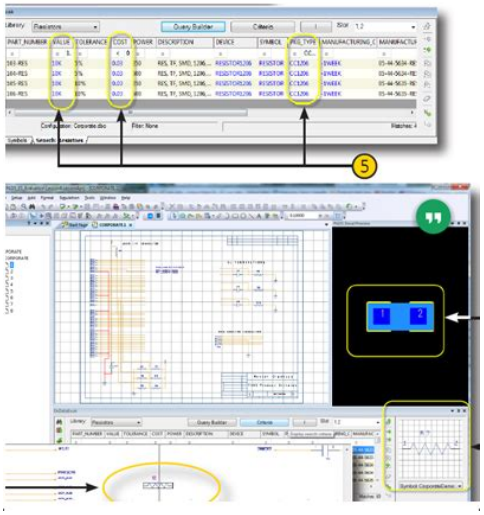
Dxdesigner Training Manual



File Name: Dxdesigner Training Manual.pdf
Size: 4766 KB
Type: PDF, ePub, eBook
Category: Book
Uploaded: 17 May 2019, 15:31 PM
Rating: 4.6/5 from 830 votes.

Status: AVAILABLE

Last checked: 1 Minutes ago!

In order to read or download Dxdesigner Training Manual ebook, you need to create a FREE account.

Download Now!

eBook includes PDF, ePub and Kindle version

Register a free 1 month Trial Account.
Download as many books as you like (Personal use)
Cancel the membership at any time if not satisfied.
Join Over 80000 Happy Readers

Book Descriptions:

We have made it easy for you to find a PDF Ebooks without any digging. And by having access to our ebooks online or by storing it on your computer, you have convenient answers with Dxdesigner Training Manual . To get started finding Dxdesigner Training Manual , you are right to find our website which has a comprehensive collection of manuals listed.
Our library is the biggest of these that have literally hundreds of thousands of different products represented.

Book Descriptions:

Dxdesigner Training Manual

This tool uses JavaScript and much of it will not work correctly without it enabled. Please turn JavaScript back on and reload this page. I am new to Mentor Graphics. Thank you in advance. No one else has this question Mark as assumed answered Outcomes Visibility Layout 430 Views Last modified on Mar 18, 2019 959 AM Tags tutorial Content tagged with tutorial basics Content tagged with basics xdx designer vx2.1 Content tagged with xdx designer vx2.1 schematic basics Content tagged with schematic basics layout basics Content tagged with layout basics mentor for starters Content tagged with mentor for starters Ratings This content has been marked as final. I am going to move this to the more appropriate productspecific community where it will receive greater visibility. The Member Resources area is for general community questions and suggestions. Recommended Content Does get fabrication layer texts for generated silkscreen work for anyone else. Silk Removing inside a selected polygon or rectangle shape. Report a trace segment width Expedition PCB Show Trace Width Report Diff Pairs All Rights Reserved. It also supports bidirectional crossprobing with all editors. Design reuse and variant design are supported, which reduces product timetomarket, improves product quality, and lowers product costs. Whether you are experienced with VPL or this is your first exposure, I'd like to invite you to download this white paper, published recently, which.Detailed information on the use of cookies on this website is provided in our Privacy Policy. By using this website, you consent to the use of our cookies. Video Library Quick and tothepoint video tutorials to get you started with Altium Designer This application note assumes this combination. Complete flexibility is found in all pages of the wizard, giving you as little or as much control as you would like over the translation settings before committing to the actual translation process.http://www.szp45.pl/wsg/userfiles/designjet-110-service-manual.xml

- **dxdesigner training manual, dxdesigner training manual pdf, dxdesigner training manual download, dxdesigner training manual free, dxdesigner training manual 2017.**

