cadence®

OrCAD Tutorial

Product Version 17.4-2019
October 2019

Document Last Updated: December 2020

© 2019-2020 Cadence Design Systems, Inc. All rights reserved.
Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Product PSpice contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

© 2020 All Rights Reserved.

© 2020 All Rights Reserved.

1

Tutorial Overview

This tutorial enables you to evaluate the power of the OrCAD tools for the PCB design process. You can run through the steps to perform the basic tasks in the PCB design process in sequence.

You will start with capturing the circuit diagram in Capture, followed by circuit simulation using PSpice, through to the PCB layout stages, and finally, complete the design cycle by generating the manufacturing output.

This document does not cover all the features of a tool. It only highlights the tasks that you need to perform in each OrCAD tool so that your design works smoothly through the flow.

Audience

This tutorial is useful for a:

- Designer who wants to use OrCAD tools for the complete PCB design flow or for analog and digital simulation flow.
- First-time user of OrCAD Capture, PSpice, and OrCAD PCB Editor.

Using the tutorial

To run through the complete tutorial, you need the following tools:

- OrCAD Capture CIS
- PSpice AD
- OrCAD PCB Editor

All these tools are available with the `OrCAD PSpice Designer and CIS` license.

© 2020 All Rights Reserved.

Sample Design

The design created by the end of this tutorial is available at the following location:

<17.4_installation_directory>\share\orcad\tutorial\tutorial.opj

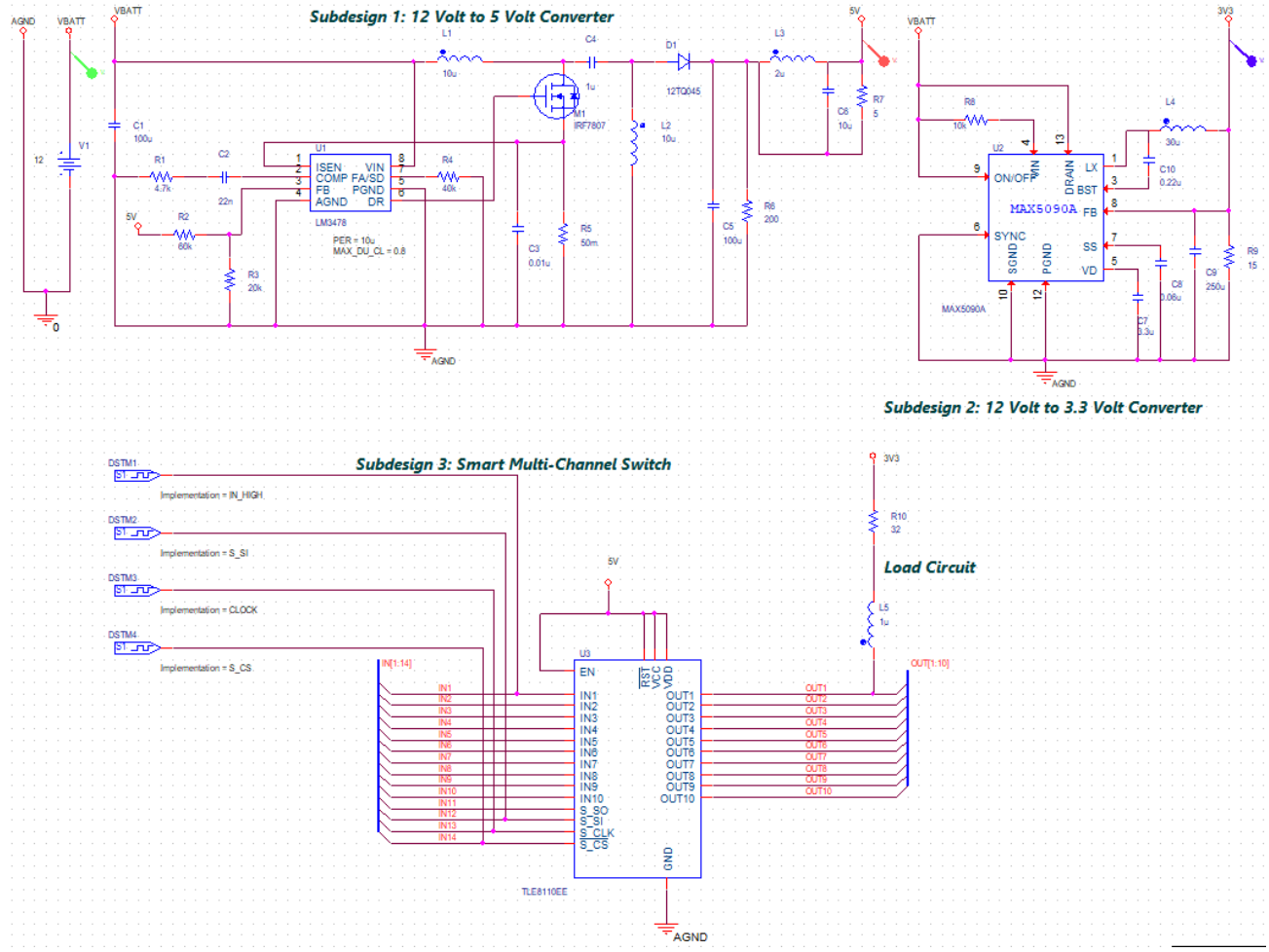
© 2020 All Rights Reserved.

2

Creating a Schematic Design

In this chapter, you will create a schematic design for a fan-control module as shown in the following figure.

Figure 2-1 Schematic design for the complete fan-control module



As shown in Figure 2-1 on page 7, there are three subdesigns in this fan module. The Table 2-1 on page 8 explains the function of each subdesign.

© 2020
All Rights Reserved.

Table 2-1 Subdesigns in the fan module circuit

Subdesign	Function of this subdesign
Subdesign 1	A step-down DC-DC converter. It uses IC LM3478, a low-side n-channel MOSFET controller for switching regulators, which converts 12 volt to 5 volt. The output of 5 volts is the internal power supply for this circuit. It provides supply voltage to IC TLE8110EE (in subdesign 3).
Subdesign 2	A step-down DC-DC converter that uses the IC MAX5090A to convert 12 volt to 3.3 volt.
Subdesign 3	A smart multi-channel switch, TLE8110EE that is powered by 5 volt supply. This switching IC, TLE8110EE can control multiple load types (fan load in this tutorial). It has input 10 channels and a serial peripheral interface (SPI). It has 10 output pins, which can control up to 10 fans.

© 2020 All Rights Reserved.

Creating a New Project

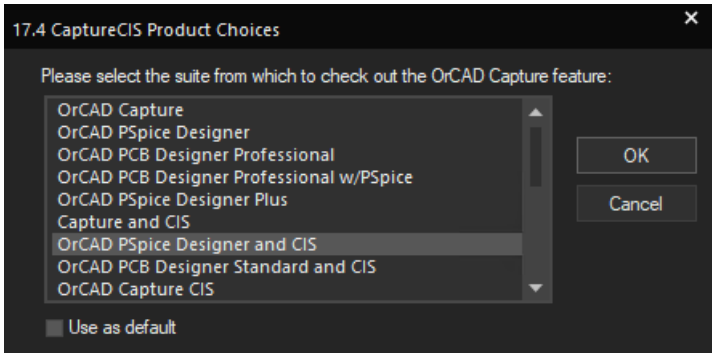
A project file (.OPJ) is a container for the design file (.DSN). In addition, a project file also includes simulation profile and layout information.

To create a new project, do the following:

1. From the Start menu, select *Capture CIS 17.4*.

   The 17.4 CaptureCIS Product Choices dialog box opens.

   Figure 2-2 Product Choices window



2. Select `OrCAD PSpice Designer and CIS`.

   The OrCAD Capture CIS window opens.

3. Select *File – New – Project*.

4. In the New Project dialog box, specify the project name as `tutorial`.

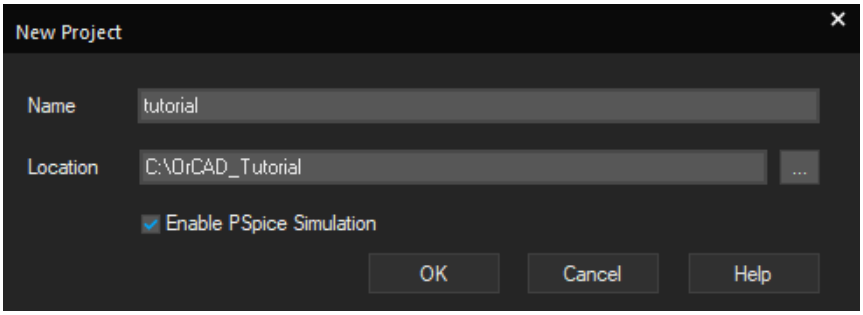
5. Specify the location where you want the project files to be created.

   For this tutorial, specify the location as: `C:\OrCAD_Tutorial`

6. Select *Enable PSpice Simulation*.

© 2020
All Rights Reserved.

Figure 2-3 New Project window

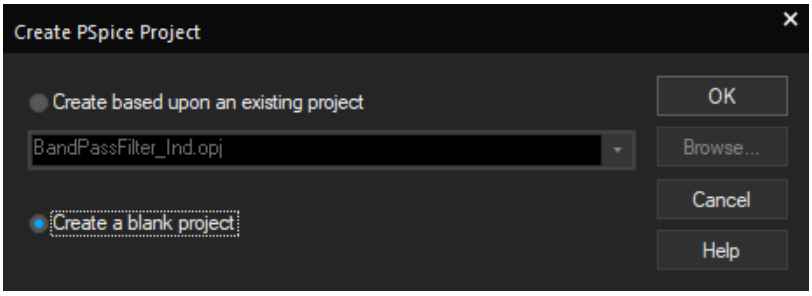


7. Click *OK*.

   The Create PSpice Project dialog box appears.

8. Select *Create a blank project*.

Figure 2-4 Create PSpice Project window



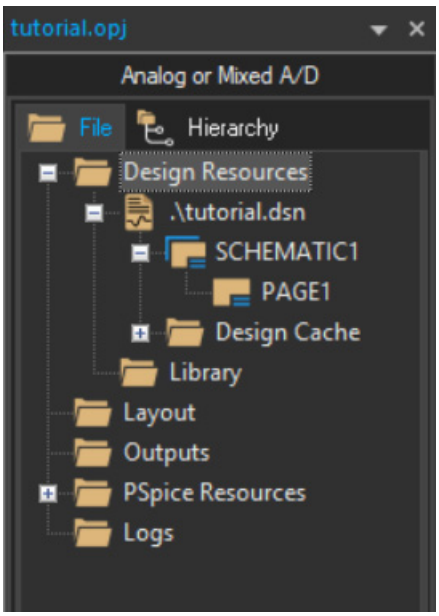
When you create a blank project, the project can be simulated in PSpice, but libraries are not configured by default. When you base your project on an existing project, the new project has same configured libraries.

9. Click *OK*.

   The `tutorial` project is created. In the project manager window, a design file, `tutorial.dsn`, is created. Below the design file, a schematic folder with the name `SCHEMATIC1` is created. This folder has a schematic page named `PAGE1`.

© 2020
All Rights Reserved.

Figure 2-5 Project Manager window



10. Select *Options – Schematic Page Properties*, and choose the page size as `B`.

© 2020
All Rights Reserved.

Creating the Design in Capture

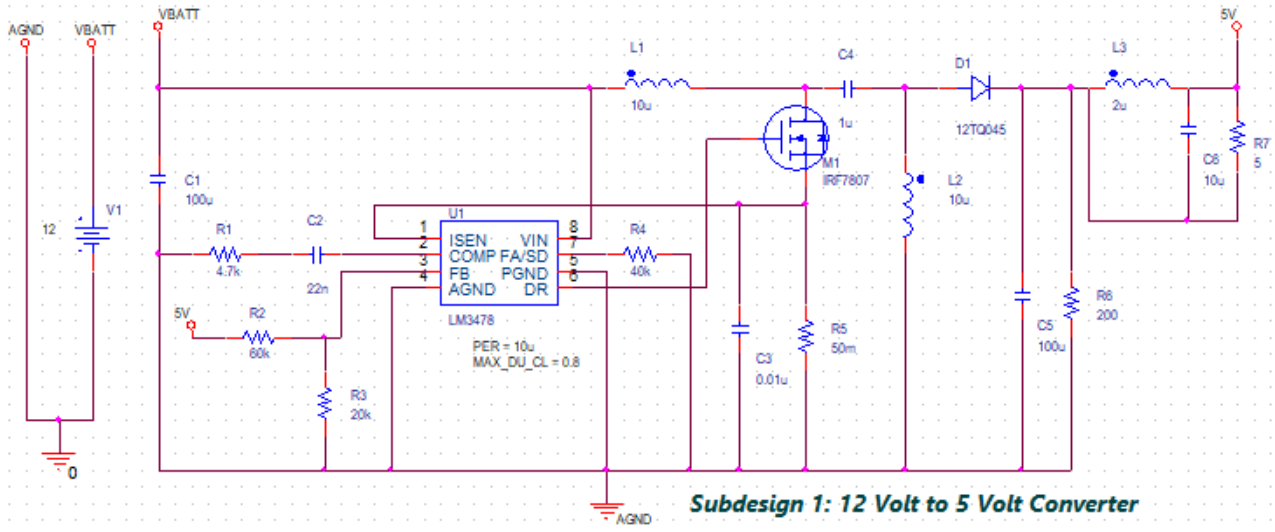
You will now create the three subdesigns of the fan module as shown in Figure 2-1 on page 7.

Subdesign No.	Instructions for subdesign creation are in the section...
1	Adding 12 Volt to 5 Volt Converter
2	Adding 12 Volt to 3.3 Volt Converter
3	Adding Smart Multi-Channel Switch Circuit

Adding 12 Volt to 5 Volt Converter

Subdesign 1 shown in Figure 2-6 on page 12 has two parts, 12 volt main power supply and 12 volt to 5 volt converter.

Figure 2-6 Subdesign 1: Main power supply and 12V to 5V converter



Part 1: Create the circuit for 12 volt main power supply

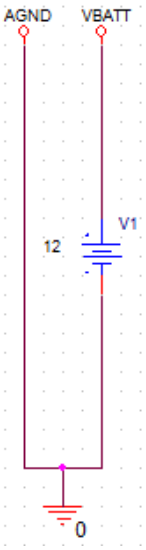
Perform the following tasks to create the circuit for the main power supply:

1. Placing DC voltage source (V1) using Modeling Application
2. Placing ground symbols

© 2020
All Rights Reserved.

3. Adding and placing power ports
4. Connecting the components

Figure 2-7 Subdesign 1A: Main power supply



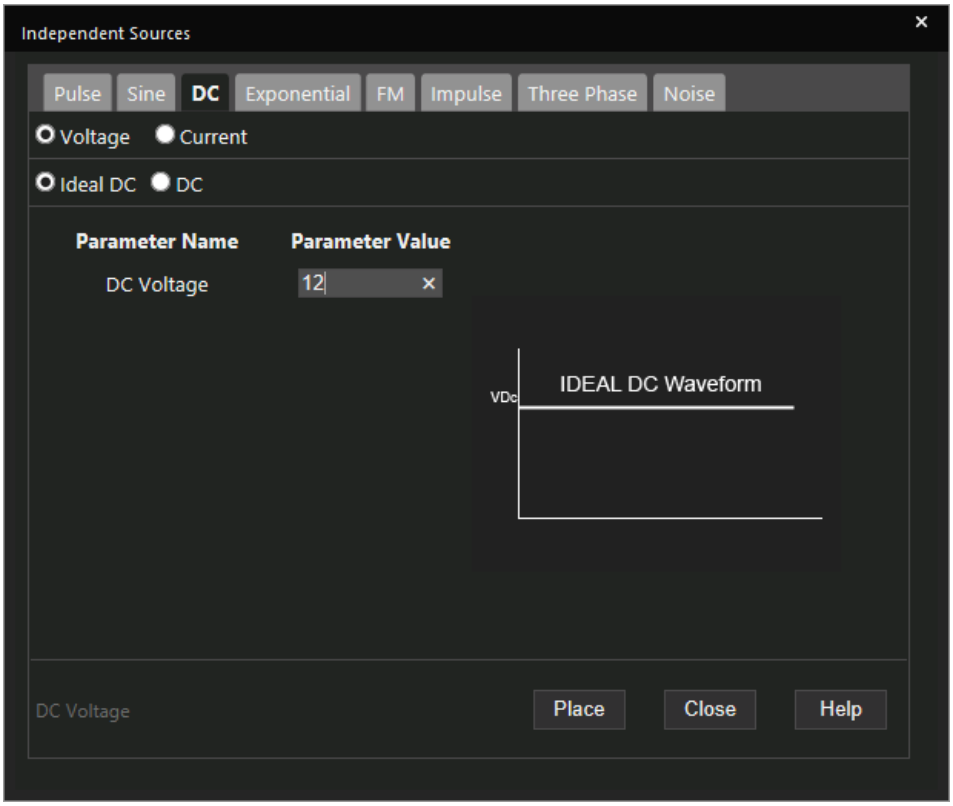
Placing DC voltage source (V1) using Modeling Application

1. Select *Place – PSpice Component – Modeling Application*, or click the Launch Modeling Application icon ( ) on the PSpice toolbar.

   The Modeling Application pane opens.
2. Select *Sources – Independent Sources*.
3. Click *DC*.
4. Select *Voltage* and *Ideal DC*.
5. Specify the value of the voltage as 12 volts.

© 2020
All Rights Reserved.

Figure 2-8 Independent Sources window



6. Click *Place*.

   The PSpice component is attached to a pointer.

7. Click to place the component as required.

8. To save the design, select *File – Save* or press *CTRL+S*.

Placing ground symbols

1. Select *Place – PSpice Component – PSpice Ground*.

2. Click the schematic page to place the part.

3. Right-click and select *End Mode* or press *Esc*.

Adding and placing power ports

Place a power port, `VCC` with its value set to `VBATT`, and anther one with its value set to `AGND` from the `CAPSYM` library. To do so:

© 2020
All Rights Reserved.

1. Select *Place – Power*, press *F*, or click the Place power icon () on the Draw Electrical toolbar.
2. Select `CAPSYM` from the *Libraries* list box.
3. Select `VCC` from the *Symbol* list box.
4. Click *OK*.
5. Click the schematic page to place this power port above the 12 volt DC source.
6. Double-click `VCC` to open the Display Properties dialog box.
7. Specify the *Value* as `VBATT` and click *OK*.
8. Repeat steps 1 through 7 to add another power port, `VCC`, and set its value to `AGND`.

Connecting the components

1. Select *Place – Wire,* press *W*, or click the Place wire icon () on the Draw Electrical toolbar.

   The pointer changes to a crosshair.
2. Draw the wire from the connection point of port `AGND`, PSpice ground, 12 volt DC voltage source, to the `VBATT` port.
3. Click any valid connection point to end the wire.

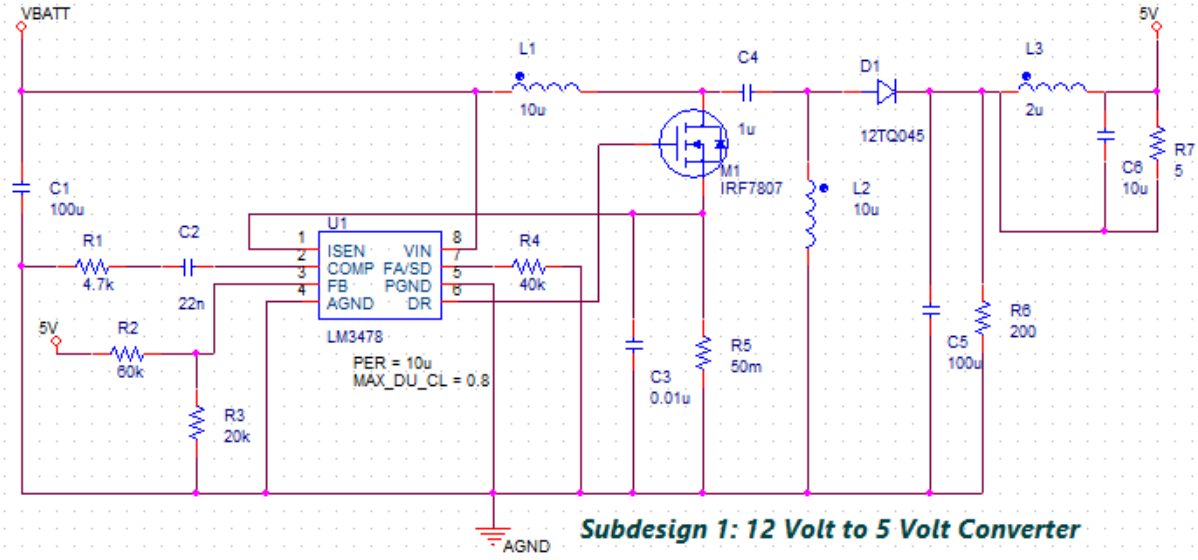
Part 2: Create the circuit for 12 volt to 5 volt converter

Perform the following tasks to create the circuit for the 12 volt to 5 volt converter:

1. Placing the DC-DC converter IC LM3478
2. Placing the MOSFET IRF7807
3. Placing the ports
4. Placing the discrete components
5. Changing the value of discrete components
6. Example - Placing the 100u Capacitor, C1
7. Placing the remaining components in subdesign 1
8. Connecting all the components in subdesign 1

© 2020 All Rights Reserved.

Figure 2-9 Subdesign 1B: 12V to 5V converter



Placing the DC-DC converter IC LM3478

1. Select *Place – PSpice Component – Search*, or click the Launch Part Search icon ( ) on the PSpice toolbar.

   The *PSpice Part Search* pane opens.

2. Specify `LM3478` in the search text box and press *Enter*.
3. Right-click `LM3478` from the search results and select *Place Symbol*.
4. Click the schematic page to place the converter IC.
5. Right-click and select *End Mode* or press *Esc*.
6. Double-click `PER` and set its value to `10u`.

   This value is modified because it sets the frequency of this IC. Changing the value to `10u` (1/10u = 100KHz) means setting the IC frequency to 100 KHz.

Placing the MOSFET IRF7807

1. Select *Place – PSpice Component – Search*, or click the Launch Part Search icon ( ) on the PSpice toolbar.

   The *PSpice Part Search* pane opens.

© 2020
All Rights Reserved.

2. Specify `IRF7807` in the search box and press *Enter*.
3. Right-click `IRF7807` from the search results and select *Place Symbol*.
4. Click the schematic page to place the MOSFET.

Placing the ports

Place the `VBATT` and the two `5V` power ports shown in subdesign 1 (Figure 2-9 on page 16) using the steps listed in the section, Adding and placing power ports.

Placing the discrete components

To place the commonly used discrete components, capacitor, inductor or resistor, do the following:

1. Select *Place – PSpice Component – Capacitor*.

   OR

   Select *Place – PSpice Component – Inductor*.

   OR

   Select *Place – PSpice Component – Resistor*.

   The part symbol is attached to a pointer.
2. Click the schematic page to place the required component.
3. To stop placing the components, right-click and select *End Mode* or press *Esc*.

