

ICU 2003 Orcad Submitted Issues

- Title:** 'Use of Network License for PSpice'
Description: 'The license for using PSpice is checked out and utilized whenever a PSpice "type" project is opened. The user may only be performing schematic changes but the license remains unusable for anyone else on the system. I think the license should be tied to actually running a simulation or perhaps viewing data in Probe. This will allow more efficient use of the PSpice License.'
- Title:** 'Improve Mechanical part handling'
Description: 'I would like the ability too add mechanical parts to my schematic and create BOMs and netlist with them in the schematic.'
- Title:** 'Improve the flow from ORCAD to ALLEGRO'
Description: 'It is awkward and unintuitive the way the 2 tools connect together. I would like to see something like project manger overlaying the 2 tools or build allegro into the ORCAD project manager. Currently the 2 tools act like 2 different and distinct point tools with a shoelace connecting them together'
- Title:** 'Add Constraint manager to the tool'
Description: 'Capture would flow much nicer with allegro in high speed design if the constraint manager tool was built into it.'
- Title:** 'Copper Flood problems in OrCAD Layout'
Description: 'If you use the line hatching as standard on dual level copper pours. I.E. copper pours of Z order 1 inside a copper pour of Z order 0. Then the copper either misses a line out or continues a line beyond the isolation barrier and shorts the copper pours together. The only way around this is to change to copper pour to a solid hatching. This is not an easy task on a multi-pour copper board. This has occurred on PCB's designed using OrCAD Layout 9.2.3 Build 923 March 18 2002.'
- Title:** 'Printing Order Of Capture schematic pages'
Description: 'It would be really good if Capture would print ALL pages in the order of page No. rather than it s current alphabetical on page name.'
- Title:** 'File Locking on Capture CIS'
Description: 'Two engineers opened the same Capture design one hooked to Layout with ITC and the other editing the schematic changing parts. Both close the Capture design and it is now corrupt and unreadable.'
- Title:** 'Layout won't connect copper pours with a track'
Description: 'I have submitted this as a complaint already and the response was "It may be fixed in a future release" MAY is not acceptable for a core deficiency. It was reported 01/17/02. Basically 2 copper pours were placed on different ends of a board on a plane layer. A routing layer trace was run from a via on one pour to a single pin component called TP6 on the other pour. The via correctly got a thermal to the pour BUT the TP6 component pin did not get a thermal and did not connect to the pour. Also the DRC DID NOT REPORT AN ERROR OR DISCONNECT!!! This is not acceptable. '
- Title:** 'OrCAD Layout ignoring design rules'
Description: 'When a PCB design has less than 8mils clearance requirements and you zoom out with the "O" key and then back in with the "I" key VERY often design rules are totally ignored until the design rule toolbar button is deselected then reselected. Design rules are then

maintained again until the next zoom out/in. I have had this problem since 9.0 and am now running 9.2 WU1 and still have it.'

10. **Title:** 'Archiving Schematic database with a subset of CIS database'

Description: 'Currently there is no way to archive a copy of the ORCAD schematic with a copy of the CIS database that has only the parts in it that are in the schematic. This is necessary to retain the integrity of the library data as it was with the design as it was released. A lot of the data in the CIS database is dynamic and constantly changing. This causes problems when pulling the design back from archive and making ECN s. This is a requirement with military schematics.'

11. **Title:** 'Capture/PSPICE Interoperability'

Description: 'Interoperability between Capture and PSpice needs improvement. There should only be one set of libraries. Presently I have to generate two separate schematics; one for PSpice simulation and the other for documentation and layout.'

12. **Title:** 'ECOs now require filenames/paths every time – Revert to Pre 14.2 method.'

Description: 'Older versions of layout performed ECOs with very little effort (by the user): as changes were made in the net list the layout program would run ECO using all the established file names and paths. In version 14.2 we must enter the filenames and paths every time and the default paths are consistently wrong! IE: if the netlist comes from a Capture folder the next dialog box to select location of new max file defaults to the capture folder not the folder in layout AND the default name keeps changing! Please go back to the older more user-friendly method.'

13. **Title:** 'PSPICE Support'

Description: 'Make PSpice a fully supported Cadence product again. Having a third-party provide support for a key product is unacceptable.'

14. **Title:** 'Interface with PADS Power PCB error prone – incorrect header'

Description: 'When I import a netlist from OrCAD to PADS Power PCB I get errors because the header of the netlist is incorrect. The header should say something like: !PADS-POWERPCB-V2 *PARTS* Even with this I do get a version error but at least it is accepted. Instead we have this as a default: *PADS-PCB* *PARTS* These lines make the netlist unacceptable to PADS Power PCB import.'