Because there is a difference in the way that project files and schematic files are named and organized between DxDesigner and Altium Designer, its worth briefly reviewing this so that you understand exactly how your schematic design and libraries files will be translated after the import process. For example, instead of using file extensions for a files type, a folder called sch in the project path indicates that files under this folder are schematic files. The individual schematic files follow the naming convention of Name.N where N is a numeric number. Likewise, a folder called sym in this project path indicates it is a symbol folder, and that all files under it are assumed to be the equivalent library files also following the same naming convention as schematic files. As you begin to import your DxDesigner files using the Import Wizard, you will be asked for your project directory name. The Import Wizard knows to look for the sch and sym folders inside the specified project path. If that directory does not exist, you will be given a warning message. Once translated, files are grouped into that PCB project. Design hierarchy is maintained, including complex hierarchy. Once the project has been compiled the schematic hierarchy will be shown. Ports Similar to power objects, a symbol with an attached attribute represents it as a port. DxDesigner symbols that contain an IN, OUT, or BI attribute are identified and translated into Input, Output, or Bidirectional ports respectively. Signal Symbols that contain a SIGNAL attribute are identified as and translated into hidden power pins.A wire or bus segment in DxDesigner can have a label attached to it. This is translated into a net label. The symbol pin is translated as sheet entries and the sheet symbol file name will point to the list of schematic sheets that matches the symbol file prefix. Boxes as defined as lower left and upper right corners translate to fourpoint

polygons.http://www.oglethorpeclub.org/~oglethor/UserFiles/designjet-110-manual.xml

Multiplepart symbols The PARTS attribute attached to the symbol indicates the number of parts this symbol represents and translates to the number subparts in Altium Designer. Annotate Symbol Type DxDesigner categorizes the symbol into four types composite, pin, annotate, and module. The most common use of symbols in DxDesigner is for sheet borders and graphical annotation. Heterogeneous Symbols Heterogeneous symbols in DxDesigner are any group of symbols that have the same HETERO attribute. When symbols are grouped under one HETERO type, they represent one device. Altium Designer translates these symbols to multiple parts or display modes under one component depending on the heterogeneous type. There are three distinct types The main difference between this type and HETERO TYPE 1 is only the context used by DxDesigner related to ICs. Click on this menu command to invoke the wizard. Rightclick command menus are available for further control over the translation process through each page of the wizard. There are additional options provided to control the automatic creation of design rules, missing vias and keepout conversions as well. Layer Mapping is simply a mapping between the names of the PADS PCB layers and Altium Designer PCB layers. Of course you can change as many mappings as you want as only suggested default mappings are given. This mapping is used by the Import Wizard to build the layer mapping for each PCB that can then be individually customized. The rationale here is that should you wish to import ten PCB designs and you want to map the layer Assembly 1 to Mechanical Layer 1, you would not have to customize each of the ten PCB designs in order to get the right layer mapping. In this instance, the default layer mapping will be saved to your Preferences. The disadvantage to using this is that Default Layer Mapping is not always intelligent with differing structures in designs, and so some manual changes may be needed afterwards.

Youll need to decide what is best for your situation. This means that for each Altium Designer schematic sheet page there is a file, an important conceptual difference to remember. There can also be multiple design documents of varying types, depending on the nature of the design you are working on. Getting started, most DXDesigner users will be interested in the schematic and PCB document types as these are the files that their designs will be translated to. Parts are defined as having a part type, a logic family, number of pins, number of gates, and signal pins. The term component is only used when the part becomes placed as a physical object in the PCB layout design. Parts in PCB designs usually correspond to physical objects gates, chips, connectors, objects that come in packages of one of more parts. Multiplepart packages are physical objects that are comprised of one or more parts. In DxDesigner, a symbol block type is the logical entity that is described graphically by attributes, pins and various properties. As block types are placed in a schematic design, DxDesigner maintains the identity of the part for back annotation, net listing, bills of materials, and so forth. At the very minimum, a part requires a part name, a part reference prefix, and a name of a PCB footprint. But it is not unlike how things work in Altium Designer except that the schematic symbol is effectively the part for all phases of design, and not just the PCB Layout portion of it. It can be initially defined at minimum as a name in a schematic library to which pins and any graphical symbols or alternative display options needed for implementation may be added. This flexibility allows a component to be represented in different ways during the design and capture process. This may not only be as a logical symbol on the schematic, but also be a footprint on the PCB or even as a SPICE definition for simulation.