Changing the value of discrete components

1. Double-click the value of any discrete component.

   The Display Properties window opens.
2. In the *Value* field, specify the required value of the discrete component.
3. Click *OK*.

Example - Placing the 100u Capacitor, C1

1. Select *Place – PSpice Component – Capacitor*.

© 2020 All Rights Reserved.

2. Click the schematic page to place the capacitor `C1` below the `VBATT` port as shown in Figure 2-9 on page 16.
3. Right-click and select *End Mode* or press *Esc*.
4. Double-click the value, `1n`.

   The Display Properties window opens.
5. In the *Value* field, specify the value of `C1` as `100u`.
6. Click *OK*.

Placing the remaining components in subdesign 1

1. Place the remaining discrete components in subdesign 1 (Figure 2-9 on page 16) using steps listed in section, Placing the discrete components and specify their value using steps from the section, Changing the value of discrete components.

PSpice Component Name	PSpice Component Value
C2	22n
R1	4.7k
R2	60k
R3	20k
R4	40k
L1	10u
C3	0.01u
C4	1u
R5	50m
L2	10u
C5	100u
R6	200
L3	2u
C6	10u
R7	5

© 2020 All Rights Reserved.

2. Place the `12TQ045` diode using the PSpice Part Search window. For steps, see the section, Placing the DC-DC converter IC LM3478.
3. Place the `AGND` port using the steps listed in the section, Adding and placing power ports.

Connecting all the components in subdesign 1

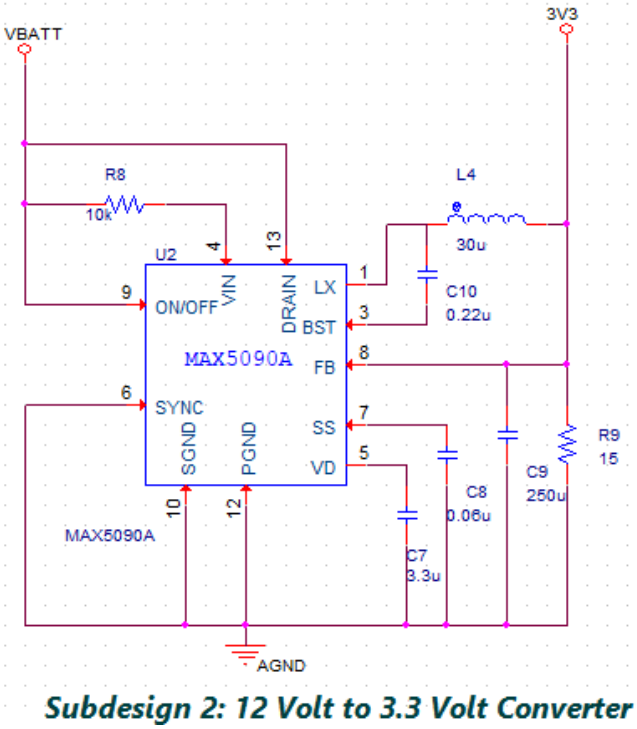
1. Draw wire from pin 1 of `LM3478` to source of MOSFET M1, `IRF7807`.
2. Draw wire from pin 6 `LM3478` to the gate of MOSFET M1, `IRF7807`.
3. Draw wire from `10u` inductor, `L1` to the drain of MOSFET M1, `IRF7807`.
4. Similarly, add wires to the subdesign 1 until all components are connected as shown in the Figure 2-9 on page 16.

© 2020 All Rights Reserved.

Adding 12 Volt to 3.3 Volt Converter

The subdesign 2 of the fan module uses IC `MAX5090A`, which is a DC-DC converter that is down converting 12 volt to 3.3 volt. The output of 3.3 volt powers the fan.

Figure 2-10 Circuit for 12 Volt to 3.3 Volt converter



Subdesign 2: 12 Volt to 3.3 Volt Converter

To create subdesign 2, do the following:

1. Place the DC-DC converter, `MAX5090A`.

   a. Select *Place – PSpice Component – Search*, or click the Launch Part Search icon ( ).

   The *PSpice Part Search* pane opens.

   b. Specify `MAX5090A` in the search box and press *Enter*.

   c. Right-click `MAX5090A` from the search results and select *Place Symbol*.

   d. Click the schematic page to place this converter.

2. Place the `VBATT`, `AGND`, and `3V3` power ports shown in subdesign 2 (Figure 2-10 on page 20) using the steps listed in the section, Adding and placing power ports.

© 2020
All Rights Reserved.

3. Place the remaining discrete components in subdesign 2 (in Figure 2-10 on page 20) using the steps listed in the section, Placing the discrete components and specify their value using steps from the section, Changing the value of discrete components.

PSpice Component Name	PSpice Component Value
L4	30u
C10	0.22u
R8	10k
C7	3.3u
C8	0.06u
C9	250u
R9	15

4. Right-click C8 and select *Edit Properties*. Set value of *IC* as 0.

5. Add wires to the subdesign 2 until all components are connected as shown in Figure 2-10 on page 20.

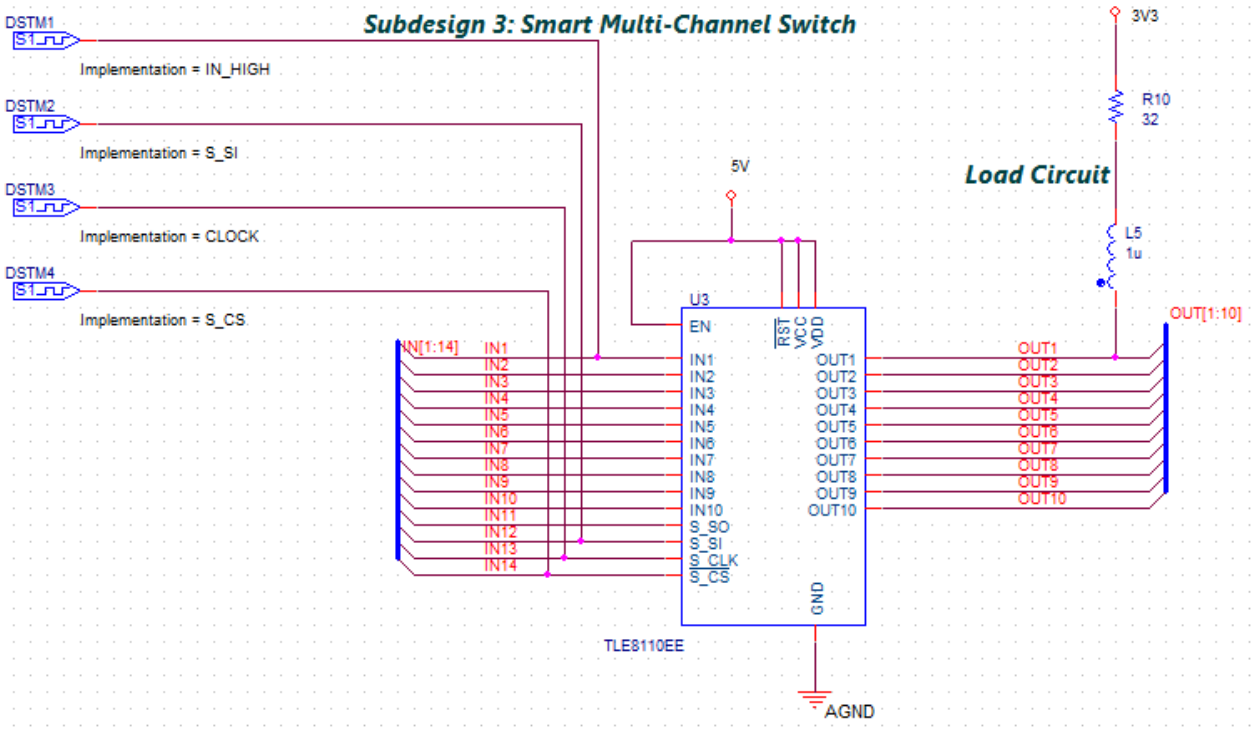
© 2020 All Rights Reserved.

Adding Smart Multi-Channel Switch Circuit

In subdesign 3, you will create the circuit for a smart multi-channel switch as shown in Figure 2-11 on page 22.

In this subdesign, IC, TLE8110EE is placed to be interfaced with an off-board micro-controller. This means that the input to the 10 channels of this IC are digital signals generated from a micro-controller. For this tutorial, these digital signals are modeled using digital stimulus sources.

Figure 2-11 Circuit for smart multi-channel switch, TLE8110EE



Perform the following tasks to create subdesign 3:

1. Placing components of subdesign 3
2. Placing and connecting buses in subdesign 3
3. Connecting components of subdesign 3

Placing components of subdesign 3

1. Place the IC, `TLE8110EE` using *PSpice Part Search*.

© 2020
All Rights Reserved.

To go through the steps, see Placing the DC-DC converter IC LM3478.

2. Place the `AGND, 5V,` and `3V3` power ports shown in subdesign 3 (Figure 2-11 on page 22).

   To go through the steps, see Adding and placing power ports.

3. Place the discrete components in subdesign 3 (Figure 2-11 on page 22) using the steps listed in the section, Placing the discrete components and specify their value using steps from the section, Changing the value of discrete components.

PSpice Component Name	PSpice Component Value
`R10` This resistor forms the RL circuit for the fan type of load.	`32`
`L5` This inductor forms the RL circuit for the fan type of load.	`1u`

4. Place the digital stimulus sources using *PSpice Part Search*.

   These are 1-bit digital stimulus sources.

   To add these, search for `DIGSTIM1` in *PSpice Part Search* and select `DigStim1` from the results.

5. Double-click the `Implementation` property of a digital stimulus source.

   The Display Properties window opens.

6. In the *Value* field, specify the value of this `Implementation` property as shown in the following table:

Digital Stimulus Source Name	Value of Implementation property
`DSTM1`	`IN_HIGH`
`DSTM2`	`S_SI`
`DSTM3`	`CLOCK`
`DSTM4`	`S_CS`

Note: The stimulus data from these sources is read from a `.stl` file, which will be

© 2020 All Rights Reserved.

created and configured in Chapter 3, “Simulating a Design,”.

Placing and connecting buses in subdesign 3

1. Select *Place – Bus*, press *B,* or click the Place bus icon ( ) on the Draw Electrical toolbar.

   The pointer changes to a crosshair.

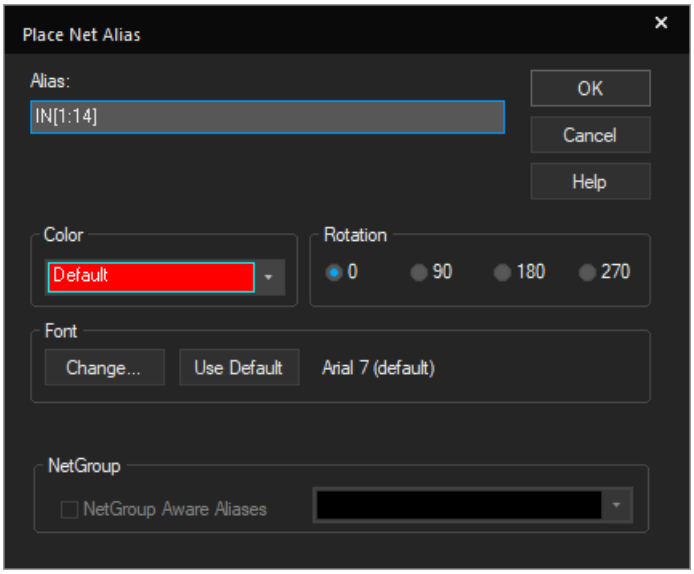
2. Draw the bus before the input pins and after the output pins of `TLE8110EE` as shown in Figure 2-13 on page 25.

3. Select *Place – Net Alias*, press *N*, or click the Place net alias icon ( ) on the Draw Electrical toolbar.

   The Place Net Alias dialog box appears.

4. Specify the value in the *Alias* field as `IN[1:14]`.

Figure 2-12 Place Net Alias window

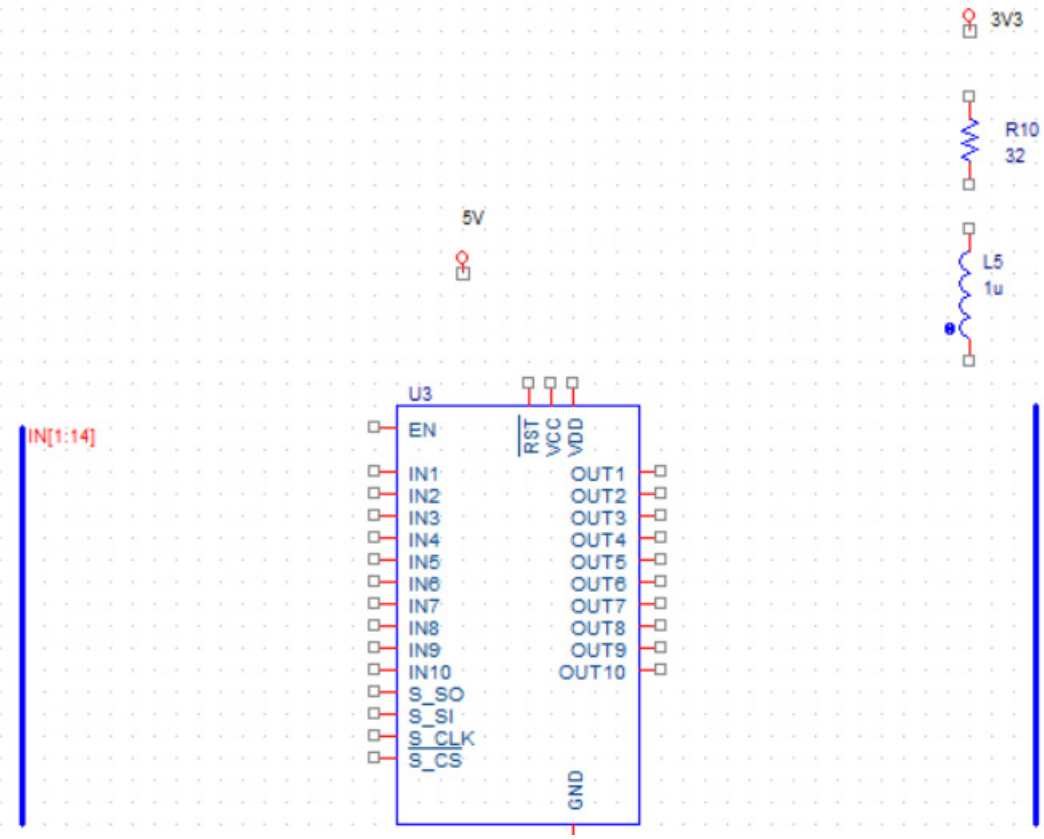


5. Click *OK*.

6. Move the cursor on the bus and place the net alias name as shown in the following figure.

© 2020
All Rights Reserved.

Figure 2-13 Placing bus and specifying net alias

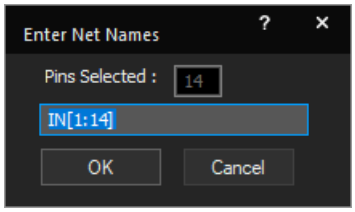


7. Select all the input pins of `TLE8110EE` (except `EN`), right-click the selection, and choose *Connect to Bus*.

   The pointer changes to a crosshair.

8. Click the bus placed before the input pins.

   The Enter Net Names window appears.



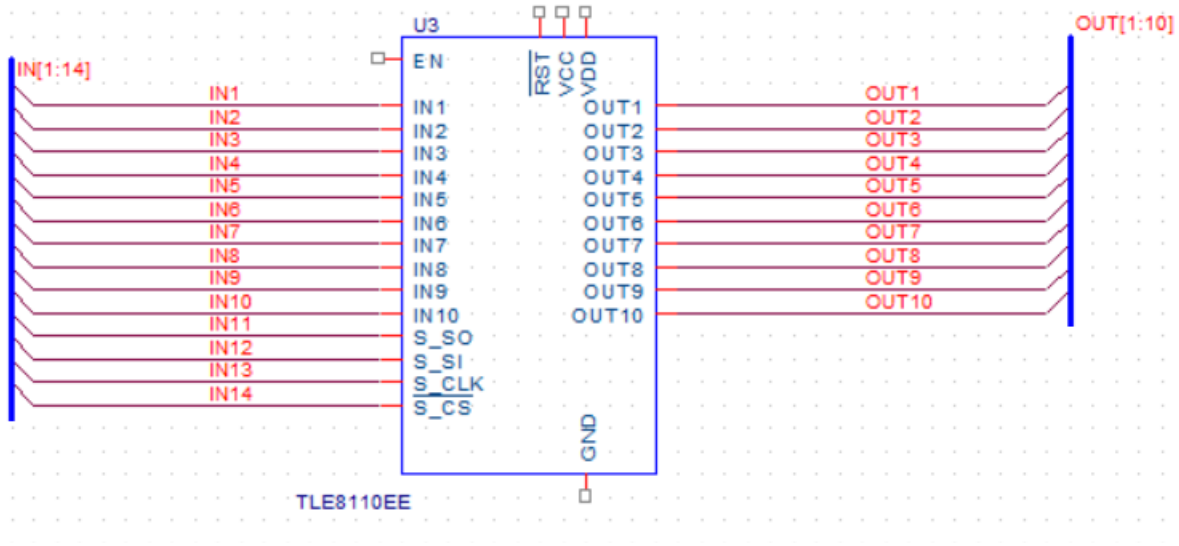
9. Click *OK*.

   Net names appear on each net from the input pins to the bus as shown in Figure 2-14 on page 26.

© 2020
All Rights Reserved.

10. Repeat steps 3 thorough 10 to place the `OUT[1:10]` net alias on the bus placed next to the output pins of `TLE8110EE`, and to connect the output pins of `TLE8110EE` to this bus as shown in the following figure.

Figure 2-14 Connecting bus with pins

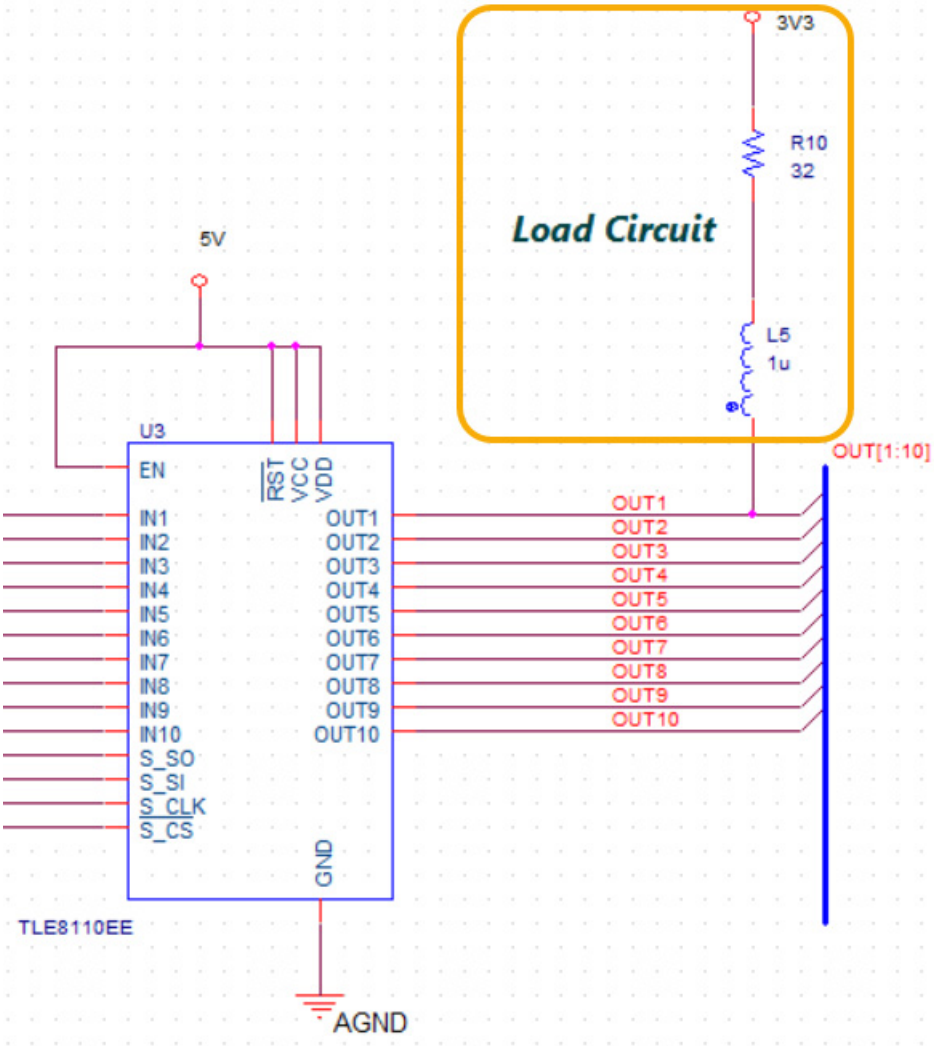


Connecting components of subdesign 3

1. Draw a wires from the pins, `RST`, `VCC`, and `VDD` to the `5V` power port.
2. Draw a wire from the `5V` power port to the input pin, `EN` of `TLE8110EE`.
3. Draw a wire from the `GND` pin to the power port, `AGND`.
4. Draw a wire from the output pin of:
   - `DSTM1` (with `implementation=IN_HIGH`) to net `IN1`.
   - `DSTM2` (with `implementation=S_SI`) to net `IN12`.
   - `DSTM3` (with `implementation=S_CLK`) to net `IN13`.
   - `DSTM4` (with `implementation=S_CS`) to net `IN14`.
5. Connect the load circuit to the net, `OUT1` as shown in the following figure.

All Rights Reserved.

Figure 2-15 Connecting the load circuit



Summary

This chapter covered the steps for creating schematic design using OrCAD Capture. In the process, you were introduced to basic design creation tasks, such as creating project, placing parts, editing property values, and connecting parts.

All Rights Reserved.

© 2020 All Rights Reserved.

3

Simulating a Design

In this chapter, you will use PSpice to simulate the design created in Chapter 2, “Creating a Schematic Design,” using OrCAD Capture. You will also learn about the transient analysis that can be performed using PSpice.

Getting the Design Ready for Simulation

To simulate a design, the PSpice simulator needs information about circuit topology, analysis type, and stimulus definitions.

The analysis type, stimulus definition, and the information related to the initial digital state of the simulation is provided by a simulation profile (*.SIM). In the following section, you will create a simulation profile.

© 2020 All Rights Reserved.