15. **Title:** 'Net connect to component pin'

Description: 'I have a hard time attaching nets to components. You have to be exactly on the net itself to get it to attach. Sometimes I have to move the component after the net is routed to get the net to properly attach.'

16. **Title:** 'VBA support for OrCAD Capture'

Description: 'OrCAD Capture still has some of the same shortcomings it had 10 years ago as well as some of the same bugs!!! Since it is clear that you are not going to support the product properly why don't you give the design community the ability to turn the product into what you can't or won't. Allow full VBA support and let us fix the problems!'

17. **Title:** 'Software / licensing installation very time consuming and error prone.'

Description: 'It is very frustrating and time consuming installing your software. There has to be a way to make it more straightforward. Every time I install a new version or we get new computers the key codes/authorization codes must be changed? I have pages of notes on how to install layout. I have to first install capture make a copy of the .lic file install the rainbow key install layout copy line from the new .lic file and then paste it in the old .lic file. Does that sound like a normal procedure for installing software???'

18. **Title:** 'Grouping (Ungrouping?)'

Description: 'The grouping function works to group items but it does not remember what was grouped when you close it down and reopen it. We use capture for harness drawings and we draw all of are connectors in a library. Each part is dimensioned 1:1 (as close as possible). Some of the parts are very complex and different parts need to be grouped. When you close it down then go to make a part similar but with a different number of terminals the groupings are lost. I am very disappointed in how this over priced "professional" software functions. I have brought all my issues up before a few years back with no response. If our customers got this type of response I don t think we would have very many customers '

19. **Title:** 'Use of Ctrl Key to select or deselect item can accidentally copy/paste items '

Description: 'I use the control key for one function in Capture. I select or deselect stuff that I may want to move or delete. If you select two items then while selecting a third item very close to the other you may end up dragging the other two items over and paste them next to the third item you are trying to select. This is because the ctrl key was held down in the process. Imagine if you have now half a schematic selected and you did the same thing. If you did not notice that you changed it before you did one other thing you would lose your only chance to undo. So now you have a half a schematic shifted a little and pasted on it self and no undos! Yeah- one may think that you would have been saving as you go but this is professionals program that should not be capable of doing that. My suggestion besides the multiple undos is to evaluate the ctrl key. I have never wanted to use the key to paste selected items somewhere else does anyone use it?'

20. **Title:** 'Packages - part properties propagation option'

Description: 'Create an option that allows the user to choose whether or not to propagate added part properties through all parts in a hetero/homo package. This should include convert views as well as normal views.'

21. **Title:** 'List of unconnected pins should be reported during netlist output.'

Description: 'Based on the netlist output it would be good to have a list of unconnected pins so I can verify this list against my design intent.'

22. **Title:** 'Generate Part problems – Not ready for primetime'

Description: 'There has been a problem with the Generate Part functionality in Capture since it first showed up. For Altera ".PIN" files (haven t tried other pin formats) the pin type information in the pinlist file is IGNORED by Capture & all pins are simply shown as "Bidirectional." There is one exception that I've seen -- if the Altera pinname is GND Generate Part makes the type "Power." There is a workaround that I've stumbled onto but I've never seen it documented in any OrCAD help. While editing the library part that Generate Part makes select View/Package then Edit/Properties to get the weird little non-Windows-standard non-resizable spreadsheet viewer. The data in this window can be cut & pasted -- BUT NOT VIA STANDARD WINDOWS cntrl-c & cntrl-v! For example I would normally use Excel to open the Altera pin file & write equations to convert a column of Altera pin types to the specific names that OrCAD requires (ex: Altera-> "bidir" converts for Capture to "bidirectional"). Now the column of correctly s

23. **Title:** 'More pin shape choices when building/editing a part'

Description: 'When building or editing a part in Capture there are no choice for a short dot pin or a short dot-clock pin. The pin that exists is way too long and takes up too much room on a crowded schematic. It would be easier with a few more choices.'

24. **Title:** 'Restoring a function from OrCAD for DOS when deleting '

Description: 'This was possible in OrCAD for DOS and I REALLY miss that function. What I m talking about is being able to delete when parts overlap; whether it is a line junction or a part. A window came up and asked: do you want to delete the line junction or part? In Capture I have a hard time deleting a line or junction that accidentally interferes with a part. It will always delete the part and not give me any choices. Can this function PLEEEEEASE be restored????? '

25. Title: 'Need symbol wizard for Capture/PSpice'
Description: 'Need to have a symbol wizard for Capture/PSpice as powerful as the symbol wizard in Microsim Schematics R8. Only having the generic rectangle for subcircuits doesn't cut it anymore.'

