L-Edit v13

Schematic Driven Layout (SDL) Tutorial

Tanner EDA Division
Tanner Research, Inc.
825 South Myrtle Avenue
Monrovia, CA 91016-3424
Tel: (626) 471-9700

TABLE OF CONTENTS

RINGVCO TUTORIAL

1. Introduction

L-Edit is a layout editing tool for use by circuit design engineers. In this tutorial you will learn about:

- Setting up cell blocks for Schematic Driven Layout (SDL)
- Importing a netlist using SDL
- Using flylines to place devices
- Tagging geometry on manually placed routes
- Setting up the automatic router
- Automatic routing
- Ripping up nets
- Importing an Engineering Change Order (ECO)

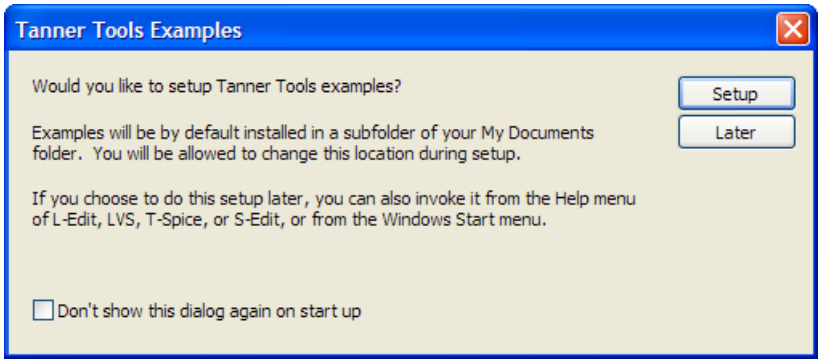
1.1. Typographic Conventions

This section describes typographic conventions that are used in this document.

Bold	Indicates elements that are a part of L-Edit, such as toolbars, menus, and buttons.
`Courier`	Indicates elements that are part of the tutorial design, such as cell names and instance names.

2. Installing the tutorial

The first time you run L-Edit after installation, the following dialog appears, asking if you wish to Setup the examples and tutorial.



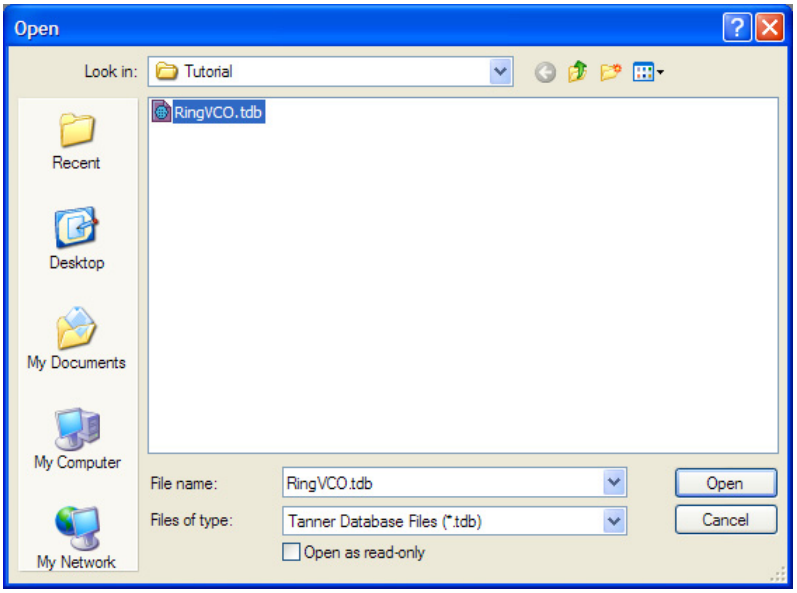
If you installed the tutorial when you first installed L-Edit, then the tutorial can be found in the location that you specified for the tutorial files. The default location for the tutorial is **My Documents\Tanner EDA\Tanner Tools v13.0\L-Edit and LVS\Tutorial**.

Tutorial and example files may be installed at any time by invoking **Help > Setup Examples…** from L-Edit, or by invoking **Setup Examples and Tutorial** from the Windows start menu for Tanner Tools.

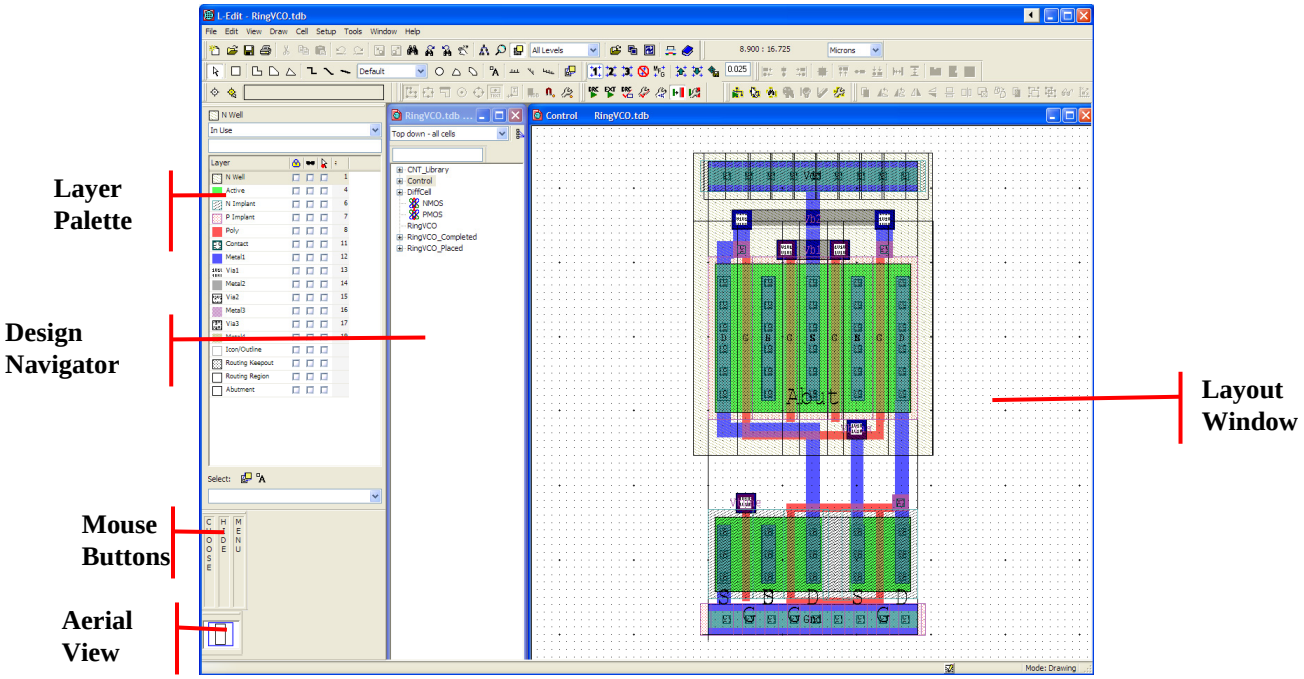
After running the tutorial, modified tutorial files can be replaced with a fresh copy of the tutorial files by invoking **Help > Setup Examples…**, and choose **Repair**, when asked to select between **Modify**, **Repair**, and **Remove**.

