

2019

PADS PROFESSIONAL STUDENT EDITION QUICKSTART GUIDE FOR VX 2.7 AND NEWER



TABLE OF CONTENTS

QuickStart Overview	3
Installation & Licensing (Video Link)	3
Lesson-1: Opening and populating a new project (Video Link)	4
Opening a new project:	4
Populating a new project	5
Lesson-2: Placing and Wiring Parts (Video Link)	6
Searching and placing parts on your schematic sheet	6
Wiring your schematic	8
Catching and fixing potential wiring mistakes	9
Placing Net Labels	11
Constraint Manager	11
Lesson-3: Forward Annotate, Create a PCB file and Draw your board shape (Video Link)	13
Forward Annotate to PCB	13
Define PCB board shape from 3D Step model	15
Lesson-4: PCB Physical Layout (Video Link)	18
Placing footprints from schematics	18
Placing Footprint in 2D/3D Side by side	19
Lesson-5: Routing the PCB (Video Link)	22
Adding Fanouts	22
Single Route	23
Sketch Route	23
Defining Internal Plane layers and nets	24
Add Layer Stackup Table	26
View Board in 3D	26
Lesson-6: Creating Fabrication and Assembly Files (Video Link)	28
Adding Dimensions	28
Exporting STEP 3D output	29
Exporting 3D PDF output	30
Configuring Batch output file generation	31

1



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.



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

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

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

O[†]C: 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.





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).

NO[†]C: 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.

earch								* Q 3
X Ø V NE555			#	QD	 . 		Preview	→ # >
Cell (al)> # SOIC127P600X175-8N-PC	م	Description (all) IC OSC SGL	TIMER 100KHZ	8-SOIC	, «	•		1 8
					,	×	<u>стике</u> сонт _т <mark>З</mark> 2-тис в	6



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 I 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.





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".







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.

I ≤ B = 2 · 2 · 4	13 🗃 Rž 📜 📖 🗤		# 1. 1	to service	0 10 10 L				
avigator 🚽 🎝 🗙 📾 Schemes	Schomo/Not Class / aver	Index	Timo	Vi	• Accientation	Routo	т	race Width (th	ı)
Net Classes	Soliona, not statistica yet	Index	(),,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,		artioriginitorito	Houto	Minimum	Typical	Expansion
EB (Lensus)	H (Mester)	27		1			4	4 8	
CCCCCHM	🗄 👪 PWR	a second and	and the second se	202	(clefoult)		8	10	1
PWR	SIGNAL_1	-					8	10	1 1
Dearances	SIGNAL_2	CIICK "PWI	<				8	10	11
Constraint Casses	SIGNAL_3	3	Plone				8	10/	1
1 Parte	SIGNAL_4	4	Signal				8	Channelle	1
	🗄 🕼 (Minimum)							Change ini	s
	E B PWR							value to "20	1
Expand Net Classes,	E 🕼 FinePitch								
then click "PWR"	E B PWB				(default)	М	5	10	1

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.

Output	* # X
Darts/ProjectBackup/backups/2018-09-20 13.55.15 - auto.zp) Started C.'MentorCraphics/PADSProVX.2.4/SOD_HOME's common/win32\bin/ces.exe /SnapshotName DxD /TopBlockName "LED_Flasher" /Context DxDesigner /PrjPath "C:\PADS_Professional_QuickStartLesson 2 - Placing and Wiring parts/LED_Flasher-prj" Finished C:'MentorCraphics/PADSProVX.2.4/SDD_HOME's common/win32\bin/ces.exe /SnapshotName DxD /TopBlockName "LED_Flasher" /Context DxDesigner /PrjPath "C:\PADS_Professional_QuickStartLesson 2 - Placing and Wiring parts/LED_Elasher.exe"	^
Finished C:\MentorGraphics\PADSProVX.2.4\SDD_HOME\common\win32\bin\ces.exe Started C:\MentorGraphics\PADSProVX.2.4\SDD_HOME\common\win32\bin\ces.exe C:\PADS_Professional_QuickStart\Lesson 2 - Placing and Winip parts\ED_Flasher.ptp Finished_C:\MentorGraphics\PADSProVX.2.4\SDD_HOME\common\win32\bin\ces.exe Finished_C:\MentorGraphics\PADSProVX.2.4\SDD_HOME\common\win32\bin\ces.exe	v
Dutput Search	

