Part/Device Creation Issue
9.2.2, Hobby Version

Hello,

Couple of issues I hope you can answer as I am stuck and can't seem to repeat what I did last week. I am new to Eagle and trying to learn the rules. I have not seen any responses from anyone about what I am doing wrong but a week before I believe Rachel and Jorge responded..now, with a simple issue I can't seem to see, I am seeing nothing from no one. How can I ever think about going to 4 layers if I can't get support on 2 layers so I can learn the Eagle rules?

1) Why is the topic of my question grayed out?
2) I am having trouble creating a single pin/pad part after having done a few previously
3) The issue is the Symbol PIN name never shows up in the editor but the Footprint PAD name does (the SYMBOL doesn't show up)
4) I just want to create a solder pad from scratch.
5) Attached is the SYMBOL attributes and the FOOTPRINT attributes and the view from the editor
6) I have been a long time user of Orcad from the DOS environment until their final 9.2 Win version…I always bought the updates. I understand the concept of schematic pin reference and the footprint pin relationship

I have Teamviewer if anyone wants to take over my PC to show me..
Thank you for any support
Jay Kobelin
jkobelin@gmail.com

Search

EAGLE 9.2 has arrived and is loaded with productivity enhancements!

Latest post on 10-12-2018 02:00 PM by 3 fact

EAGLE Forum now also in German Language Available 🔗

by 🔺 RichardHammerl on 02-06-2017 04:10 AM | Latest post 3 weeks ago by 3 okradew40

2 REPLIES   4170 VIEWS

Welcome to the Autodesk EAGLE Forum 🔗

by 🔺 sam.sattel on 01-19-2017 03:27 PM | Latest post yesterday by 🔺 edwin.robledo

2 REPLIES   2612 VIEWS

SamacSys plugin in Eagle 9.2.2

by 6 rg.jones 49m ago

0 REPLIES   6 VIEWS

pads with multiple holes

by 3 k.mcnabb yesterday | Latest post 4 hours ago by 3 k.mcnabb

3 REPLIES   24 VIEWS

PART SYMBOL NOT AVAILABLE DURING DEVICE BUILD

by 2 jkobelin yesterday | Latest post 5 hours ago by 2 jkobelin

3 REPLIES   33 VIEWS
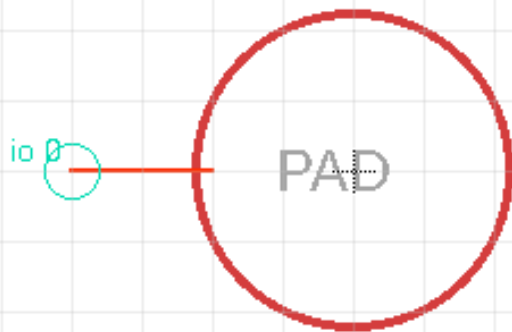
Script for showing parts 🖼

by 5 vyvojhw 6 hours ago | Latest post 5 hours ago by 5 vyvojhw

4 REPLIES   27 VIEWS

All 3 replies are mine

>VALUE

>NAME

io 0

PAD

Properties

Pin

Name	PAD
Position	-0.4 0
Angle	0 ▾
	☐ Mirror
Direction	io ▾
Swap Level	0 ▴▾
Length	middle ▾
Function	none ▾
Visible	both ▾

OK   Cancel   Apply

SYMBOL

Visible Layers

Filter: All Layers ▾

👁	#		Name	✕
	88	🟪	SimResults	
	89	🟪	SimProbes	
	90	🟧	Modules	
	91	🟩	Nets	
	92	🟦	Busses	
👁	93	🟩	Pins	
👁	94	🟥	Symbols	
👁	95	⬜	Names	
👁	96	⬜	Values	
	97	⬜	Info	
	98	🟨	Guide	