3. Getting started with L-Edit Schematic Driven Layout (SDL)

1. First, we need to open the design database for the tutorial. Invoke **File > Open…** and browse to the **RingVCO.tdb** file in the tutorial folder. The default location for the tutorial is **My Documents\Tanner EDA\Tanner Tools v13.0\L-Edit and LVS\Tutorial**, although you may have installed it in another location. Press Open in the **Open** dialog.



2. The RingVCO tutorial will load into L-Edit, and look like this. On the left side of the application window, we have the **Layer Palette**. In the center left is the **Design Navigator**, which provides a tree view of the cells and the hierarchy of the design and in the center right is the layout for a portion of our design.



3. This project consists of a design called `RingVCO`, which contains T-Cells named `NMOS` and `PMOS`, Contact Cells named `CNT_Active`, `CNT_M1M2`, `CNT_M2M3`, `CNT_M3M4`, `CNTPolyM1`, `CNT_Substrate`, and `CNT_Well`. Additionally, the design contains cell blocks named `Control` and `DiffCell`.

4. Setting up cell blocks for SDL

4. Let's examine the `Control` cell to view the requirements for a cell block to be used with SDL.

   - Double-click the `Control` cell in the **Design Navigator** to open the cell layout if it is not already open.
   - Each device that is listed in your netlist should be a cell with pre-defined layout or as a T-Cell prior to running SDL. The device name in the netlist must match the cell name in your TDB file for that device to be instantiated during SDL from your netlist. Cells that do not exist or do not match by name may be optionally generated upon importing of the netlist for later layout.

5. First, let's look at the ports that are used in the block.

   - Hide everything in the layout but ports by CTRL+middle-mouse clicking the **Switch to drawing ports** icon  or by right-clicking the **Switch to drawing ports** icon and selecting **Hide All**.
   - There are seven ports in the layout. Ports `Vdd` and `Gnd` are placed on the **Metal1** layer, Port `Abut` is placed on the **Icon/Outline** layer, and Ports `Vb1`, `Vb2`, `Vbias`, and `Vtune` are placed on the **Metal2** layer.
   - Each I/O port must be placed on a routing layer for the SDL router to connect to it and should be placed such that it is near the edge of the cell or can be easily accessed with routes. If they are placed too close to one another, the SDL router may not be able to connect to all the I/O ports. Ideally, the ports should be placed on a specific routing grid so that the SDL router can directly access the ports without having to branch off-grid.
   - The dimensions of the ports will define the width of the routes that will connect to that port. In this example, all the ports are box ports. In this case, the smaller dimension of the port will determine the route width. If the smaller dimension is less than the minimum specified route width, the minimum route width will be used. For example, assuming the minimum route width specified for **Metal1** and **Metal2** is 0.35u, `Vdd` and `Gnd` will be connected to using a 1.00u wire, as their smallest dimension is 1.00u. Likewise, `Vb1`, `Vb2`, `Vbias`, and `Vtune` will be connected to using a 0.65u wire. If two ports with different dimension are to be connected together, the smaller of the two dimensions will be used.
   - Unhide the objects previously hidden by CTRL+middle-mouse clicking the **Switch to drawing ports** icon  or by right-clicking the **Switch to drawing ports** icon and selecting **Show All**.

6. Next, let's look at the keepout region defined for the `Control` cell.

   - Hide all layers with the exception of the keepout layers by selecting both **Keepout – Metal2** and **Keepout – Metal3**, right-clicking, and invoking **Hide** > **Hide all but selected**.

- One keepout layer for each routing layer may be defined in the layout. The keepout layer defines the region(s) where the routes may not go. In this example, the routes may not pass over the active region of the transistors.
- Unhide the layers previously hidden by right-clicking any layer and invoking **Show > Show all**.

7. Now, we need to make sure that there are contact cells defined

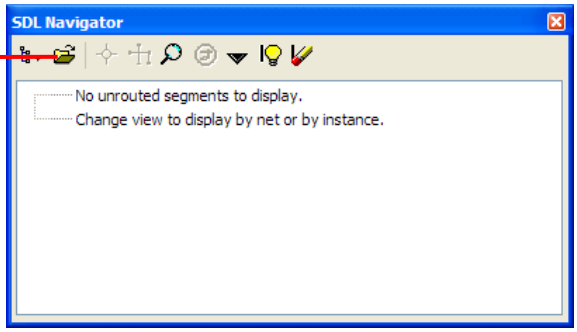
   - Contact cells `CNT_M1M2`, `CNT_M2M3`, `CNT_M3M4`, and `CNTPolyM1` are pre-defined in the layout and will be assigned later in the router setup in order to define how connections between layers are to be made.

5. Importing a netlist using SDL

8. First, let's look at the netlist we will be importing into the layout.

   - Open cell `RingVCO` by double-clicking the cell name in the **Design Navigator**.
   - Invoke **Tools > SDL Navigator > Show SDL Navigator** or right-click on the toolbar and select **SDL Navigator**.
   - Select the **Load Netlist** icon on the **SDL Navigator** toolbar to open the **Import Netlist** dialog.