Prerequisite to Simulation Setup

Before creating the simulation profile, you need to specify the stimulus definition to be used for simulating the circuit. For this, you need to create a stimulus file and then specify its location in the Simulation Setting dialog box (used to create the simulation profile). To create the stimulus file, copy the following information in a text editor (`.txt`) file and save it as `stimulus.stl`.

```
.STIMULUS IN_HIGH STIM (1, 1)
+ 0 0
+ +0 1

.STIMULUS S_SI STIM (1, 1)
+0ms 0
+808.400000000u 0
+ +200ns 0
+ +200ns 0
+ +200ns 0
+ +200ns 0
+ +200ns 1
+ +200ns 1
+ +200ns 1
+ +200ns 0
+ +200ns 0
+ +200ns 0
+ +200ns 1
+ +200ns 0

.STIMULUS CLOCK STIM (1, 1)
+ 0 0
+ +0 1
+REPEAT FOREVER
+ +100n 0
+  +100n 1
+ ENDREPEAT

.STIMULUS S_CS STIM (1, 1)
+0u 0
+807.763636u 1
+808.333363636u 0
+811.56335227u 1
+812.13370351u 0
```


© 2020 All Rights Reserved.

Creating a New Simulation Profile

To view the behavior of this circuit over time, transient analysis will be used. To perform this analysis, you will specify this analysis type when creating the simulation profile.

To create a new simulation profile, do the following:

1. Select *PSpice – New Simulation Profile*, or click the *New Simulation Profile* icon ( ) on the PSpice toolbar.

   The New Simulation dialog box opens

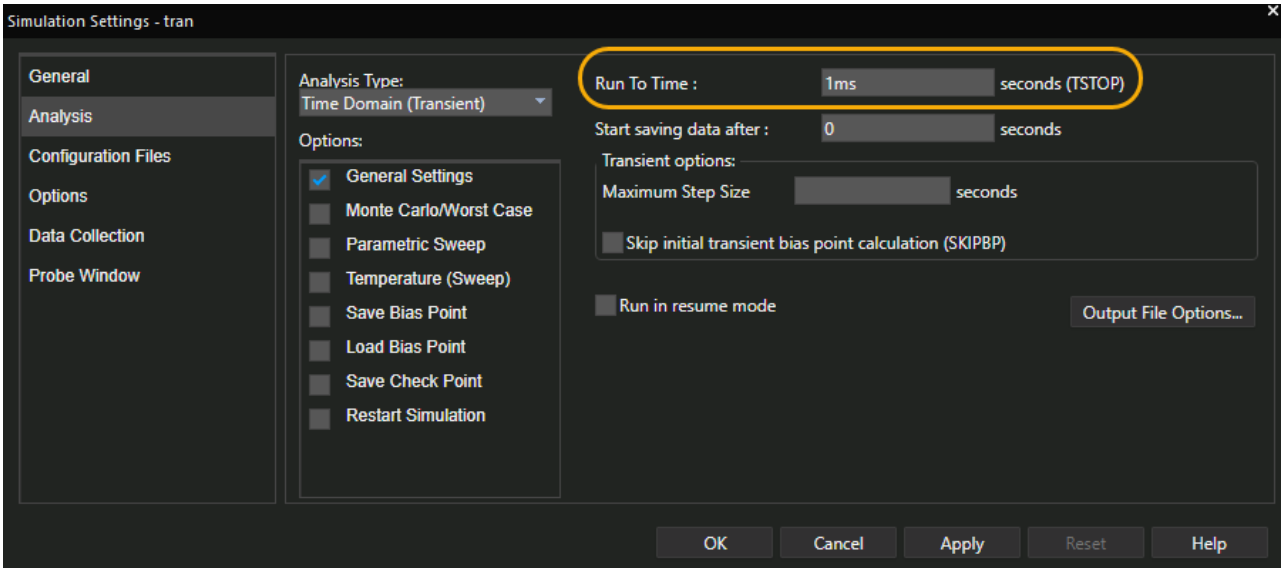
2. Specify the name of the new simulation profile as `TRAN`.
3. In the *Inherit From* drop-down list, ensure that `none` is selected and click *Create*.

   The Simulation Setting dialog box appears with the *Analysis* tab selected.

4. In the *Analysis Type* drop-down list, `Time Domain (Transient)` is selected by default. Retain this default setting.
5. Specify the following options to run a transient analysis.

   - In the *Run To Time* text box, specify the time as `1ms`.

Figure 3-1 Simulation Setup - Analysis tab



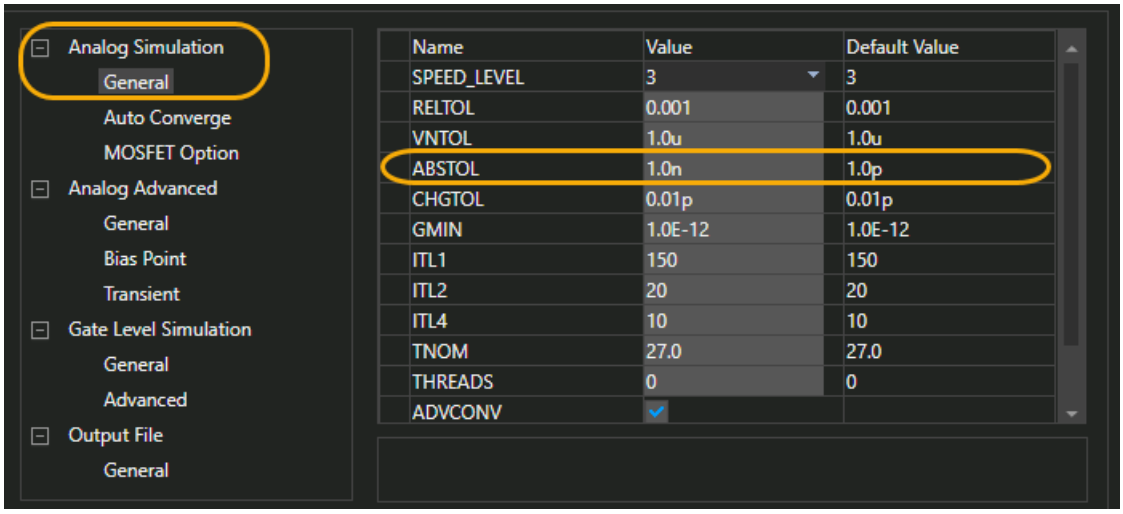
6. In the *Options* tab:

   - To prevent any convergence issues, increase the tolerance to `1n`. To do so:

© 2020
All Rights Reserved.

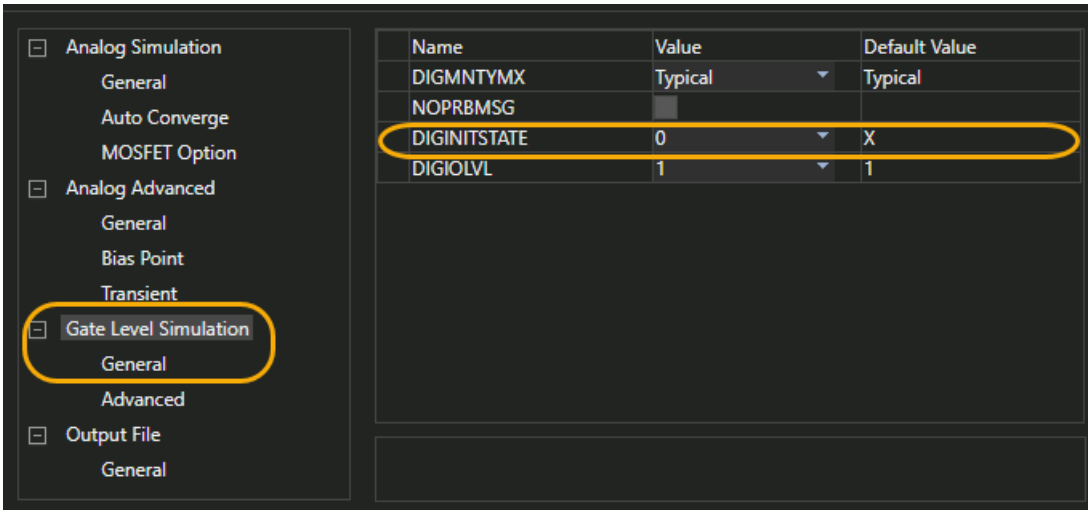
Set *ABSTOL* as `1n`.

Figure 3-2 Simulation Setup - Options tab



- ❑ To set the initial digital state of the simulation to `0`, make the following modification:

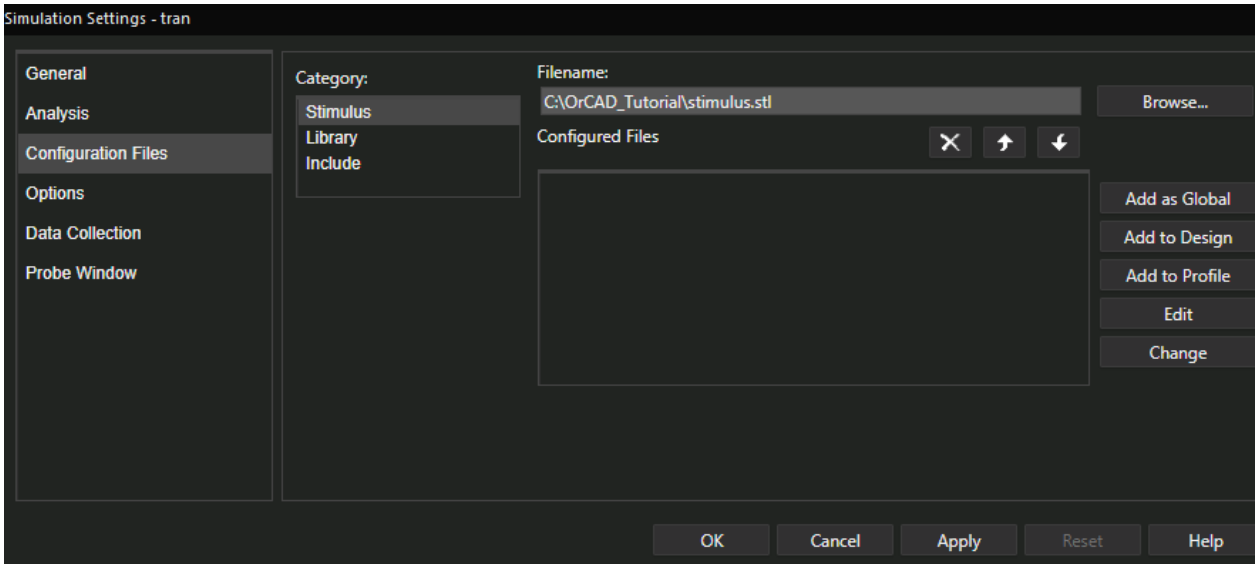
  Set *DIGINISTATE* as `0`.



7. In the *Configuration Files* tab, select `Stimulus` from *Category* list.
8. Click the *Browse* button to navigate to the `stimulus.stl` file.

All Rights Reserved.

Figure 3-3 Simulation Setup - Configuration Files tab



9. Click the *Add to Design* button.

10. Click *OK* to save your modifications and to close the dialog box.

When the simulation is run from the schematic, the simulator reads the SPICE models connectivity information (netlist) from the design files, and the analysis type and the stimulus details from the simulation profile.

© 2020
All Rights Reserved.

Running the Simulation

To simulate the design, choose *PSpice – Run* or click the Run PSpice icon ().

The Schematic1-TRAN - PSpice window opens.

Viewing Output Waveforms

To visualize the circuit behavior and determine the validity of your circuit design, you can plot the output waveforms in the Probe window. By analyzing the output waveforms you can evaluate your circuit for performance.

For PSpice to display output waveforms in the Probe window, you need to place markers in your circuit design in Capture to indicate the points where you want to see simulation waveforms displayed in PSpice.

Markers can be placed:

- before simulation to limit results written to the waveform data file, and automatically display those traces in the active Probe window.
- during or after simulation, to automatically display traces in the active Probe window.

To add markers:

➡ Choose *PSpice – Markers* or use the icons provided on the PSpice toolbar.

Important

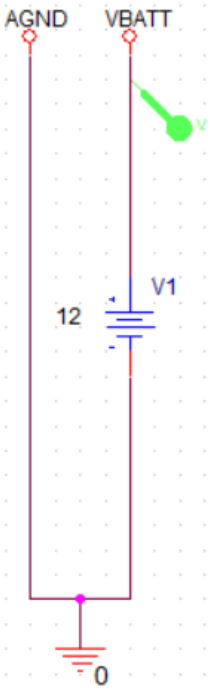
To view the markers in the simulation results, the schematic design must be open.

You will now add Voltage markers to view the output waveforms in the Probe window. To do so:

1. Select *PSpice – Markers – Voltage Level*, or click the Voltage/Level Marker icon () on the PSpice toolbar.

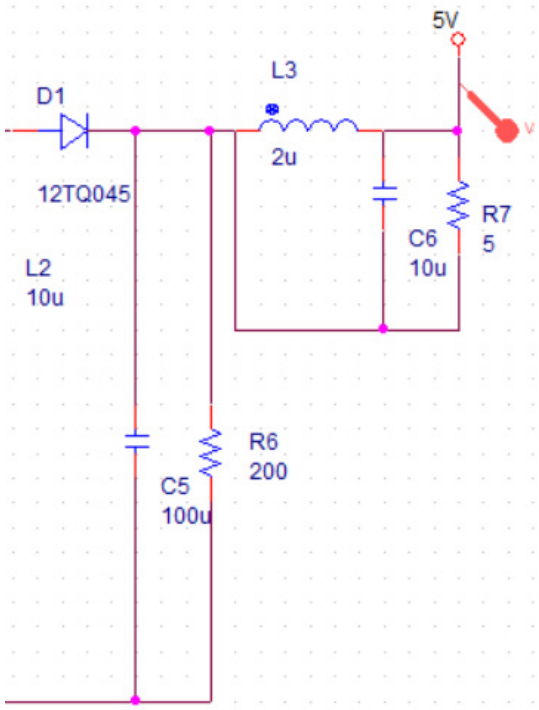
© 2020 All Rights Reserved.

2. Place the marker at the main power supply, that is at the 12 Volt `VBATT` power net.



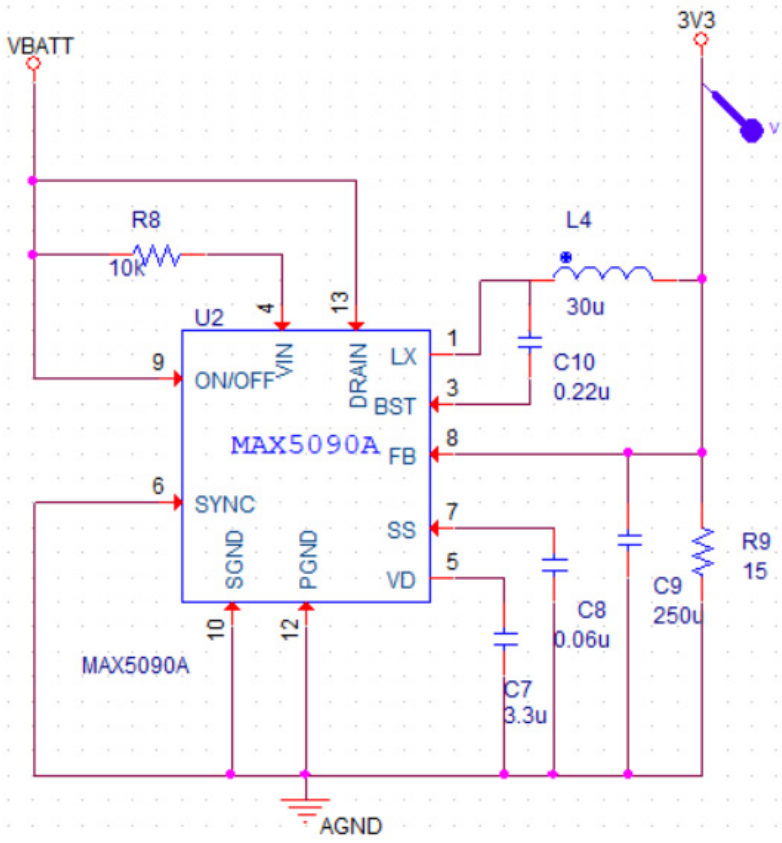
All Rights Reserved.

3. Place the marker at the end of the first subdesign (12V to 5V converter), that is at the `5V` power net, as shown in the following figure.



© 2020
All Rights Reserved.

4. Place another voltage marker at the end of the third subdesign (12V to 3.3V converter), that is at the `3V3` power net, as shown in the following figure.



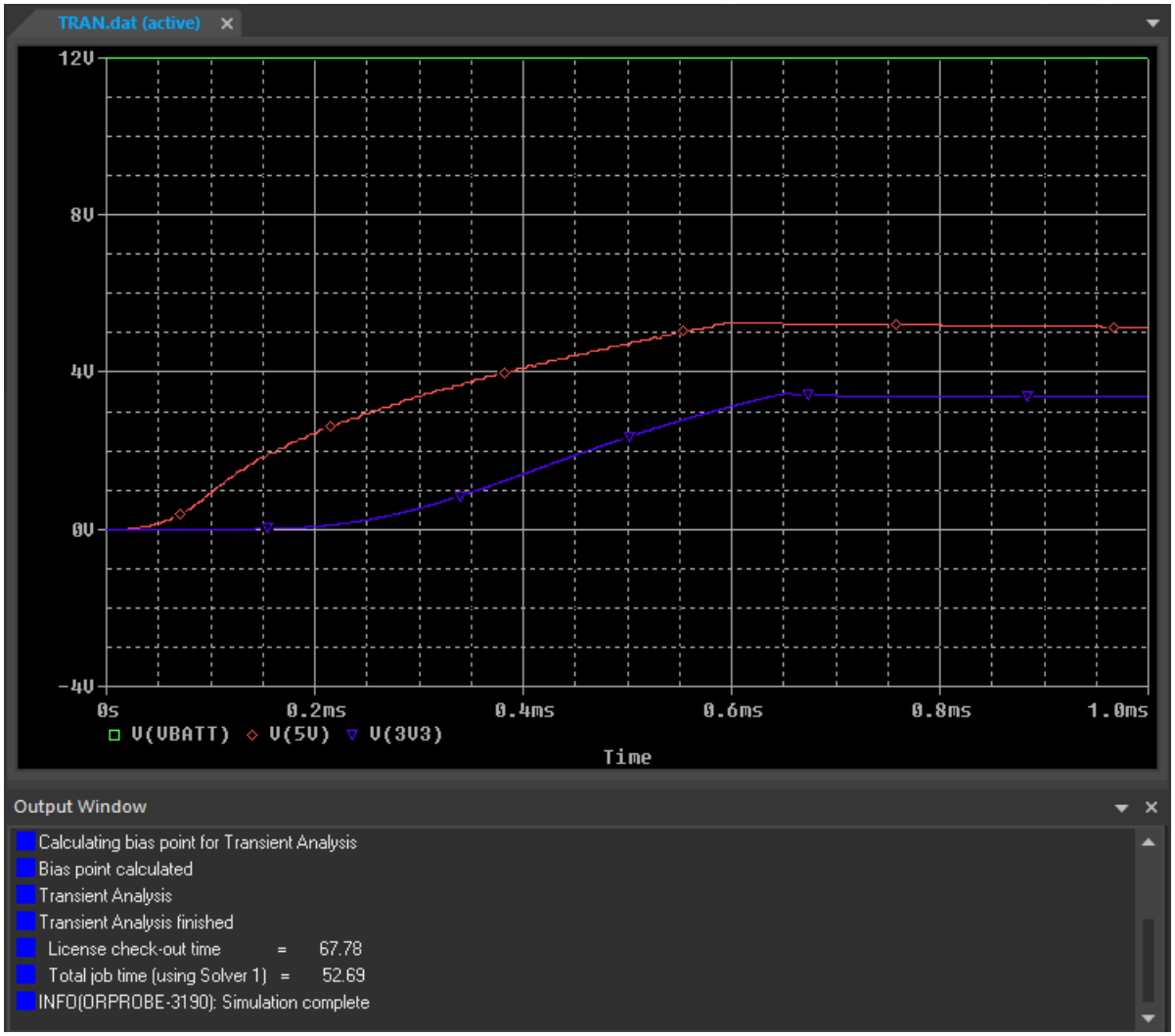
Note: If you add markers before simulating the design, the output waveforms are displayed automatically in the Probe window after the simulation is complete.

5. To view the output waveform at the marker location, double-click the marker.

   The output waveform appears in the Probe window in PSpice as shown in the following figure.

© 2020
All Rights Reserved.

Figure 3-4 Output Waveform - Probe window

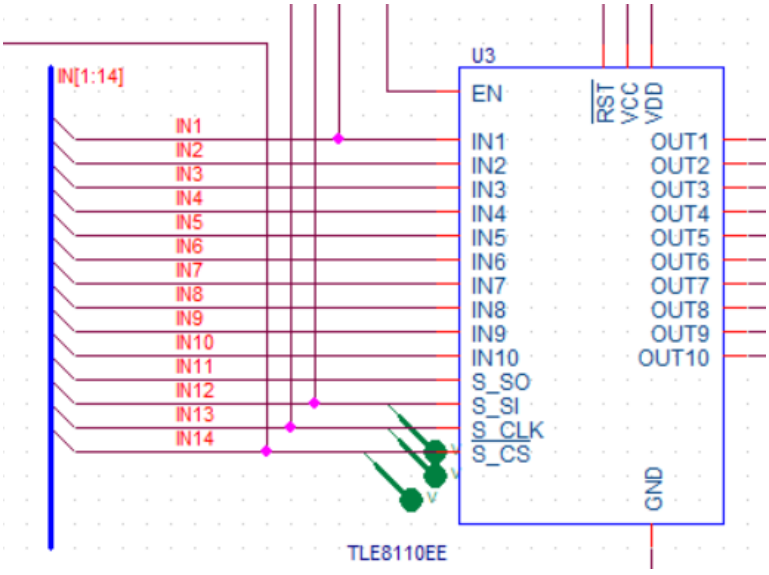


6. To view the waveform of the digital stimulus sources at the input of the switching IC `TLE8110EE`, place voltage markers on the following nets:

   - ❑ `IN12` (connected to pin S_SI)
   - ❑ `IN13` (connected to pin S_CLK)

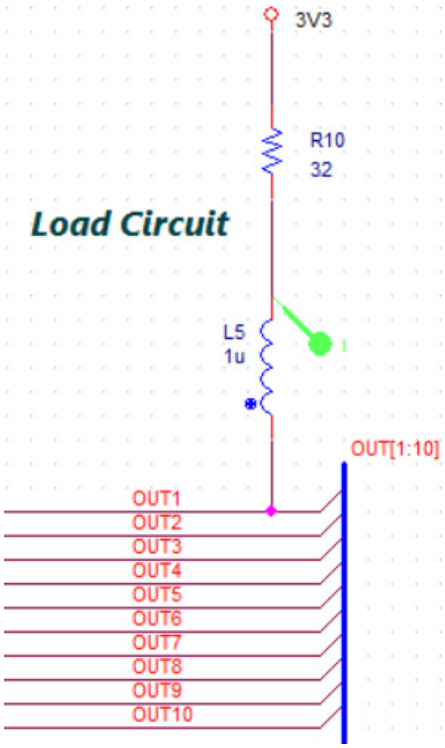
© 2020
All Rights Reserved.