A scope is effectively a query that you build to define all the member objects that are governed by that rule, giving you full control. For the sake of a baseline comparison, lets review the Rules Hierarchy of PADS Layout. These rules rely on a predefined list format where higher numbers on the list have precedence over those that are lower Conditional layer rules can be applied for an additional level of precedence. For example, a Default with a conditional layer rule would be higher in the precedence hierarchy than a simple Default rule. You can even define multiple rules of the

same type, but each targeting different objects. Queries are easily accessed for any rule Figure 22. Advanced Query options are also available to help you write your own, more complex queries. Toggle the corresponding Enable option for the rule in the relevant list. In addition to scoping, there is also a userdefined priority setting. The combination of rule scoping and priority is very powerful and gives an unprecedented level of control that allows you to precisely target the design rules for your board. During compilation checks are made to see how relationships are defined, to validate the relationship between the models and the symbols and to bundle them into a single integrated library. This file can not be directly edited after compilation, offering portability and security.This is 2dimensional representation only, and can be quite different from the actual component itself Once placed as a footprint into a PCB file, it is given a designator and value comment Remember, you can also browse through the Help contents, and use F1 and Whats This at any time in a dialog for more details. You may receive communications from Altium and can change your notification preferences at any time. You may receive communications from Altium and can change your notification preferences at any time. Please fill out the form below to get your free trial started.

You may receive communications from Altium and can change your notification preferences at any time. For more information, click here. You may receive communications from Altium and can change your notification preferences at any time. You may receive communications from Altium and can change your notification preferences at any time. We have the perfect program for you. You may receive communications from Altium and can change your notification preferences at any time. To participate you need to register. Registration is free. Click here to register now. For a better experience, please enable JavaScript in your browser before proceeding. It may not display this or other websites correctly. You should upgrade or use an alternative browser. Thanks in advance Nishal Thanks in advance NishalI need the English version. Because I just understand English. Thanks But thank you very much.Thanks in advance Maurizio Im looking for a good tutorial, book or training manual, for dxdesigner how create symbols, schematics,.By continuing to use this site, you are consenting to our use of cookies. Share space Manage All Content 1351 Documents 188 Discussions 363 Questions 799 Polls 1 Videos 0 Events 0 Thumbnails Details Hierarchy Questions All questions Open questions Answered questions Questions with helpful answers Type to filter by text Filter by tag Sort Sort by date created newest first Sort by date created oldest first Sort by latest activity newest first Sort by latest activity oldest first Sort by title 1 2 Previous Next Title Author Latest activity Views Custom menu or buttons in Variant Manager.All Rights Reserved. Wordpress is free, so they aren’t disrespecting their customers every moment of every day and charging them thousands of dollars for it . I’d been using Altium Designer previously and giving them Hell about minor issues, honestly, but the company decided to standardize on Mentor Graphics for PCB design.

I want to share my opinions about it with you whoever “you” are. Jesus H. Christ, I get a better user experience with CadSoft EAGLE, the pencil and quadrille paper I was designing with 30 years ago, or even the 80’s version of “HiWire” the very first CAD I ever used. I could deal with a Windows 3.1 level of user interface, even a DOS 5.0 user interface, but with DX Designer I feel like my productivity would be better if I was actually PISSING my designs into a snowbank. Half of the time when I click on the “Help” button it will crash the application, and the times that it doesn’t the information you need is just not there. Don’t EVEN bother with the vestigial contextsensitive help icon, it will just open a browser window that tells you to look somewhere else for information. This “feature” alone puts Mentor Graphics at the top of my list of nominees for “Death by Mass Rabid Weasel Attack”. It’s reminiscent of the scene in “Office Space” where they guys are trying to figure out how to launder money by looking up the definition in the dictionary; they found words but not answers. This will likely take you several tries and that’s after you manage to zoom in on the component, more on that later. If you are lucky enough to get that darned menu open, then you get to select the “Transform” option, then the “Rotate CCW” option, which will rotate it 90 degrees.

Here’s another really kludgey part of DX Designer component pins had better be three grid increments apart or you are going to see your nets do some pretty weird snakedances on the way and then they will probably still evaporate like Karl Rove’s wetdream of a permanent Republican majority after your terminating mouseclick. Every pin seems to have this mysterious 3grid “line of death or disfigurement” around it. Kissoff your hopes of making a fairly compactyetclear schematic in DX Designer or get ready to do the Snake Dance!

