PADS® PROFESSIONAL

2019

PADS PROFESSIONAL STUDENT EDITION QUICKSTART GUIDE

FOR VX 2.7 AND NEWER

TABLE OF CONTENTS

QUICKSTART OVERVIEW (VIDEO LINK)

The purpose of this Quick-Start is to provide you with a simple introduction to PADS Professional. Through a series of short exercises, you will learn how to make a simple PCB design using PADS Professional.

Although PADS Professional supports various PCB design flows, this guide focuses on a simplified PCB design process where the following topics are introduced.

OPENING AND POPULATING A NEW PROJECT ▶ PLACING AND WIRING PARTS ▶ FORWARD ANNOTATING ▶ PCB PHYSICAL LAYOUT ▶ ROUTING THE PCB ▶ GENERATING MANUFACTURING FILES

Before proceeding, please make sure you've downloaded, installed and activated PADS Professional evaluation software version 2.7 or newer.

INSTALLATION & LICENSING (VIDEO LINK)

After placing a request to evaluate PADS Professional, you should've received an email with links to

- The PADS Professional online installer

and

- Software evaluation license

tip: **If you did not receive this email, please contact your local authorized Mentor Graphics Distributor *via https://www.pads.com/buy.***

1. Double click the online installer to begin the software installation.

note: **You will need to remain connected to the internet until the installation completes. Installation time will vary depending on the speed of your internet connection.**

2. Click **"Next".**
3. Review and accept the license agreement if you agree.
4. Click **"Next"** to accept the recommended installation location.
5. When the installation completes, click **"Done".**

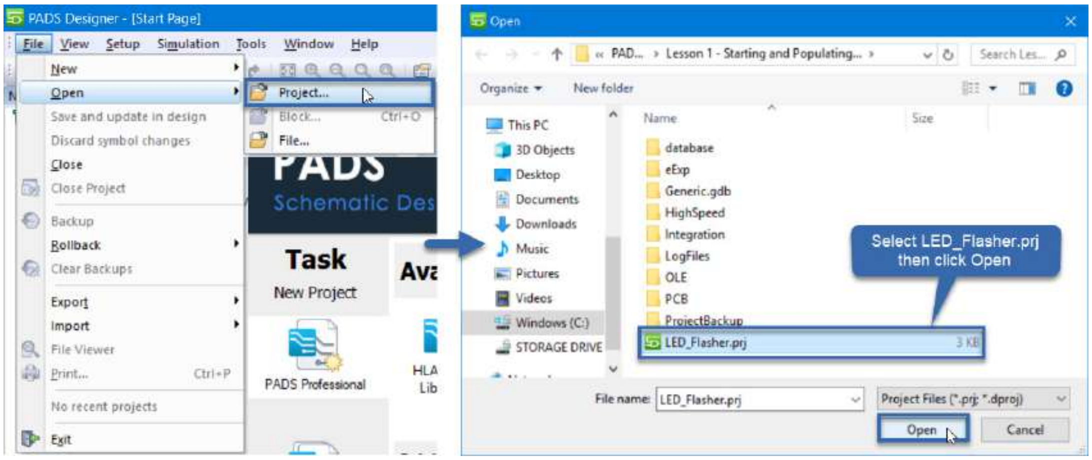
You should now see the PADS Professional Schematic and PADS Professional Layout icons on your desktop.

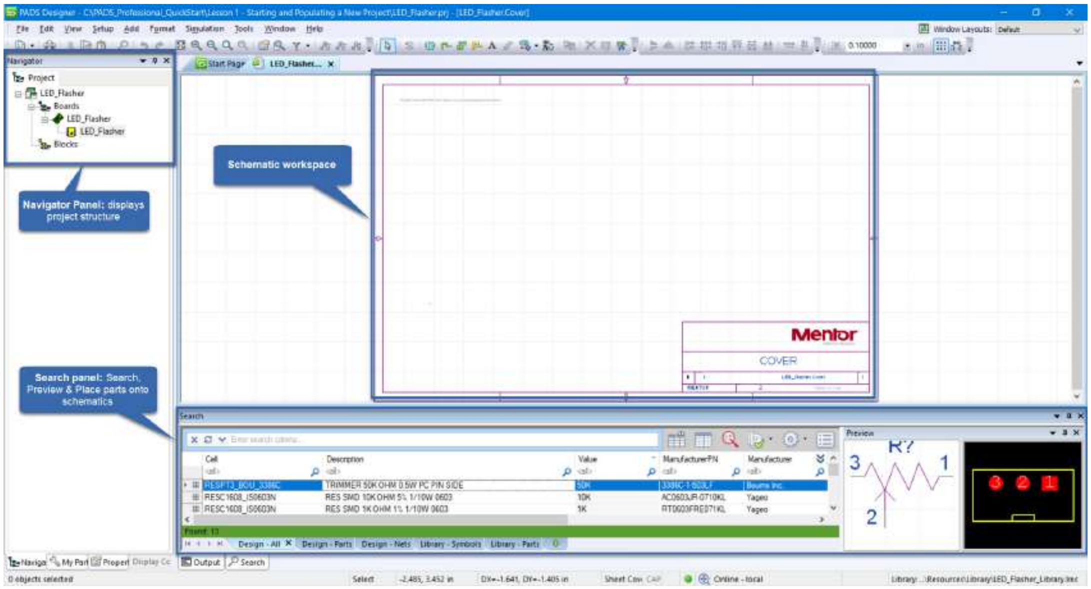
6. Download the license file named **PADSPro.dat** from the link provided to you via email, and save this file in **C:\PADS_Professional_QuickStartVX.2.7.**
7. Close all windows and proceed to **Lesson 1.**

LESSON-1: OPENING AND POPULATING A NEW PROJECT (VIDEO LINK)

Opening a new project:

1) From the Windows Startmenu, navigate to the **PADS Pro Tools VX 2.x.** folder and click on **PADS Pro Designer VX 2.x.**
2) Once PADS Designer opens, go to: **File ► Open ► Project.**
3) Navigate to **C:\PADS_Professional_QuickStart_VX.2.7\Lesson 1 - Starting and Populating a New Project\LED_Flasher.prj**, select the file named **"LED_Flasher.PRJ"** and click **Open.**

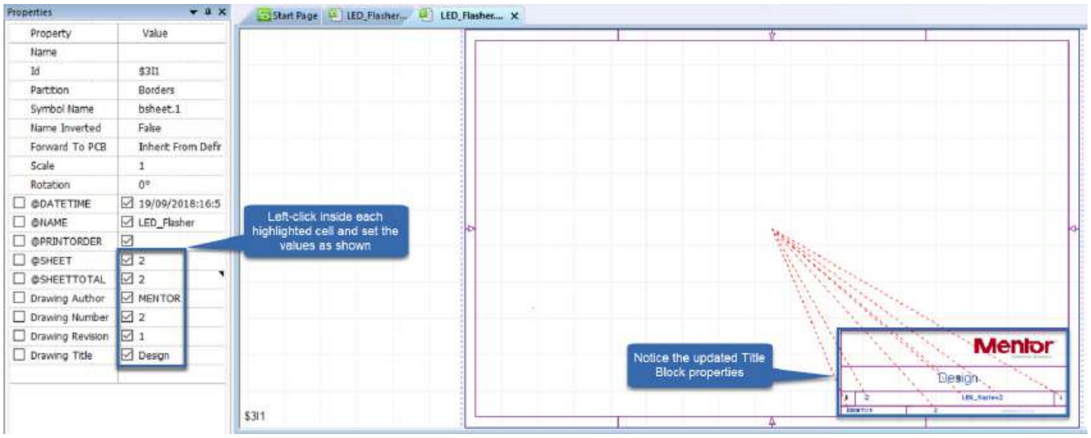




Populating a new project

1) Add a new sheet to your project by going to: **File ► New ► Sheet.**

2) The **Properties** panel will automatically display. Left-click inside each highlighted cell and overwrite system generated values as shown below.



3) Left-click to focus on the **Navigator** panel.

4) Within the **Navigator** panel, Right-click the file named **"3"** and click **"Rename"** from the menu.

5) Type **"Design"** and press **Enter** key.

6) This concludes Lesson 1: You've successfully added a new Schematic sheet to your design and customized its Title Block.

7) Go to: **File ► Close Project**, keep **PADS Designer** open, then proceed to **Lesson 2.**

tip: **PADS Professional automatically saves your work.**

LESSON-2: PLACING AND WIRING PARTS (VIDEO LINK)

Searching for and placing parts on your schematic sheet.

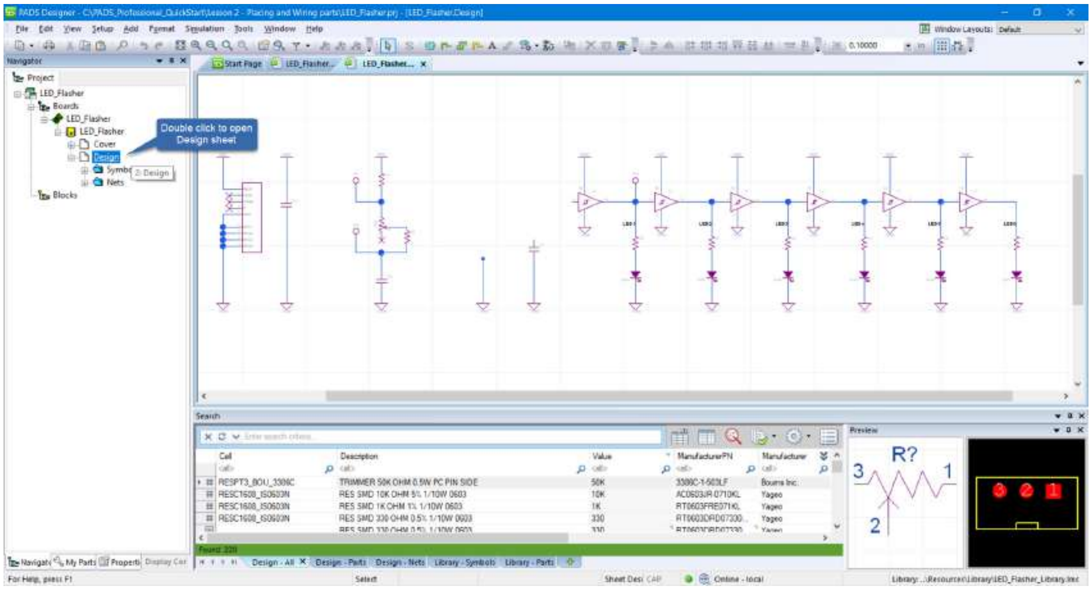
1) If not already open, launch PADS Designer and go to: File ► Open ► Project

2) Navigate to **C:\PADS_Professional_QuickStart_VX.2.7\Lesson 2 - Placing and Wiring parts\LED_Flasher.prj**, select the file named **LED_Flasher.PRJ** and click **Open.**

note: **This project is prepopulated with an incomplete schematic page, which you will complete as part of Lesson 2.**

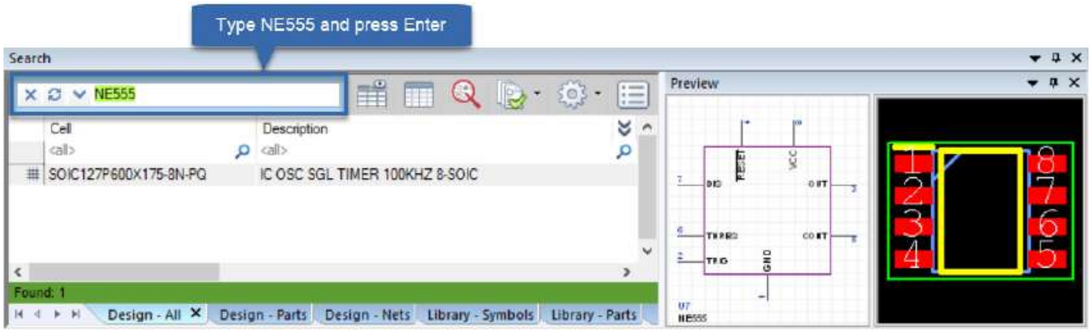
3) Within the **Navigator** panel, Double-click the **Design** sheet to view its contents.