- ❑ `IN14` (connected to pin S_CS)



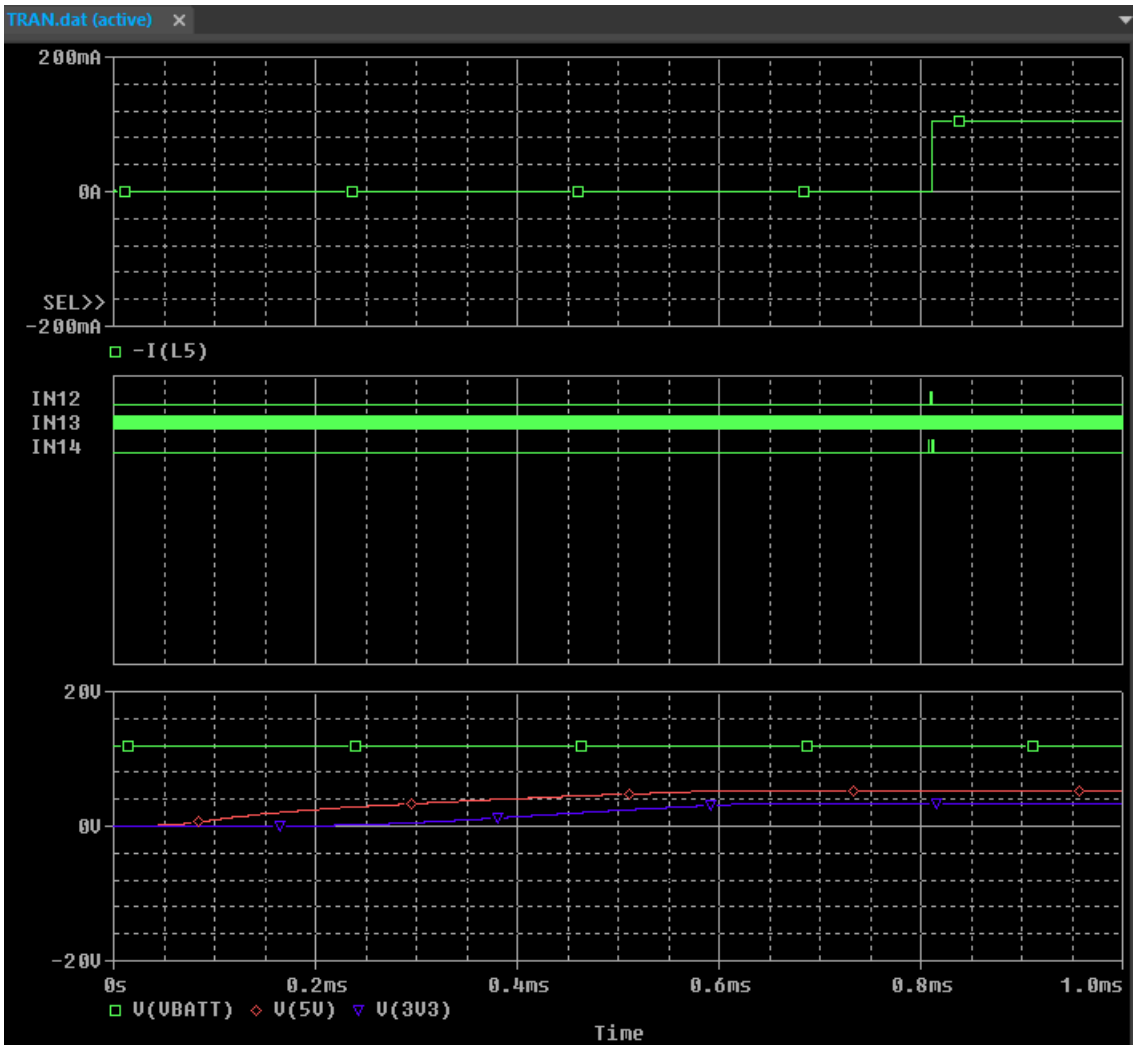
7. In PSpice, select *Plot – Add Plot to Window*.

8. In Capture, place a current marker (using the Current Marker icon, ) on the pin of the inductor as shown in the following figure.



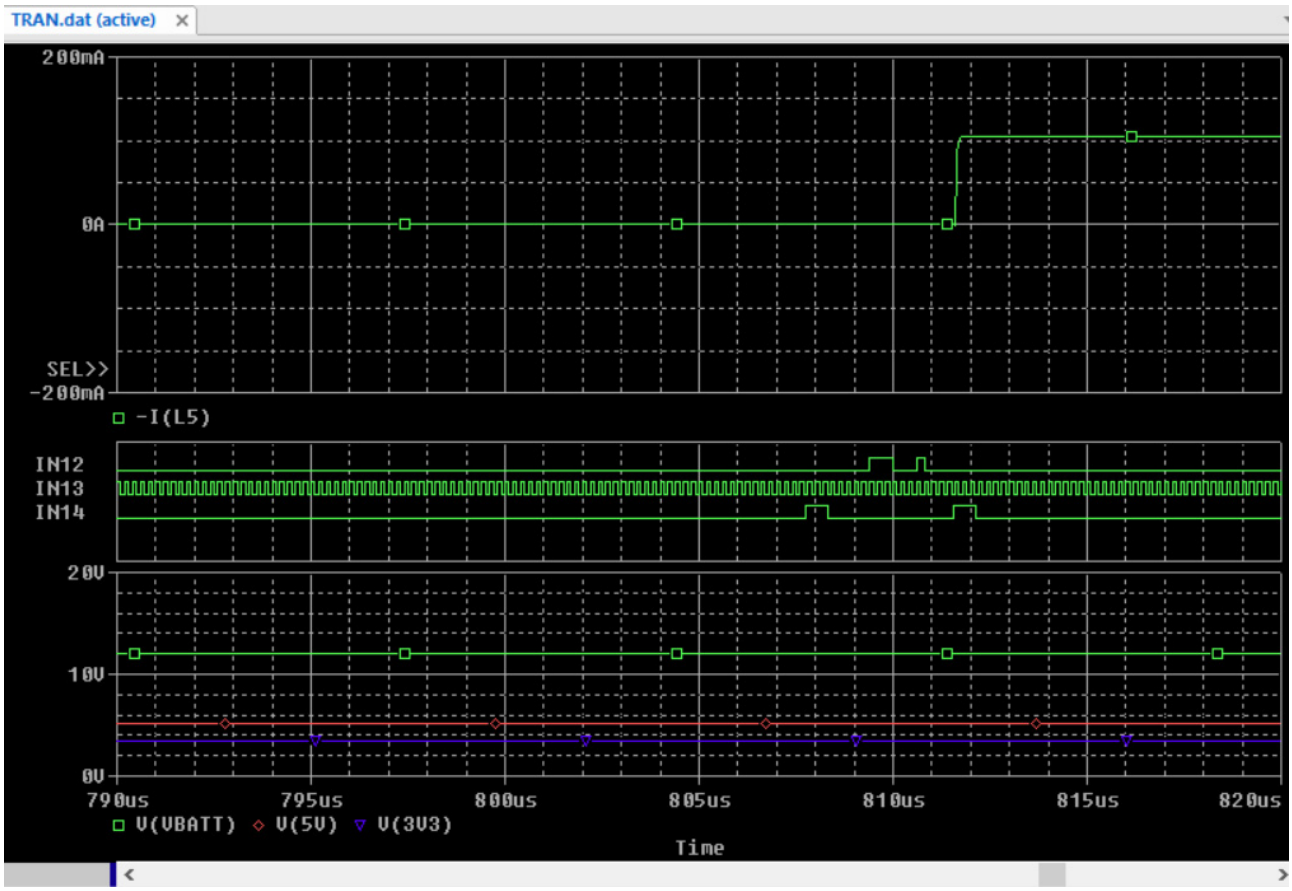
All Rights Reserved.

The changes in the waveform are displayed in the following figure.



© 2020
All Rights Reserved.

The command to turn on the fan is sent to the `S_SI` pin. This command is read and executed at the low to high transition of the `S_CS` signal. The input signal through the `S_SI` pin is sent from the duration `808us` to `811us`, as shown in the following figure.



Summary

This chapter covered the steps for simulating the fan module design using OrCAD PSpice. In this chapter, you were introduced to various tasks involved in the simulation process, such as creating a simulation profile, running simulation, placing markers, and analyzing simulation results.

© 2020
All Rights Reserved.

© 2020
All Rights Reserved.

4

Preparing for PCB Layout Creation

Now that you have verified the performance of your logical circuit through simulations, you can start designing the physical layout of the PCB board for this schematic design.

Before you create the PCB layout for this schematic design, you need to ensure that the design has no open or unconnected signal, footprint information is available for all components, and electrical constraints, if any, are specified.

Adding and Placing Connectors

To connect the fan module with a system, connector components are required to be placed in the schematic design.

To add and place connectors in this schematic design, do the following:

1. Open Capture.
2. Select *Place – Part*, press *P*, or click the Place part icon ( ).

   The *Place Part* pane opens.
3. To add `Connector.olb` to the project, click the *Add Library* icon ( ).

   The Browse File dialog box opens.
4. Browse to `<installation_directory>\tools\capture\library\Connector.olb`.
5. Select `Connector.olb` and click *Open*, or double-click `Connector.olb`.

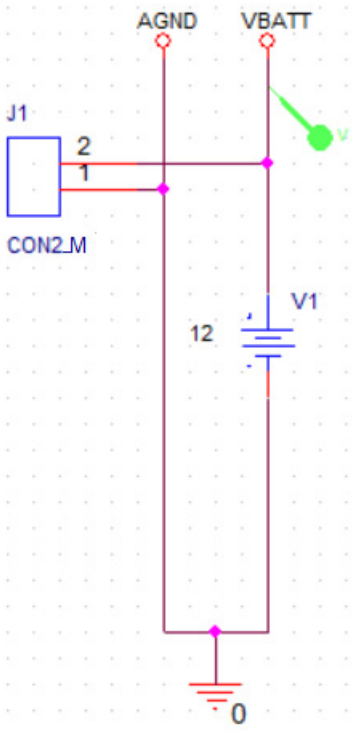
   The `CONNECTOR` library appears in the *Libraries* list box.
6. Search for `CON2_M` from the *Part* list box.
7. Click the *Place Part* icon ( ) or press *Enter*.

   The part symbol is attached to the pointer.

© 2020 All Rights Reserved.

8. Click the schematic page where you have placed the 12 volt DC source and place the connector `J1`.
9. Right-click and select *End Mode* or press *Esc*.
10. Right-click this connector and select *Rotate* and the connect it as shown in Figure on page 44.

Figure 4-1 Connector at the main power supply



Similarly, add a 14-pin connector (`CON14_M`) to the input and output channels of the smart multi-channel switch IC.

To add the connectors in the smart multi-channel switch circuit, do the following:

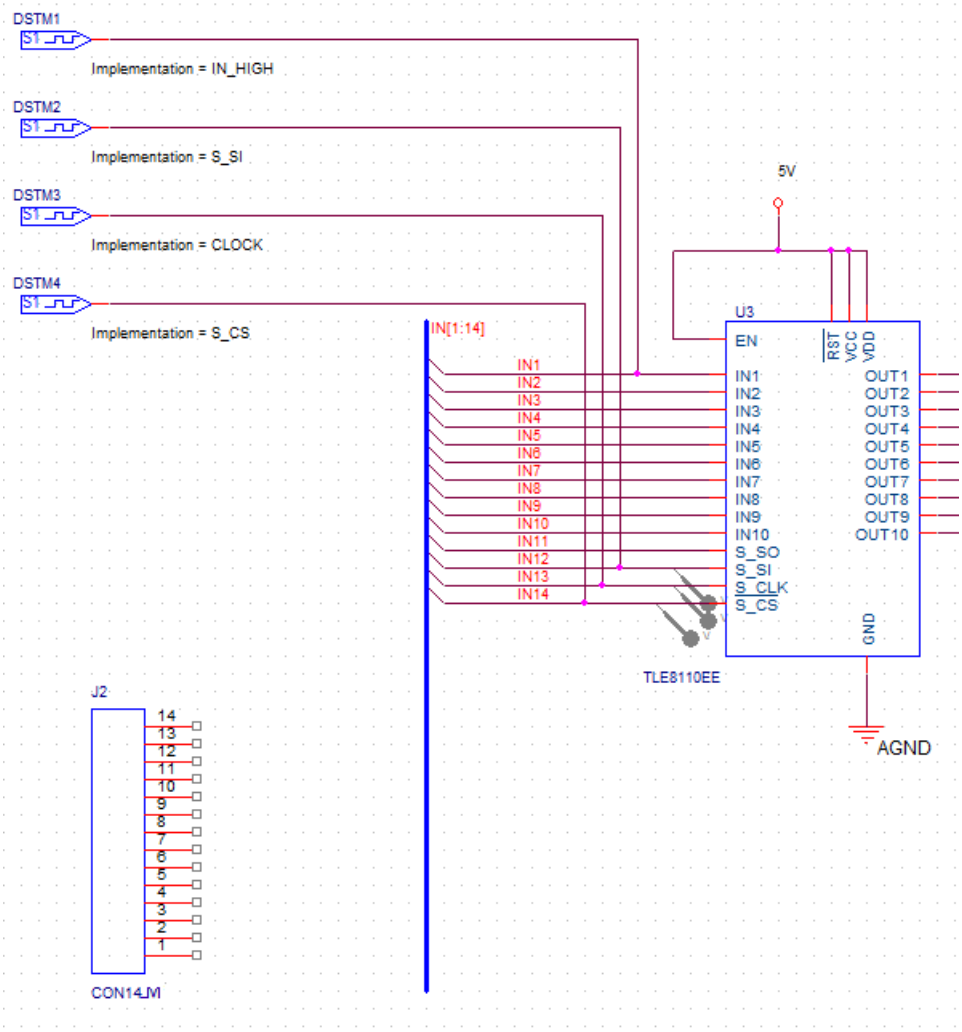
1. In the *Place Part* pane, search and select `CON14_M` from the *Part* list box.
2. Click the *Place Part* icon ( ) or press *Enter*.

   The part symbol is attached to a pointer.
3. Click the schematic page before the IC `TLE8110EE` and place the connector `J2` as shown in Figure 4-2 on page 45.
4. Right-click `J2` and select *Rotate*.

© 2020
All Rights Reserved.

5. Extend the bus before the input pins of `TLE8110EE` as shown in Figure 4-2 on page 45.

Figure 4-2 Placing connector, J2



6. Select all the pins of connector `J2`, right-click the selection, and choose *Connect to Bus*.

   The pointer changes to a crosshair.

7. Click the bus that you had extended in step 5.

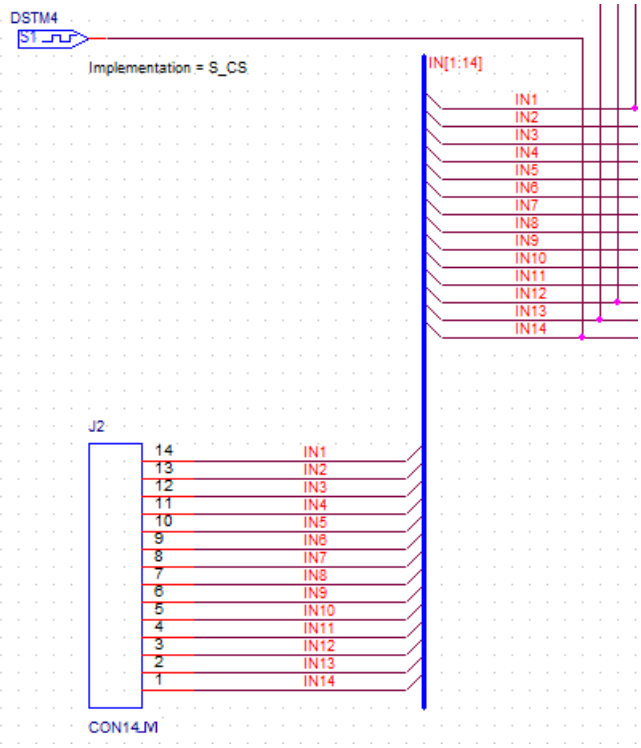
   The Enter Net Names window appears.

8. Click *OK*.

   Net names appear on each net from the connector pins to the bus.

© 2020
All Rights Reserved.

Figure 4-3 Connecting J2 to bus at the input of TLE8110EE



9. Similarly, place another connector `J3` and do the following:

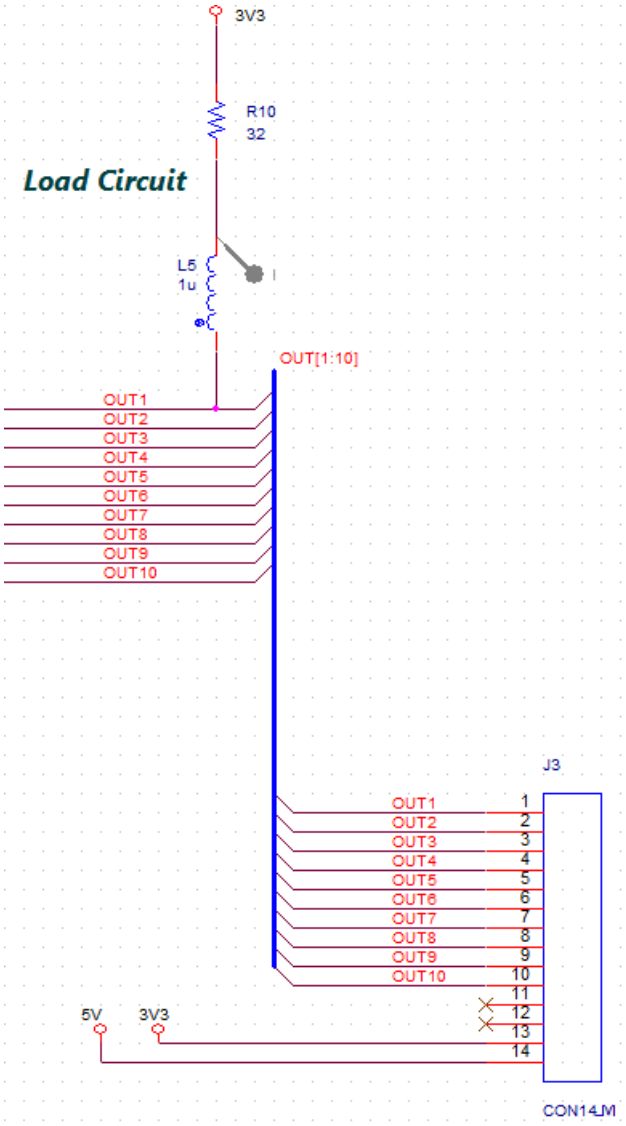
   a. Connect its first 10 pins as shown in Figure 4-4 on page 47.

   b. Click the No Connect icon ( ) or press *X*, and connect it to pins 11 and 12 of connector `J3`.

   c. Connect `3V3` and `5V` power ports to pins 13 and 14 of connector `J3`.

© 2020
All Rights Reserved.

Figure 4-4 Connecting J3 to bus at the output of TLE8110EE



All Rights Reserved.

Updating Footprints

As the first step, update footprint associated with all the resistors in the design.

Updating Footprints associated with Resistors

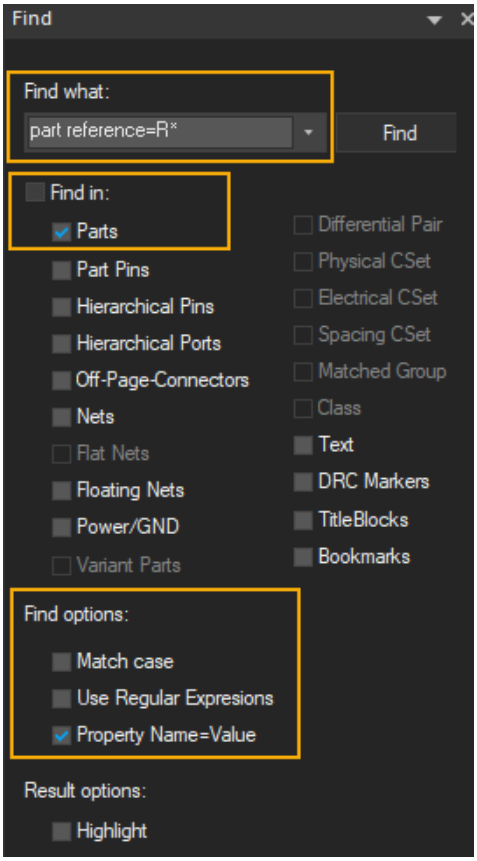
To assign footprints to all the resistors, do the following:

1. Select *Edit – Find* or press `CTRL+F`.

   The Find pane appears.

2. Specify `part reference=R*` in the *Find what* field.
3. Select the *Parts* check box under *Find in*.
4. Select the *Property Name=Value* check box under *Find options*.

Figure 4-5 Specifying search criteria in Find pane

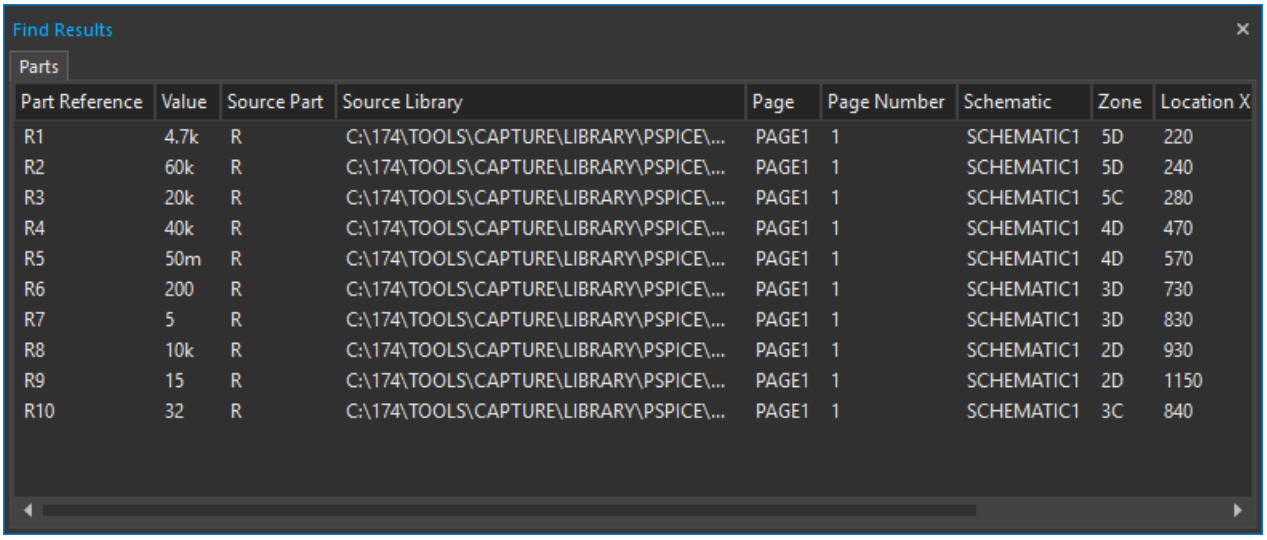


© 2020
All Rights Reserved.

5. Click the *Find* button.

   The Find Results window appears with all the resistors in the design.