26. Title: 'PowerPCB Interface - Netlist and diff pair rules'
Description: 'Differential pairs and net classes should translate with the PowerPCB netlist option. Also back annotate should be easier.'

27. Title: 'Multiple Parametric Sweeps'
Description: 'There has been a number of occasions when I want to sweep more than one component at a time. I don't know how difficult it would be to implement but I feel it would be a great enhancement if multiple parametric sweeps were available. Monte Carlo runs are too random for what I want to do. '

28. Title: 'Schematic "layers for design notes use"'
Description: 'It would be great if schematic sheets had one or more "design note" layers. The notes would be used by designers who maintain the schematic. Those layers should have selective visibility and printability. For example the schematic that is printed for customer service could have helpful notes on it concerning where to look for various problems. The one printed for production could have special instructions and calibration notes. The designers need all kinds of notes on requirements circuit responses etc. '

29. Title: 'Combine Property String effect lost when merging INCLUDE fil'
Description: 'When using the MERGE Include file to fill in the part properties for a BOM the COMBINE property string effect is lost. For example we have a property called LOAD. The designer enters "NO LOAD" in that field of the schematic when the part is to be skipped during production. It is included in the combine property string so that other parts of the same part number will be listed on a separate line on the BOM. Using the part number for the combine of the INCLUDE file causes the original combine to be ignored. In that case the BOM shows all parts of that part number combined with the LOAD attribute of the lowest reference designator. We are forced to not use MERGE include file. Instead we have to use Update Properties to stuff all desired fields from the Item Master List into the schematic. That is undesirable because it makes the information permanent in the schematic instead of floating with the master item list.'

30. Title: 'INCLUDE Fil Merge direction is wrong'
Description: 'When merging an INCLUDE file an error is generated for every part in the INCLUDE file that is not present in the Capture design. It should be the other way. An error/warning should be generated for each part in the Capture design that is not found in the INCLUDE file. I use my Item master list as the INCLUDE file. It has several thousand part numbers in it. Consequently I get several thousand errors.'

31. Title: 'Reduce number of hand movements in OrCAD Capture'
Description: 'When entering a schematic into OrCAD Capture there are more hand movement than needed. Greater use of the mouse buttons would help. An example is that you need to hit the right key and then move the cursor over esc an left click to end placing a part. A double right click would be faster. The same holds true when drawing lines to connect parts.'

32. Title: 'Improved sensation - PSpice.'
Description: 'Of course is that PSpice provides basic instruments to calculate the various types of circuits or complex projects more than true. I would suggest to create some additional instruments(external devices display panels maybe even separate voltmeters or wattmeter's not markers etc) that could be give more visualization or more understanding of calculating s process. That there were 2 modes of operation for example professional mode and tutorial mode or mode for students or simple mode. It would be very useful to students who are interesting

such software and could give them better sensation of working process. Another reason I think is that such software would let work with it much more people including persons who don't have special derivation who are only fan. Having basic knowledge(not special) you would be estimating some circuits that are interesting for you. It expand borders of using PSpice.'

33. Title: 'Capture File Lock '

Description: 'There needs to be some type of lock applied to the Capture files when they are opened to prevent two or more users from being in the same schematic at the same time. At the very least maybe some type of warning message saying that the file is already in use do you really want to open it.'

34. Title: 'Save Changes prompt appears when NO change has been made'

Description: 'OrCAD Capture currently displays a very confusing and inaccurate prompt when exiting the program having made NO changes. In OrCAD Capture merely opening a .dsn file and then closing it will elicit a "Do you want to save changes.." prompt from the program even if not a single change was made! This can lead to real confusion as to whether some value or component was actually changed. Bear in mind that OrCAD can be open for hours just for viewing purposes; when it is then closed hours later and this prompt appears it causes unnecessary doubt when in fact NO change was made to the file. In this situation the application should simply just close down with no prompt whatsoever since nothing was altered within the file.'