Select to open Import Netlist dialog

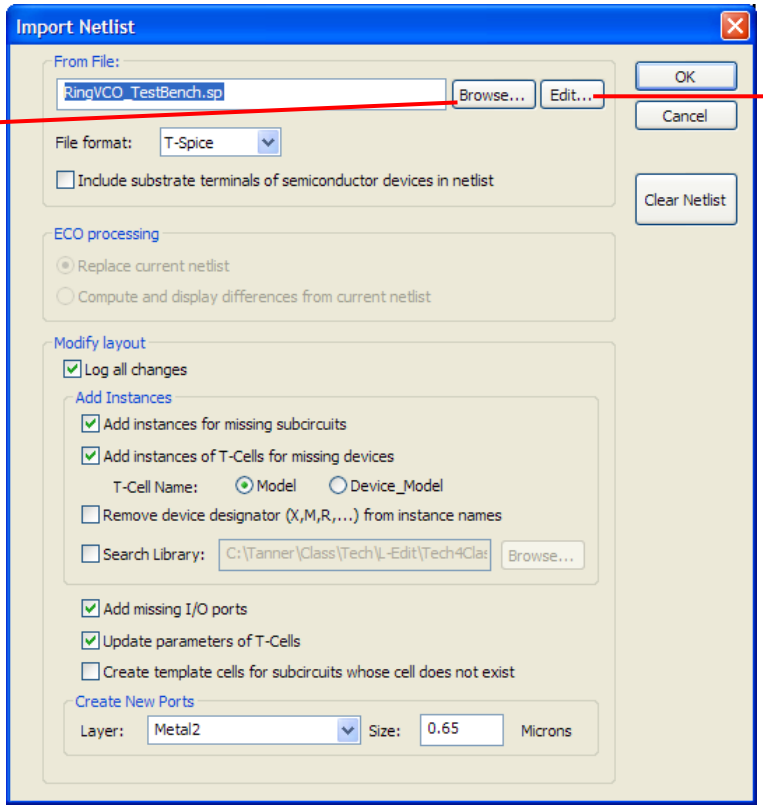


   - Select **Browse…** to locate and load in the netlist `RingVCO_Testbench.sp` into the **From File:** field. The file is located in the following location:

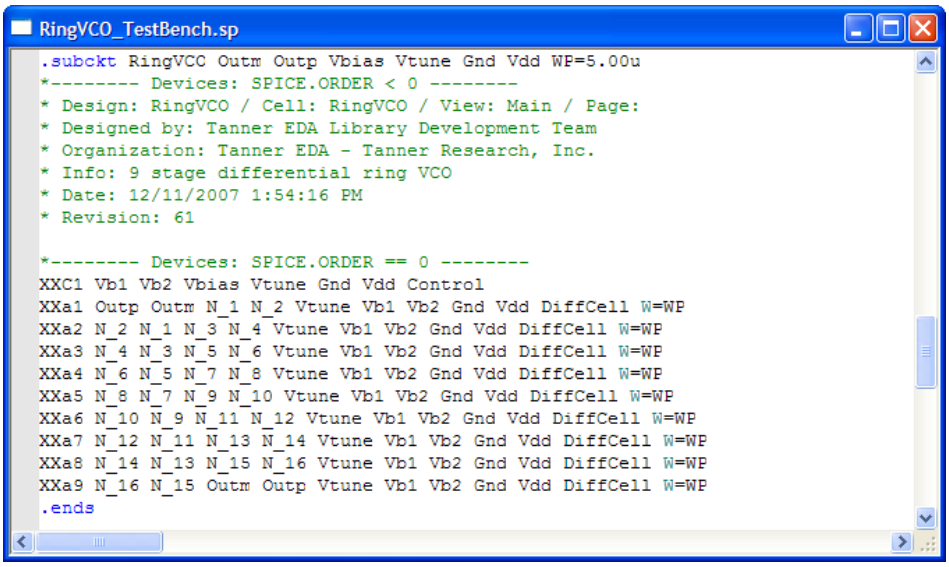
     My Documents\Tanner EDA\Tanner Tools v13.0\L-Edit and LVS\Tutorial\RingVCO_TestBench.sp

Browse to locate netlist.

Press Edit to view netlist.



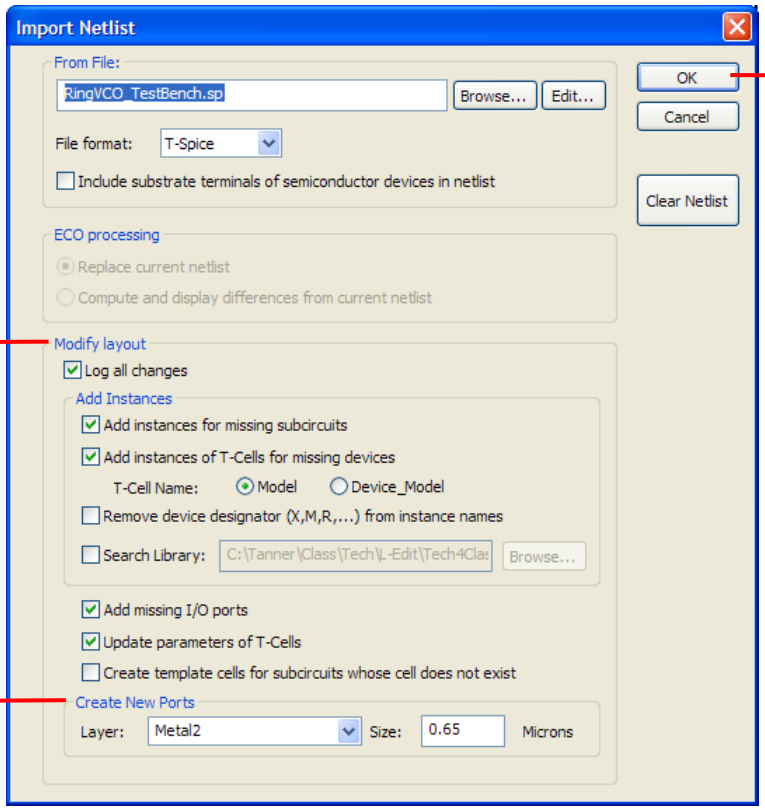
- Press **Edit…** to view the netlist. Scroll to the middle of the netlist to view the subcircuit definition for `RingVCO`. The devices listed inside the subcircuit will be instantiated into the `RingVCO` cell upon importing the netlist using SDL. Notice there is one `Control` block and nine `DiffCell` blocks.