tip: **Within the schematic workspace, scroll the middle mouse button forward to zoom in, backwards to zoom out. This will improve the visibility of objects on your schematics.**

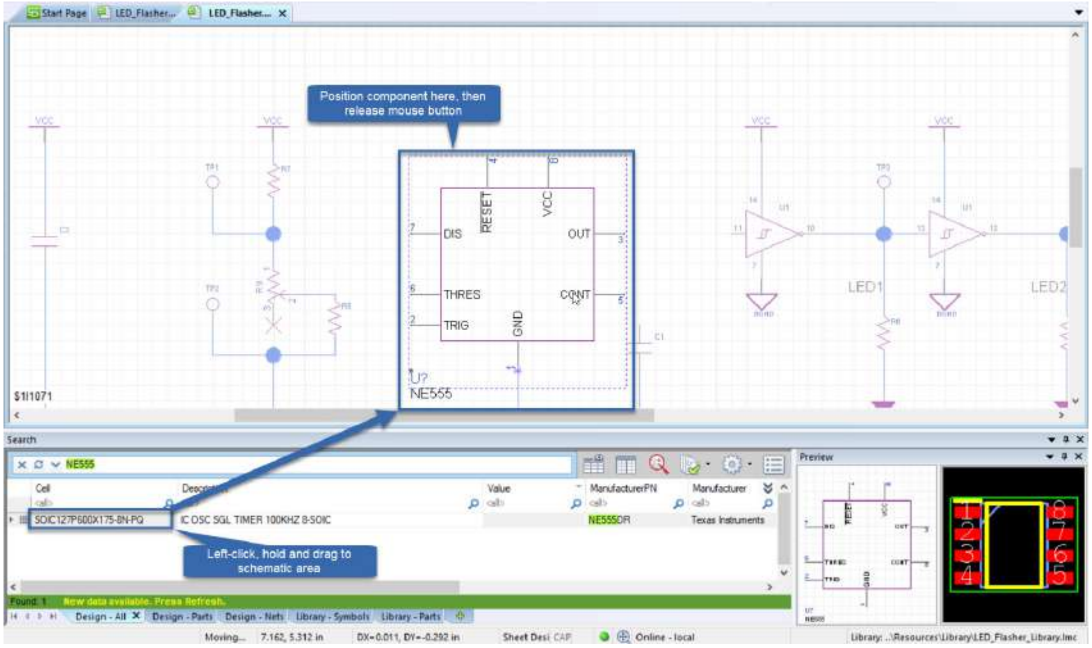


4) Within the Search panel search box, type: "NE555" and press Enter to see the NE555DR component as shown below (***you may need to click once on the search result to see the part's symbol and footprint***).

note: **Upon first opening PADS Designer, you may see the Output panel on top of the Search panel. Simply click once on the Search panel tab to bring it into focus.**



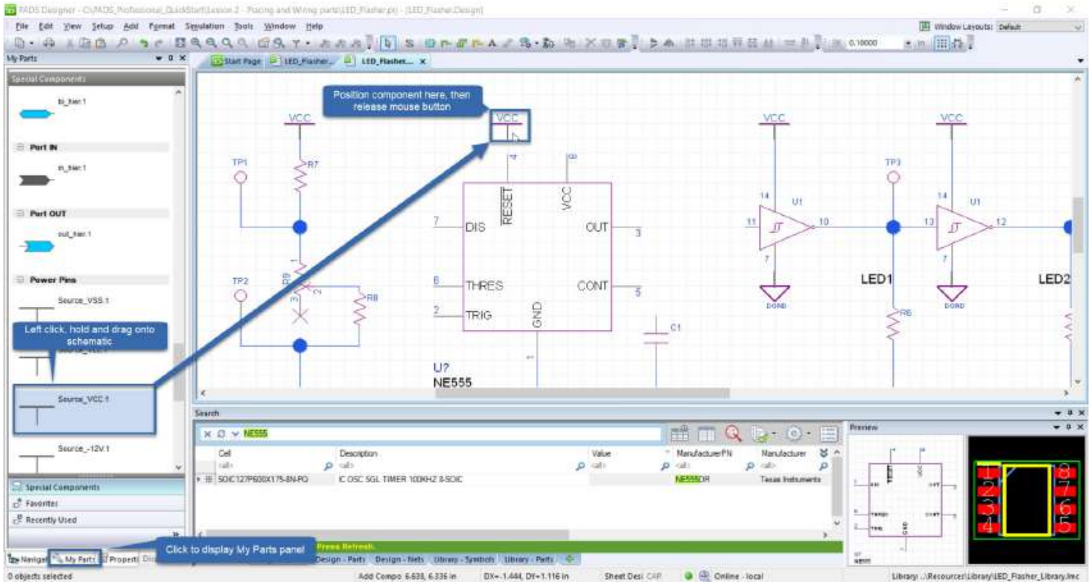
5) Drag and drop the NE555 part from the search area onto the Design sheet (***try to position your symbols as shown in images to facilitate the wiring process***).



note: **Press & hold the middle mouse button while moving your mouse to pan the schematic workspace.**

6) From the **My Parts** panel, scroll to locate the **Special Component** named **Source_VCC.1.**

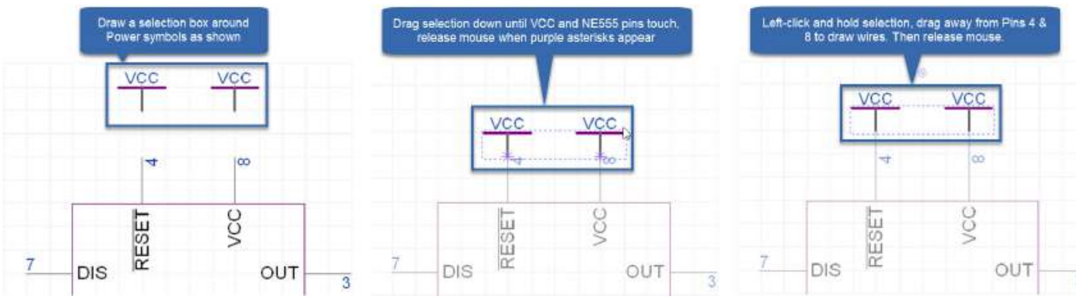
7) Drag and drop **Source_VCC.1** onto the **Design** sheet.

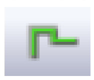


8) Repeat **Step 7** to add a second **Source_VCC.1** component to your schematic (***Or hold down the CTRL key while dragging the same symbol***).

Wiring your schematic

1) Follow the steps in the image below to wire your Power components (The two VCC symbols) to the NE555.



2) Click the Net icon  on the top toolbar area (or Press lowercase **n** key) to activate the wiring tool.

3) After your mouse cursor turns into a crosshair, **Left-click and hold** on **Pin 6** (THRES) of the **NE555** symbol.

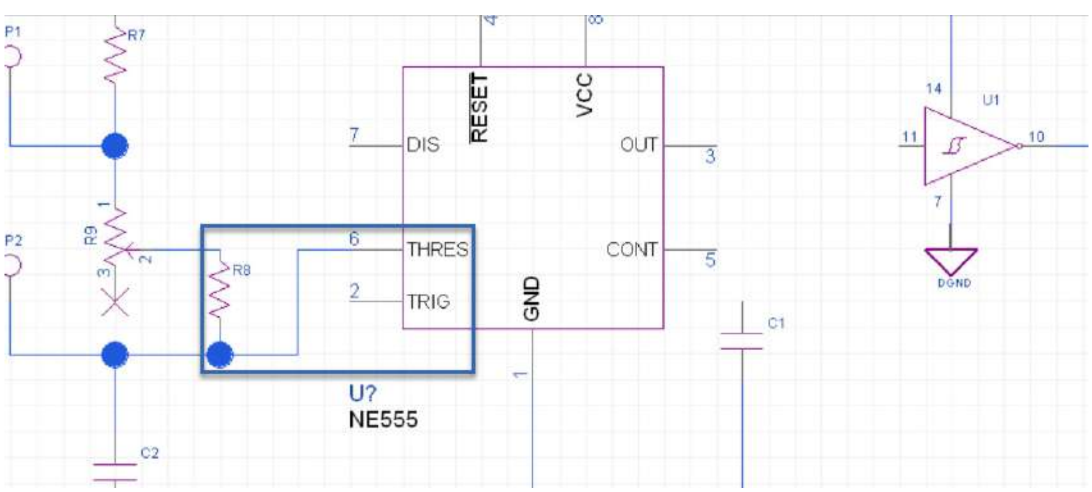
4) Then, move your mouse cursor 2 grid steps to the left and press the **Spacebar.**

5) While still holding down the left mouse button, move your mouse cursor down 4 grid spaces and press the **Spacebar** again.

6) Finally, move your mouse cursor onto the bottom pin of **R8** and release you left mouse button.

7) Press the Escape key to cancel the command.

8) You should end up with a wire similar to the one shown below.



tip: **Hover your mouse over the Net icon for 5 seconds to see a short animated tutorial.**

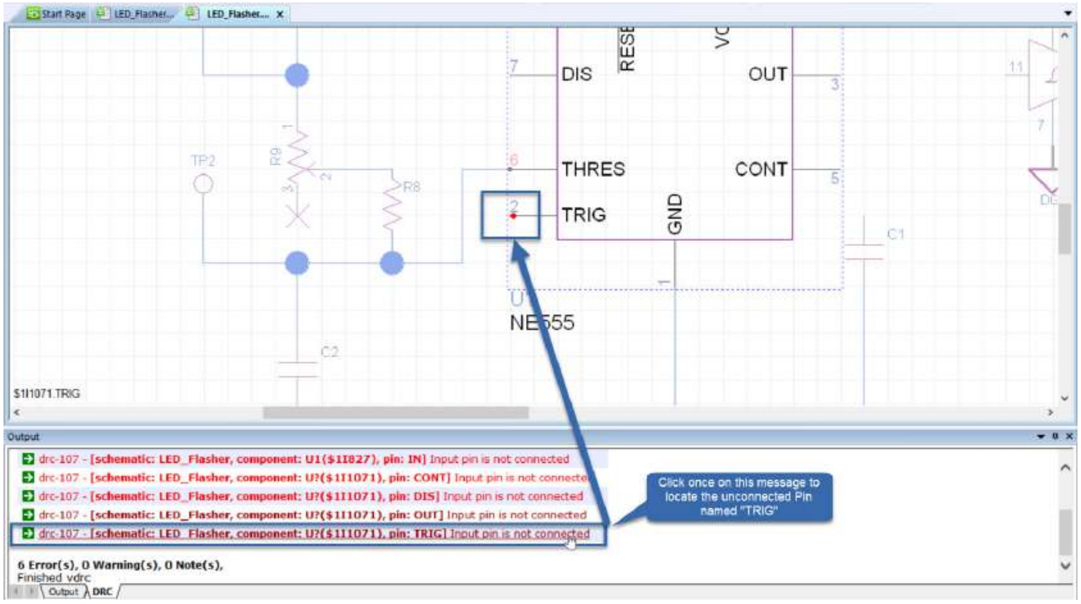
Catching and fixing potential wiring mistakes

9) Go to: **Tools ► Verify.**

10) In the **Verify** window, click the **"OK"** button.

