

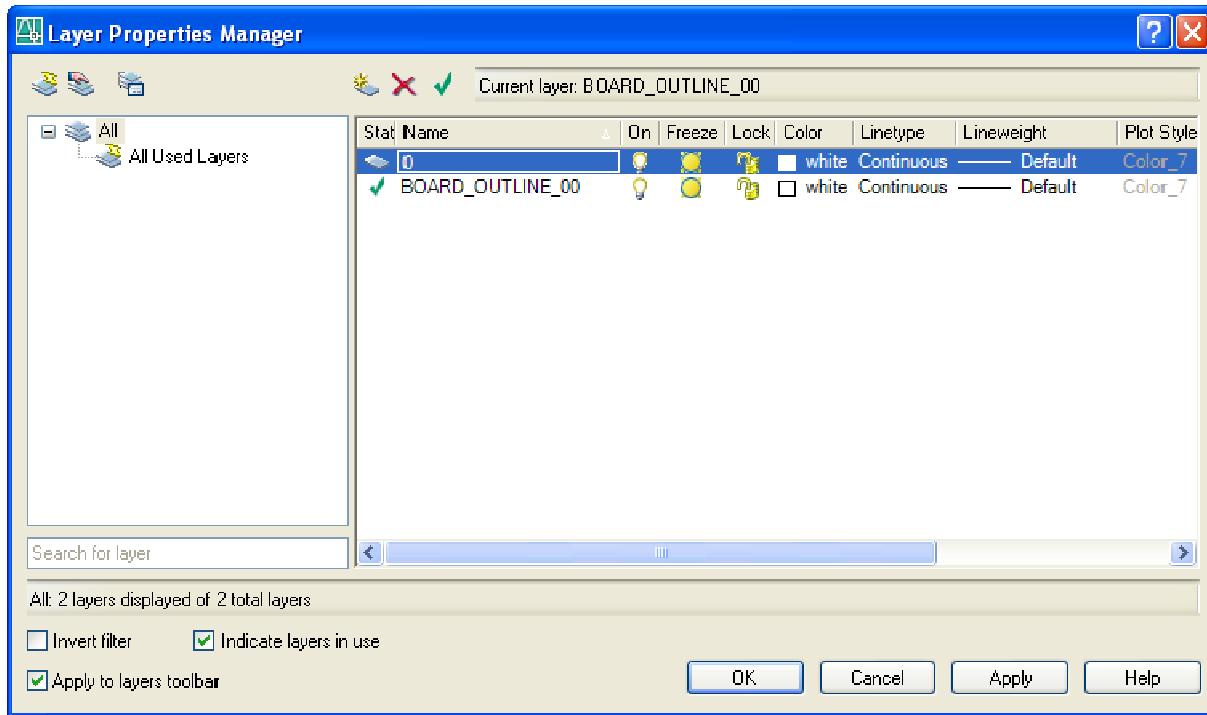
TECHNICAL APPLICATION NOTE

Overview




- This document outlines the steps involved in creating a drawing in AutoCAD and translating the information via PADS DXF option into PADS PowerPCB. It is assumed the user is familiar with the AutoCAD product.

Course of action:

- Open PowerPCB and set up your design default, such as grids, units, display, and layers.
- Choose File/Export and export the design in DXF format.
- Start AutoCAD
- Choose File Open within AutoCAD and import the DXF file you just created in PowerPCB
The PowerPCB working area appears within the AutoCAD drawing area.
- Go into *Data/Layers* menu to verify a “New” layer or layer names,
Example: BOARD_OUTLINE_00. Below is a picture of how it will appear in ACAD.
This will also show all the other Powerpcb appropriate layer naming structure.



- At this point you can then exit the menu and set BOARD_OUTLINE_00 as your default layer using the *Layer Control* button.
- We are now ready to draw the board outline, and we will use the *Draw Polyline* routine to do this. Simply draw a rectangular shape on the screen. You will also have the option of setting the width as well when doing this. Complete the board outline with the *Close* command that appears on your prompt as one of the options when drawing a polyline object.
- Remember that only one “closed” polyline object can be used for the board outline since PADS PowerPCB only allows one to be present. Cutouts can be done as 2D-Line draws so that they can come over correctly to our software.
- The following picture is how the “properties” of the board outline should look in ACAD. Use the *Edit/Properties* menu and select the outline you had just created to have this occur.