New Layer   Show Layers   Hide Layers

Layer Sets ▾   New Set   Remove Set

OK   Cancel

>VALUE

96

>NAME

95

93

io 0

94

PAD

94
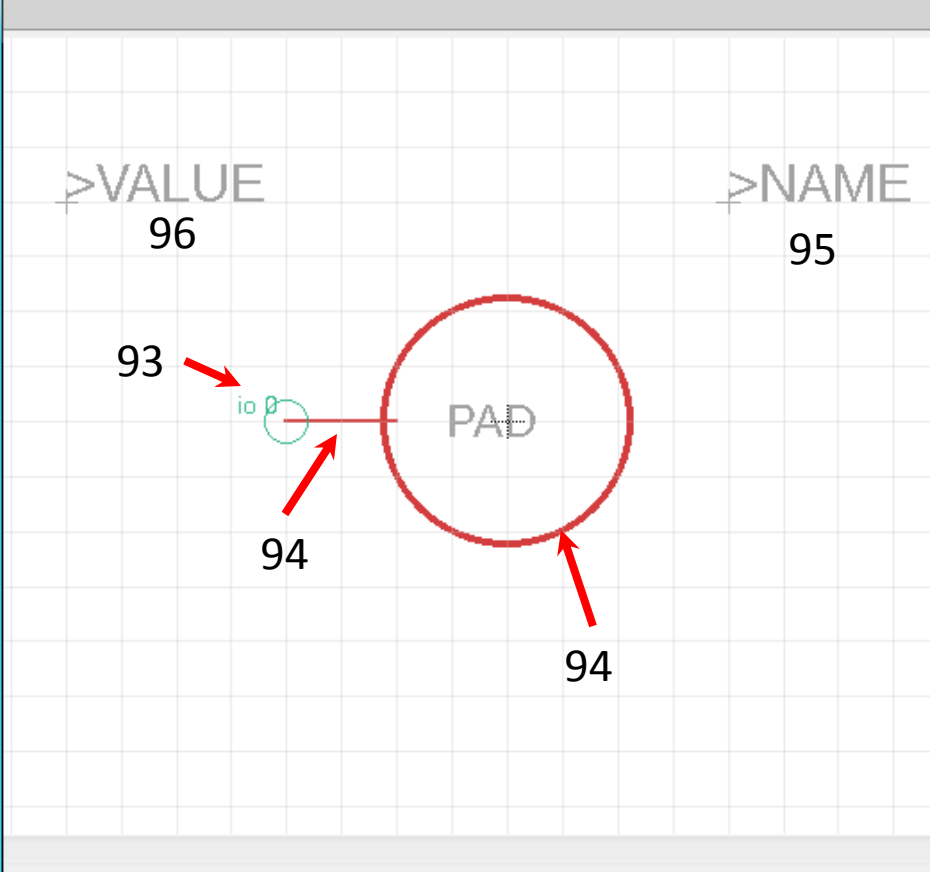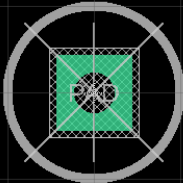
SYMBOL

>VALUE >NAME

Properties

Pad

Name	PAD	
Position	0	0
Angle	0	
Mirror	☐	
Diameter	auto ▼	
Drill	43 ▼	
Shape	square ▼	
Thermals	☑	
Stop	☑	
First	☐	

OK   Cancel   Apply

Visible Layers

Filter: All Layers ▼

#	Name
1	Top
16	Bottom
17	Pads
18	Vias
19	Unrouted
20	Dimension
21	tPlace
22	bPlace
23	tOrigins
24	bOrigins
25	tNames
26	bNames
27	tValues
28	bValues
29	tStop
30	bStop
31	tCream
32	bCream
33	tFinish
34	bFinish
35	tGlue
36	bGlue
37	tTest
38	bTest
39	tKeepout
40	bKeepout
41	tRestrict

