File
online help
Layer Control
All Colors
Instance
Instance
Block
Std. Cell
Cover Cell
Physical Cell
IO Cell
Area IO Cell
Black Box
Black Blob
Module
Net
Cell
Blockage
Row
Floorplan
Partition
Bump
Power
Grid
Track
Congestion
Multiple Color
Miscellaneous
Wire&Via
PC(M0)
CA(V01)
M1(M1)
V1(V12)
M2(M2)
V2(V23)
World View
Click to select single object. Shift+Click to de/select multiple objects.
Q
SelNum:0
(229.925, 474.349)
Routed

1.7. Export files

Finally, we need to export a DEF format file for Cadence. This file contains physical cell placement and automatic routing information as well as electrical net information. Select ***File -> Save -> DEF*** from the main menu, change the ***Output DEF Version*** to **5.5**, fill in an appropriate file name, and then click ***OK***.

1.8. Exit Encounter

Select ***File -> Exit.***

1.9. Run ADDVIAS code

Your DEF file missing some Vias for your layout, so you have to run this script to add vias in your def file

```
addvias.pl YOUR_DEF_FILE.def YOUR_DEF_FILE_VIAS.def
```

If it appers "command not found", try source it again

```
. /proj/cad/startup/profile.ic-5
```

```
addvias.pl YOUR_DEF_FILE.def YOUR_DEF_FILE_VIAS.def
```