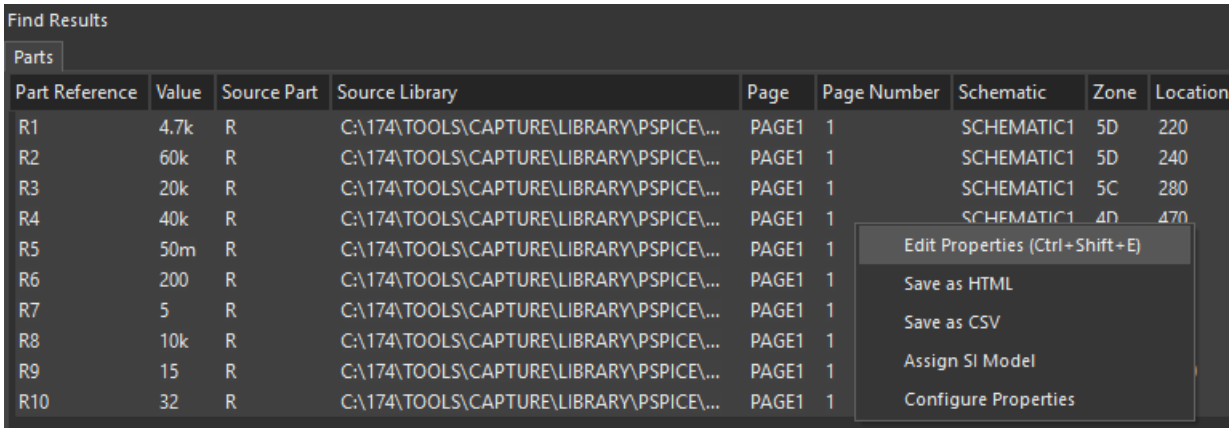
Figure 4-6 Viewing search results in Find Results window



6. To select all rows in the search result, click the first search results row, press `SHIFT` and then click the last search results row.

7. To modify the properties of the selected search results, right-click the selection and choose *Edit Properties* or press `CTRL+SHIFT+E`.

Figure 4-7 Editing properties of search results

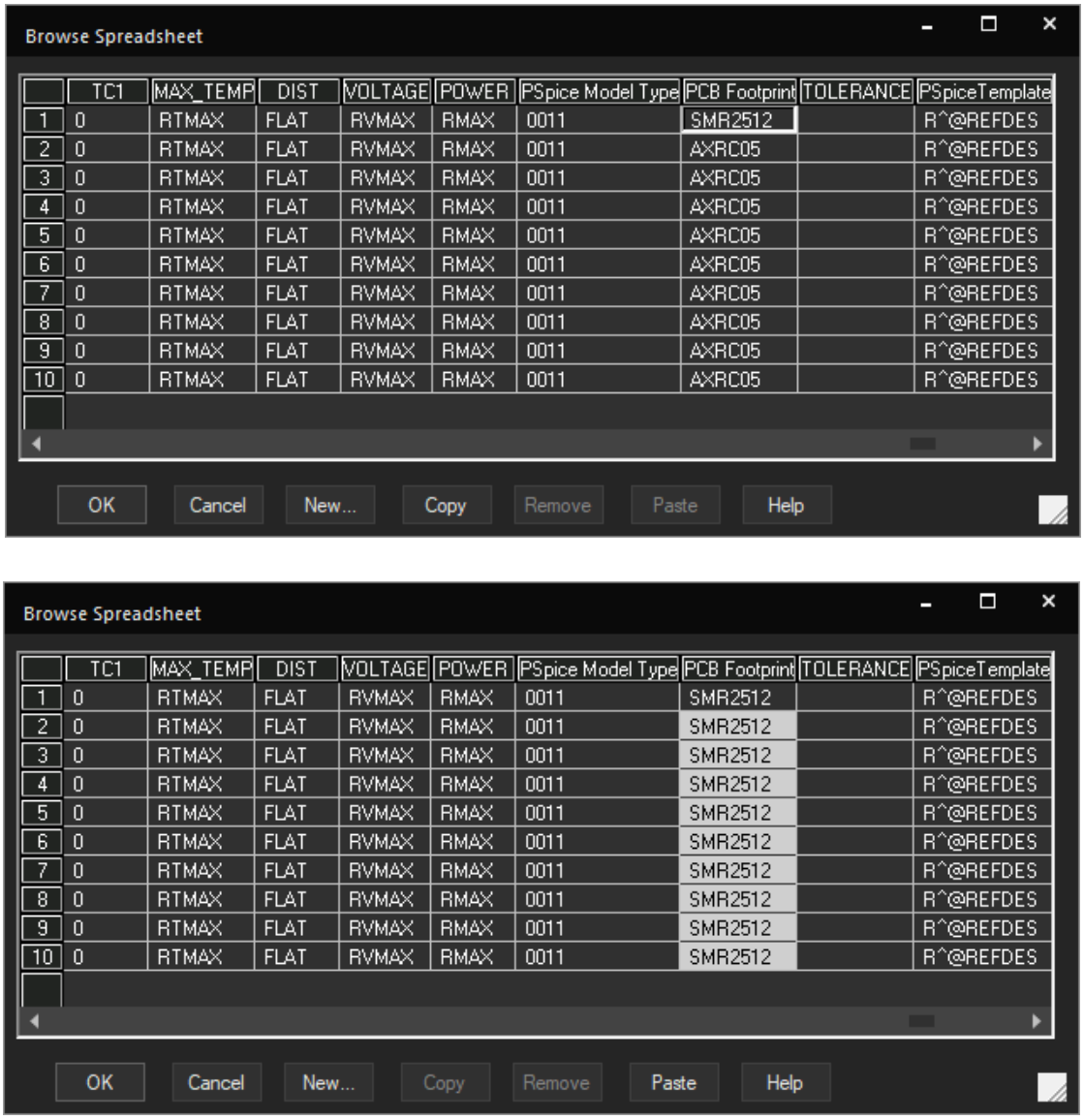


The Browse Spreadsheet window opens.

© 2020
All Rights Reserved.

8. Change the value from `AXRC05` to `SMR2512` for *PCB Footprint* corresponding to each resistor.

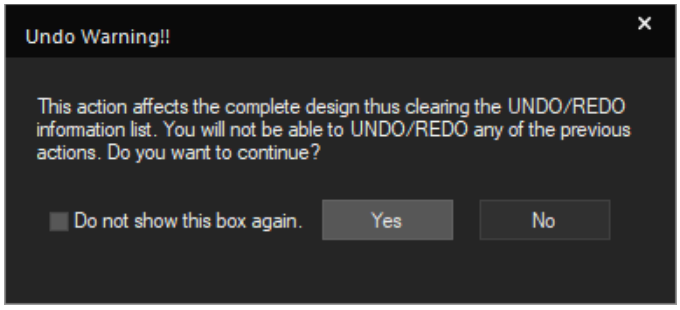
Figure 4-8 Changing PCB Footprint values in Browse Spreadsheet window



9. Click *OK*.

© 2020
All Rights Reserved.

An undo warning appears to confirm the change.



10. Select the *Do not show this box again* check box and then click *Yes*.

Updating Footprints associated with Capacitors

To update footprints associated with all the capacitors in the design, do the following:

1. Repeat step 2 to step 10 listed in the section, Updating Footprints associated with Resistors with the following changes:
   - In step 2, specify `part reference=C*` in the *Find what* field.
   - In step 8, change the value to `SMC0603` for *PCB Footprint* corresponding to each capacitor.
2. Save the design.

Updating Footprints associated with Inductors

To update footprints associated with all the inductors in the design, do the following:

1. Repeat step 2 to step 10 listed in the section, Updating Footprints associated with Resistors with the following changes:
   a. In step 2, specify `part reference=L*` in the *Find what* field.
   b. In step 8, change the value to `SML0805` for *PCB Footprint* corresponding to each inductor.
2. Save the design.

© 2020
All Rights Reserved.

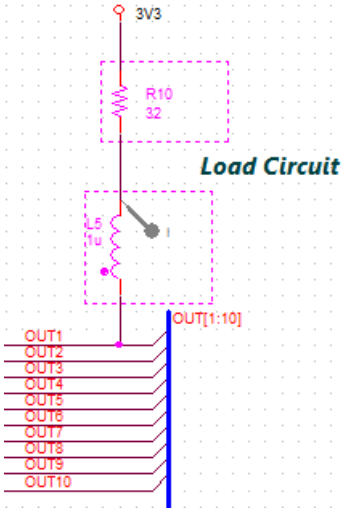
Configuring the PSpiceOnly Property

These components are added only to represent the fan loads and are not required for the physical layout. To ignore them in board design, the `PSpiceOnly` property is specified.

To assign the `PSpiceOnly` property to the inductor and the resistor of the load circuit, do the following:

1. Select the inductor and resistor in the load circuit.

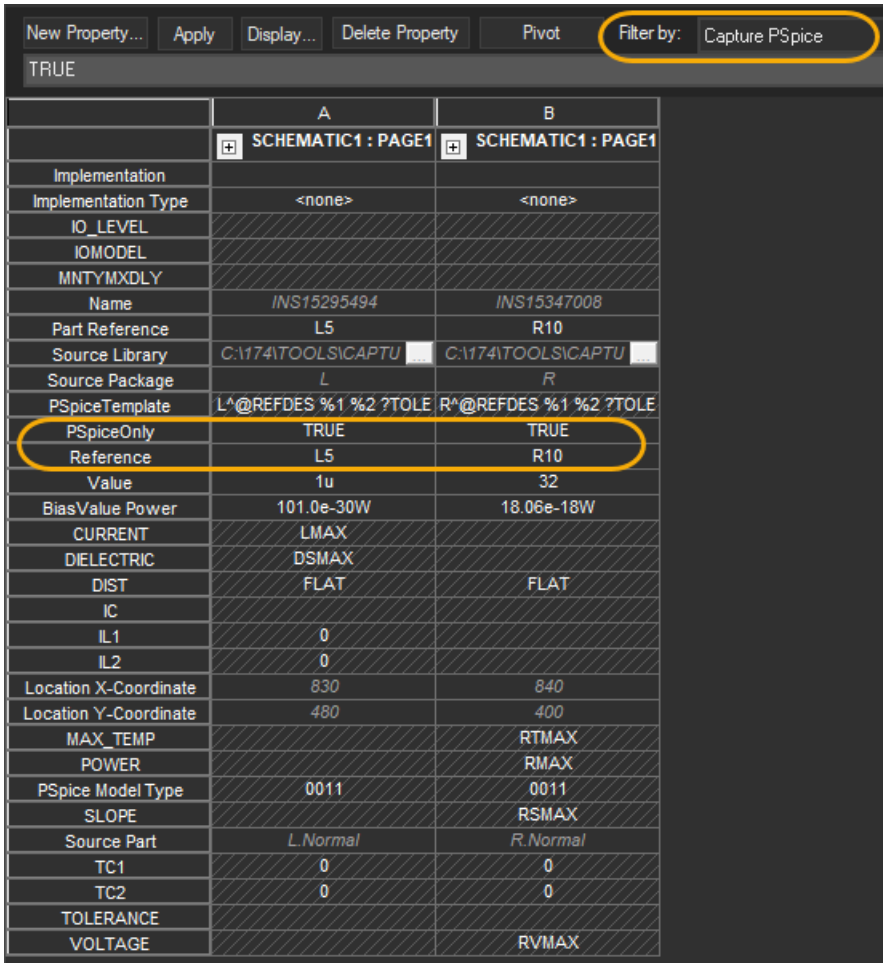
Figure 4-9 Selecting components of load circuit



2. Right-click and select *Edit Properties*.
3. Click the *Parts* tab in the *Property Editor* window.
4. From the *Filter by* drop-down list, select `Capture PSpice`.
5. Specify `TRUE` in the `PSpiceOnly` property for the selected inductor and resistor.

All Rights Reserved.

Figure 4-10 Parts tab in Property Editor window



	A	B
	SCHEMATIC1 : PAGE1	SCHEMATIC1 : PAGE1
Implementation		
Implementation Type	<none>	<none>
IO_LEVEL		
IOMODEL		
MNTYMXDLY		
Name	INS15295494	INS15347008
Part Reference	L5	R10
Source Library	C:\174\TOOLS\CAPTU	C:\174\TOOLS\CAPTU
Source Package	L	R
PSpiceTemplate	L^@REFDES %1 %2 ?TOLE	R^@REFDES %1 %2 ?TOLE
PSpiceOnly	TRUE	TRUE
Reference	L5	R10
Value	1u	32
BiasValue Power	101.0e-30W	18.06e-18W
CURRENT	LMAX	
DIELECTRIC	DSMAX	
DIST	FLAT	FLAT
IC		
IL1	0	
IL2	0	
Location X-Coordinate	830	840
Location Y-Coordinate	480	400
MAX_TEMP		RTMAX
POWER		RMAX
PSpice Model Type	0011	0011
SLOPE		RSMAX
Source Part	L.Normal	R.Normal
TC1	0	0
TC2	0	0
TOLERANCE		
VOLTAGE		RVMAX

6. Save the design.

All Rights Reserved.

Adding Constraints

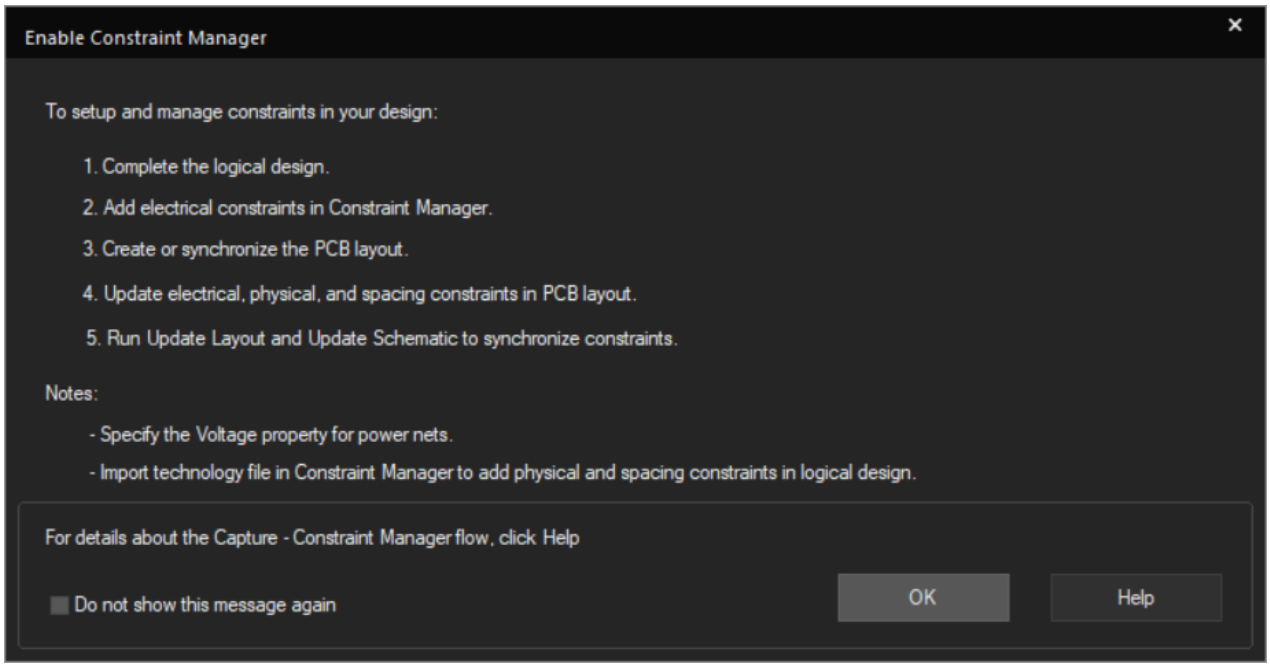
To specify the minimum value of the total etch length of each net, Constraint Manager is launched from Capture.

To add this electrical constraint in the schematic design, do the following:

1. Select *PCB – Constraints Manager* or click the CM icon ( ) on the PCB toolbar.

   An information window appears to explain the Capture-Constraint Manager flow.

Figure 4-11 Enable Constraint Manager window



2. Click *OK*.

   The Migrate Constraints dialog box appears.

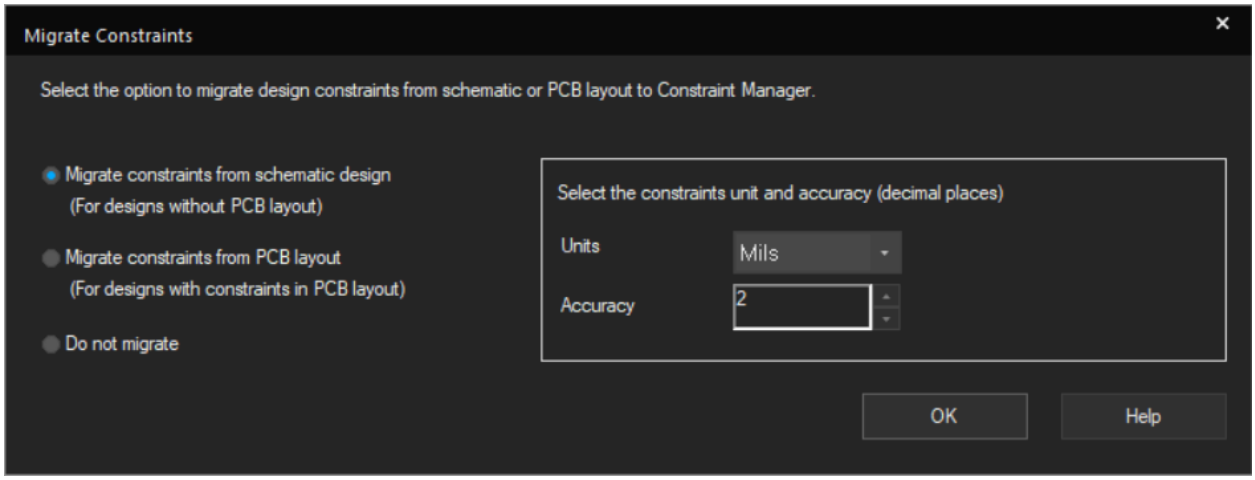
3. Select *Migrate constraints from schematic design*.

4. Specify the unit to be used for physical and spacing constraints in the Constraint Manger window.

   PCB Editor uses `Mils` as the default unit. For this tutorial, select `Mils` from the *Units* drop-down list.

© 2020
All Rights Reserved.

Figure 4-12 Using option to migrate constraints from schematic design



5. Click *OK*.

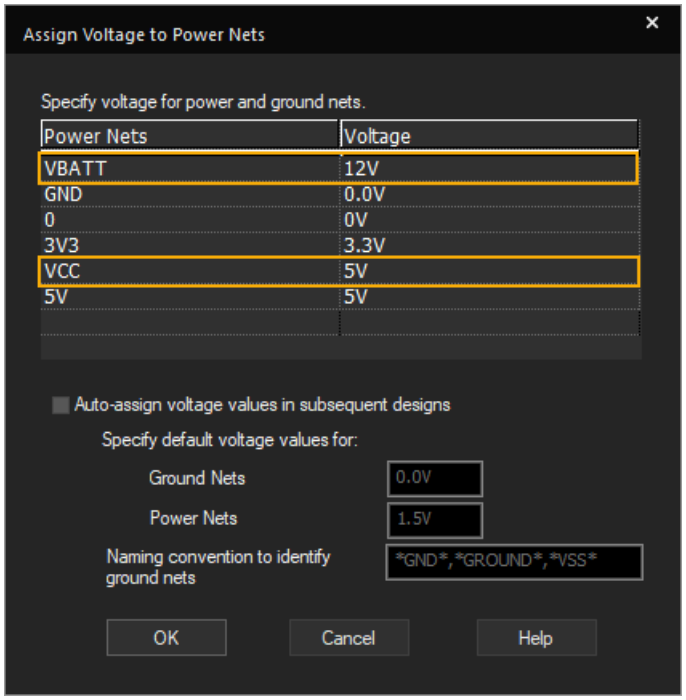
   The Assign Voltage to Power Nets window opens. This has predefined voltage values for all the power nets.

6. Modify these voltage values as follows:

   - ❑ VABTT = 12V
   - ❑ VCC=5V

© 2020
All Rights Reserved.

Figure 4-13 Modifying predefined voltage values



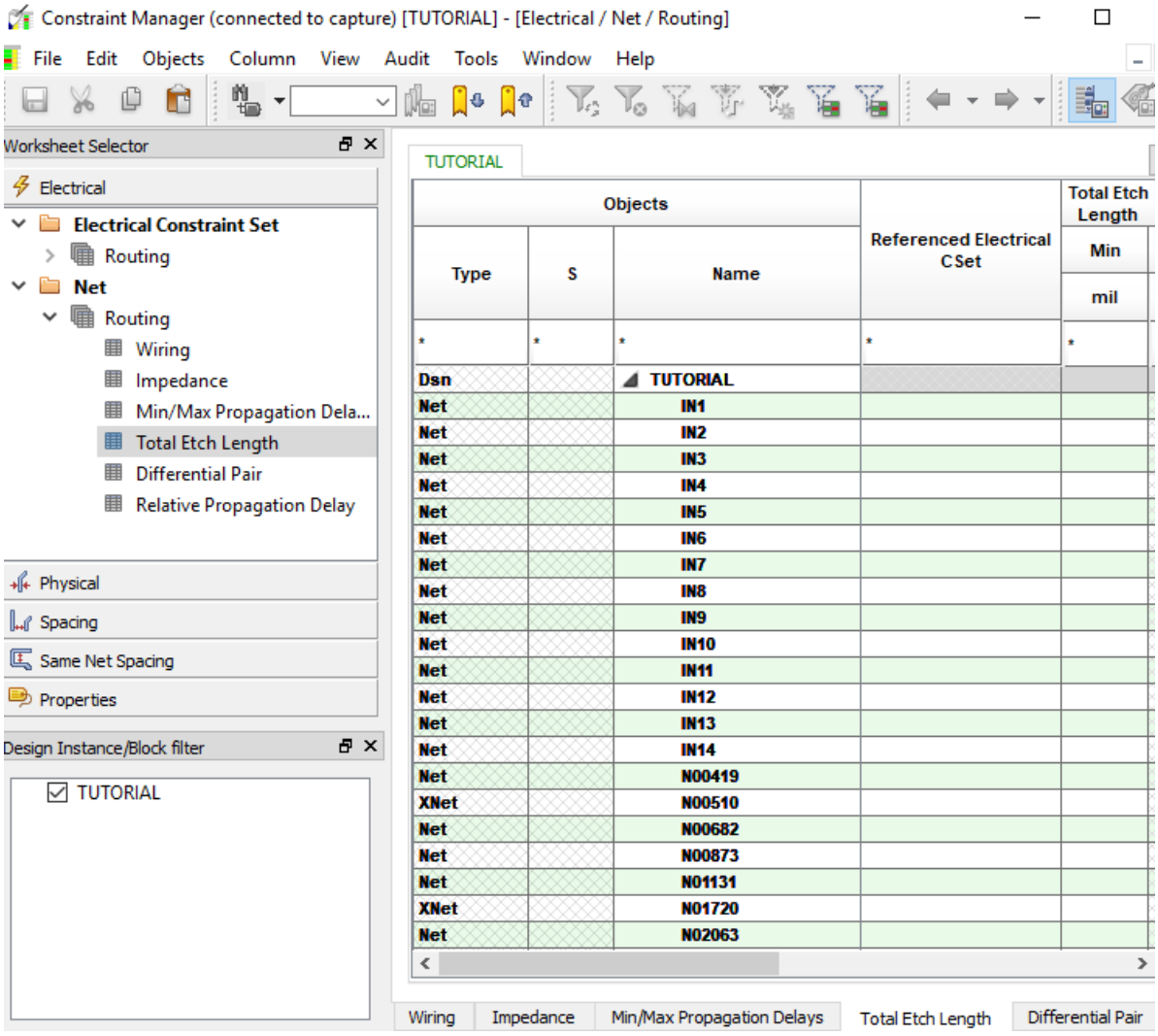
You can open this dialog box from *SI Analysis – Identify DC Nets*.