This alone merits the “The Angry Mobs Decapitated Me” dismerit badge for whoever made this the Mentor Graphics cannon. Well, DX Designer makes this a “Thing of the Future” by defining “zoom” to mean “zoom to center of sheet”. Positioning your mouse and using the scroll button gets you a well tempered finger from Mentor Graphics rather than a clear view of the circuit element you want to see.Can you do it one time, then paste copies of all the nets and labels on each memory device. Not well or easily, unless you copy them off a different page because the idiot DxD product deletes the “global” attribute when you do this within the same page.Yes, DX Designer is that unclean. It is hard to imagine this level of incompetent applications programming after all these years, the product’s user unfriendliness and unwieldiness and the level of corporate “up yours, Bud” attitude that this product radiates as being anything except an embodiment of everything that’s going wrong in America right now. Bring in the guillotines! Companies that put out this kind of excrement need to be shut down and their executives and managers blacklisted. This gang has turned a once pretty okay product into crap, and nobody should pay them to do that again they should be sentenced to screwing up orders in a fast food restaurant where the angry customers can call them personally to task for their sins, forever and ever amen. These kind of subhumans need to have their birth certificates savagely revoked they steal from all Humanity, even while knowing better.You’ll be glad you did. Dx Designer will blow you away! You can follow any responses to this entry through the RSS 2.0 feed.Magical composition of fluid emotion! Andy while there are times where I would prefer an “in situ” symbol editor the automatic seeding from DxDesigner to the Symbol Editor and then autoupdate from the symbol editor to DxDesigner is extremely productive no design reload or refresh.

I have been using DxDesigner for years and aside from the disk crash I have not lost a single design. I have never read a manual, don’t use the help files but I do have that antiquated instrument — the phone — updated with a headset and I dial my trusty customer support engineers and voila — don’t need no stinking manual or training classes. If you had worked with the customer support engineers at Mentor Graphics you would have written a completely different article — how wonderful it is to find a company who has not outsourced their customer support. However, you have chosen one of the most bizarre methods I have seen to date to ask for help. Is this a desperate plea to join the unemployed. There are many thousands of companies — and countless engineers using DxDesigner to create market dominating products today and you are raising your hand and saying “not me”. I would caution against that approach — there are many very good engineers inhabiting the unemployment ranks due to no fault of their own. I think you have walked out onto the thin ice and while I do applaud the vivid imagery and tightly coupled metaphors you might want to head back to shore now. It’s truly appreciated. Hyperbolic invective is indeed sadly a declining artform, bemoaned by Mark Twain a century ago thanks for your comments on that as well! Do you pay for what you reasonably expect and then get those features, or are you expected to make a lifestyle change that requires retraining and reeducation to make use of the product. I know it CAN be done because it HAS been done, but not by Mentor Graphics! This is where the external text editor comes in to fix many hours of component creation in the native “Component Wizard” that DOESN’T force pin numbers to the required upper case. We returning users are given no such guidance until a lot of time is wasted utterly by BAD DOCUMENTATION and BAD PROGRAMMING in the Mentor Graphics product!

Do the fastfood establishments in your neck of the woods withhold ketchup unless you ask for it. It strikes me as “bizarre” to think so, but maybe your boss has much deeper pockets than mine. Umm, did you parse that statement fully before you typed it. I don’t think I’ve ever claimed that Mentor Graphics is the only ballless corporation foisting inadequate products onto customers, but feel free to provide a quote that says otherwise. I’ll publish it, promise! Likewise,if they are that lame as to steal from their stockholders the cost of running a phoneroom to offset the bullshit nature of their documentation, aren’t the just as worthy of the noose. Hello, Mentor Graphics. I could do WITHOUT needless irritation and recursive rework required by DxD. MG’s ignoring of basic programming tenets is a VERY major barrier for me overcome before I can encourage others to adopt their package. How many times have you done it, anyway I believe the answer is a definite “Yes” but obtaining the prerequisite knowledge is not addressed by my statement. Clearly, if Mentor Graphics is to be successful expanding their customer base beyond their current users they really should heed your feedback. No disrespect intended but I have my own agenda. Annoying but I have adjusted to asking for the ketchup at the same time I make my payment and I do agree that the penny pinching had caused an unneeded behavioral change. Without diving into politics too far I think the root cause is with the bankers and I am all for auditing the Federal Reserve If there is something worth boiling my blood over, I think that is getting close to the right domain. I am not a young dog and I have used almost every EDA solution available and some that are no longer available. The constant innovation has caused a few foul words to spill but at the same time it has saved my hind quarters numerous times. You might have convinced our competitors to stay away from DxDesigner and we would have had another competitive edge.