- SDL may be run on multiple blocks using the same top-level netlist file. If the subcircuit name in the netlist does not match a cell name identically, the top-level schematic will be used from the netlist.
- Close the netlist window.

9. Now, let's import the netlist using SDL.

- Select the **Load Netlist** icon  again on the **SDL Navigator** toolbar to open the **Import Netlist** dialog.
- Select the following **Modify layout** and **Create New Ports** settings in the **Import Netlist** dialog as shown.



Change **Modify layout** settings as shown.

Change **Create New Ports** settings as shown.

Press **OK** to import the netlist.

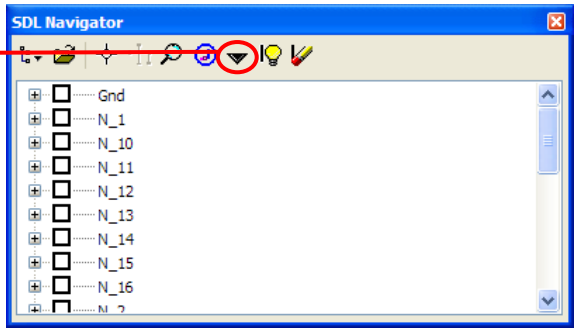
- Press **OK** to import the netlist.
- Close the resulting log file.

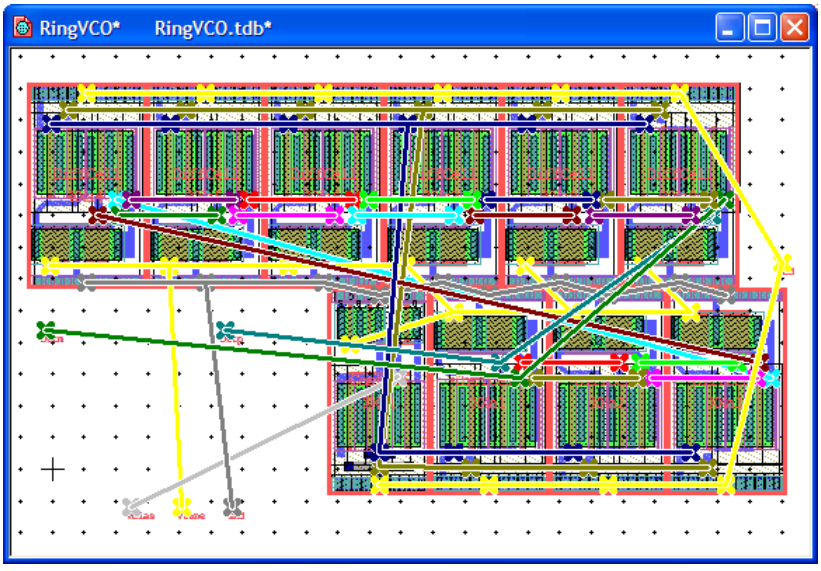
6. Using flylines to place devices

10. Flylines can now be used to facilitate placing the blocks such that the layout is compact and the routes are optimized.

    - The devices are automatically placed next to each other in the layout and are all selected.
    - The **SDL Navigator** will contain the checklist of all nets in the layout that need to be routed.
    - To ensure all devices in the `RingVCO` cell are selected, press **CTRL+A** to select all. There should be 6 ports and 10 instances selected.
    - To view the flylines for all nets, select the **Command Menu** icon on the **SDL Navigator** toolbar and invoke **Add Selection Flyline**. Colored flylines should appear on the layout for each net segment as shown below.

Select **Command Menu** followed by **Add Selection Flyline**.





    - Select and drag each port or instance to create the shortest routing paths possible for the design. The flylines will update as you move the ports and instances.
    - If you prefer to work with one or two nets at a time during placement, clear the flylines by selecting the **Remove All Markers** icon on the **SDL Navigator**

toolbar. Select the net(s) in the **SDL Navigator** checklist that you want to work with, and press the **Flyline** icon  on the toolbar. Alternately, you may right-click the net name(s) and invoke the **Flyline** selection.

- To toggle the flylines off and on, select the **Toggle Markers** icon  on the **SDL Navigator** toolbar. To clear the flylines, select the **Remove All Markers** icon  .
- You can view cell `RingVCO_Placed` for one example of how the instances and ports may be placed.

7. Tagging geometry on manually placed routes

11. Once the ports and instances are placed in the layout, critical nets may be manually routed to ensure they have the shortest path and will not be routed by the SDL auto router.

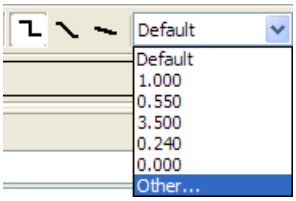
    - Open cell `RingVCO_Placed` by double-clicking the cell name in the **Design Navigator**.
    - In this design, `Vdd` and `Gnd` are considered critical nets that should be manually routed.

12. Manually placed routes should be tagged with their net name so that all geometry associated with the net segment can be ripped up if needed.

    - In this cell, the `Vdd` net segment was connected by overlapping the `Vdd` rails on all instances. Therefore the `Vdd` checkbox is checked to indicate the completion of routing for that net segment. No geometry was placed; therefore the net segment does not need to be tagged.
    - Geometry was added to the `Gnd` rail at the top of the layout to connect the `XXC1 Control` block to the `XXa9 DiffCell` block. To view the geometry already placed and tagged, right-click net segment `Gnd` in the **SDL Navigator** checklist and invoke **Select Net**. There should be 3 boxes and 2 instances selected and all are tagged with the net name **"Gnd"**, as can be viewed in the **Status Bar** at the bottom left of the L-Edit window.

