Customers are required to follow certain criteria for all designs whether they are ultimately done in EECAD or by the customers themselves. These criteria, approved by EES Management, are listed below:

EECAD's "MUST" List

General EECAD "MUST" Guidelines

1. All designs **MUST** be assigned a drawing number at the start of the project. This includes designs installed in the machine as well as test stands or similar types of fixtures.

   - Requests for drawing numbers **MUST** be submitted via the EECAD job request form at https://mis.jlab.org/mis/apps/job_request/eecad/submit.cfm

   - The submission **MUST** include the necessary System, Subsystem, and Title information so that the appropriate drawing number can be assigned to the design.

2. All active designs, including prototypes, **MUST** be stored on the JLAB queue at \\jlabgrp\eecad\users private directories\your name. The minimum requirements include the project schematic, PCB layout, and the associated component library. Additional project information such as the BOM, gerber data, and any related firmware files or mechanical drawings **MUST** also be included in subfolders before the design is completed and installed. Folders will be created as needed to accommodate the various projects demands. Contact Lori, x-7332, or Roger, x-7083, for assistance.

3. Existing signed-off drawings/designs in need of modification **MUST** utilize an ECO form, \\Jlabgrp\eecad\ECO archives, to document the changes being made. Contact EECAD for additional guidance.

4. Drawings submitted to Document Control **MUST** have all the appropriate title block information filled in (reference the Document Control Group manual or contact EECAD for guidance).

5. Electronic copies of all signed-off drawings and related files **MUST** be stored in the appropriate system directory located at: \\Jlabgrp\eecad.

6. A unique design library **MUST** be created for each project design. This library preserves the components used in the design regardless of any changes and/or updates made to the Master Libraries.

7. The most up-to-date sheet formats **MUST** always be used. They can be found at \\jlabgrp\eecad\PCAD\Schematic_formats and \\jlabgrp\eecad\PCAD\PCB_formats. These formats have standard font styles, pad styles, print setups and Gerber generation setups included in them. Using them will save a substantial amount of time when printing documentation and generating fabrication data.

8. Standard parts from the Master Libraries **MUST** be used in all designs. They can be found at [\\Jlabgrp\eecad\PCAD\EECAD_LIBS](\\Jlabgrp\eecad\PCAD\EECAD_LIBS) or one's local PCAD folder if set up for automatic weekly library updates.

   - If a component does not yet exist, a request **MUST** be made for its creation via the EECAD job request form. Include a web link or data sheet and the specific pattern footprint desired.

   - If a draft schematic symbol or PCB pattern has already been created, it **MUST** be submitted for review and inclusion in the Master Libraries.

   - Completed component(s) will be returned to the requestor for review within two days.

9. Capital letters **MUST** be used on all drawings and PCB layouts. Lower-case letters are acceptable only when using scientific notation; i.e. u for micro, k for kilo, n for nano.

10. A completed documentation package **MUST** consist of the following:

    - Fabrication drawing - this tells the board house how to make the board and corresponds to the .fab gerber file.

    - Artwork drawing(s) - this can be important if the PCB has to be recreated and there is no other source file. It is also a good historical document.

    - Schematic

    - Assembly drawing(s) - consisting of views of installed components and a Bill of Materials detailing all the components necessary to build this board. Also included should be any special instructions concerning mounting hardware, order of assembly, etc.

    - The BOM includes the bare PCB, sockets, front panel, and any other components specific to the board. Typically this is done as an Excel spreadsheet and copied onto the drawing format. The BOM template can be found at [\\Jlabgrp\eecad\PCAD\Templates](\\Jlabgrp\eecad\PCAD\Templates).

    - Any firmware or similar documentation associated with the design.

11. Prototypes and/or designs never intended for installation in machine-related systems may be assigned a –PR designation in the revision block. It is recommended that this designation be used when multiple PCB fabrication runs will be needed before a final design is achieved. The project coordinator/engineer is responsible for making this determination. Following this procedure permits the development of a design without requiring immediate sign-off of the documentation and the submission of numerous ECO's as the design develops. However, the below rules MUST be strictly followed.

    - These prototype designs **MUST** be used for testing/debugging only.
    - Prototype and pre-production PCB soldermasks are to be colored RED for the initial fabrication. For the second and subsequent pre-production PCB fabrication runs, the soldermask color is to be BLUE.
    - Most designs **SHOULD NOT** fall into the above category and are to be given the standard Rev- assignment. First-run or prototype fabrications of this type **MUST** follow the above soldermask color codes. Any subsequent production PCB soldermasks **MUST** be GREEN.
    - If no changes are made to a prototype board and it is installed in a machine-related system, the revision block **MUST** be changed to "—" and the documentation **MUST** be submitted to

the Document Control Department. A pen and ink change **MUST** be made to the board on the silk screen indicating the appropriate drawing revision.

<u>Schematic Design "MUST" Guidelines</u>

1. Component text **MUST** always be horizontally oriented. Generally, pin numbers on 2-pin discrete components **SHOULD NOT** be visible. However, there may be exceptions to this recommendation for components such as those having an SOT footprint or similar multi-pin pattern.

2. Setting up component and net attribute parameters allows ERC to be run at the completion of the schematic and before beginning a PCB layout. ERC **MUST** always be run at the completion of the schematic design. Any detected ERC error or warning **MUST** be fully evaluated to determine the seriousness of the violation.

<u>PCB "MUST" Guidelines</u>

1. Familiarization with component and net attributes is essential for successful PCB designs. Setting up these parameters allows the interactive use of DRC. During manual routing, DRC may be turned ON at any time. DRC **MUST** always be run at the completion of the PCB design. Any detected DRC error or warning **MUST** be fully evaluated to determine the seriousness of the violation.