I hesitate to cause reflection and a reversal of sentiment as you might be working for one of our competitors but I also love a good debate. It’s been fun. There can be no doubt that we are not competitors.Why did they do it when there are even free schematic capture tools out there that are better. It seems to have been designed to make you as angry as possible. It randomly moves bits of net around as you draw, you end up with unconnected net on top of each other aaaah!. Its fine if you place all you components and then connect them and don’t ever move or copy anything, and you’ve got pleanty of time and patience and you like a challenge. If this is not you then I suggest an alternative tool or that you have a supply of valium to hand! I have to suspect that your willingness to tolerate DxD’s amazing suckitude is based purely on your having become inured to it. After five years and three major revisions, I’m still amazed at how buggy it is, how unpolished and downright hostile the user interface is.It’s like you’ve stepped into 1994. They sell on momentum only, and the fact that the folks that have been using it for all these years are woefully behind on what currently exists in the market. I don’t know how they could have screwed up their product this much without folding. It just isn’t worth the hassle of trying to use it. My boss has been looking at Mentor Graphics DxDesigner and PADS as the product to move to from our current schematic capture and board layout tool but I think Altium may be the way to go and your arguments for and against PADS and DxDesigner give me a leg to stand on. The last thing my company needs right now is to be crippled by a program that apparently is meant to be sold to huge, slow, already dead but don’t know it yet companies that have lots of people around with nothing better to do than make phone calls to support engineers and to “google” for help as one of you said.

In the current business climate you have to be quick or you will soon be out of a job and business. It can be hard to dissuade “Management” from making demonstrably wrong decisions. I’d used PADS in the dim past when it was a halfway decent system. Mentor Graphics has utterly screwed the pooch with the DxDesigner respin. I am unequivocal in my opinion that it is the least desirable product in the field today. Buggy, crashes. oh, miserable. Even mentorgraphics “Logic” which was supposed to be superceded years ago by DxD is still around yet we were told many years ago it would be killedoff by DxD. I’m an Orcad fan myself and I spent 10 months using DxD and finally gave up. I have a list of things that a piece of dust can do better. DxD is unworthy of existence and my

blood's boiling just writing about it so I'll say no more. Andy Everything about it is horrible. It sucks! and they charge BIG bucks for this! I will change to Altium. Why yout company change to Mentor whats the reason I cannot understand this stupid stap. We wnt back to PCAD and just zoom along even without fancy features that crash the program. An absolutely bad designed software, with lots of bugs and a terrible help system. When I work with this piece of crap I have to smoke lots of cigars to keep relaxed. The people from Mentor have absolutely no compassion for the people that are actually going to work with their tools. Because of mentor I began drinking and taking pills!!! I prefer Altium and Allegro by 20dB more than the Mentor dissaster tools. If you hate life use Mentor Graphics!! If you want to pay more for less try Mentor Graphics!! I hate them and their product and their utter disdain for users THAT MUCH. The PADS support says it will not support any Altium product beyond Altium 9 I think but that is it just doesn't support the 3D modeling of Altium. I have had no problem importing Altium 14 basic schematic designs into Pads Logic.