7. Click *OK*.

The Constraints Manager window opens.

© 2020
All Rights Reserved.

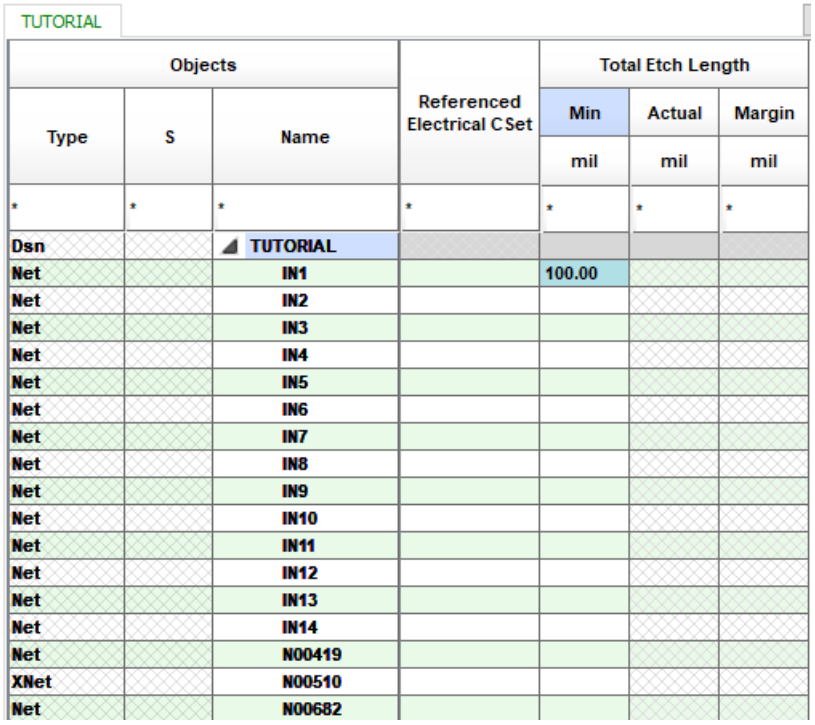
Figure 4-14 Constraint Manager window



8. Specify the minimum total etch length for the `IN1` net as `100` mils as shown in the following figure.

© 2020
All Rights Reserved.

Figure 4-15 Specifying minimum total etch length value in schematic design



TUTORIAL

Objects			Referenced Electrical CSet	Total Etch Length		
Type	S	Name		Min	Actual	Margin
				mil	mil	mil
*	*	*	*	*	*	*
Dsn		TUTORIAL				
Net		IN1		100.00		
Net		IN2				
Net		IN3				
Net		IN4				
Net		IN5				
Net		IN6				
Net		IN7				
Net		IN8				
Net		IN9				
Net		IN10				
Net		IN11				
Net		IN12				
Net		IN13				
Net		IN14				
Net		N00419				
XNet		N00510				
Net		N00682				

9. To specify the same value for *Total Etch Length* in all the nets, select the next row up till the last net in this window. Release the mouse and specify `100` in the last row.

© 2020
All Rights Reserved.

All the nets and xnets in the design are assigned the same value.

TUTORIAL

Objects			Referenced Electrical CSet	Total Etch Length		
Type	S	Name		Min	Actual	Margin
				mil	mil	mil
*	*	*	*	*	*	*
Dsn		TUTORIAL				
Net		IN1		100.00		
Net		IN2		100.00		
Net		IN3		100.00		
Net		IN4		100.00		
Net		IN5		100.00		
Net		IN6		100.00		
Net		IN7		100.00		
Net		IN8		100.00		
Net		IN9		100.00		
Net		IN10		100.00		
Net		IN11		100.00		
Net		IN12		100.00		
Net		IN13		100.00		
Net		IN14		100.00		
Net		N00419		100.00		
XNet		N00510		100.00		
Net		N00682		100.00		
Net		N00873		100.00		
Net		N01131		100.00		
XNet		N01720		100.00		
Net		N02063		100.00		
Net		N02685		100.00		
Net		N03504		100.00		
Net		N03592		100.00		
XNet		N04181		100.00		
XNet		OUT1		100.00		
Net		OUT2		100.00		
Net		OUT3		100.00		
Net		OUT4		100.00		
Net		OUT5		100.00		
Net		OUT6		100.00		
Net		OUT7		100.00		
Net		OUT8		100.00		
Net		OUT9		100.00		
Net		OUT10		100.00		

10. Save the design.

Summary

This chapter covered the steps for preparing the schematic design for designing the physical layout of the PCB board. In the process, you were introduced to tasks, such as placing

© 2020
All Rights Reserved.

connectors, adding footprint information, and adding electrical constraints using Constraint Manager.

© 2020 All Rights Reserved.

5

Creating a Board Design

Once the logic design is completed, the next step is to create the layout of the PCB. This chapter walks you through the basics of the layout creation steps, such as placement, routing, and generating output data to create a layout of the fan-control module design in the PCB Editor.

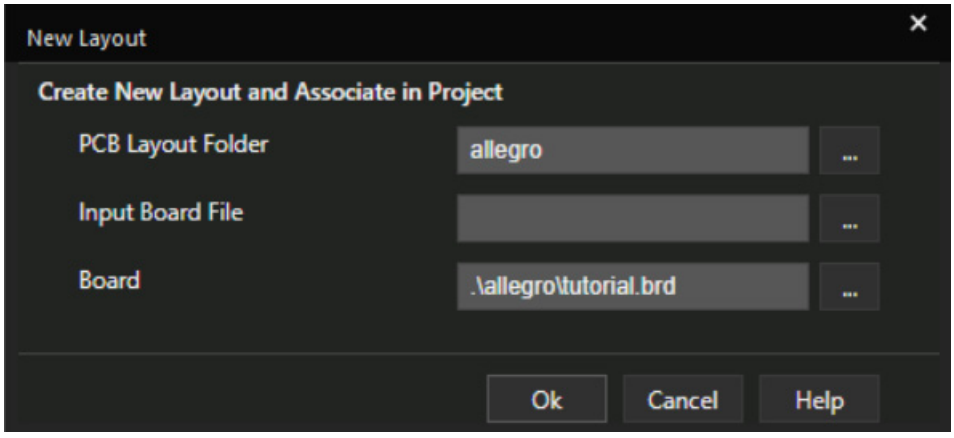
In this chapter, you will learn how to create a layout design `tutorial.brd` for the logic design you created and simulated in the previous chapters.

Creating a Blank Board

The first task is to create a new blank board design. Perform the following steps to create a blank board design:

1. Open `tutorial.opj` in OrCAD Capture.
2. Choose *PCB – New Layout*.

Figure 5-1 Creating a blank board design in Capture



The new board design will be created in the default PCB Layout folder `allegro` inside the working project directory.

© 2020
All Rights Reserved.

3. To create a blank board design, leave the *Input Board File* name field blank.
4. Specify the output board file name as `tutorial` and click *OK*.

   The Cadence 17.4 Product choices dialog box is displayed.
5. Select the *OrCAD PCB Designer Professional* w*/PSpice* option and click *OK*.

   A blank design `tutorial.brd` is opened in PCB Editor. The logic design data is transferred to the layout and is saved in the `allegro` directory.

Creating Design Outline

Design outline specifies the boundary within which components can be placed. It is necessary to create a design outline when transferring design data for ECAD-MCAD evaluation.

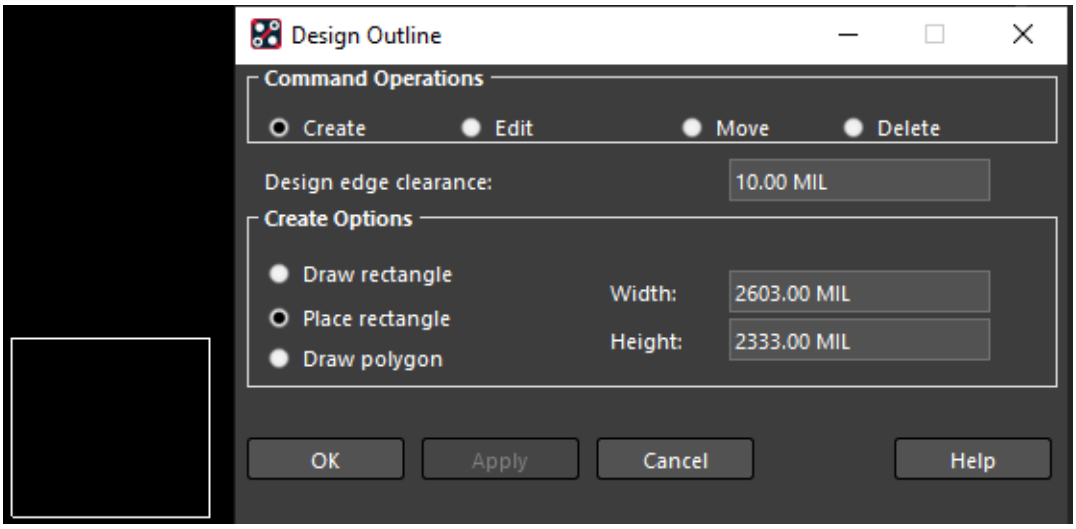
1. In PCB Editor, choose *Outline – Design*.

   The *Active Class and Subclass* fields are by default set to *Board_Geometry* and *Design_Outline* in the Options tab.
2. In the Design Outline form, select *Place rectangle* and set *Width* and *Height* values to `2603` and `2333 mils,` respectively.
3. Set *Design edge clearance* value to `10 mils`. This value defines the space between the board outline and package and route keepin boundaries which is required to accommodate manufacturing tolerances, testing, and assembly.

   A rectangular design outline is attached to the cursor.

© 2020 All Rights Reserved.

Figure 5-2 Setting up design outline parameters



4. Click anywhere in the design canvas to place the outline.
5. Click *OK* to close the Design Outline form.

   A rectangular board outline is created with package and route keepin areas.
6. To view the entire board in the design canvas, choose *Display – Zoom – Fit* to center the board outline in the design window.

© 2020
All Rights Reserved.

Figure 5-3 Design outline with package and route keepin areas

7. Choose *File – Save* to save the design.

Placing Components

After creating the design outline you can start with component placement. All types of component symbols (discrete, ICs, mechanical, and format) can be placed either manually or by using automatic placement in the PCB Editor.

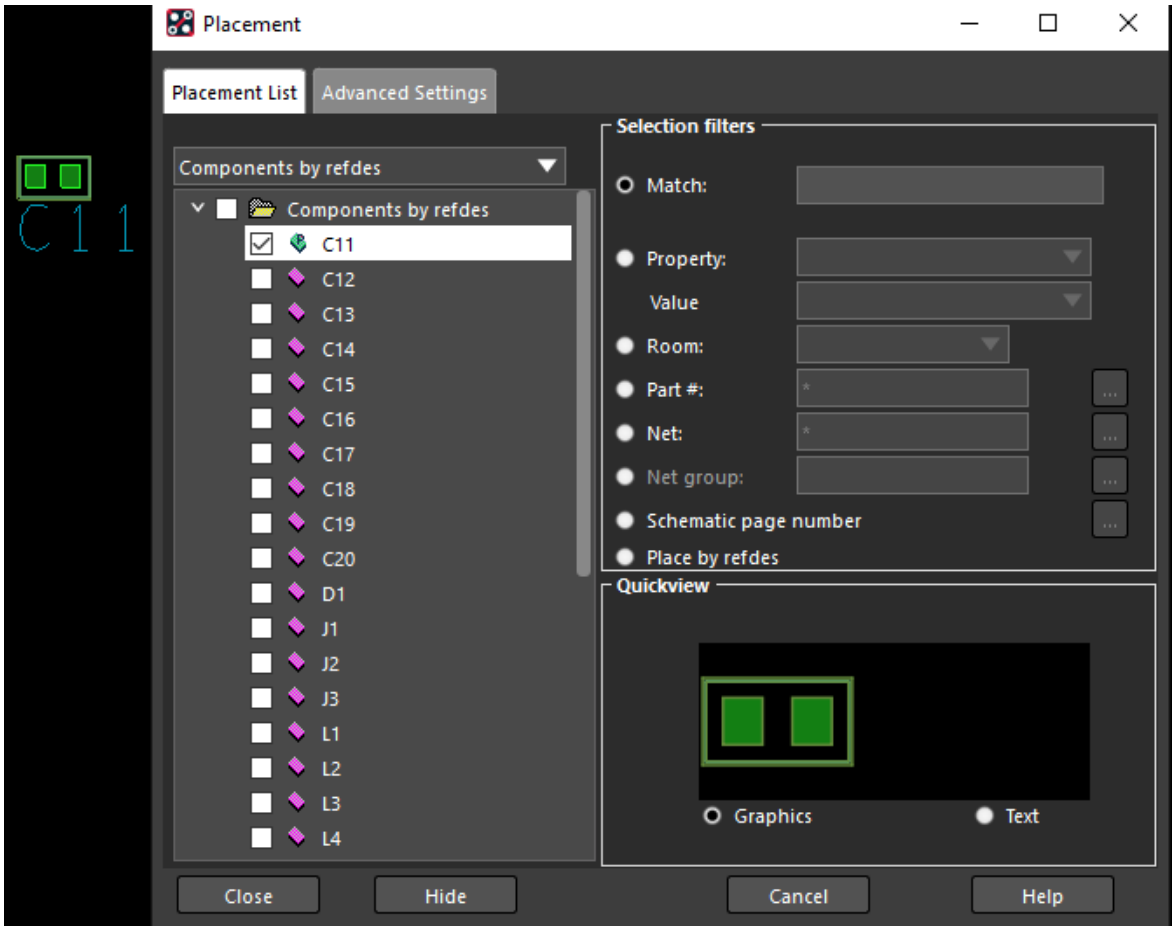
For this tutorial, place components interactively using the `place manual` command either in horizontal or vertical orientation using standard grid size of `25 mil`, which makes them easy to align and assemble on the board.

1. To start the placement, first enable the placement editing environment. Choose *Setup – Application Mode – Placement Edit*.
2. Choose *Place – Components Manually*.

   By default, Placement form shows all the components listed by their reference designators. You can choose any reference designator and place it in the design canvas.

© 2020
All Rights Reserved.

Figure 5-4 Selecting components from Placement form



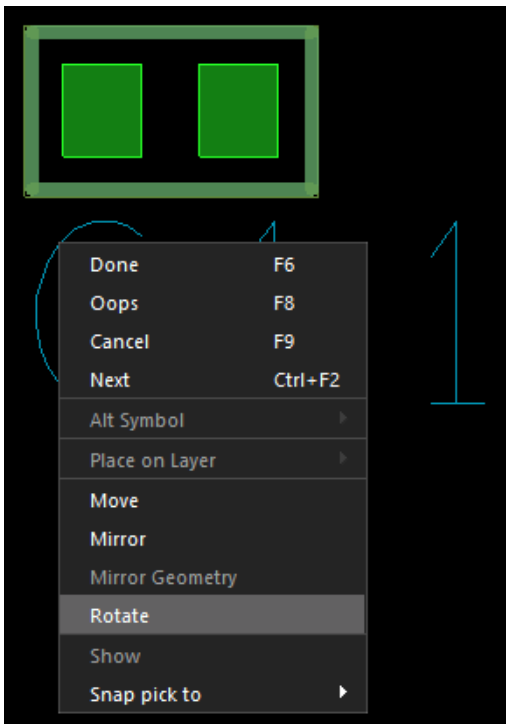
3. Select the capacitor symbol `C11`.

   The symbol gets attached to the cursor and is also visible in the *Quickview* window.

4. Right-click to choose *Rotate.* Use the handlebar to rotate the symbol by 90 degrees in anti-clockwise direction.

© 2020
All Rights Reserved.

Figure 5-5 Rotating component before placing



5. Left-click in the design canvas to place the capacitor symbol.
6. Similarly, select each reference designator one by one to place all the component symbols.

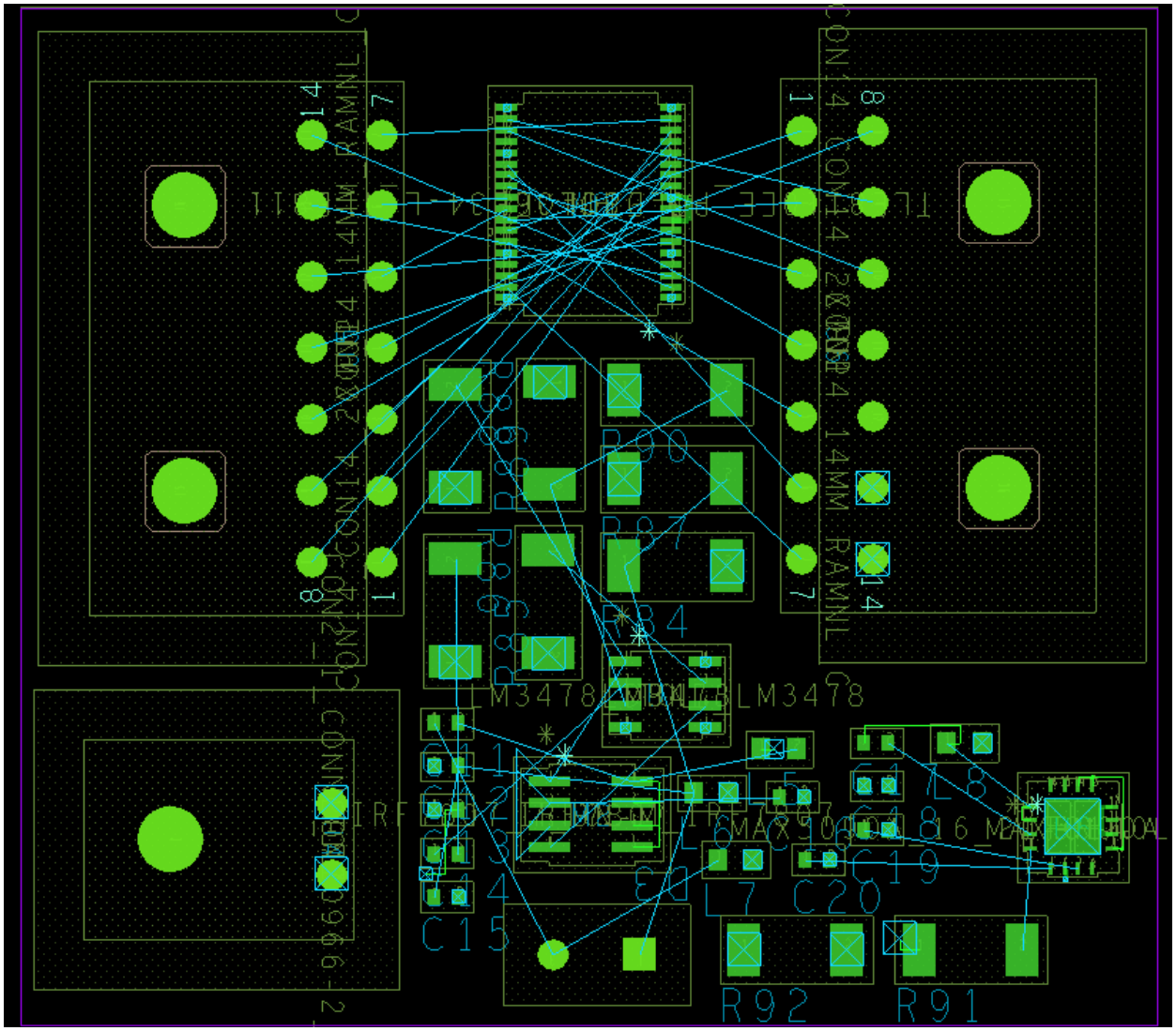
Tip

Place the symbols according to their logically connectivity. You can refer to schematic design to know which components are logically connected and must be placed closely.

The following image depicts a sample placement.

© 2020
All Rights Reserved.

Figure 5-6 Sample placement



7. Choose *File – Save* to save the design.

Note: The placement is the most important step of the PCB design and impact all the subsequent steps of the layout design. If you choose to place components differently, the final routed board will also be different.

© 2020
All Rights Reserved.

Setting Up Constraints

To meet manufacturing requirements you can configure the default constraint values in Constraint Manager either by creating constraint sets or by directly modifying the constraint values.

Before routing the board, specify the design constraints for routing. For this tutorial, specify two constraints for routing power nets: minimum thickness of the cline and maximum length of cline in neck mode.

1. Choose *Setup – Constraints.*
2. In the *Physical* domain, choose *Net – All Layers.*
3. In the *All Layers* worksheet, select the net `5V`.
4. Change the value of *Min Line Width* to `15 mil` and *Max Neck Length* to `100 mil`.
5. Similarly, select the nets `3V3` and `0` and change the values for *Min Line Width* and *Max Neck Length.*

   New values are displayed in blue color for the power nets. When routing, any violation to these values will create a DRC error.

Figure 5-7 Setting overrides for physical constraints

Objects			Referenced Physical CSet	Line Width		Neck	
Type	S	Name		Min	Max	Min Width	Max Length
				mil	mil	mil	mil
*	*	*	*	*	*	*	*
Net		OUT10	DEFAULT	5.00	0.00	5.00	0.00
Net		VBATT	DEFAULT	5.00	0.00	5.00	0.00
Net		0	DEFAULT	15.00	0.00	5.00	100.00
Net		3V3	DEFAULT	15.00	0.00	5.00	100.00
Net		5V	DEFAULT	15.00	0.00	5.00	100.00

6. Close Constraint Manager.
7. Choose *File – Save* to save the design.

© 2020
All Rights Reserved.