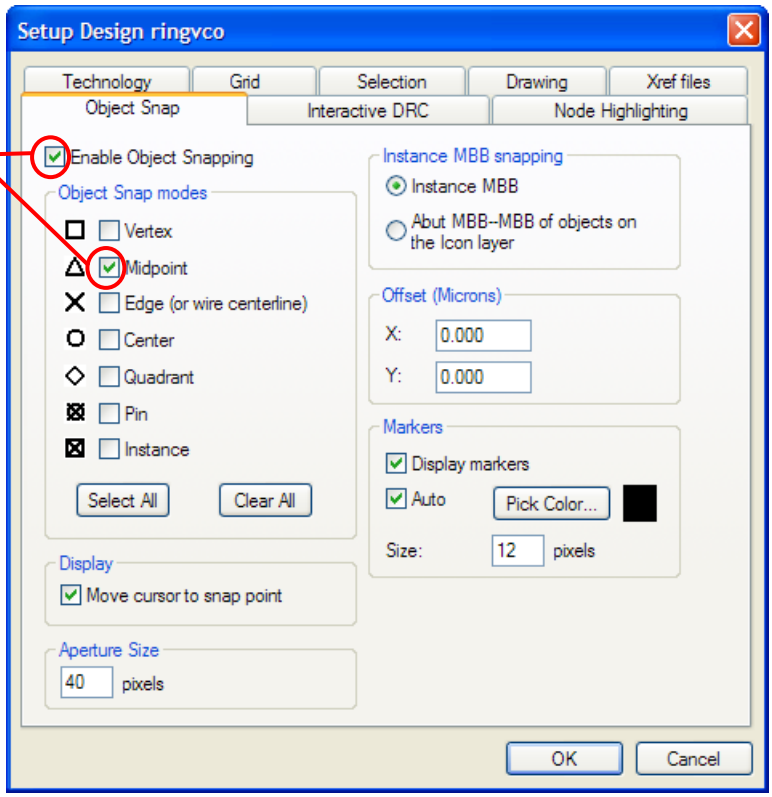
13. Let's complete the routing of the Gnd net segment and tag the geometry we place.

    - Expand the ⊞ symbol to the left of the `Gnd` net segment in the **SDL Navigator** to view the pins associated with that net segment. Notice that `XXa1/Gnd` and `XXC1/Gnd` are the only pins not checked off.
    - Select the `Gnd` net segment in the **SDL Navigator** and press the **Flyline** icon  on the toolbar. This shows the connection that needs to be made between the top and bottom `Gnd` rail.
    - We want to make a wide metal (1.00u) connection between the rails on Metal1 along the left side of the layout. To do this, select the **Metal1** layer from the **Layer Palette** and the **Switch to drawing orthogonal wires** icon  on the L-Edit toolbar.
    - Select the dropdown from the toolbar to set the wire width and select `1.000`.

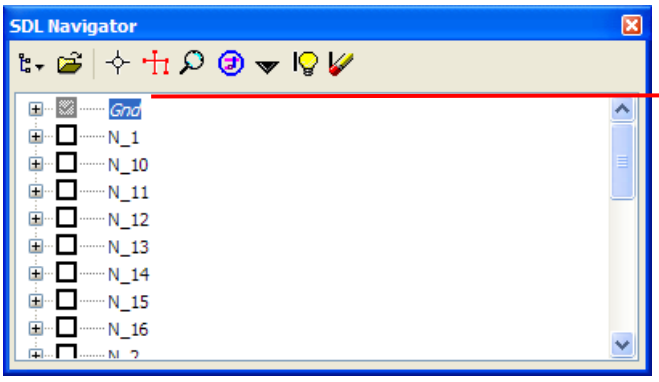


- Enable object snapping to **Midpoint** by selecting the **Setup Object Snap** icon on the L-Edit toolbar.

Select **Command Menu** followed by **Add Selection Flyline**.

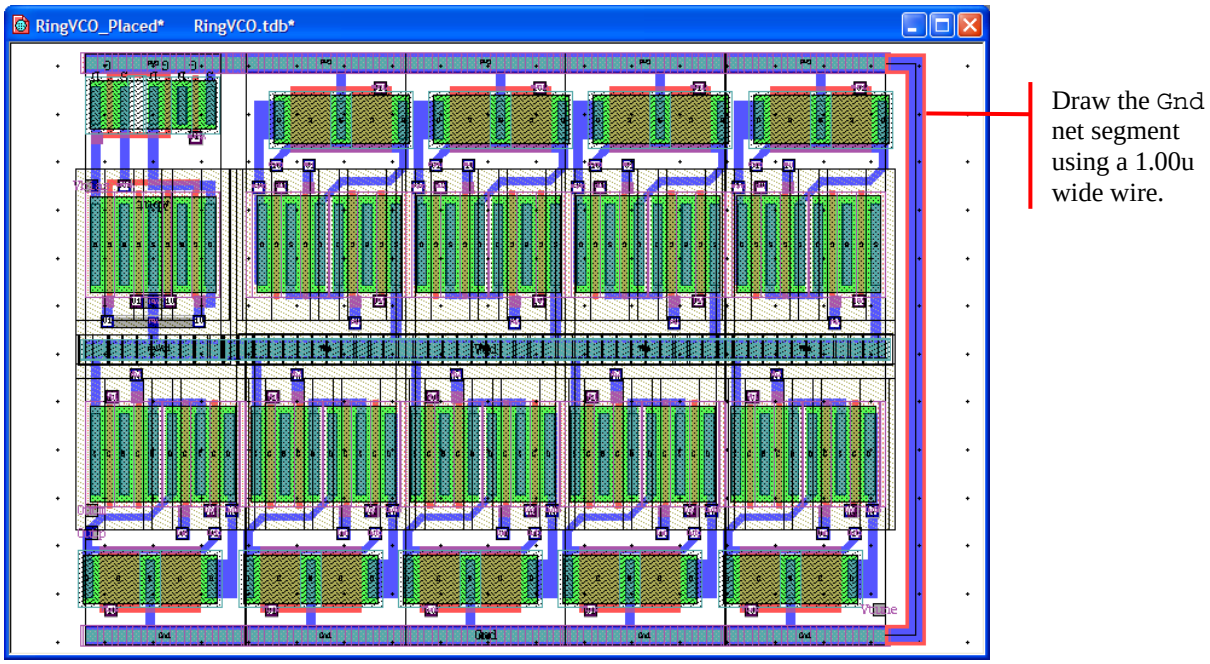


- Right-click the `Gnd` net segment in the **SDL Navigator** and select **Active Net**. Alternately, you can click on the `Gnd` net segment in the **SDL Navigator** with the middle-mouse button to activate the net. The net name font will change to italics to indicate that it is the active net. All geometry placed while the net is active will be tagged as the `Gnd` net segment.