DX designer also has a translator for Altium DXP Its bad enough that I'm just a newbie and I'm still learning the ropes, but when I have to spend hours maybe even days trying to debug HDL Designer before I can even start my real graded work, it makes me wonder why I want to do this for the rest of my life. At least as a writing major I don't have to worry about Word forgetting what the letter "e" is. Sometimes I wonder if the compiler is determined by the tide because one hour stuff works like magic and the next I might as well have shit pretty pictures on the screen. The best thing about HDL Designer is that it is impossible to edit this work on anything but the lab computers with their fancy crapware, so I have to sit in the dark dungeon of the lab, at 4am in the morning, wishing I was a writing major or somebody digging ditches. It gets especially fun when I can't fix the mistakes that HDL Designer makes so not compile on HDL Designer.You think after a couple years and multiple patches and updates Mentor would have tried to fix some of the problems that make it so unusable. After using Dxdesigner for a year and a half and listening to mentor say "it's getting fixed in the next release", only to have the next release come out and be equally bad, we've finally given up. I'd rather spend time remaking a thousand symbols in a new tool than to waste another minute using DxDesigner. It's hard to believe how bad this program is until you've actually used it. I kept saying, maybe once I got used to it, it will be ok. But after a year and a half all you learn are that more and more things are broken. It's a complete steaming pile of poo. Usually we get a response back in 6 months saying "we know this is a bug, but we've closed it as we're not going to fix it." Yes, so when parts magically unmirror and the nets move to different pins and there's nothing to alert you your design is now wrong, yes this isn't worth fixing according to mentor.

Even using visio for schematics and making your netlist and bom manually in excel would be more effiecient than using DxDesigner. I used to love my job. I don't even want to be an engineer any more. I'm still an Altium proponent where expensive commercial ECAD is concerned I love being able to rearrange device pins on the fly to make a schematic clearer. At home I still use CadSoft Eagle bought a license for a longsincecratered home business some years back, but it does most small projects well enough to not be excruciating. I am starting to actively search for open source solutions. "Fab Lab Tulsa" is opening soon, and I'm hoping to offer volunteer instruction in exchange for access to the cool tools. There are other, better options to the clusterkludge moneyholes that Mentor puts out.Maybe I'll take a look at their test drive over at.Originally they where the only circuit tools available for MACS. Eventually they expanded to include PCs. They never got as large as the major players, maybe because the MAC market stunted their growth. They were part of the core group of circuit tool companies who were in on the ground floor in developing autorouter technology. And unlike the most of the others they were not purchased by the major players who were buying all the companies with autorouter technology to get a lock on that capability. McCAD's autorouter is extremely capable and probably as good as anyones. It is an interesting history and George the only guy you talk to for support will be happy to tell you about the layout competition of the 1980's in which PADS won in spite of the fact that all the routes to the edge connector of their

board were all shorted together and no one noticed it until after the competition was over. Had someone noticed that it might have been a different story in the industry. I like their tools and am a fan but have no affiliation with their company. Presently, ModelSim is crashing inexplicably on me.

Thanks for your comments, I got a chuckle out of them in spite of the recognized shared pain. I came across an open sourced tool “KiCAD” a few years ago and it looks very interesting for schematic capture as well as PCB layout. I haven’t used it for a full design iteration, so I don’t have first hand knowledge. I wonder if anyone can comment on it. There appears to be an ongoing effort to correct bugs and improve the program. Anyone out there used it successfully I’ll have to kick KiCAD’s tires real soon now! As a newbie it took a while to get the hang of “joining the dots” but Protel itself wasn’t the problem. Anyway, these days I’m very comfortable with Altium and reckon it’s darn good with some quirks but not many bugs. I can do designs real quick when given the schematic pencil drawing. Then things went sour. My company I work for got bought out and the new company has standardised on Mentor 7.9.2 everywhere. There is still a possibility that we might ignore Mentor locally but for collaborative worldwide projects there probably won’t be a choice for me. Why am I sad about this 1 I may retire in a few years time and I’m not keen on spending that period learning a program whereby I then promptly forget it, just when I may be useful with it. 2 I’ve probably spent nearly 2 weeks making sample schematics and pcbs with Mentor and I have discovered that I really really hate it. Why do I hate it 1 After 2 weeks I should have been able to draw schematic lines and pcb tracks. Just doing those simple tasks were difficult. So hard that I still don’t know how to place a pcb track and I don’t consider myself a dummy. There seems to be 2 camps of the way things pan, grabbing the sheet and moving it or moving the sheet in the opposite direction of the mouse movement. Altium and other photo packages are in the former camp. I find grabbing the sheet to be the easiest and BEST way to move around a schematic or pcb.