Routing Nets

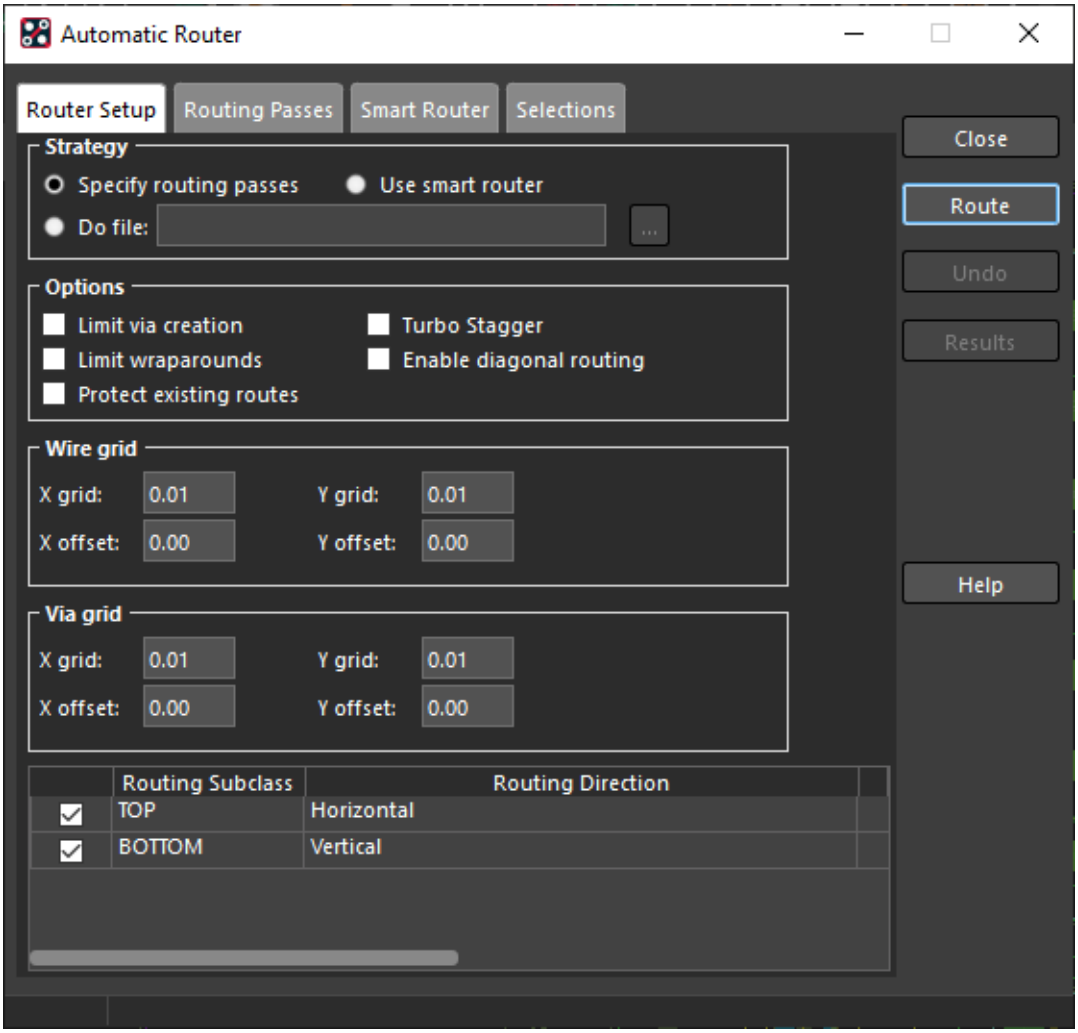
Connecting the components in a layout is called routing. PCB Editor provides both manual and auto routing capabilities.

For this tutorial, use smart router for complete auto-routing.

1. Choose *Route – PCB Router – Route Automatic.*

   The Automatic Router opens and shows options to configure parameters for routing. For this tutorial, use the default values.

Figure 5-8 Automatic Router setup form



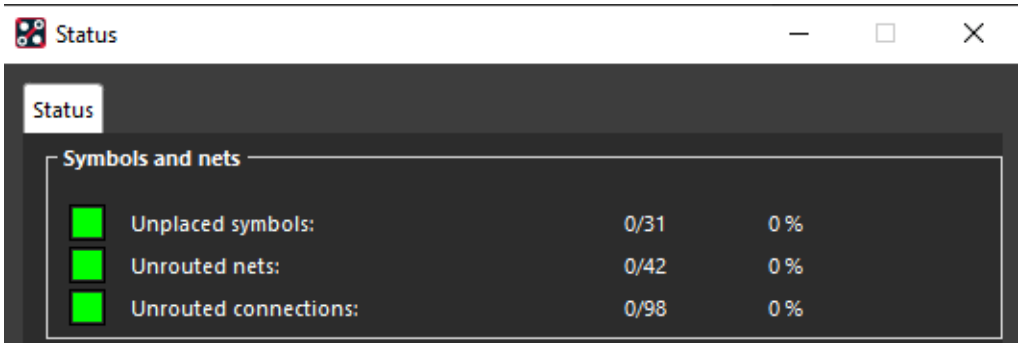
2. To start auto-routing, click the *Route* button.

© 2020
All Rights Reserved.

The auto-route process starts and takes a few seconds to complete. The auto-router considers the design outline and keepin areas while routing nets.

3. To verify that routing is completed successfully, choose *Check – Design Status*.

Figure 5-9 Design status form



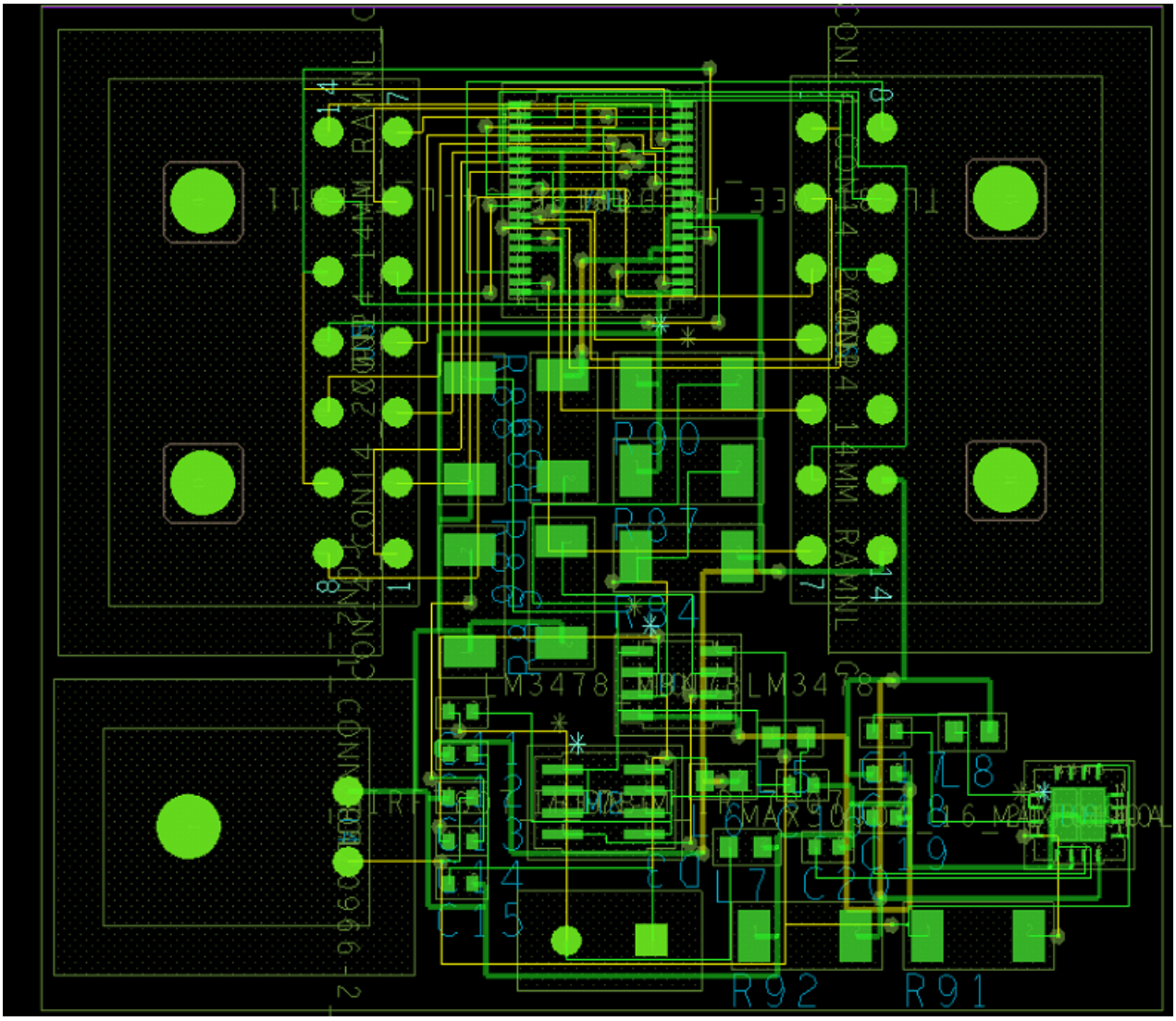
The Status window confirms that all the nets are routed.

4. Click *OK* to close the Status window.
5. Close Automatic Router.

The following image shows a routed board design.

© 2020
All Rights Reserved.

Figure 5-10 Auto-routed board design sample



6. Choose *File – Save* to save the design.

Renaming Components

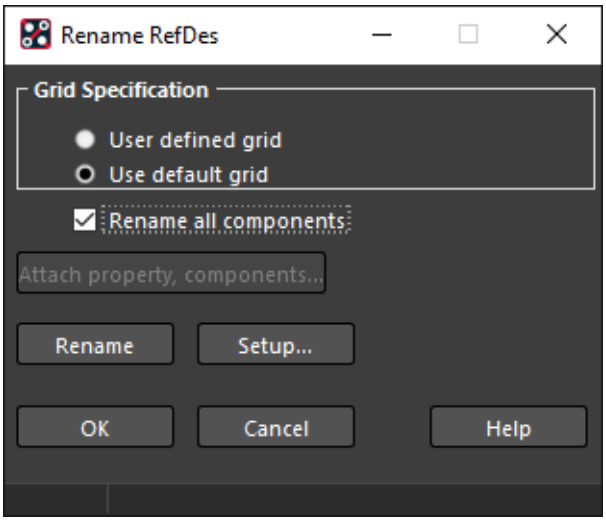
After you have completed the placement and routing you can reorder the reference designators of components on the board in a specific pattern. This steps makes the testing and assembly process easier.

1. Choose *Manufacture – Auto Rename Refdes – Rename*.

© 2020
All Rights Reserved.

The Rename Refdes form opens showing grid settings and option to select all the components for renaming action.

Figure 5-11 Rename reference designators options



2. Click the *Setup* button to specify more options.

   The Rename Ref Des Set Up form opens.

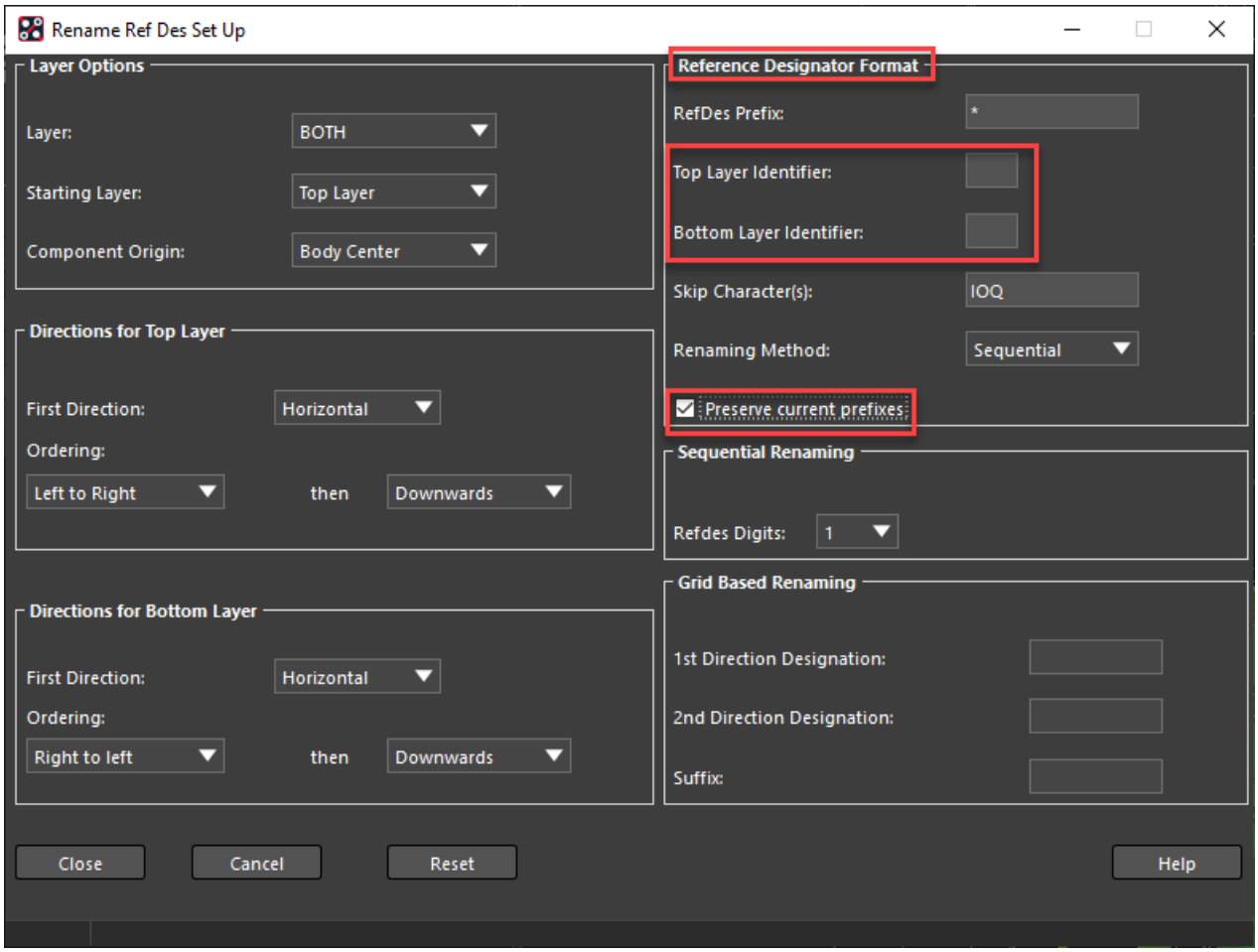
3. In the *Reference Designator Format* section:

   a. Remove layer identifier for TOP and BOTTOM layers.

   b. Enable the *Preserve Current prefixes* check box.

© 2020
All Rights Reserved.

Figure 5-12 Rename reference designator settings

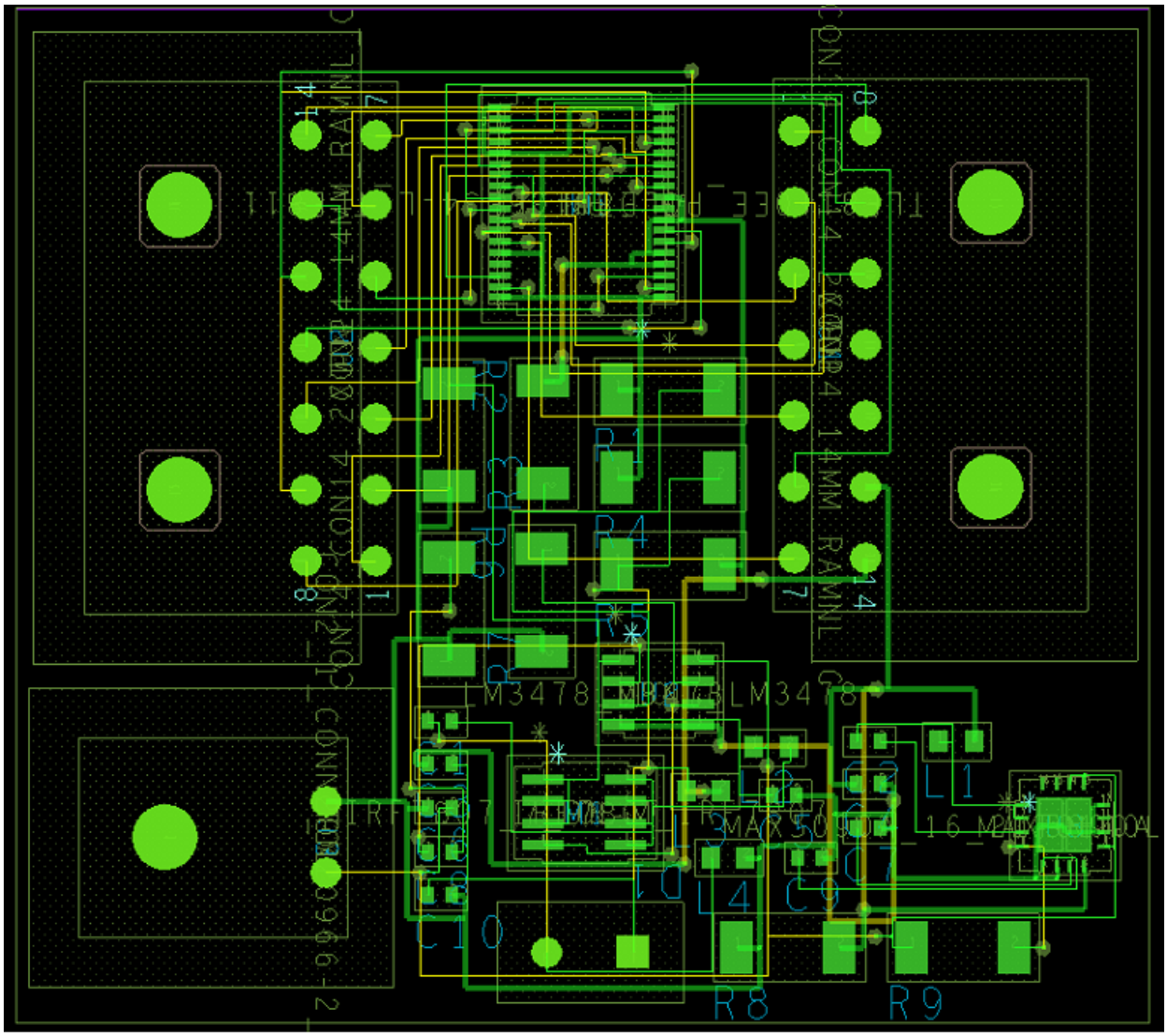


4. Close the *Rename Ref Set Up* form.
5. Click the *Rename* button in the Rename RefDes form.

   Renaming of reference designators starts from the upper-left corner of the board in the horizontal direction and the numbers are increasing in the downward direction.

All Rights Reserved.

Figure 5-13 Board design with renamed reference designators



6. Click *OK* to close the Rename RefDes form.
7. Choose *File – Save* to save the design.

© 2020
All Rights Reserved.

Visualizing Design in 3D

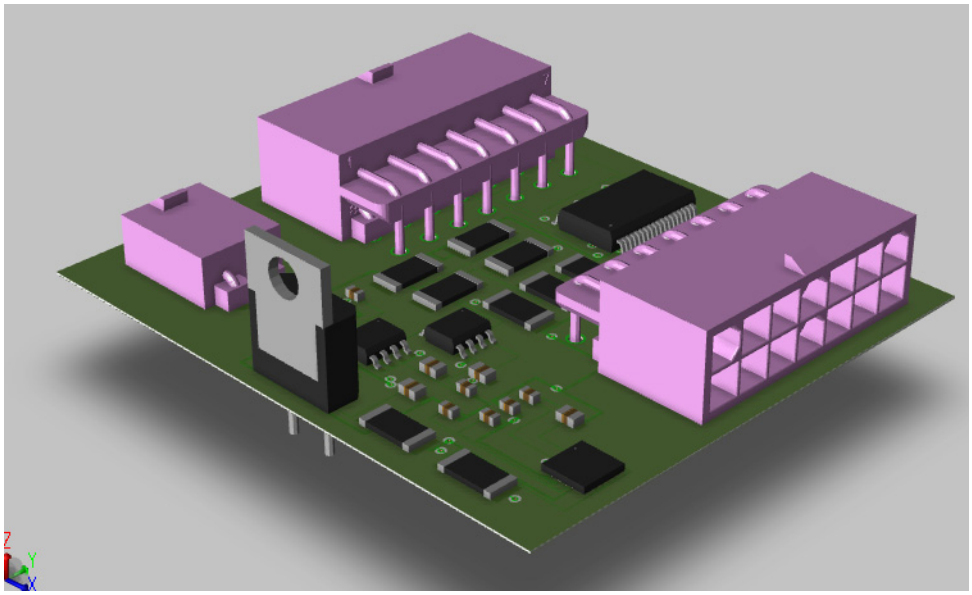
The built-in 3D visualization tool of PCB Editor lets you preview the board design at any time during the design. You can open the design in 3D Canvas and verify the design as a complete assembly.

To analyze the design in the 3D Canvas, 3D models must be assigned to all the symbols. You can map a 3D model either at a symbol level or at the design level. PCB Editor, by default, supports STEP models for 3D visualization. For the components used in this tutorial, 3D models are assigned at the symbol level.

1. Choose *Display – 3D Canvas*.

   A progress bar displays while loading the design into 3D Canvas.

Figure 5-14 3D view of board design



2. Choose *View – Camera* options to view the design in different perspectives.
3. Close 3D Canvas.

Generating Manufacturing Files

The final task is to generate various types of output files of the physical design data. You can create Gerber files, Excellon NC Drill files, DXF files, IPC2581, ODB++, and printer/plotter

© 2020
All Rights Reserved.

files.These files are standard files and are required by the fabrication houses to manufacture a PCB.

For this tutorial, create three types of output files:

- Artwork (Gerber)
- NC Drill
- IPC2581

Creating Artwork

To create artwork files, PCB Editor reads film control records to determine the number of artwork files to produce, their names, and list of classes and subclasses to include in each artwork file.

To specify classes and subclasses for an artwork file, use Color Dialog to set the visibility of required classes and subclasses.

1. Choose *Setup – Colors.*

   The Color Dialog opens.

2. In the *Layers* tab, click the *Off* button for *Global Visibility*.

   The visibility of all the classes and subclass are turned off.

3. Expand the *Stack-Up – Conductor* folder, select *Soldermask_Top* and *Pastemask_Top* layers and enable the check box for *Pin* only.

   The soldermask and pastemask layers of pins becomes visible in the design canvas.

4. Similarly, in the *Geometry* folder, select *Soldermask_Top* and *Pastemask_Top* layers and enable the checkbox for *All* objects.

   The visibility of soldermask and pastemask layer is set on both board and package geometry.