Activate the `Gnd` net to tag geometry.

- Draw a wire connecting the upper Gnd rail to the lower Gnd rail.



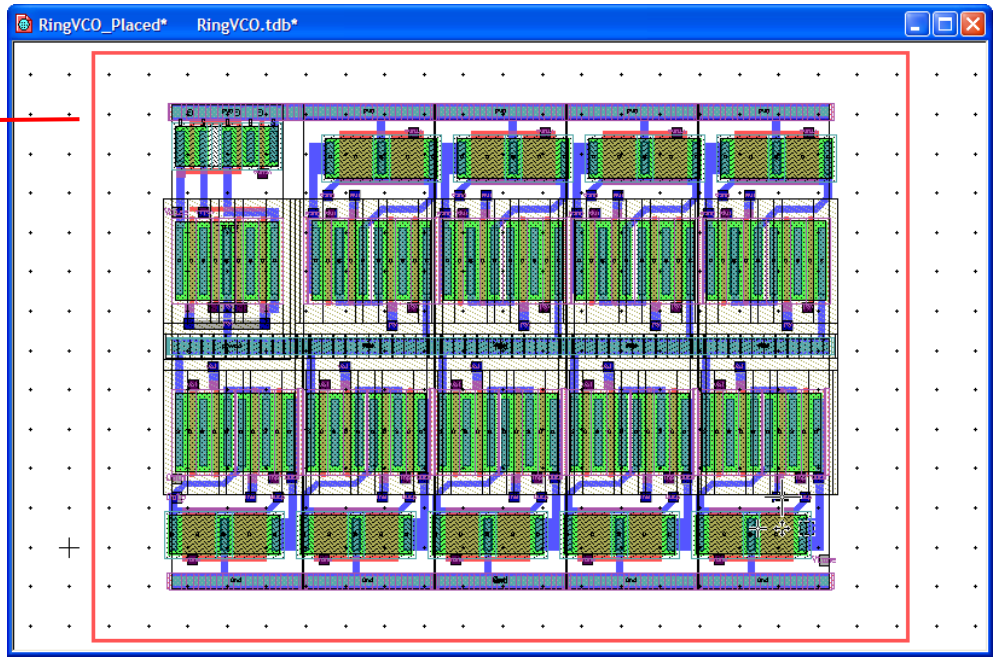
- Turn off the active net by clicking on the Gnd net segment in the **SDL Navigator** with the middle-mouse button.
- If geometry for a net segment is placed without first activating the net for tagging, the geometry can still be tagged with the net name. Just make the desired net the active net using the steps shown above, select the geometry you would like to tag, and invoke the **Command Menu** icon  on the **SDL Navigator** toolbar followed by **Tag Selections with active net**.
- Check off the Gnd net segment checkbox in the **SDL Navigator** to indicate that the entire net segment has been completed.

8. Setting up the automatic router

14. We will want to create a routing region to define the area in which the automatic router will be constrained to.

    - Select the **Routing Region** layer from the **Layer Palette** and the **Switch to drawing boxes** icon  on the L-Edit toolbar.
    - Draw a box covering the entire layout. This will allow the router to place routes outside of the minimum bounding box of the I/O ports.

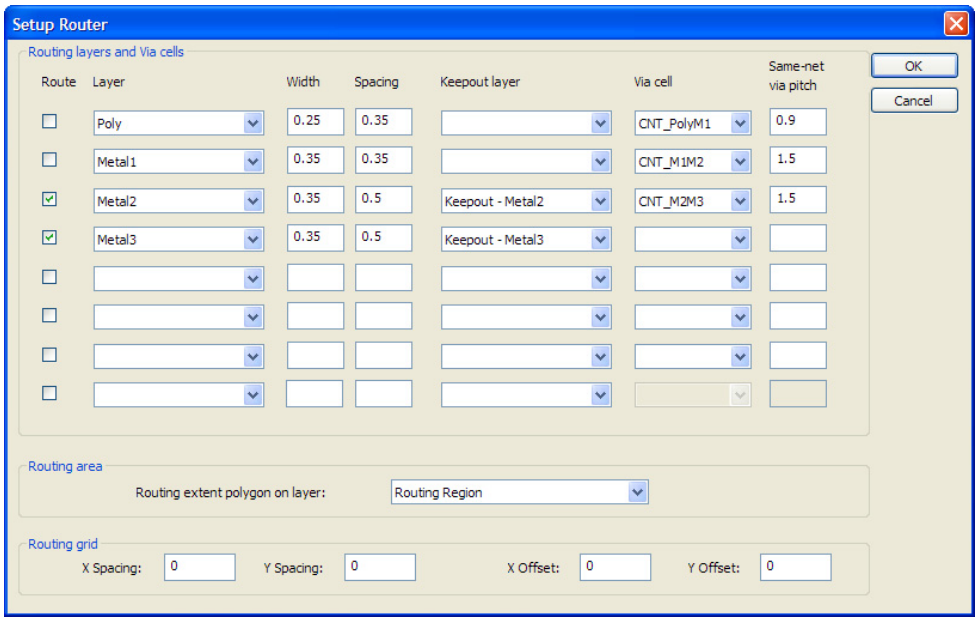
Draw a routing region using the **Routing Region** layer.



15. Prior to using the SDL automatic router, some routing information must first be defined.

    - Open the Setup Router dialog by selecting the **Command Menu** icon  on the **SDL Navigator** toolbar and invoking **Setup Router…**.