New Layer   Show Layers   Hide

Layer Sets ▼   New Set   Rem
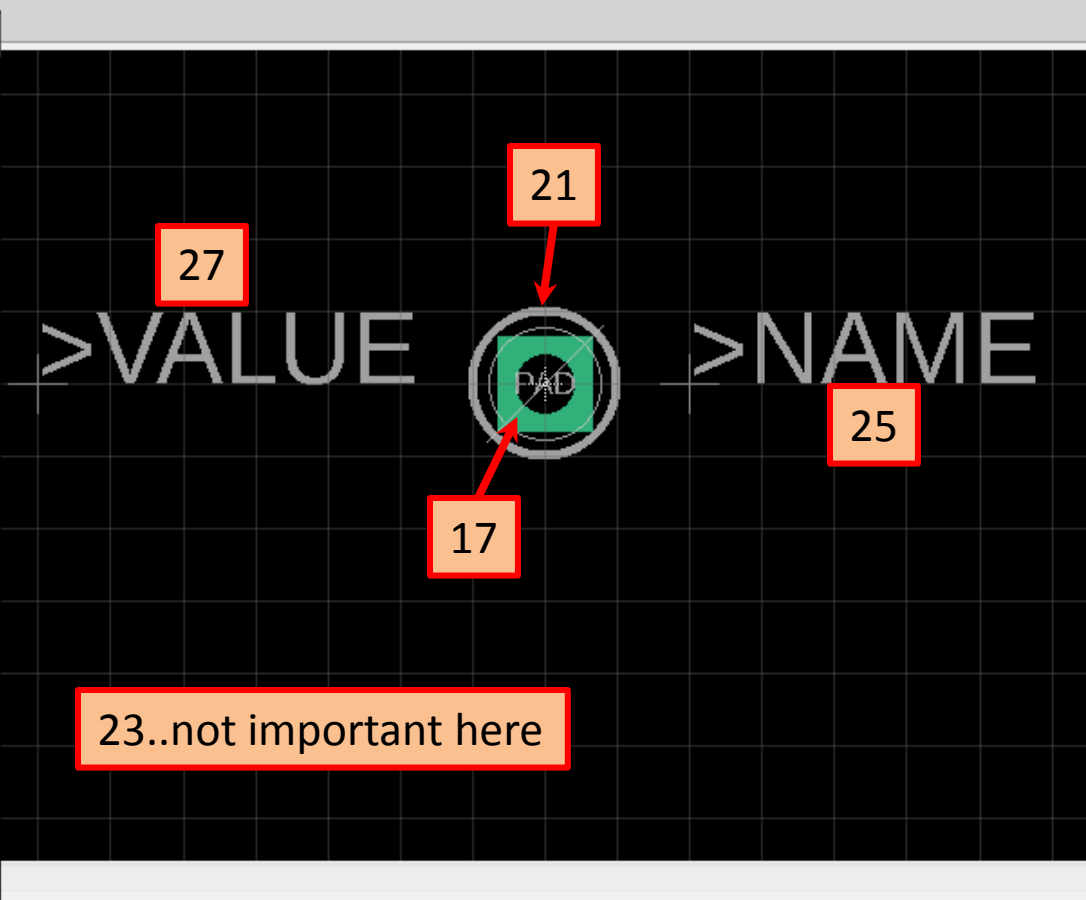
OK

FOOTPRINT

Visible Layers

Filter: All Layers

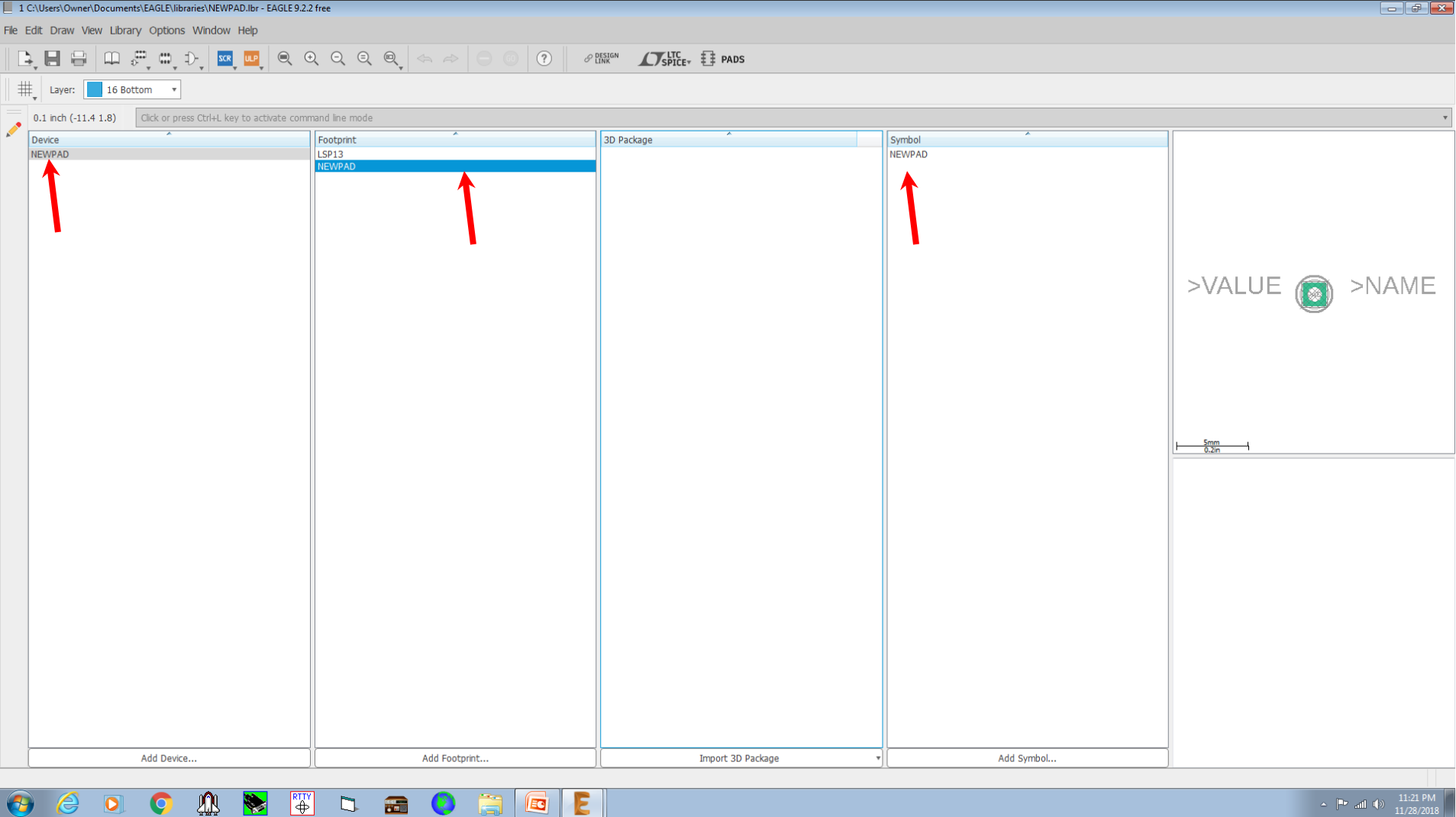|	#		Name	✕
◉	1	🟥	Top	
	16	🟦	Bottom	
◉	17	🟩	Pads	
	18	🟩	Vias	
	19	🟨	Unrouted	
	20	🟨	Dimension	
◉	21	⬜	tPlace	
	22	⬜	bPlace	
◉	23	🟩	tOrigins	
	24	🟩	bOrigins	
◉	25	⬜	tNames	
	26	⬜	bNames	
◉	27	⬜	tValues	
	28	⬜	bValues	

New Layer   Show Layers   Hide Layers

Layer Sets   New Set   Remove Set

OK   Cancel

27

21

25

17

23..not important here

>VALUE   PAD   >NAME

FOOTPRINT

File   Edit   Draw   View   Library   Options   Window   Help

Layer: 16 Bottom

0.1 inch (-11.4 1.8)   Click or press Ctrl+L key to activate command line mode

Device
NEWPAD

Footprint
LSP13
NEWPAD

3D Package

Symbol
NEWPAD

>VALUE   >NAME

5mm
0.2in