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**".

reate New Design	×	
Design Create new design Central Library:		
Design Technology: PCB Template:	This	PCB template includes manufacturing notes
4 Layer QuickStart_Template	Browse	
ОК	Cancel	

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".

PADS	Professional Layo	ut	
?	New changes are Annotation. Do y Annotate now?	e ready for Forward you want to Forward	
	Yes	No	
	Don't show the	is message again	

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

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.



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.

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
- 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.



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"**.

Nap Hole Features					×
Define the mapping between the PCB. Holes of type <discard> will view to cross-probe with this dialo</discard>	holes in the selected face not be added to the board g.	and the type of hole th d outline. You can click	ey should be , on a hole fe	acome in th sature in th	ie e 3D
Interred Hole Data	Hole Type	Hole Attribute		High	light
Delete Previously Created Items Cavilies Contours Mounting Holes	Route Border	(th) Se Ho	it the Hol le Attribul	e Type a te as sho	and own
		OK Cano	el	Apply	۲



- 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.

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 4 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.



TADS Protestand Layout CalM25 Protestional Oal	tissen Lesson 4 - Pot tissen graw gradysi 🕐 🕐 💷 🖤	Physical Legendrich galpat generative P Bi III = 1	w Brb C Loc Quedistant	2)	1			ay: 1,420, 395 d	udy: 0, 0 (M)	- o x
Component Explorer + 3 X	S 200 famor	Designal			(= W U	140 Fist	e			
tell plane Convitor Convitor	Left click once designator to sele		J11 VEUS DATA- DATA- ID GND MTG1 MTG2 MTG3 MTG4		13	ŀ		as F3 to rotate floorp		
	ID_Faster(Cove	r) LED_Rasher/Desig	4		Move mouse to layout space		->			
Companient Explaner Hiet Explorer Hazard Explorer	d 🔛 LEED, Fla	iher 8 2180 Plan	en Designi	There are a contact	1		1			P.
Thep 12 3	10044-00	ruter 100	s ruih	E repto Segnere	THE LEADER	Employ A	a ratio spectational	10 Shap to Get		72/

Placing Footprint in 2D/3D Side by side

- 1) Close the Schematic window. 😤 2:LED_Flasher(Design) 📃 💷 🎫
- 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.





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 multibody 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.





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

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.



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.
- 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**.

Plane Assign	ments					
Layer\Net	TT	Layer Usage	Plane Type	Plane Class	Plane Data State	
📕 Layer 1		Signal	Positive	(Default)	Dynamic	
Layer 2		Signal 💌	Positive	(Default)	Draft	
E 🚍 Laver 3		Plane	Positive	(Default)	Dynamic	



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

Plane Assignments		Nets	
		Excluded:	Included:
Click here once	Layer Usage	Net Name	/ A Net Name
ELayer 1	Signal	\$1N1123	
Laver 2 🙀	Plane 👻	(Net0) Select D0	GND
Laver 3	Signal	CLK_OUT	
Lauer 4	Signal	DGND	
- advert		LED1	Then click arrow to add
		LED2	net and click "OK"
		LED3	
		LED4	
		LED5	
		LED6	
		VCC	~
			OK Cancel 🧶

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

Layer\Net			Layer Usage	Plane Type	Plane Class	Plane Data Sta	ate	
📕 Layer 1			Signal	Positive	(Default)	Dynamic		
🗉 🛑 Layer 2			Plane	Positive	(Default)	Dynamic 🥣	Click and set this option	
DGND 📀		•	Check thi	s radio button	(Inherited)	Inherited	to "Dynamic"	
	Ĺ	€_Us Add/re	e route border as move nets from p	plane shape lane layer	ОК	Cancel	Apply 🔗	