Polyline




General ^

Color	<input type="checkbox"/> ByLayer
Layer	BOARD_OUTLINE_00
Linetype	————— ByLayer
Linetype scale	1.0000
Plot style	ByColor
Lineweight	————— ByLayer
Hyperlink	
Thickness	0.0000

Geometry ^

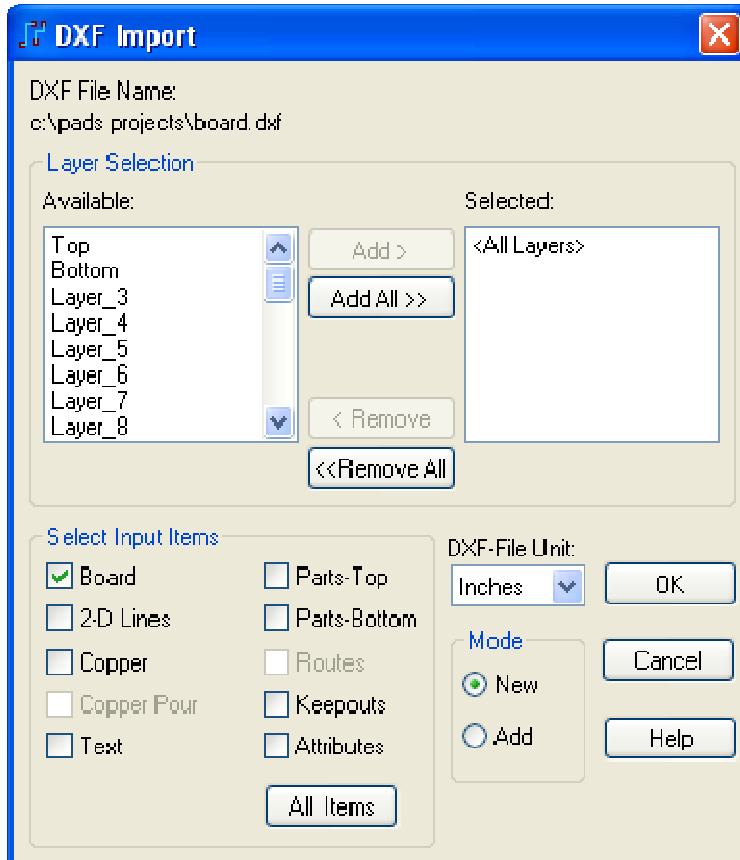
Vertex	1
Vertex X	12.1426
Vertex Y	24.8853
Start segment width	0.0000
End segment width	0.0000
Global width	0.0000
Elevation	0.0000
Area	296.8896
Length	69.5950

Misc ^

Closed	Yes
Linetype generation	Disabled

- Once the board outline has been created and everything has been set correctly, we then can create a DXF file that can be used by the PADS DXF translator to bring it into PowerPCB. Use *File/Export* from the ACAD menu and select the option for a DXF type file. Set the decimal accuracy to 10 when asked to assure correct precision of data.

- At this point PADS PowerPCB can be invoked and a empty screen should appear once you are in the software. Using *File/Import* select the file type to be (*.DXF) and point to the location of the DXF file that was created from ACAD and select it. Proceed by selecting OK/Open so that the following menu appears on your screen.



- In this menu only the *All Layers* and *Board* items need to be selected and the *DXF file unit* set to what was used in ACAD. Select the OK button to proceed which should then bring in the board outline that was done in ACAD.

Note: Selected input items will vary depending on what you have drawn and are trying to translate to Powerpcb.

Possible Problem Situations

- *Text is visible on the board, but you can not see it in preview or when you verify for a laser print.*
Check the width of the text. If text width is zero, change it to proper width to meet your specifications. You should at that point be able to see text in preview and also when you then verify.

THE FOLLOWING IS A LAYER TO LAYER CONVERSION CHART FOR PAD'S POWERPCB TO AUTOCAD

Layers

PADS DXF attempts to preserve as much of the PADS information as possible. The translator maintains all your defined colors except the background color.

PADS Entity	AutoCAD Entity/Layer:
Text	Text on layers TEXT_nn.
2D Lines	Polylines of real width on layers 2D_LINE_nn.
Copper	Polyline outline of real width on layers COPPER_nn.
Copper Cutout	Polyline outline of real width on layers COPPER_CUTOOUT_nn.
Copper Pour	Polyline outline of real width on layers POUR_HEADER_nn. Create a block called POUR_HEADER_0. Copper pour flood data brought in from PADS can be seen by performing a Hatch using a solid fill.
Copper Pour Cutout	Polyline outline of real width on layers POUR_VOID_nn.

Hatch Outline	Polyline outline of real width on layers POUR_OUTLINE_nn.
Hatch Void	Polyline outline of real width on layers POUR_VOID_nn.
Thermal Relief	Polyline segments of real width on layers POUR_VIATHERM_nn for vias, on layers POUR_PADTHERM_nn for terminals.
Board	Polyline outline of real width on layer BOARD_OUTLINE_00. Create a Block called BOARD_1 from the outline shape.
Board Cutout	Polyline outline of real width on layer BOARD_CUTOUT_00 (If a board cutout exists it must be combined with the board outline as a Block called BOARD_1). There is some type of problem in AutoCAD in which the Block for a board cutout and board outline must be made, broken and then remade again in order for it to import into PowerPCB correctly.
Connections	Lines on layer LINK_00
Padstacks	Polyline outline of real width on layers PADS_INNER, PADS_BOT, PADS_TOP, DRIL_PLTE_THRU_nn, DRIL_NPLTE_THRU_nn, DRIL_PLTE_PRTL_nn, DRIL_NPLTE_PRTL_nn.
Component Name	Text on layers PART_NAME_TOP and PART_NAME_BOT
Parttype Name	Text on layers PART_TYPE_TOP and PART_TYPE_BOT
Combined Items	Combined 2-D lines/Text are converted into a block.
Decals/Parts	Become blocks on layers PART_TOP, PART_BOTTOM.
Outlines/Text	Sub blocks on layers BODYnn.
Parttypes	Blocks on layer PARTINFO
Traces	Polylines of real width(s) on layers TRACE_nn.