Add Device...   Add Footprint...   Import 3D Package   Add Symbol...

DEVICE

>VALUE ▣ >NAME

5mm
0.2in

Package	Variant	
NEWPAD	"	⚠

Technologies	Attributes
NEWPAD	

Footprint OK

DEVICE

New ▾   Connect

Prefix

Value   ⦿ Off   ◯ On

DEVICE

>VALUE >NAME
| 5mm |
| 0.2in |

?

Package	Variant	
NEWPAD	"	⚠

Error

Technologies	Attributes
NEWPAD	

After selecting "CONNECT"

Connect (NEWPAD)

Pin	Pad	Connection
Name	**Name**	**Pin** / **Pad**
	P$	

NO NAME

| Connect | Append | Disconnect |

Copy from:

OK   Cancel

DEVICE

File   View   Options   Window   Help

Name

▼ Libraries
  ▼ 📂 libraries
    ▶ 📓 BNC_PCB.lbr
    ▶ 📓 HALF WATT RESISTOR.lbr
    ▶ 📓 IC14_NEW_TRY.lbr
    ▶ 📓 IC16.lbr
    ▶ 📓 IRF540.lbr
    ▼ 📖 NEWPAD.lbr
        ⋮ NEWPAD
      ▼ Footprints
          ⋮ NEWPAD
      ▼ Symbols
          ⟝ NEWPAD
  ▶ 📁 Managed Libraries

LIBRARY STRUCTURE