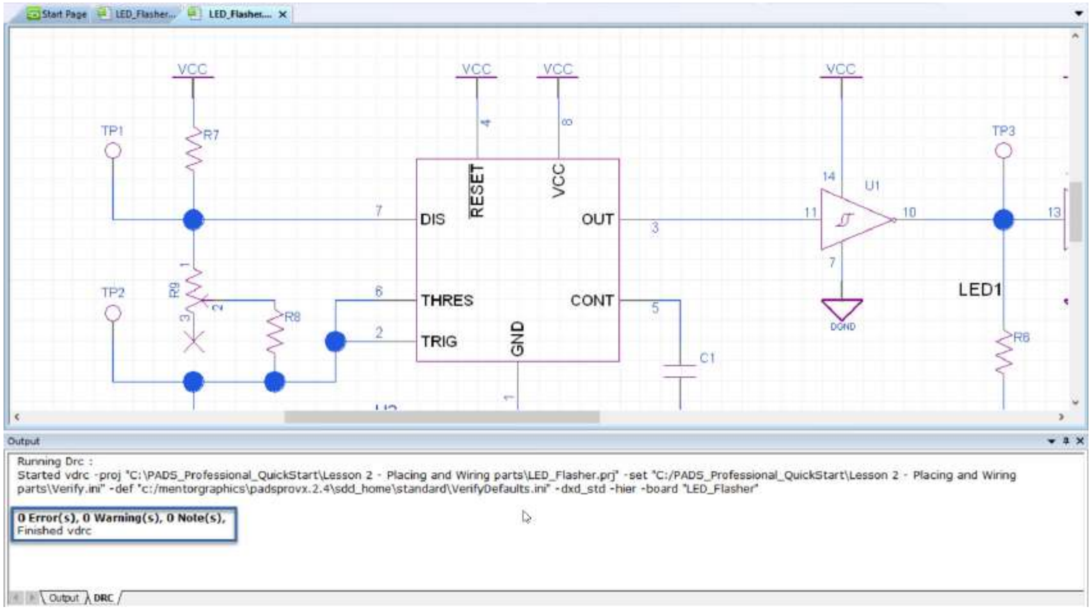
11) The **DRC** panel will appear, displaying a list of potential electrical issue.

12) Click on the last error message in the DRC panel to see the unconnected pin in your schematic workspace.



tip: **While working on your design, you can customize the type of conditions that should be monitored and how they should be reported in PADS Designer by adjusting options shown in the Verify panel (accessible via *Tools ► Verify*).**

13) Fix these errors by repeating Steps 2-6 of this section, to finish wiring the rest of the design as shown.

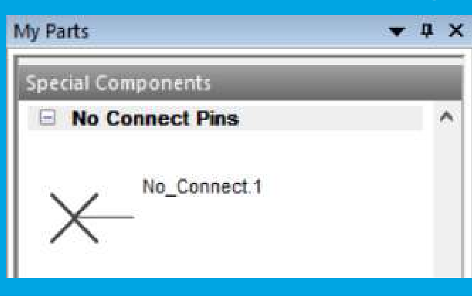


14) Go to: Tools ► Verify, and Click **"Ok"**.

15) Notice the errors are gone.

16) Right click on the DRC panel and click **"Close"**.

tip: **You can also place "No Connect" special components on pins that are purposefully left unconnected without changing your design verification rules.**

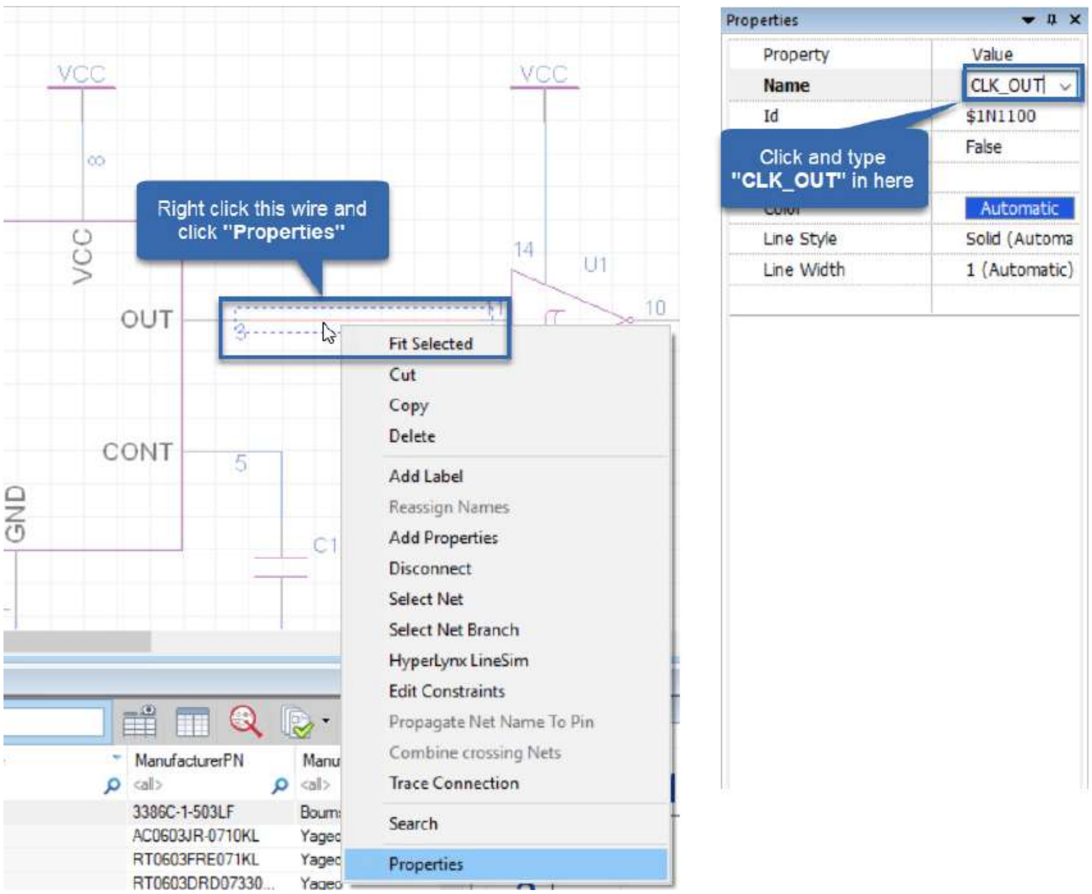


Placing Net Labels

1) Right click the wire connecting **Pin 3 of the NE555** and **Pin 11 of the OpAmp**, then click **Properties.**

2) Within the **Properties** panel, change the value of Name to **CLK_OUT** and press **Enter.**

3) Notice that there is now a label on this wire. Using this method allows you to easily name connections in your design.



4) Click any empty space in your schematic workspace to clear all filters.

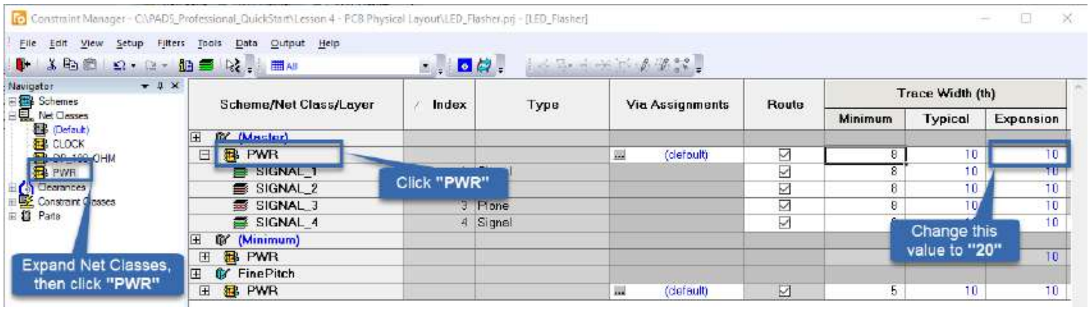
Constraint Manager

Using the constrain manager, you can predefine design rules that will get forward annotated to the PCB layout workspace.

1) Click **Tools ► Constraint Manager.**

2) Within the Constraint Manager's **Navigator** panel, select **Net Classes ► PWR.**

3) Expand **Master,** then **PWR,** and set the **Trace Width (th)** ► Expansion values to **"20".** Note that everything under PWR in the Expansion column in Master will be changed.

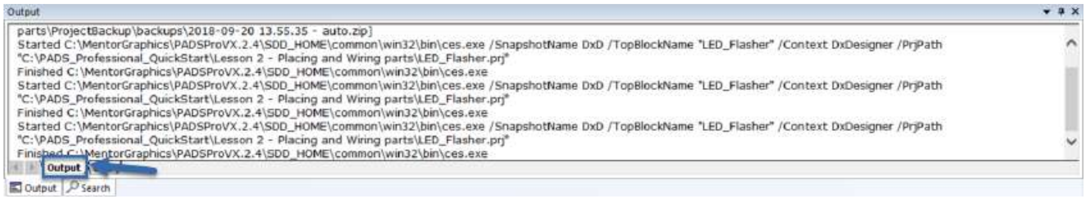


tip: **At any time, you can toggle between English and Metric units within the *Schematic Editor* properties accessible from: *Setup* ► *Settings* ► *Schematic Editor*.**

4) Check the box to not show warning again in the same session, then click **OK.**

5) Close the **Constrain Manager** window.

6) Right-click and close the **Output** panel.



7) Go to: **File** ► **Close Project.**

LESSON-3: FORWARD ANNOTATE, CREATE A PCB FILE AND DRAW YOUR BOARD SHAPE (VIDEO LINK)

During forward annotation, each component in the logical PADS Professional Designer workspace is mapped to a physical footprint in the PADS Professional Layout workspace. This includes packaging, assigning reference designators and creating an associated PCB file in one operation.

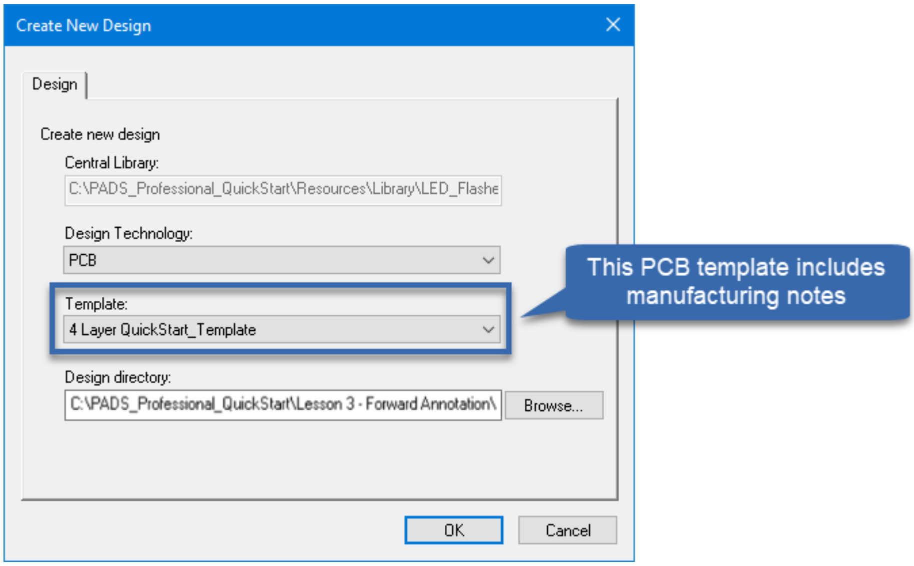
Forward Annotate to PCB

1) If not already open, launch PADS Designer and go to: **File ► Open ► Project.**

2) Navigate to **C:\PADS_Professional_QuickStart_VX.2.7\Lesson 3 - Forward Annotation\LED_Flasher.prj**, select the file named **LED_Flasher.PRJ** and click **Open.**

3) Within PADS Designer, go to: **Tools ► PADS Professional Layout.**

4) Set the Template drop down menu to **"4 Layer QuickStart_Template"** as shown, then click **"OK".**



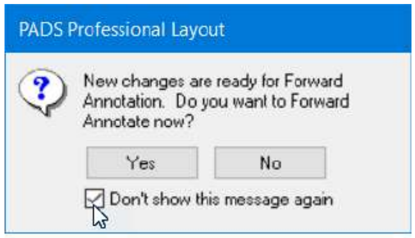
note: **The following steps only need to be performed once. PADS Professional Layout interface will automatically open and load your base PCB file.**

5) Click **"Yes"** to create the design directory when prompted.

6) Click **"Ok"** to import the layout template.