5. Click *OK* to close the Color Dialog.

6. Choose *Export – Gerber*.

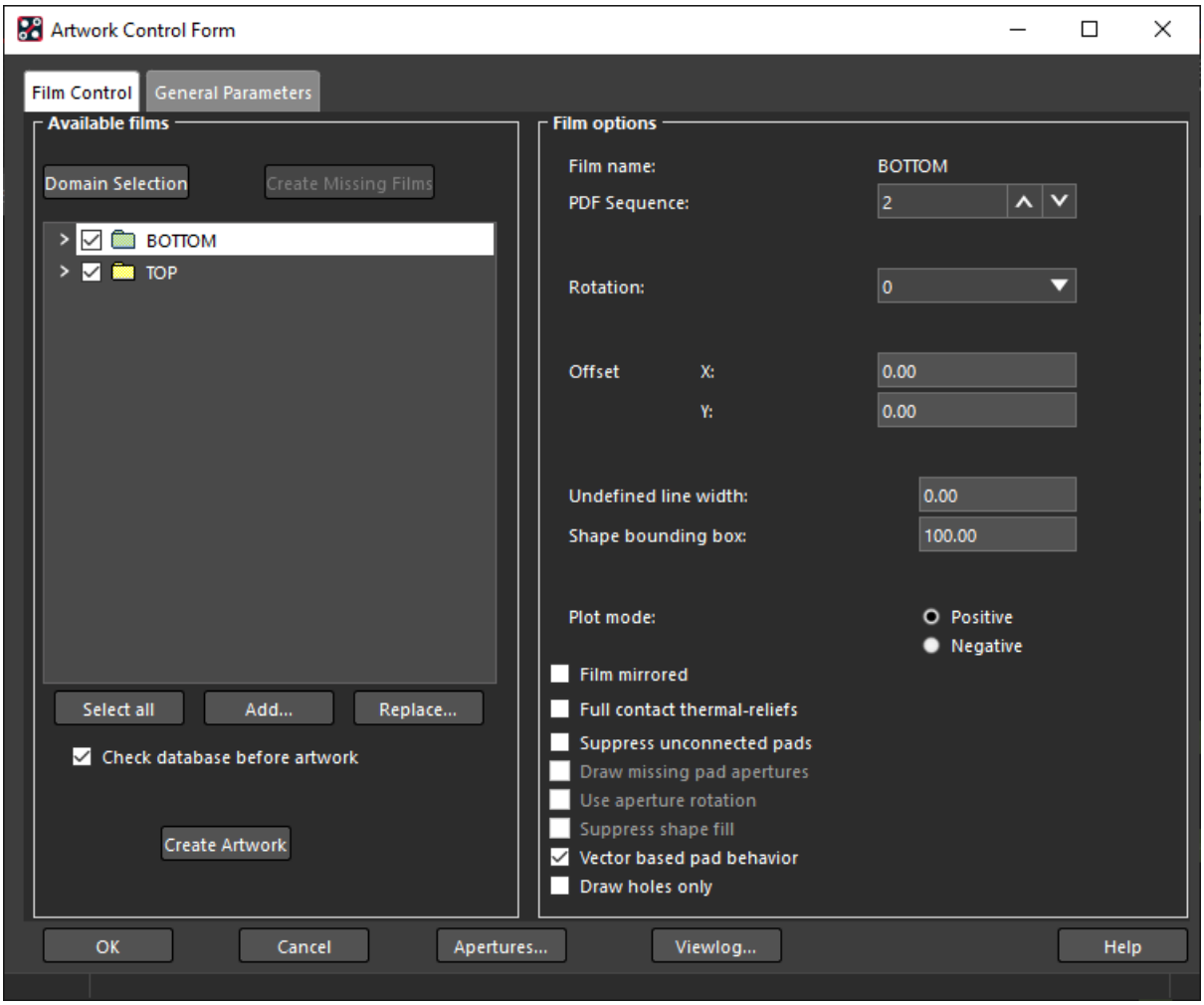
   The Artwork Control Form opens which reads the cross-section and auto-generates one film record for each etch subclass and includes etch, pins, and vias.

7. In Artwork Control Form, select both the TOP and BOTTOM layers.

8. Click *Create Artwork* to generate artwork.

© 2020 All Rights Reserved.

Figure 5-15 Artwork generation settings



Two artwork files (`TOP.art` and `BOTTOM.art`) are created in the `allegro` directory.

9. Click *Viewlog* to review the log file.
10. Click *OK* to close the Artwork Control Form.

Creating NC Drill

NC Drill output files are created for numerically-controlled (NC) drills and router and helps in assessing the cost of PCB manufacturing. The drill output files includes drill legend tables and drill files.

© 2020
All Rights Reserved.

Generating Drill Legend

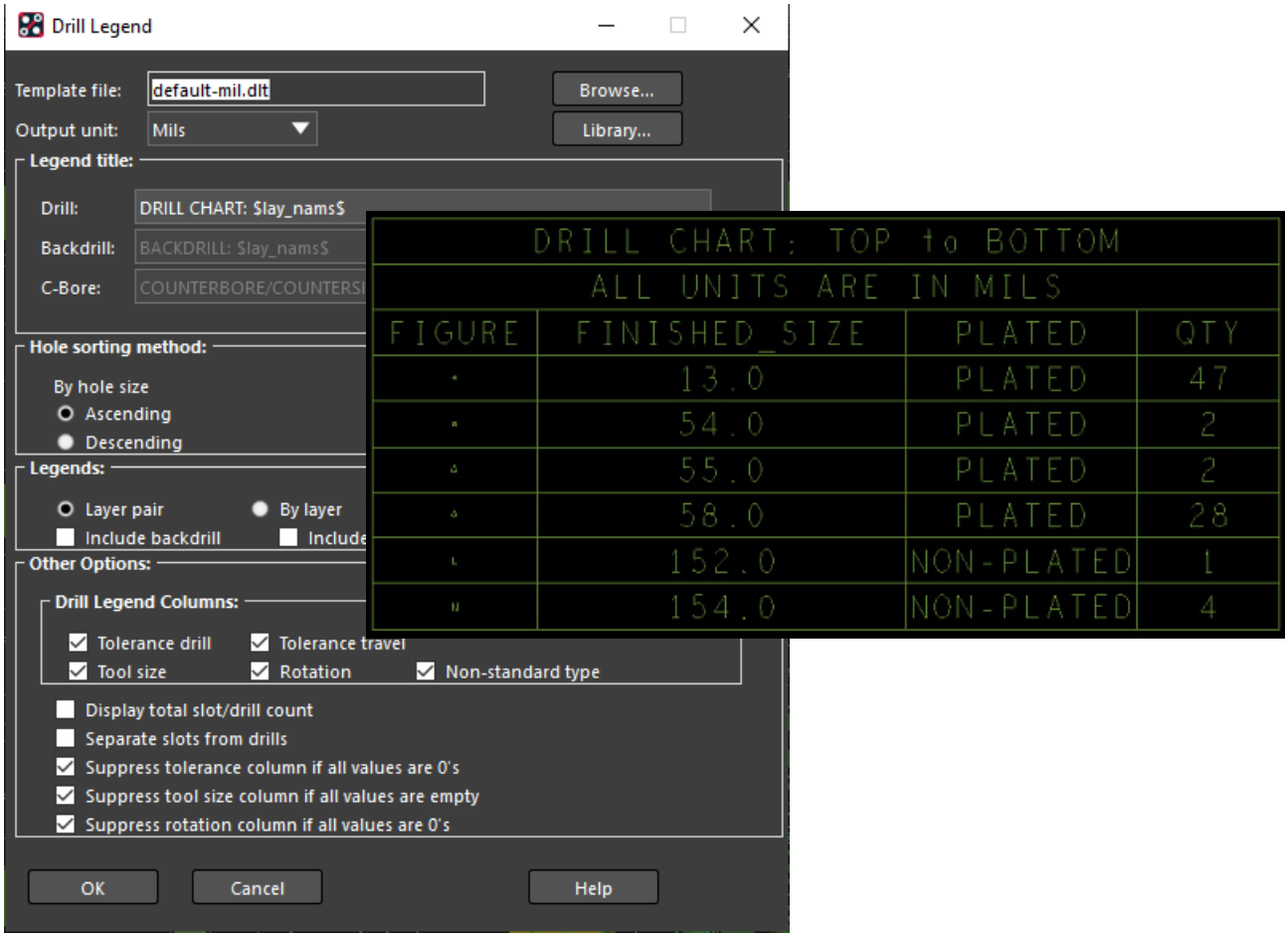
Drill legend tables are used in fabrication drawing and shows the number, type, and tolerance of plated and non-plated holes in the design.

1. Choose *Setup – Colors,* click *On* to enable *Global Visibility* and click *OK* to close the dialog box.
2. Choose *Manufacture – Create Drill Table.*

   The Drill Legend form opens.
3. Click *OK* to generate the drill legend symbol.

   The drill legend symbol gets attached to the cursor.
4. Left-click to place the drill legend in the design canvas.

Figure 5-16 Drill legend parameters and table



© 2020
All Rights Reserved.

Generating NC Drill

NC drill file is created based on the parameters specified for the drill coordinate data format.

1. Choose *Export – NC Drill*.

   The NC Drill form opens.

2. Click *Parameters* to open NC Parameters form.

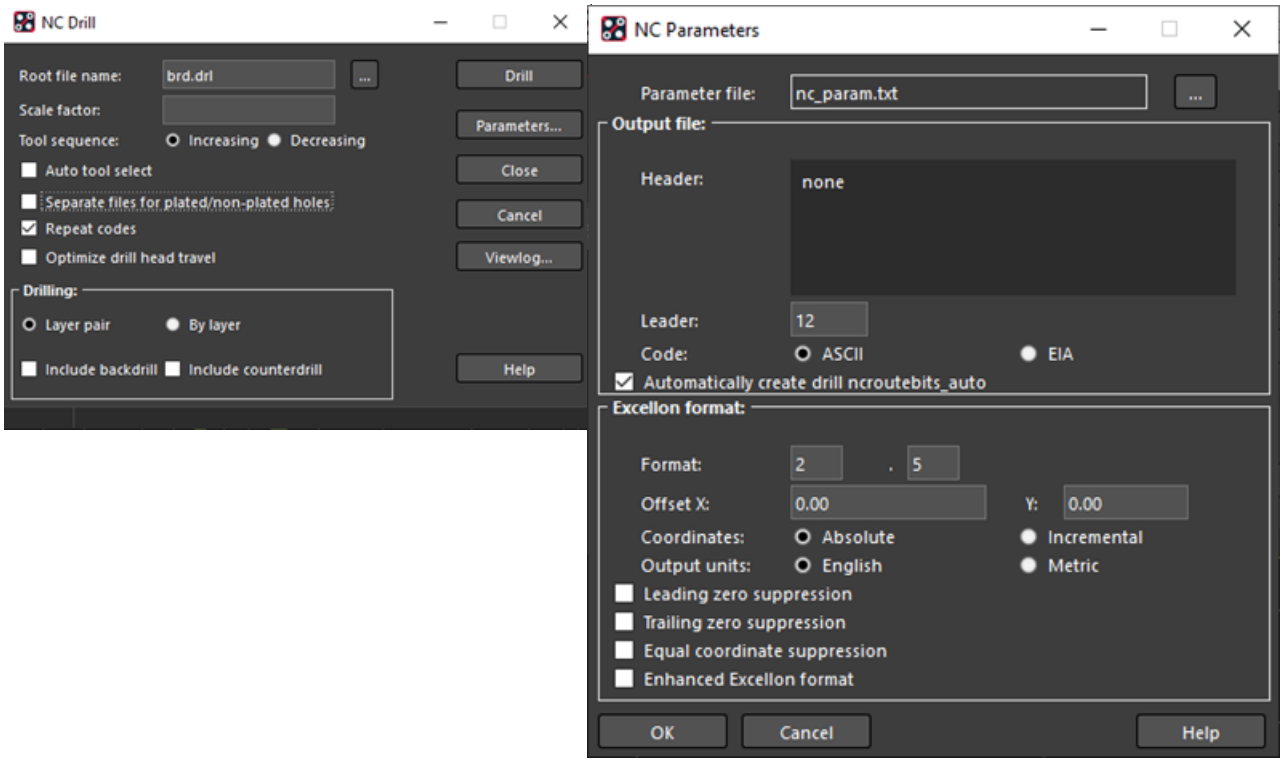
3. Enable the *Enhanced Excellon format* checkbox.

   A header in the NC Drill and NC Route output files is generated that uses Excellon commands.

4. Click *OK* to save the parameters.

5. Click *Drill* to generate the drill file.

Figure 5-17 NC Drill setup options



   The NC Drill file (`tutorial.drl`) is created in the `allegro` directory.

6. Click *Viewlog* to review the log file.

7. Close the form.

© 2020
All Rights Reserved.

Creating IPC2581 Files

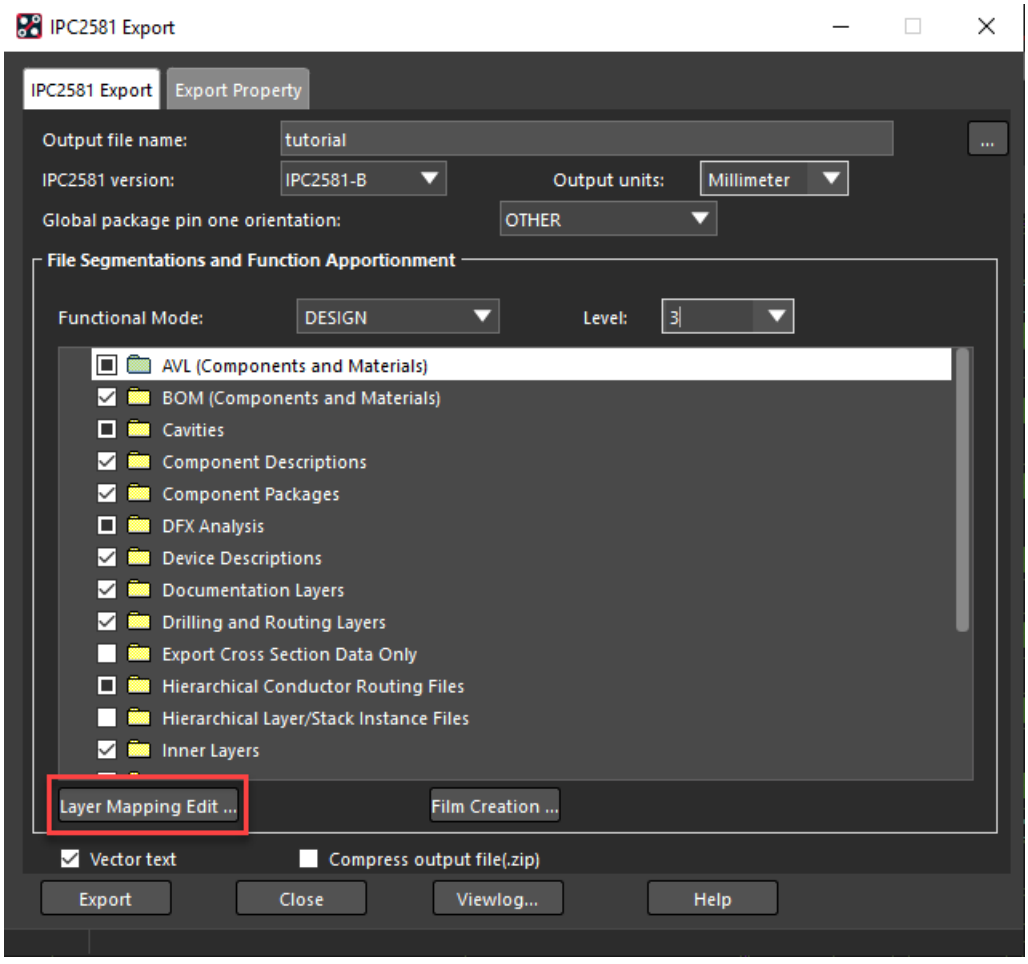
IPC2581 is an XML-based data exchange format used for providing physical design data for fabrication and assembly of PCBs.

For this tutorial, create IPC2581 output using default values.

1. Choose *Export – IPC2581*.

   The IPC2581 Export form opens.

Figure 5-18 Generating IPC2581 output



2. Leave the *IPC2581 version* to default, which is set to the latest version *IPC2581-B*.
3. Select *Output units* to `Millimeter`.
4. Set the *Functional Mode* to `DESIGN` and *Level* to `3`.

© 2020
All Rights Reserved.

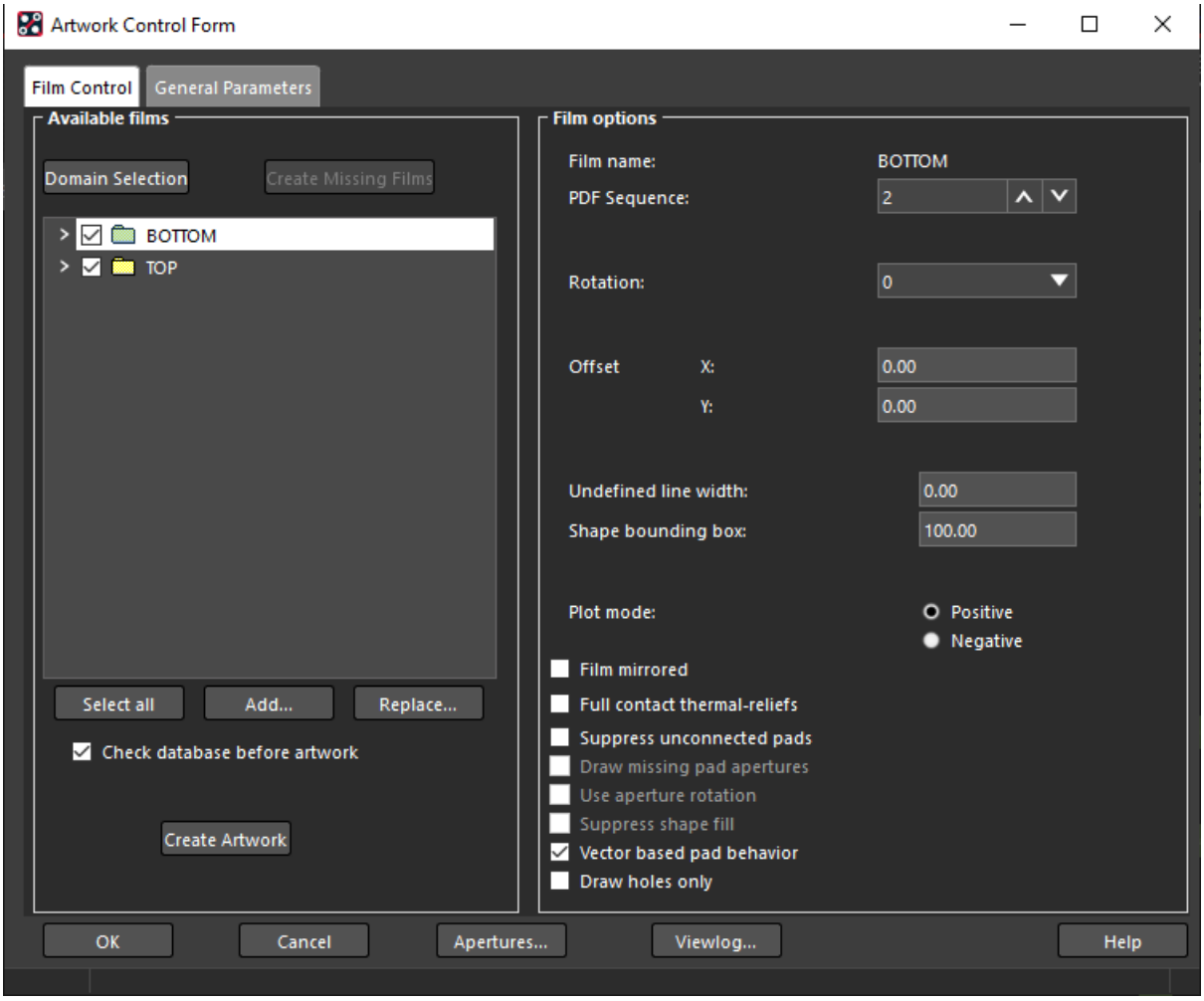
Five functional modes are supported and each mode consists of three levels that define the complexity and detail of the output file.

5. Click *Film Creation* to add class and subclass for film records.

   The Artwork Control Form opens.

6. Select both the TOP and BOTTOM layers and click *OK* to close the form.

Figure 5-19 Film record creation settings



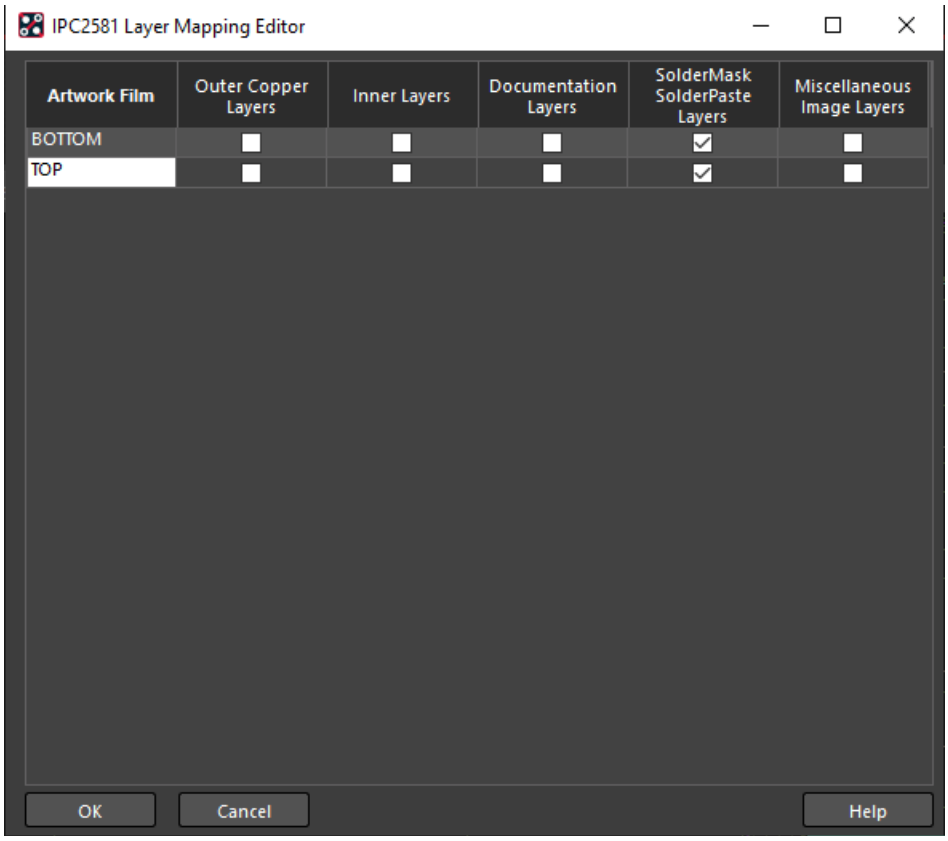
Tip

When exporting IPC2581, it is recommended to enable *Dynamic unused pads suppression* option in Cross-section Editor to suppresses unconnected pads for the selected object types (pin/via) on the selected inner layers.

© 2020
All Rights Reserved.

7. Click *Layer Mapping Editor* to specify layer type for each artwork film.
8. Select the check boxes for *Soldermask and SolderPaste Layers* for both the artwork films and click *OK* to close the form.

Figure 5-20 Layer mapping editor



9. To generate IPC2581 file, click *Export*.

   An XML file (`tutorial.xml`) is created in the `allegro` directory.
10. Click *Viewlog* to review the log file.

Summary

This completes the task of creating a board design and generating manufacturing files for the PCB. In this chapter, you were introduced to the flow of tasks required for creating a board design using PCB Editor. For detail description of layout creation tasks, refer to PCB Editor information set.

© 2020
All Rights Reserved.