Vias	Blocks on layer VIA
Signals	Linetype with name and information.
Layer information (display colors, type of layer, thickness, etc.)	Text on layer DBLAYERS
Teardrops	Blocks on layers TEAR_nn
Decal names	SYM_NAME
General PCB information	Text on layer PCB_PARAMS
Jumper silkscreen	Polyline on level JUMPER_BOX

Keepout:

A keepout should be drawn as a closed Polyline on layer KEEPOUT_nn (KEEPOUT_01 would create a keepout on layer 1). The settings for the keepout are based on a unique line type Keepout_ABCDEFGH. The keepout should also be made into a Block in the format of Keepout_x (Keepout_1, keepout_2, for each successive keepout that you create, this does not determine the layer it is placed on, keepout_nn does).

A - is the tick mark for Placement

B - is the tick mark for Component Height

C - is the tick mark for Component Drill (this can only be used if you assign the keepout to All Layers)

D - is the tick mark for trace & copper

E - is the tick mark for copper pour and plane area

F - is the tick mark for via and jumper

G - is the tick mark for testpoint

H - is the value for the component height when used in conjunction with A & B (the value is set in database units, 1 mil = 38100 database units)

For the Keepout tick marks, a 0 means no check mark and a 1 means there is a check mark.

Note: Either the Copper Pour and Copper Pour Cutouts, or the Hatch Outlines and Hatch Voids, are exported to the DXF file. This depends on whether the Pour Outline or the Hatch Outline is selected in the Copper Pour Preferences in PowerPCB.

Note: Assigning unique colors in AutoCAD may result in translation back to PADS as black if the color is not one of the PADS 16 colors.

Following is a cross-reference of the DXF layer names used in the various releases of PADS products. We don't recommend trying to bring an old DXF file into a new release of PowerPCB???

nn refers to a layer number 00-31???

PERFORM

PowerPCB V1.1-V1.3

PowerPCB V1.5 – V3.6

TXTnn	TEXTnn	TEXT_nn
LINnn	2D_LINenn	2D_LINE_nn
COPnn	COPPERnn	COPPER_nn
CCOnn	COPPER_CUTOOUTnn	COPPER_CUTOOUT_nn
CPRnn	POUR_UNFLOOD	POUR_VOID_nn
CPOnn	POUR_FLOOD	POUR_HEADER_nn
HOLnn	HATCH_OUTLINEnn	POUR_OUTLINE_nn
HVOnn	HATCH_VOIDnn	POUR_VOID_nn
THVnn	VIA_THERMALnn	POUR_VIATHERM_nn
THRnn	PAD_THERMALnn	POUR_PADTHERM_nn
BRD00	BOARD_OUTLINE	BOARD_OUTLINE_00
CONnn	CONNECTIONnn	LINK_00
PADXinn	PADS_INNER	PADS_INNER
PADXBnn	PADS_BOTTOMnn	PADS_BOT
PADSTnn	PADS_TOPnn	PADS_TOP
PNM00	PART_NAME_TOP and PART_NAME_BOTTOM	PART_NAME_TOP and PART_NAME_BOT
CMB00	PART_TOP	PART_TYPE_TOP
COM01	PART_BOTTOM	PART_TYPE_BOT
BDYnn	BODYnn	
DRLTPnn	DRIL_PLTE_THRUnn	DRIL_PLTE_THRU_nn
DRLTN	DRIL_NO_PLTE_THRUnn	DRIL_NPLTE_THRU_nn
DRLNP	DRIL_PLTE_PRTLnn-nn	DRIL_PLTE_PRTL_nn

DRLNN
TRKnn
VIAi
PKGnn
PCB30
KEY00
CTBnn

DRIL_NPLTE_PRTLnn-nn
TRACKSnn
VIAi
PARTINFO
PADS_PARAMETERS
DRILL_SYMBOL
COMPONENT_TEXTnn

DRIL_NPLTE_PRTL_nn
TRACE_nn
VIA
PART_INFO
PCB_PARAMS
DRIL_SYMBOL
PART_TOP_TEXT_nn and
PART_BOT_TEXT_nn
DBLAYERS
TEAR_nn
JUMPER_BOX
JUMPER_INFO