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

ayer\Net			Layer Usage	Plane Type	Plane Class	Plane Data State	
📕 Layer 1			Signal	Positive	(Default)	Dynamic	
Elayer 2			Plane	Positive	(Default)	Dynamic	
CGND	-	•			(Inherited)	Inherited	
Layer 3			Plane	Positive	(Darault)	Dynamic	
WCC		•			(Inherited)	Inherited	
🚍 Laver 4			Signal	Positive	(Default)	Dynamic	
	+	+					



- 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.





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.

	-		
Export succeeded.	Save	Close	1

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.**

Туре:	3D PDF (*.pdf)	~		
Save in:	.\Output\		_	
Name:	LED_Flasher.pdf			

- 2) Set Type to **3D PDF.**
- Click "Save" to generate the 3D PDF file in your project's Output directory. Close the Export window.

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.

 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.



Select order o	Configuration file	DDB++ Output		LE	D_Flasher_ODBSetup and click OK
	Configuration file				and click OK
	and any attention off				
MICH.	OC: LED_Passner.ktr	ODB++ setup file: Sys: ODBS etup			
	k ODB++	Dutout ontions Loss LED FLASHE	R OD9Setup		
Se	stup icon	Dutout patt Sys: ODBSetup			
	vs: Drill.dsf	Annual		(and	
AND DESCRIPTION		Output job name: Designodb	Generale separa	le variant cutputs	Include variant data
	Sys: ODBSetup.ocf	Log life path: \LogFiles			Append
🔽 👩 🔽 📰 s	Sys: ExtendedPrint.pcf	Export options		Pins, Pads, Vias and Pa	ckage options
	200602300044			Non-functional pad re	moval
M y S	Sys: CC2Out.eccz			Pins: None	~
s 💭 🤉 🐜 📢	Sys: BOMSettings.bcf	Generated Silkscreen data	Round corners	Vist: None	-
Incent of		Part numbers	Advanced Packaging Data	and the second second	
				Package outline layer	
		Advanced	Options	Placement Liutine	~
and a star		Define layer mapping			*
		Design Layers	ODB++ file	Mapped Laser	
Delete po	evicusiv generated files Delete ge	DXF_Fab_notes	i i	ISR_DXF_Fab_notes	E
(C) server pro		DVF_Fab_titleblock1		ISR_DXF_Fab_titleblock1	
		DFA_Bound_Top		ISR_OFA_Bound_Top	C
		User Design Notes		ISR_User Design Notes	
		Stackup - All Layers		ISR_Stackup · All Layers	E

- 5) Click "OK", then "Generate"
- 6) Click "NO" to save current settings if prompted.



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.

📕 📝 📕 = 🛛 Output			- 0	×
File Home Share	View			~ ?
$\leftarrow \rightarrow \checkmark \uparrow$ \blacksquare « PCB	> Output	~ Ŭ	Search Output	,
	Name		Туре	^
Quick access	BOM		File folder	
🛆 OneDrive	CCZ		File folder	
> 🔄 This PC	📜 Gerber		File folder	
> 🕩 Network	JDF		File folder	
	NCDrill		File folder	
	ODBpp		File folder	
	PDF		File folder	
	LED_Flasher.pdf		Adobe Acrobat D	
	LED_Flasher.step		STEP File	~
11 items				



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



© 2019 Mentor Graphics Corporation

All Rights Reserved 8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777. Telephone: 503.685.7000 Toll-Free Telephone: 800.592.2210 Website: <u>www.pads.com</u> Support: <u>http://support.mentor.com</u>

TRADEMARKS: The trademarks, logos and service marks (Marks) used herein are the property of Mentor Graphics Corporation or other third parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the respective third-party owner. The use herein of a third-party Mark is not an attempt to indicate Mentor Graphics as a source of a product, but is intended to indicate a product from, or associated with.