- Verify the following settings in the Setup Router dialog:



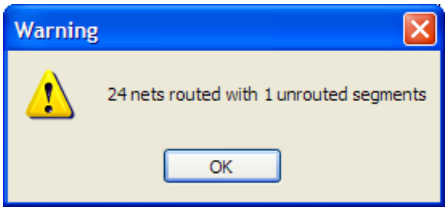
- For this example, we will be routing in **Metal2** and **Metal3** as can be seen via the **Setup Router** dialog **Route** checkboxes. The first routing layer will generally be used to route in the horizontal direction and the second will be used in the vertical direction.
- The **Poly** and **Metal1** layers are not used as routing layers but are specified in order to define parameters for connecting routes to those layers. They are used mainly to define which via or contact cell should be used for connection to ports on that layer.
- The **Width** field defines the minimum width to be used for routes on that layer. As mentioned earlier in the tutorial, the size of the port box will define actual width of the route to be used. If connecting to a point port, the width in this dialog is used. If connecting to a line port, the width of the line port is used unless the line port is smaller than the width specified in this dialog. When connecting to box ports, the smaller dimension of the box port is used unless that dimension is smaller than the width specified in this dialog.
- The **Spacing** field defines the minimum spacing to be used between objects placed on that layer. Routes will also be spaced away from any object on the same layer that is present at the current level of hierarchy that is being routed.
- The **Keepout layer** defines the area in which each routing layer should not route. **Keepout – Metal2** and **Keepout – Metal3** were defined in the lower level cells and cover the active region of the transistors. This keeps the routes from crossing over the transistors.
- **Via cell** defines via cell to use for the specified layer when transitioning to the next layer. The last routing layer does not require a via cell to be defined as it does not have a subsequent layer it needs to connect to.
- **Same-net via pitch** defines the center-to-center spacing to be used when placing vias on wide metal traces or on the same net.

- **Routing extent polygon on layer** defines the layer that defines the area in which the router is allowed to place routes. Here we selected the **Routing Region** layer that we previously created.
- **X Spacing**, **Y Spacing**, **X Offset**, and **Y Offset** are all used to define a custom grid for routing. In this case, all are set to a value of 0 allowing the router to automatically calculate the grid to use.
- Select **OK** in the Setup Router dialog to accept any changes.

9. Automatic routing

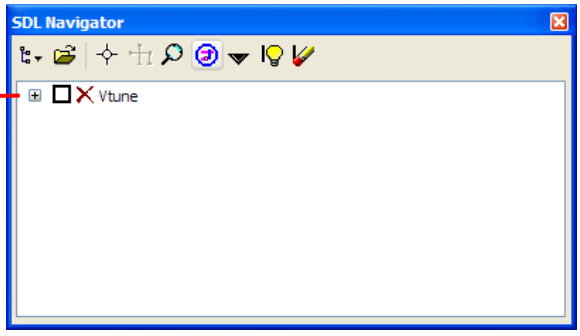
16. Once the routing setup is entered, the SDL automatic router may be used to route a single net or all nets that have not been completed.

- In cell `RingVCO_Placed`, select the **Route All** icon in the **SDL Navigator** to automatically route the remaining nets. Nets that have been checked off in the checklist will not be touched by the router.
- A warning will appear stating the number of nets that were routed and indicating that a net segment was unable to be routed.



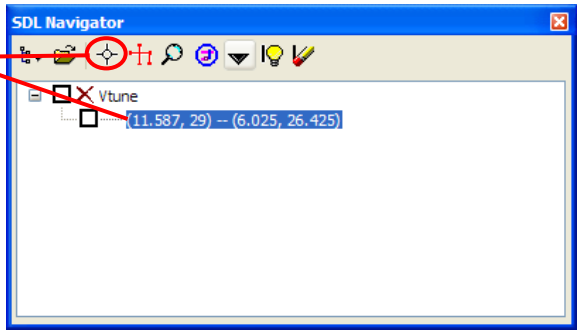
- All geometry that was placed by the automatic router has been tagged with the name of the its net.
- The SDL Navigator view will automatically change from viewing **By Net** to viewing **By Unrouted Segment**.
- Switch to view **By Net** and notice that each net segment is marked with a visual indication of whether the route completed or failed. To change the SDL Navigator view, select the Netlist view icon and select to view **By Net**, **By Instance**, or **By Unrouted Segment**. Completed nets are marked with ✓ and uncompleted are marked with ✗.
- Make sure the view is switched back to view **By Unrouted Segment**.
- Expand the ⊞ symbol to the left of the `Vtune` net segment in the **SDL Navigator** to view the coordinates of the portion of the net that could not be routed.

Expand the + to view coordinates.



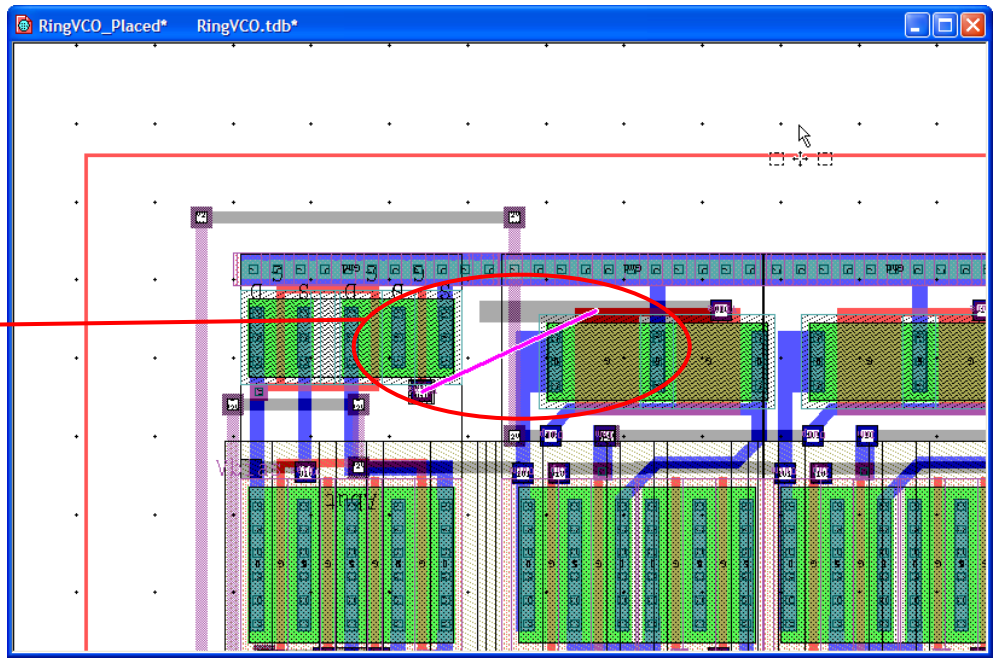
- Click in the layout to deselect all the routes that were added.
- Select the coordinates and press the **Marker** icon  to view the connection that could not be completed.