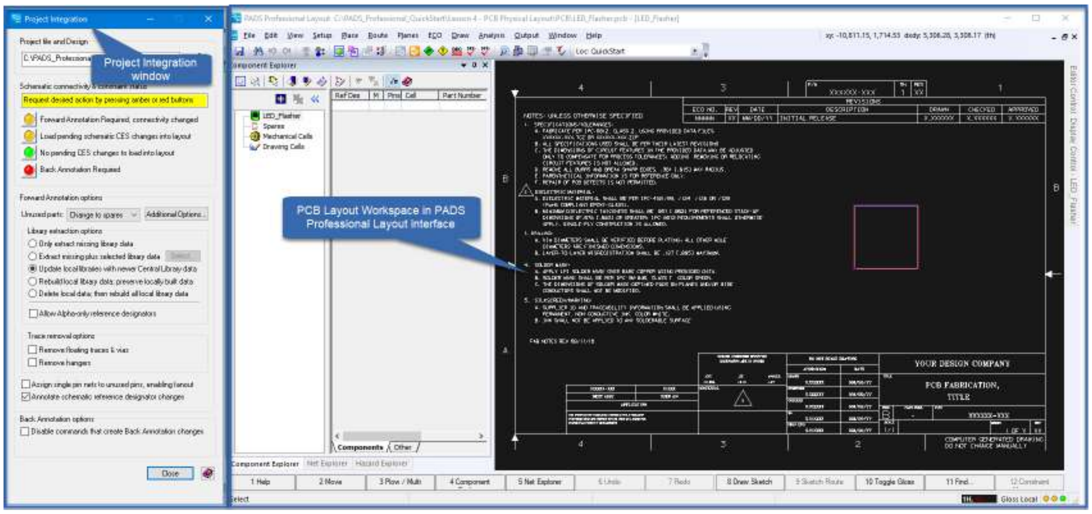
7) Click **"Ok"** to dismiss the message about Back annotation being disabled.

8) Check the box to not show the prompt to forward annotate again, then click **"Yes".**

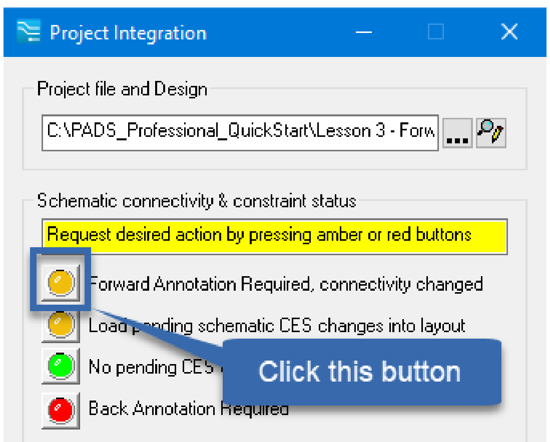


9) PADS Professional Layout interface should have automatically opened, loaded an empty PCB, and displayed the **Project Integration** as shown.

tip: **The PCB file name is automatically set to the PADS Designer project name.**



10) Within the **Project Integration** window, click the top amber button to forward annotate.



note: **You may see a series of notifications about tasks being processed. They will automatically disappear when finished. You may also see a note that Forward Annotate completed with warnings. Simply close the output panel.**

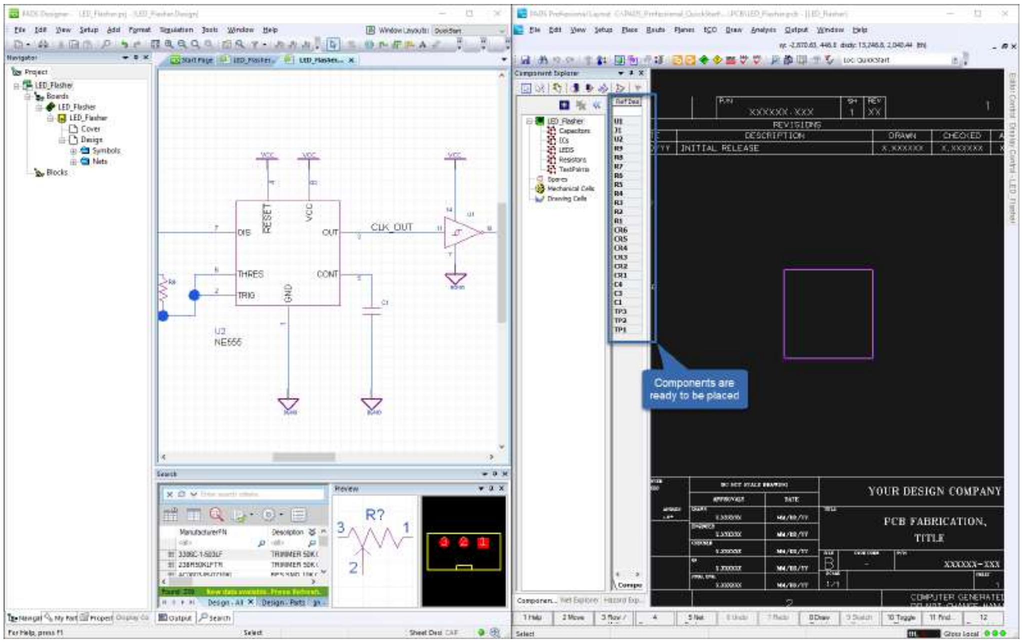
11) Click **"Ok"** to dismiss the warning message.

12) The amber lights should now be Green, indicating that your schematic and PCB data have been synchronized.

13) Click the **"Close"** button to dismiss the **Project Integration** window.

14) You now have 2 windows open: The **PADS Designer** interface displaying your schematic, and the **PADS Professional Layout** interface displaying the contents of your PCB.

tip: **Feel free to move these 2 windows to different monitors if you have more than 1 monitor. At this stage, components have been imported into your PCB workspace and are ready to be placed on the board. If not visible, enable the *Component Explorer* through the *Place* ► *Component Explorer* command.**



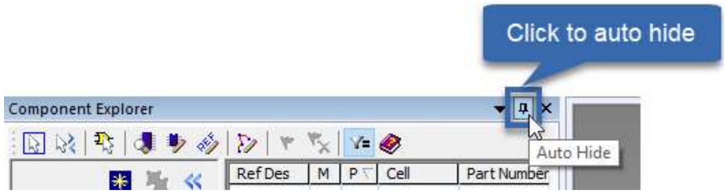
15) Close the PADS Designer interface

but leave the PADS Professional Layout interface open.

Define PCB board shape from 3D Step model

Although it is possible to manually draw a complex 2D board shape within PADS Professional Layout, in some case it can be quicker to define a 2D board shape based on a 3D model.

1) Within the PADS Professional Layout interface, click the **bookmark** icon to hide the left panel group and expand the visible workspace area.



2) Switch to **3D view** by going to: **Window ► Add 3D view.**

3) Within the**3D view,** import the Step model by clicking **3D ► Import Mechanical Model**

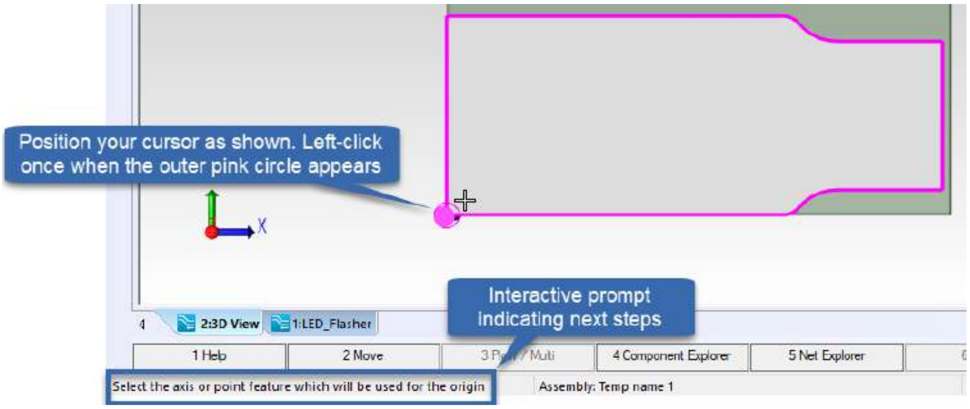
4) Navigate to **C:\PADS_Professional_QuickStart_VX.2.7\Resources\Step models**, select **Board outline.step** and click **"Import".**

5) Go to: **3D ►Create Board.**

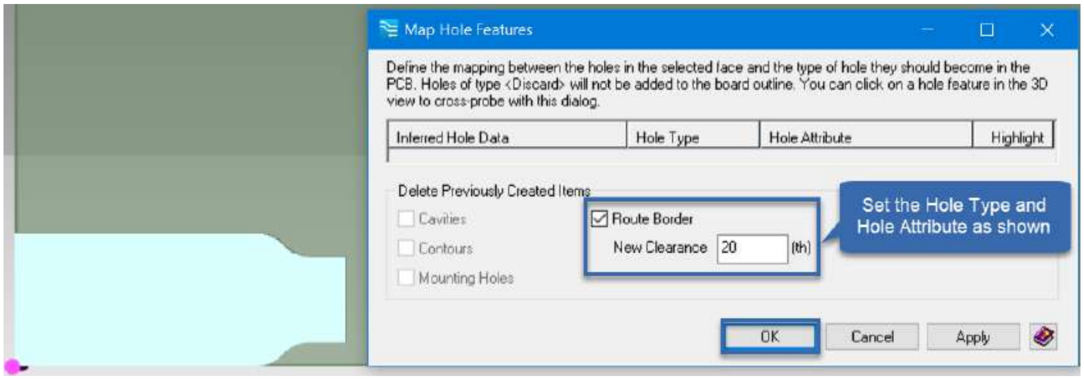
6) Left-click once on the imported Step model.

tip: **The top face outline will turn pink on mouse-over, to indicate which face you are about to select.**

7) Position your cursor as show. Left click once when the pink outer circle appears. This will define the board origin.

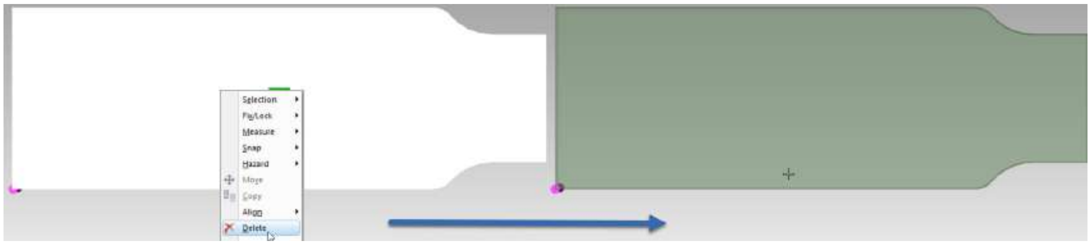


8) Within the **Map Hole Features** window: Check the box next to **"Route Border"**, then set **New Clearance to 20** and click **"OK"**.

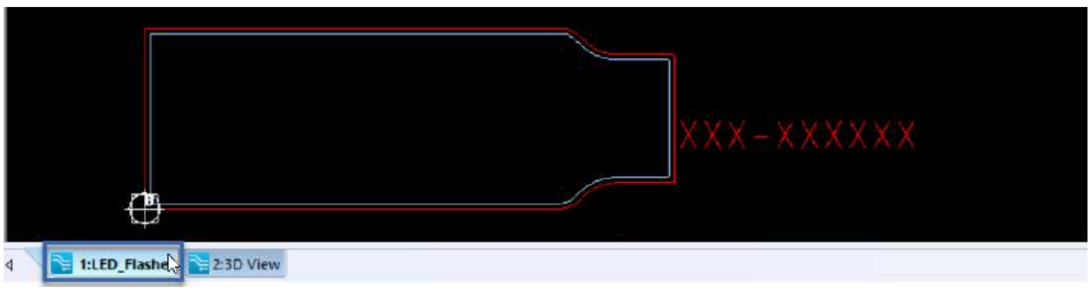


9) Right-click the Step body and click **"Delete"**

10) Click **"Yes"** to dismiss the prompt. Notice your new board outline.



11) Click once on the view tab named **1:LED_Flasher** to view the **board outline** in **2D View.**



12) Press **CTRL + B** key combination to fit your board outline to the PCB layout space.

13) Go to: **File ► Save**, then Close the **PADS Professional Layout interface.**

14) Proceed to the next exercise.

LESSON-4: PCB PHYSICAL LAYOUT (VIDEO LINK)

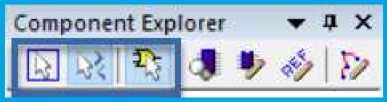
Placing footprints from schematics

In this lesson, you will open the schematic and PCB workspaces side by to aid in footprint placement. Some footprints have already been grouped and placed on the board for you.

note: **Although not required, guiding footprint placement from the schematic workspace can speed up the board layout in projects with higher component counts.**

1) Launch PADS Professional Layout, then go to: **File ► Open.**
2) Navigate to **C:\PADS_Professional_QuickStart\Resources\Step models,** select the file named **LED_Flasher.PCB**. Click **Open.**
3) Your workspace has been pre-configured to allow easy placement of parts from the **Design** schematic page.