35. Title: 'Individual page saves are not possible'

Description: 'OrCAD Capture currently employs an illogical and counter productive method of saving changes to schematic pages. When editing a page and a Save is executed the program displays a prompt stating "Data has changed in this schematic. Changes across ALL pages of schematic X will be saved. Would you like to save ALL pages?" Essentially the program is giving the user two choices: Save the entire document or nothing. There is no provision for saving JUST THE PAGE YOU RE CURRENTLY EDITING. This is frankly ridiculous. At the very least the buttons could be changed to read "SAVE ALL PAGES" "SAVE THIS PAGE" and "CANCEL". Ideally the prompt should be changed to read "Save changes to this schematic page?" Buttons="SAVE" AND "CANCEL" and a "SAVE ALL PAGES" command could be added to a pull down menu for safety s sake. Please change this! '

36. Title: 'Multiple Undo s'

Description: 'OrCAD Capture currently has the ability to execute only one level of UNDO. This is completely unacceptable in a product that is touted as being a professional CAD tool. All other competing CAD tools I use (Mentor ePD PowerLogic Visio) have multiple levels of UNDO. For that matter even Microsoft Word has multiple levels of UNDO! This is a necessary feature in a tool of this sort due to the complexity of current designs and the inherent "let s draw the circuit this way since we can always go back" nature of modern schematic capture. The current state of OrCAD Capture allows zero flexibility in this regard and severely limits its usefulness as a "professional" CAD tool. Our firm has stopped using the program for new designs for this and other reasons. Unless it is changed it will be relegated to only opening legacy files.'

37. Title: 'Need alternative platform'

Description: 'Please provide tools on alternative platforms. My company will _never_ buy another Windows-based product! Your tools (and Tanner s) are the only reason we still have any Doze boxes left. Preferences: FreeBSD 4 Linux Solaris'

38. Title: 'Part Symbol Generation'

Description: 'The ability to generate a symbol from a text file would speed designs and make them more accurate. The file should be a .CSV file that can be output from Excel. Entering in a 1000 pin device can take all day when you include checking. Many IC vendors provide text files

of pinouts or these can be extracted from PDF datasheets. The information in the file would include Name Number Shape Type.'

39. **Title:** 'Hierarchy viewable within Capture'

Description: 'The ability to view hierarchical netnames in the schematic and on printouts is essential for designs repeating hierarchical blocks. A Hierarchy viewer that would take the information after the netlist has been generated would suffice.'

40. **Title:** 'Recent files list in capture'

Description: 'list is limited to 4 if you open a .bom file it goes on the list. increase the list to at least 10'

41. **Title:** 'Unrouting traces to vias'

Description: 'Multiple connections to a single via: difficult to unroute or move a single trace'

42. **Title:** 'Moving a via with multiple connections'

Description: 'When multiple connections intersect a single via and that via is moved not all lines get moved and multiple vias are created. '

43. **Title:** 'Add real macro feature'

Description: 'add a real macro feature (not one that you have to write code for) for capture and possibly spice. The macro feature as used with the old DOS system was perfect and a great time saver. Also allow a real block drag such as with the DOS version.'

44. **Title:** 'Capture multi-node selection'

Description: 'It would be great help if when trying to select a node component net etc and there are more than one item in very close proximity that a pop-up select box appears to allow the selection of the correct item.'

45. **Title:** 'Capture part movement and net reconnect'

Description: 'On moving components that have been connected to nets i.e. IC's the nets reconnect together in very dangerous ways. The only way around this so far is to disconnect the nets first move the component and then reconnect manually. Capture before version 9 used to drag IC s etc correctly (i.e. it would hold the pin connection firm and rubber band the net beyond this point). It would better to ask for net join or avoid on placing the component.'

46. **Title:** 'DXF Export of schematic should not include PSpice in drawing'

Description: When export schematic drawing to dxf format the PSpice information should not appear on the dxf file drawing.

47. **Title:** 'Password protection option with Schematic drawing and PCB Layout'

Description: Schematic drawing and PCB Layout can be Password protected.'

48. **Title:** 'Layout Footprint replacement deficiency'

Description: 'If a new part is used to replace a current part on a design Layout will not replace the footprint for the correct outline. The only way is to use AutoECO/Override Attrs. But this causes additional changes that may not be useful at that time It overrides all of the attributes in an existing board quote for the help file. The best solution is to add checks for footprints compatibility and the provision of selecting new footprints for the error check or a not so good solution is add an option to autoeco like AutoECO/Override Footprints .'

49. **Title:** 'Pin and Gate Swap functionality'

Description: 'Pin and Gate swap functionality needs enhancement. It is not well documented in OrCAD Capture. Accessing the screens to input pins gate swapping group is not obvious and

requires some educated guesses as to how swapping will be performed. Correcting these problems is pretty easy.

- 1) Add readable documentation, which better explains how it works and the differences between the two.
- 2) Add commands that allow ease of access to the popup screen for inputting pin/gate swapping options.'