1. The design drawing number, revision, and design name **MUST** be included on the PCB Silkscreen. Typical font sizes for this text are 80–125 points in height and **SHOULD** be larger in size than the text for component Reference Designators.

2. Prototype and pre-production PCB's **MUST** make use of the -PR revision, which is to be included on the silkscreen. If it is deemed appropriate for the prototype board to be installed in the machine or facility, a pen and ink change shall be made to the -PR, indicating the most recent revision number which **MUST** match the drawing information submitted to Document Control.

3. Prototype and pre-production PCB soldermasks **MUST** be colored RED for the initial fabrication. For the second and subsequent pre-production PCB fabrication runs, the soldermask color is to be BLUE. All production PCB soldermasks are to be GREEN.

Schematic "SHOULD" Guidelines

Adhering to the following guidelines will insure that drawings are easier to work with. Items that are marked "must" are to be followed at all times. Items marked "should" indicate good drawing style. Items marked "shall" or "use" indicate techniques that if not done will make it difficult for others to edit or following the drawings.

1. Schematics **SHOULD** have a logical flow. Generally, inputs **SHOULD** be on the left and outputs on the right. Uniform and consistent spacing of symbols with respect to the sheet and each other is important.

2. All fonts used in a schematic **SHOULD** be "TrueType" only. Notes **SHOULD** be entered using the TrueType "Partstyle" font. Capital letters **SHOULD** be used on all drawings. Lower-case letters are acceptable when using scientific notation; i.e. u for micro, k for kilo, n for nano.

3. Unless there is more than one type of GND net on a schematic, one **SHOULD NOT** leave the net name turned on at each symbol. It is not necessary to leave the "type" attribute turned on for each symbol of the same connector.

4. "Snap to Grid" **SHOULD** be turned on. This is the default mode in all EECAD sheet outlines. A grid size of 100 mils **SHOULD** be used.   A grid size less than 50 mils **SHOULD NOT** be used for component placement and wiring in a schematic. A grid size of 25 mils may be used to position text adjacent to components.

5. Attempt to minimize the crossing of lines. Four-way wire connections **SHOULD** be avoided.

6. Active components generally have a separate symbol used to connect power, ground, and bypass capacitors.

    • In single-sheet designs or where dictated by critical design considerations, these "power" symbols and their respective bypass capacitors **MAY** be placed on the same sheet, preferably near the right or bottom of the sheet.

    • Larger, more complex designs **SHOULD** have the power symbols placed on the higher-numbered or final sheet(s) of the schematic.

7. Unconnected gates or those with grounded inputs **MUST** be included in the schematic. These gates **SHOULD** be included on the same sheet as the used gates in the IC package. Alternatively, they **SHOULD** be placed on the higher numbered sheets, which are generally associated with connections the power gates.

8. A table of contents on the first sheet is very helpful for complex drawings made up of several sheets.

9. "Sheet Connectors" **SHOULD** be used to link global nets across multiple sheets. Text associated with Sheet Connectors **SHOULD** be placed such that it is readable.

10. Operational amplifiers **SHOULD** generally be placed with the negative terminal up and the input on the left. Specific circuit topology issues **SHOULD** drive exceptions to this guideline.

11. Whenever possible, negative voltage symbols and/or "ports" **SHOULD** be oriented in a downward direction.

12. Critical nets **SHOULD** use attributes such as "Width" to define design rules that can later be used by DRC to ensure compliance to those parameters.

13. Embedded component attributes, such as "Part Number", "Voltage", or "Power", **SHOULD** be utilized to define specific device parameters and aid in the generation of a good BOM early in the design stage of the project.

PCB "SHOULD" Guidelines

2. Time spent on component placement and planning a routing strategy can help to minimize the number of vias and layers required to route the design. One **SHOULD** pay very close attention to this phase of the layout process cannot be overstressed.

3. For thru-hole layouts, a grid size of typically no less than 25-mils **SHOULD** be used for component placement and trace routing. When working with surface-mount layouts, whenever possible use no less than a 5-mil grid for both component placement and trace routing.

4. For power and ground distribution, Power and GND planes **SHOULD** be utilized rather than copper pours. Copper pours **MAY** be used on signal layers to prevent copper thieving and/or to provide additional capacitance and noise shielding. EECAD can provide additional guidance.

5. Planes and copper pours **SHOULD NOT** be routed to the edge of the PCB. Planes **SHOULD** be kept a minimum of 40-mils from the edge of the board. The "standard" JLAB PCB outlines have this feature already incorporated on the plane layers. Copper pours **SHOULD** be kept away from the board edge by a similar distance.

6. Flipping a component serves a purpose in the PCB editor. One **SHOULD NOT** flip a component unless it is intend to be mounted on the reverse side of the board.

7. When routing a board, one **SHOULD** make sure that one of the Route modes is used, not "Add Line" mode. Lines are not intelligent unless the "Reconnect Nets" command is executed.

8. The default font size for most component Reference Designators is 60 points in height. The font size **MAY** be modified as necessary to accommodate space limitations. Text less than 40 points **SHOULD NOT** be used.

9. Tolerances **SHOULD** be included for hole sizes in the drill symbol table. Recommended tolerances are as follows: +/-.003 for sizes < .055, +/-.004 for sizes .055 -.124, and +/-.005 for .125 and above.

10. Provide dimensions of the overall size of a board, including any cutouts or odd shapes. Dimensions are typically done with respect to the lower left corner of the board, but design parameters may dictate other measurement requirements. Also, provide a dimension from the dimension origin to the nearest mounting hole if possible. Dimension precision **SHOULD** be kept to two digits, unless higher precision is required.