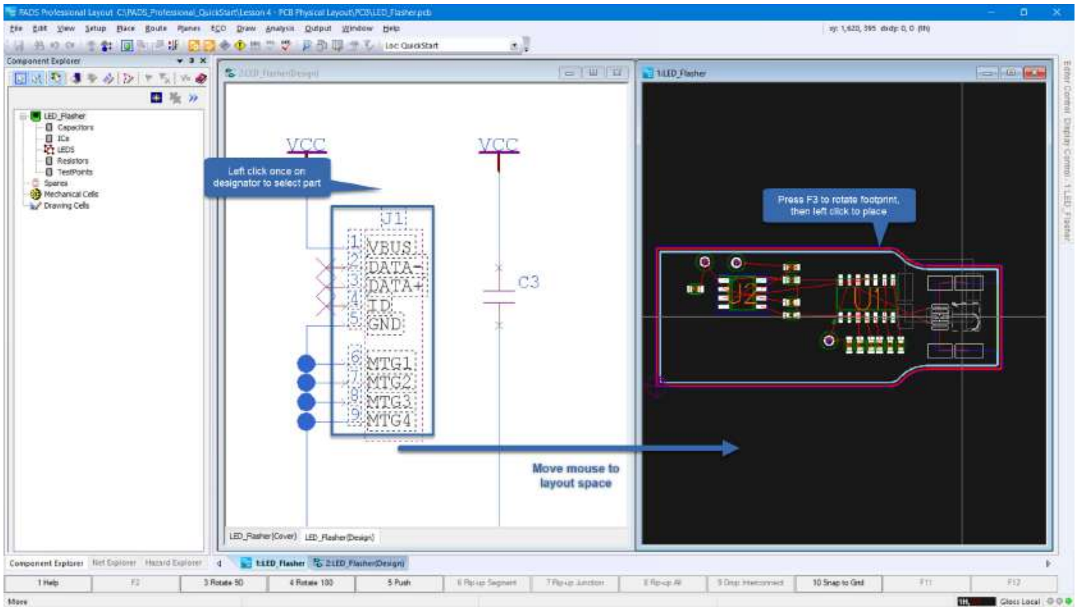
tip: **Displaying the schematic view within your layout space does NOT consume a schematic license. This method allows users to place parts on their board in an organized manner. The following settings are already enabled for you. *Windows ► Add Schematic View*. Within the *Component Explorer* panel the following options are enabled.**



Parts can also be placed as groups or sequentially by first selecting them from the *Component Explorer Panel.*

note: **If steps 4-6 don't work for you, toggle the *Component Explorer* visibility by clicking its bookmark icon twice.**

4) Within the Schematic window, use the middle mouse button to zoom into the component **J1.**
5) Left click **J1** once such that it is selected (no need to hold down the mouse).
6) Move your mouse cursor over to the PCB layout space and notice how the associated footprint to **J1** is appended to the cursor and ready to be placed.
7) Press the **F3** key to rotate **J1**'s footprint, **position,** and **left click** to place the footprint as shown.



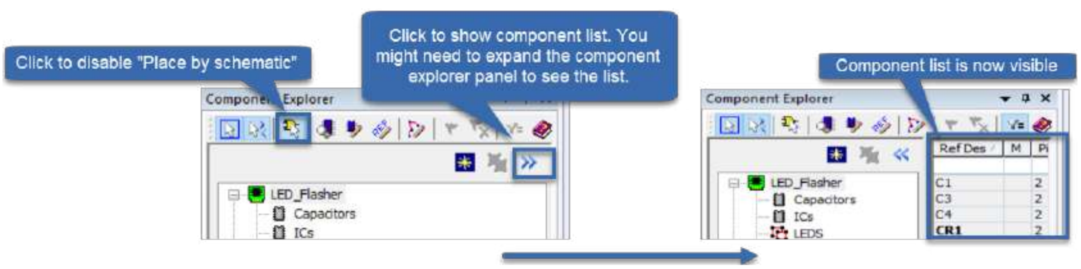
Placing Footprint in 2D/3D Side by side

1) Close the Schematic window. 

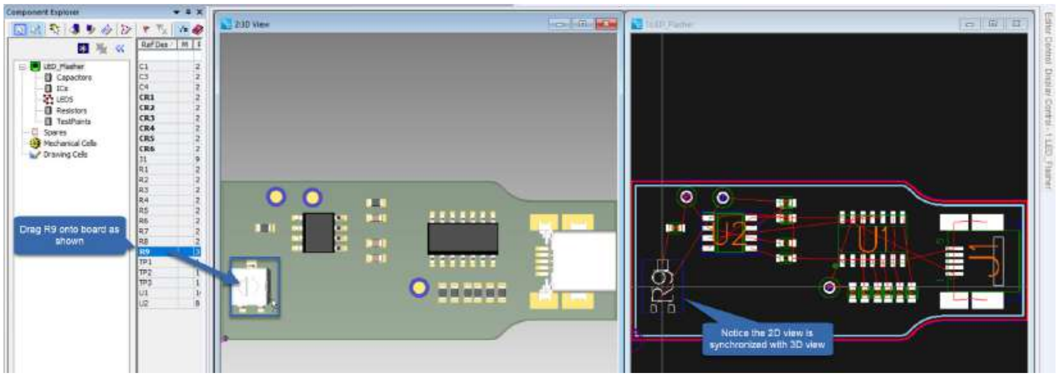
2) Go to: **Windows ► Add 3D view**.

3) Go to: **Windows ► Tile Vertically.**

4) Within the **Component Explorer** panel, click **"Place By Schematic"** and **"Show list".**

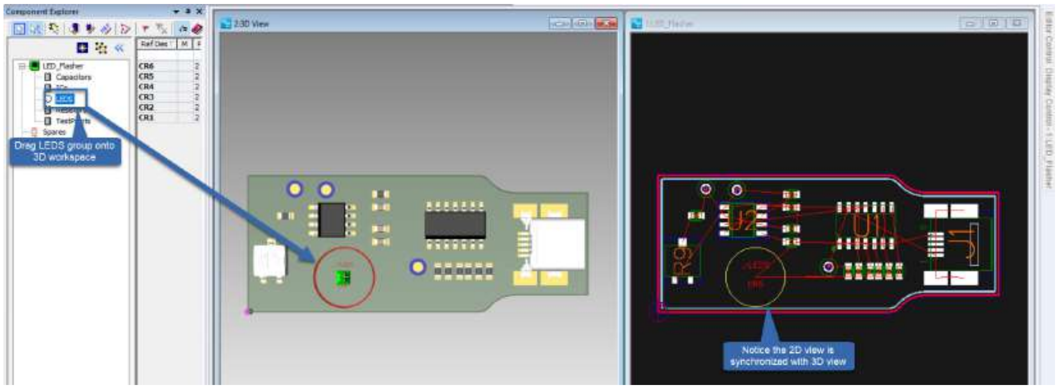


5) Within the Component Explorer list, drag **R9** onto the board in **3D View,** as shown below.



tip: **You can view your board in 3D and 2D side by side simultaneously. This can facilitate alignment and placement of more complex 3D geometry, especially if placing multi-body STEP assemblies. STEP model color information will be imported if defined from within your MCAD program. Zooming IN/OUT or Panning will move both 2D/3D views.**

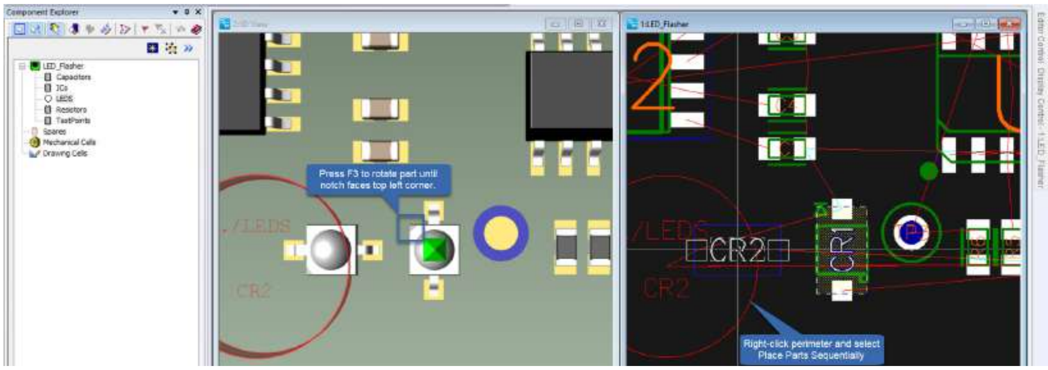
6) Within the **Component Explorer,** drag the **"LEDS"** group onto the **3D view.**



7) Within the **2D view,** right-click the group object's perimeter and click **Place Parts Sequentially.**

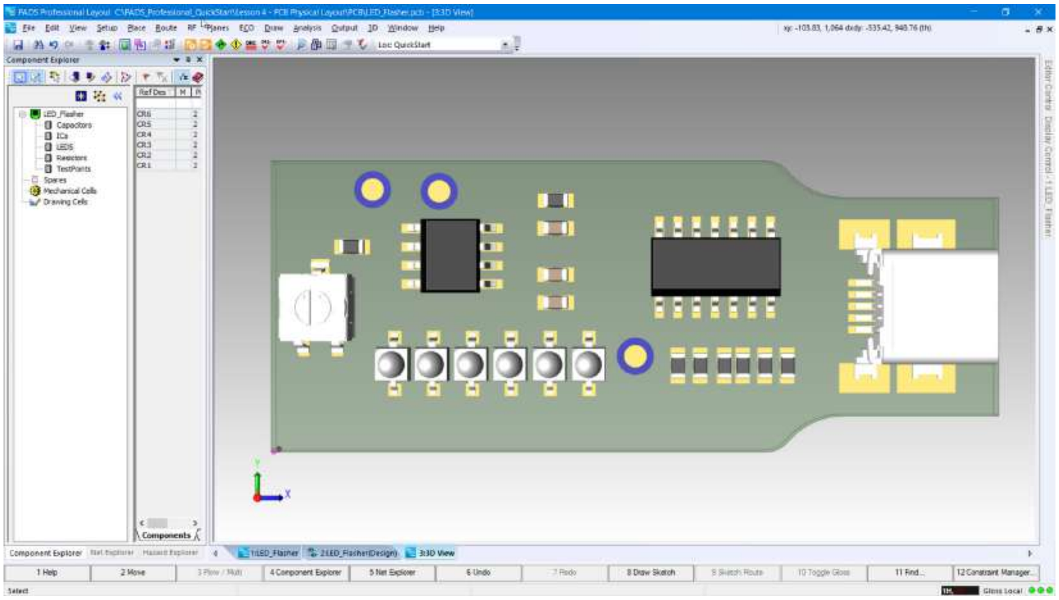
8) Press **"F3"** to rotate LEDs before placing. Make sure the notch (visible in 3D) is facing the **top left** corner.

tip: **Part grouping can be defined from the schematic workspace to simplify part placement during layout. You can also select a part and use arrow keys to precisely position its footprint.**



9) Continue to rotate and place remaining LEDs, as shown below.

tip: **Parts that are not yet placed are written in bold text in the *Component Explorer*.**



note: **You may notice some LEDs turning yellow during placement. Live DRC is activated and currently set to prevent you from placing parts in positions that violate any relevant placement DRC rule. Components with violations will turn yellow.**

10) Go to: **File ► Close**, and click **"Yes"** when prompted to save.