Select coordinates and press Marker



- The marker may be different than the flyline if a portion of the net segment was routed but not completed. Notice in this case that the marker indicates a missing route between a port and a wire.

Marked segment that was unable to be routed



- Instead of routing all nets at once, a single net or selected nets may be routed by selecting the net(s) in the **SDL Navigator**, right-clicking the net name and invoking **Route**.
- Make sure to check off the nets that are completed so that subsequent automatic routing does not affect those completed routes.

10. Ripping up nets

17. If you are not happy with the path of a route, you can ripup the nets you want to reroute.

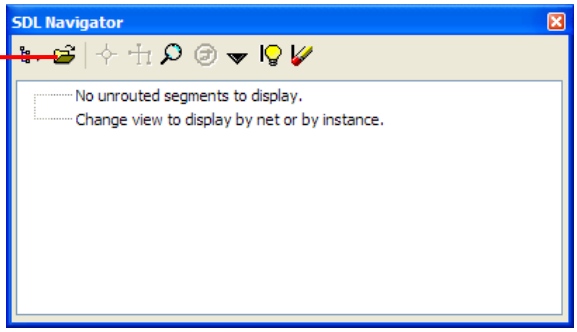
    - Switch back to view **By Net** in the **SDL Navigator**.
    - Right-click the `Outp` net segment in the **SDL Navigator** and invoke **Ripup Net**. A message will appear indicating how many objects were deleted from the net and the routing status indication for the net is removed. All geometry that was tagged for that net has been removed.
    - The net can now be re-routed manually or by using the automatic router.

11. Importing an Engineering Change Order (ECO)

18. Many times changes to the design or an ECO will be issued on a design after the layout has been created. If the layout was created using SDL, an ECO netlist may be loaded to view the differences between the new netlist and the previous netlist.

    - Close cell `RingVCO_Placed` and reopen cell `RingVCO`. Select the **Load Netlist** icon  on the **SDL Navigator** toolbar to open the **Import Netlist** dialog.

Select to open Import Netlist dialog



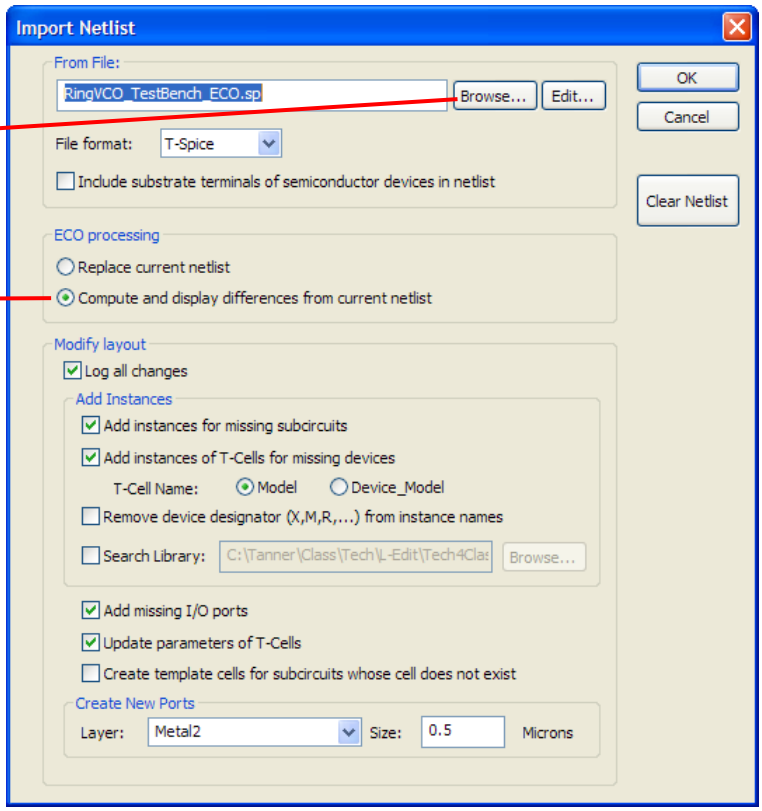
    - Select **Browse…** to locate and load in the netlist `RingVCO_Testbench_ECO.sp` into the **From File:** field. The file is located in the following location:

      My Documents\Tanner EDA\Tanner Tools v13.0\L-Edit and LVS\Tutorial\RingVCO_TestBench_ECO.sp

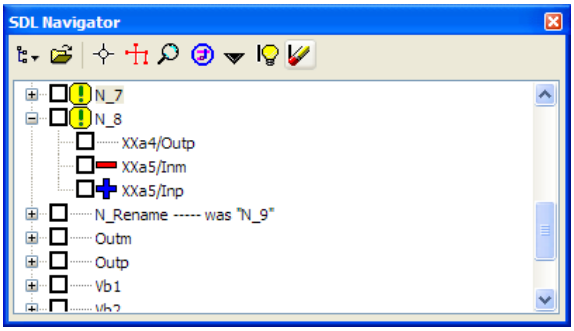
- The ECO processing section will now be enabled. Select **Compute and display differences from current netlist**.

Browse to locate ECO netlist.

Select to Compute and display differences from the current netlist.



- Select **OK** to bring in the ECO changes. Close the resulting log file.
- Expand the ⊞ symbol to the left of the `N_8` net segment in the **SDL Navigator**. The ⚠ icon indicates a difference between netlists for that net segment. A ✚ indicates that a pin was added in the new netlist while a ▬ indicates that a pin was removed.



- Notice that renamed nets are indicated with both the new net name and the old net name. In this case, net segment `N_9` was renamed to `N_Rename`.