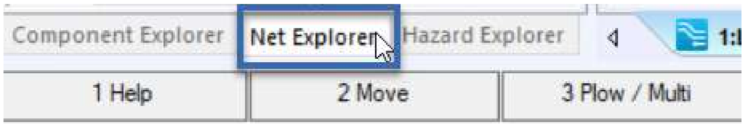
LESSON-5: ROUTING THE PCB (VIDEO LINK)

Adding Fanouts

1) Within PADS Professional Layout, go to: **File ► Open.**

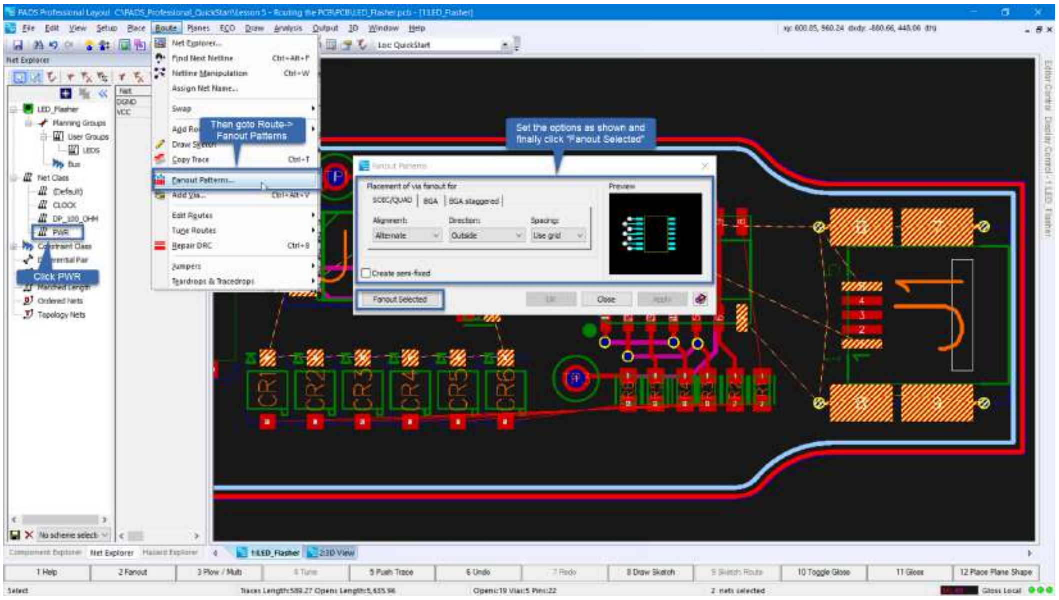
2) Navigate to **C:\PADS_Professional_QuickStart_VX.2.7\Lesson 5 - Routing the PCB\LED_Flasher.prj,** select the file named **LED_Flasher.PCB,** and click **Open.**

3) In the bottom left corner of your screen, click on the **"Net Explorer"** tab to activate the **"Net Explorer"** panel.



tip: **If the *Net Explorer* tab is not available Go to: *Route* ► *Net Explorer*. Then drag and drop the *Net Explorer* panel onto the *Component Explorer* to tabulate them.**

4) Within the Net Explorer panel, select the **PWR** Net Class and notice that all objects assigned the **DGND** and **VCC** nets are highlighted on your board.



5) Go to: **Route ► Fanout Patterns,** ensure the settings are as shown, then click **"Fanout Selected".**

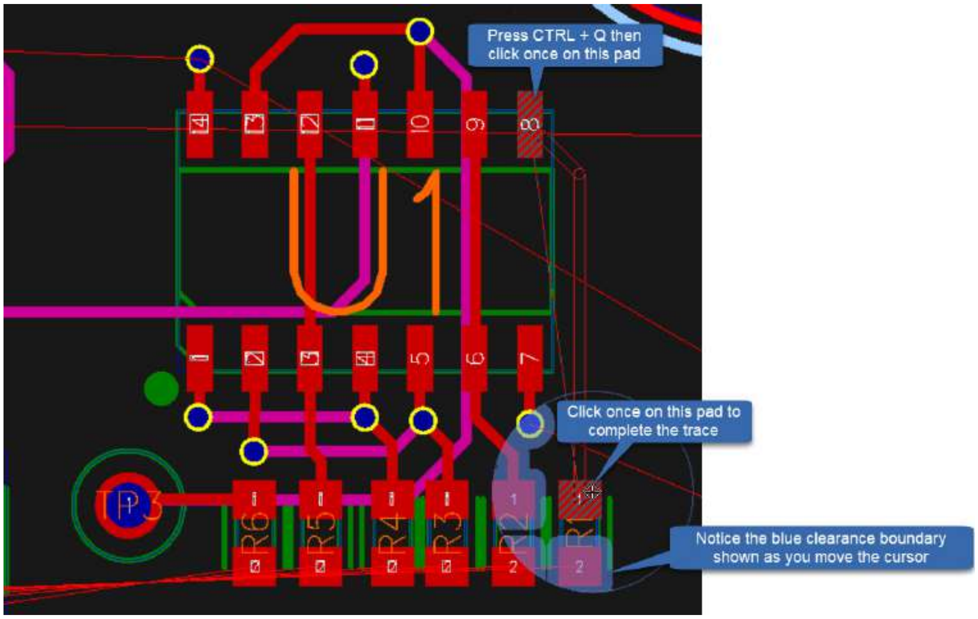
6) Click **"OK"** to dismiss the Fanout report, then **click close** to exit the **Fanout Patterns** window.

tip: **You can group your components by creating the appropriate class type. Then select the class and run the Fanout command.**

Single route

Next, let's route the connection between **Pad 8** of component **U1** and **Pad 1** of **R1.**

1) Left click once in the empty black space to clear any selection filter.
2) Zoom into **U1**.
3) Go to: **Route ► Add Routes ► Plow** (or Press **CTRL+Q**) to launch single trace routing.
4) Left click once on **Pad 8** of **U1,** move your cursor down, then click on **Pad 1** of **R1** to complete the trace.



5) Press the **Escape** key twice to terminate the route command.
6) Left click once in the empty black space to clear the filter.

tip: **When the route command is active, right-click to access a drop down menu to change trace width, routing layer, routing modes, and more. Interactive route tuning tools are available from the route menu.**

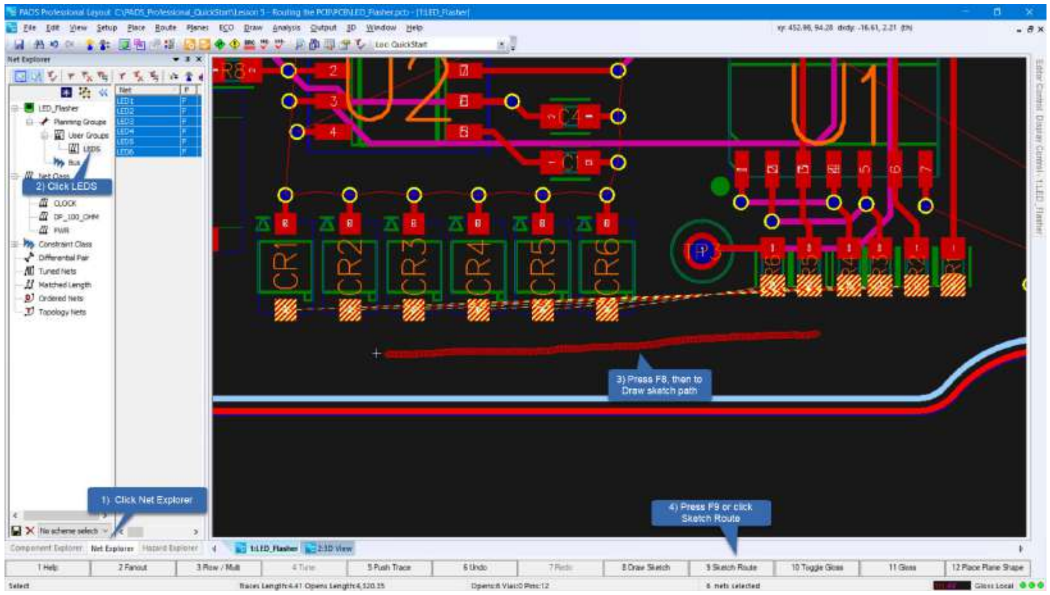
Sketch Route

Sketch Route allows you to select a set of nets, draw a sketch path, then let the routing engine route the selected nets based on the user defined path.

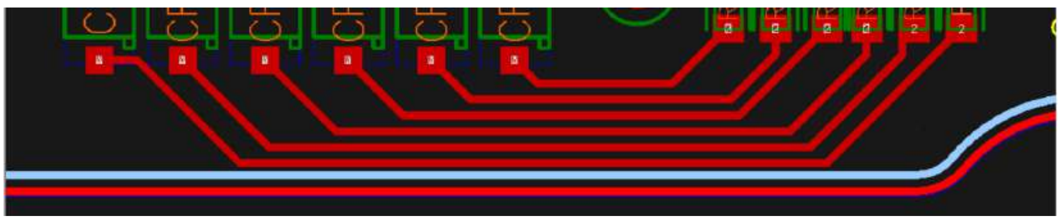
1) In the bottom left panel area, click the **Net Explorer** tab.
2) Within **Net Explorer,** under **user groups** click the **LEDS** group to select all LED nets *(Alternatively, draw a selection rectangle across the LED1-6 ratsnests directly in the PCB workspace).*

3) Click **Draw Sketch** on the bottom toolbar area (Or press **F8** key). Then left click once to start drawing the sketch path and once again to stop drawing.

4) Click **Sketch Route** (Or press **F9** key).



5) Allow the route engine to route the traces.

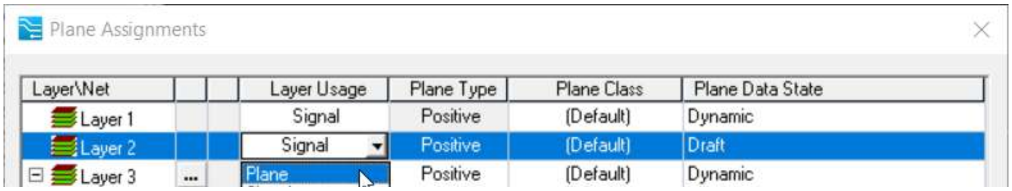


6) Save your work.

Defining Internal Plane layers and nets

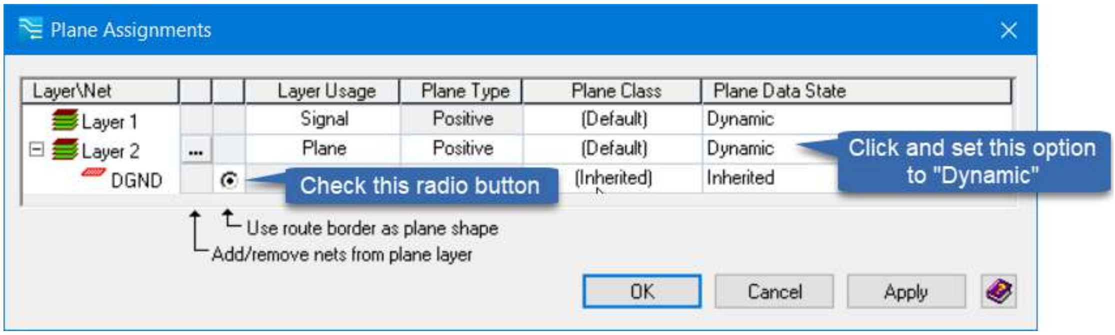
1) Go to: **Planes ► Plane Assignments.**

2) Within the **Plane Assignments** window, under the **Layer Usage** column for **Layer 2,** click on the keyword **Signal** and select **Plane.**

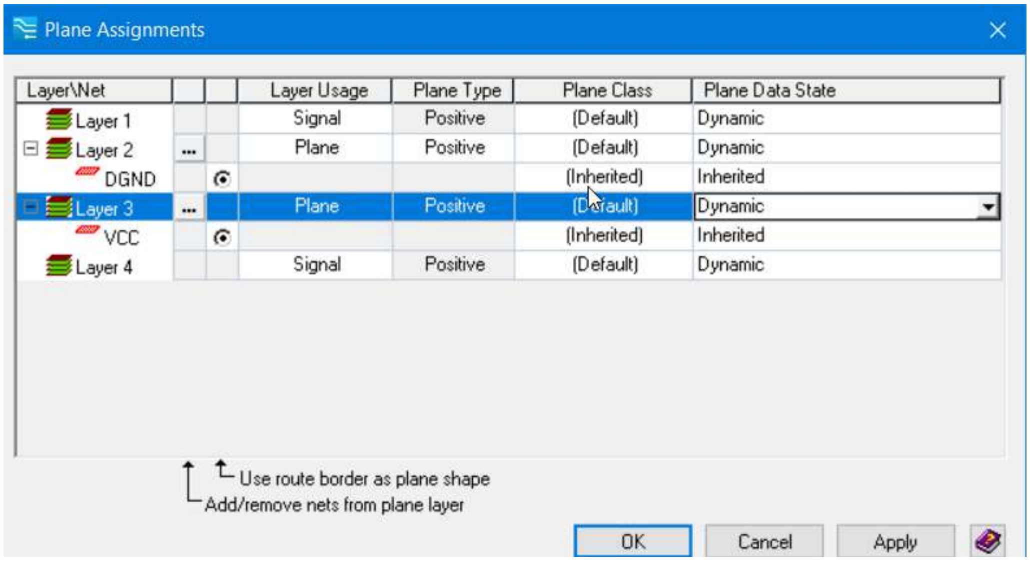


3) Click the dotted box next to **"Layer 2"** table entry, then assign it the **DGND** net as shown below.

4) Check the **radio button** to automatically add **Pullbacks,** then set the Plane Data State to **Dynamic.**



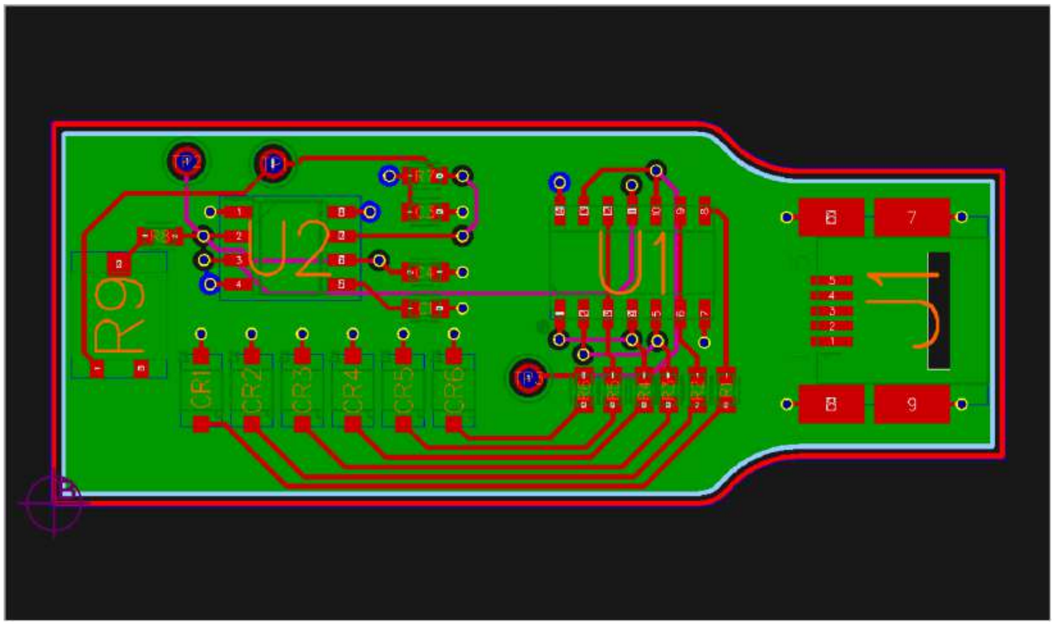
5) Repeat steps 2-4 to define **Layer 3** as a **Plane layer** assigned to the **VCC** net.



6) Click **"OK"** to apply your changes and dismiss the Plane Assignment window.

7) Go to: **View ►Fit Board** (Or Press **CTRL + B**).

8) You should have results similar to the image show below.

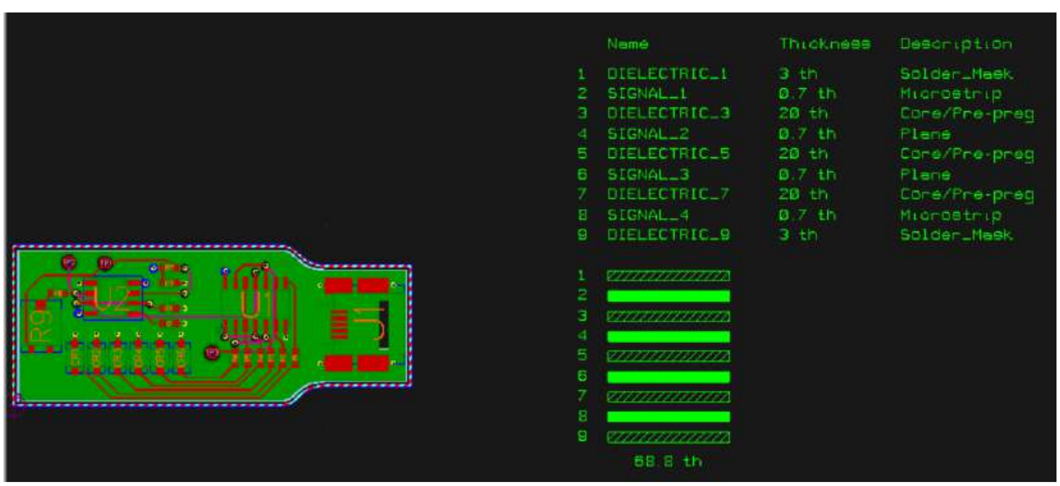


Add Layer Stackup Table

1) Go to: **Place ► Layer Stackup.**

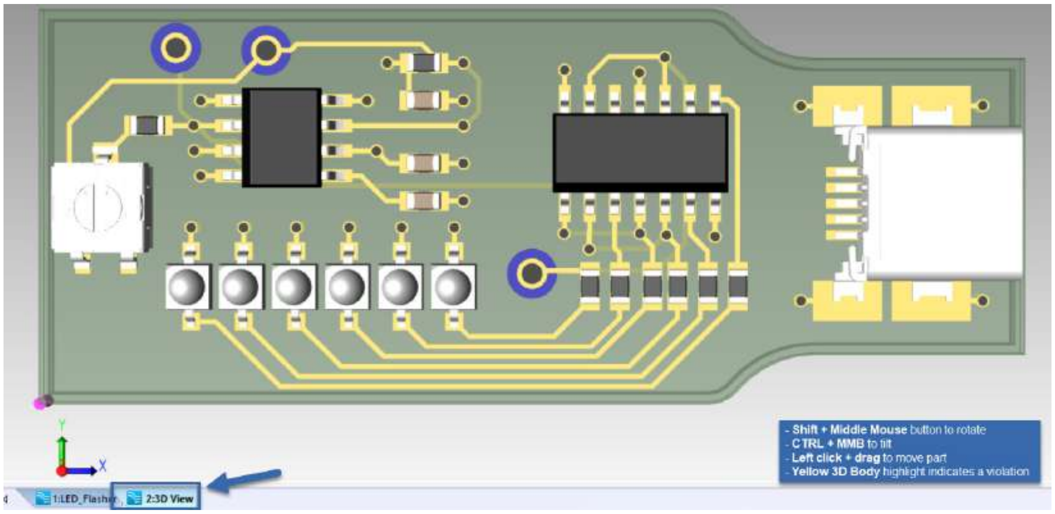
2) Click **OK.**

3) Left click to place the **Layer Stackup** table as shown below.



View Board in 3D

1) Click the **2:3D View** document tab to see a 3D rendering of your board.



tip: **While in 3D mode, you can also place parts including multi-body mechanical assemblies (such as product enclosures and mounting brackets). Live DRC rules can be customized to enforce desired placement behavior.**

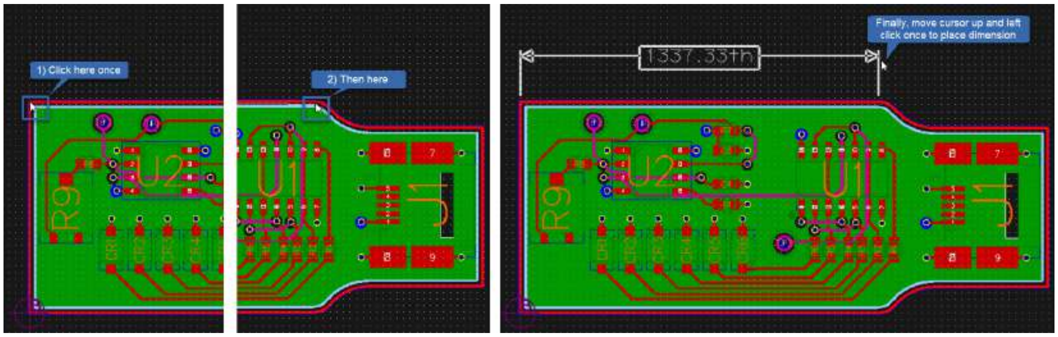
2) Go to: **File ► Close**, Click **"Yes"** to save your changes.

LESSON-6: CREATING FABRICATION AND ASSEMBLY FILES (VIDEO LINK)

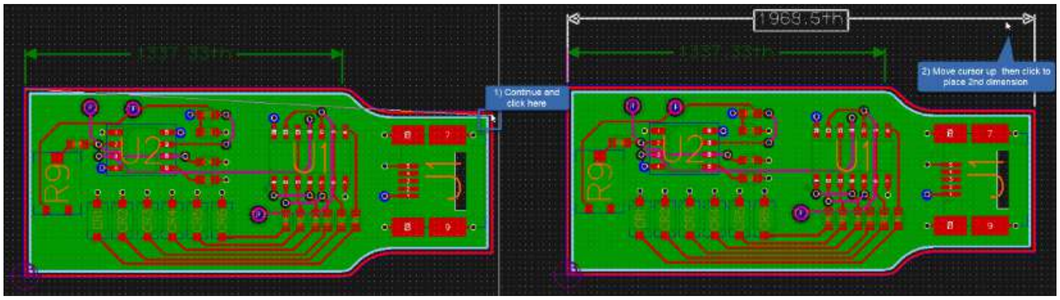
Board fabricators will typically require a minimum set of design files in order to build your board. In this exercise, we focus on adding dimensions to the board and generating a set of fabrication and assembly files.

Adding Dimensions

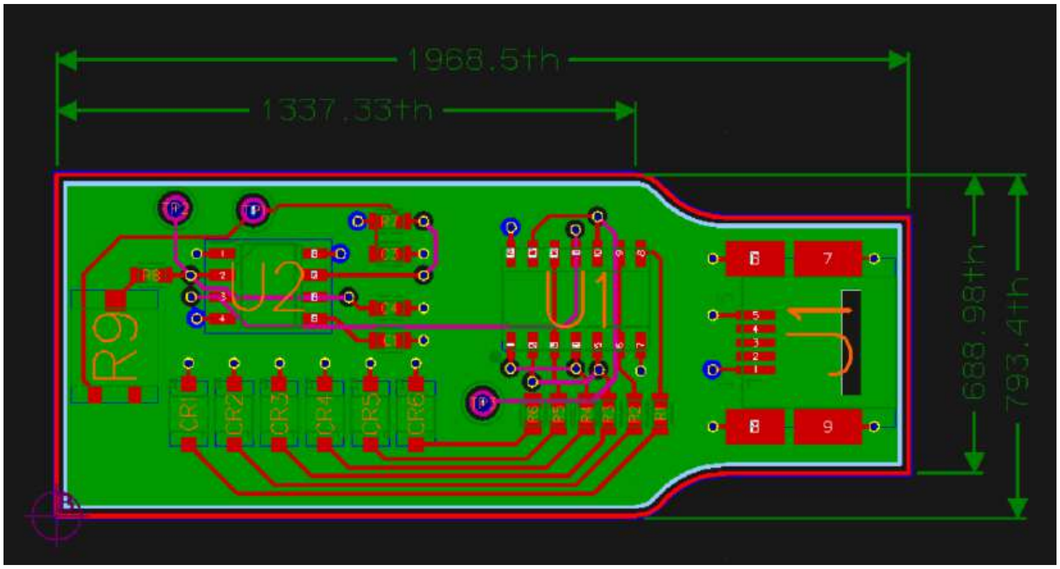
1) Within PADS Professional Layout, go to: **File ▶ Open.**
2) Navigate to **C:\PADS_Professional_QuickStart_VX.2.7\Lesson 6 - Creating Fabrication and Assembly Files\LED_Flasher.prj**, select the file named **LED_Flasher.PCB,** and click **Open.**
3) Go to: **Draw ▶ Dimension ▶ Stacked.**
4) Left click once on the top left corner of the board, then nearest edge to the right.
5) Move your cursor up to a desired height and click once again to place the dimension.



6) While the command is still active, click the furthest top right edge of the board and move your cursor up to place a second stacked dimension.



7) Press the **Escape** key twice to cancel the command.
8) Repeat steps **3-7** to dimension the board's height.



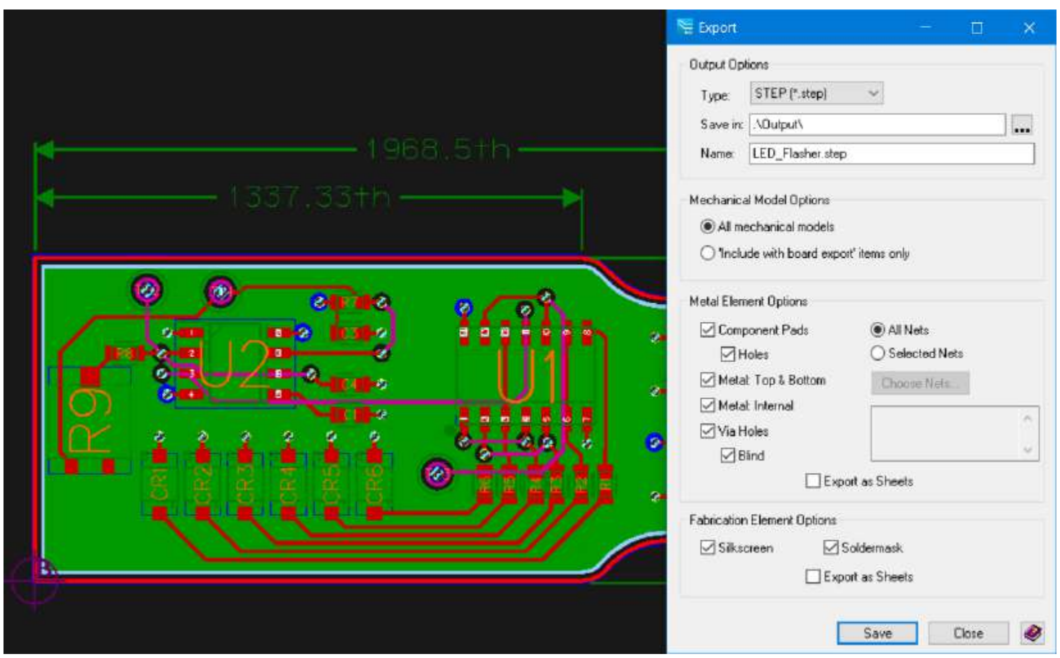
Exporting STEP 3D output

You can export **3D STEP** outputs of your board. This file can then be imported into MCAD software.

1) Go to: **3D ► Export.**

2) Set **Type** to **STEP.**

3) Under **Metal Element Options,** enable the options shown to allow those objects to be included in the STEP 3D export (Particularly copper traces, component pads and silkscreen information).



4) Click **“Save”** to export the 3D Step file to the **Output** directory of your project.

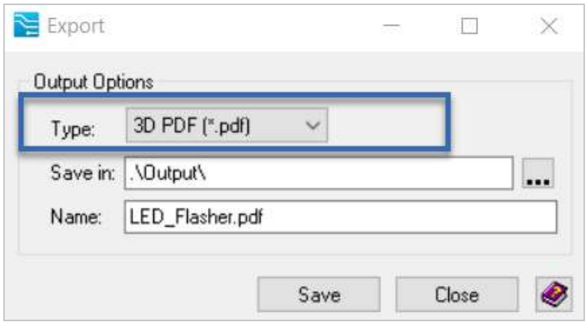
5) When the indicator stating **“Export succeeded”** appears, your STEP files have been successfully generated.

6) Click **“Close”.**

Exporting 3D PDF output

To generate a PDF document that includes a self-contained 3D model of your design.

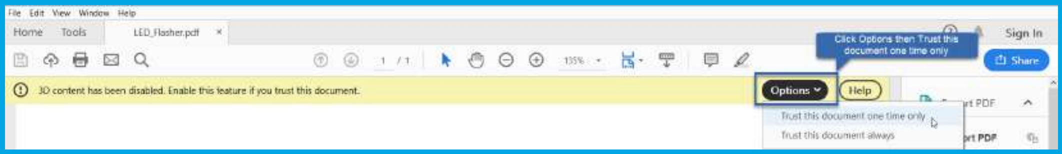
1) Go to: **3D ► Export.**

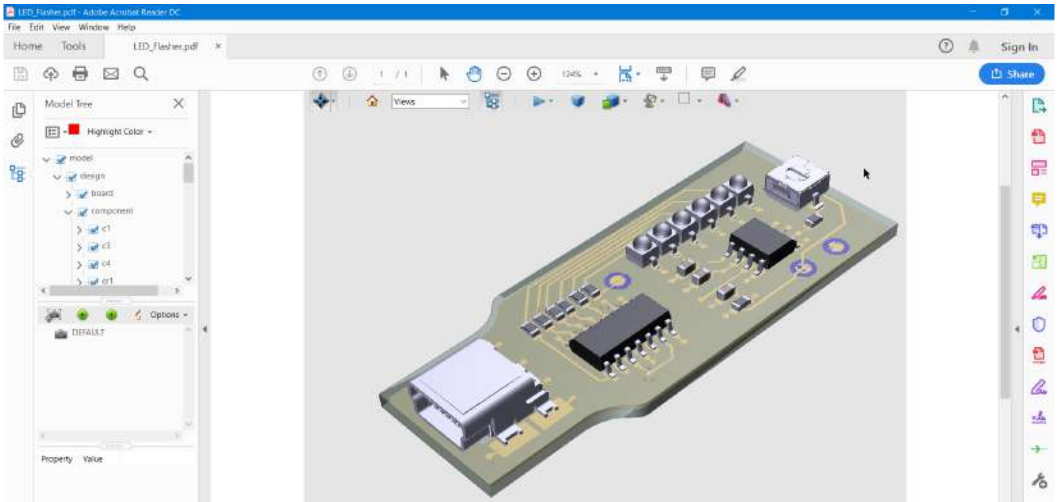


2) Set Type to **3D PDF.**

3) Click **“Save”** to generate the 3D PDF file in your project’s **Output** directory. Close the **Export** window.

tip: **The Output directory in your case is located in *C:\PADS_Professional_QuickStart_VX.2.7\Lesson 6 - Creating Fabrication and Assembly Files\PCB\Output*. Open the file named *LED_Flasher.PDF* to see the 3D PDF. Within the PDF document, you may need to allow execution of 3D content to allow execution of 3D content.**



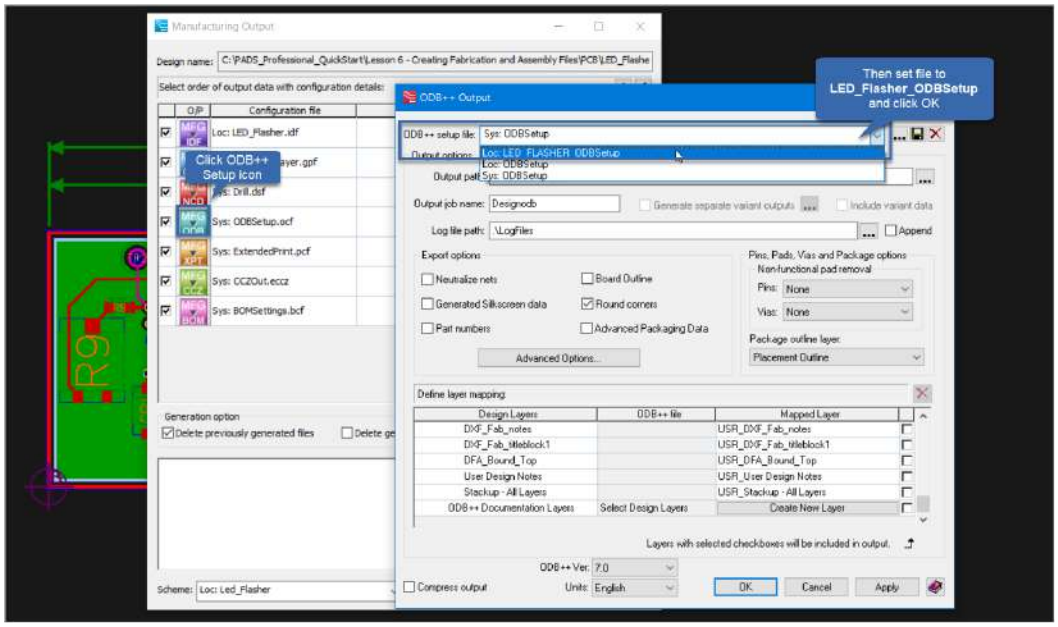


Configuring Batch output file generation

Instead of individually generating output files, most file types can be batch generated by PADS Professional.

> tip: **In PADS Professional, simply setup your output configuration for each output type (For example go to: *Output ► Gerber or Bill of Materials* and setup then save your desired options. You can later batch generate all configurations by going to *Output ► Manufacturing Output,* and click *Generate.***

1) Go to: **Output ► Manufacturing Output.**
2) Within the **Manufacturing Output** window, click the **ODB++ setup** icon.
3) Within the **ODB++ Output** window, set the **ODB++ Setup** file drop down to **LED_Flasher_ ODBSetup.**
4) This will load pre-configured ODB++ settings such as layer mapping and overlay configuration. You can save time by sharing these files with other users or make them part of a project template.

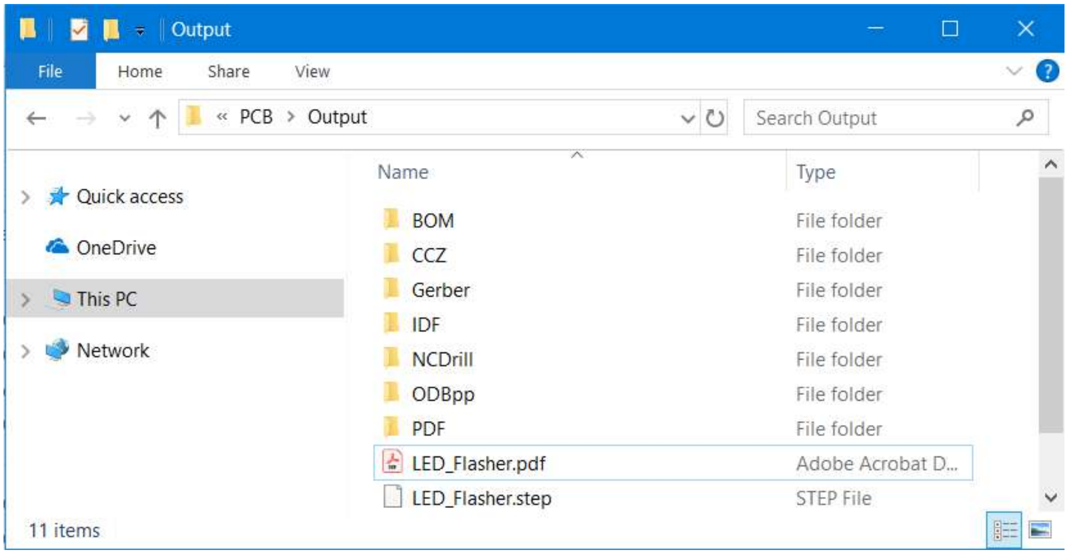


5) Click **"OK"**, then **"Generate"**

6) Click **"NO"** to save current settings if prompted.

tip: **You can also go to: Output ► ODB++, then click Generate. This method will automatically preview the ODB++ files for you to review before sending to the fabrication house.**

7) The system will begin generating the appropriate files and store them in your project **output** directory.



Congratulations, you've successfully completed the PADS Professional QuickStart guide.

note: **For experienced PCB designers looking for an in-depth evaluation of this product, please access the full evaluation manual from here *C:\PADS_Professional_Eval\PADSProfessionalEvaluationGuide.pdf***

Mentor®
A Siemens Business

8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.

Telephone: 503.685.7000 Toll-Free Telephone: 800.592.2210

Website: www.pads.com Support: http://support.mentor.com