If you are using a printed copy of this procedure, and not the on-screen version, then you MUST make sure the dates at the bottom of the printed copy and the on-screen version match.
The on-screen version of the Collider-Accelerator Department Procedure is the Official Version.
Hard copies of all signed, official, C-A Operating Procedures are available by contacting the ESSHQ Procedures Coordinator, Bldg. 911A

C-A OPERATIONS PROCEDURES MANUAL

15.4.2 Electrical Systems Design Room Work Procedure

(Electrical Systems Design Group Procedure C-A-EDG-001)

Note: This document was formerly a C-A Group Procedure. The content of the group procedure was reviewed by the Technical Supervisor. All approvals and/or issue dates of the original group procedure are maintained for present use.

Text Pages 3 to 23

Attachments

Hand Processed Changes

HPC No.	Date	Page Nos.	Initials

Approved: ***Signature on File*** _____
Collider-Accelerator Department Chairman

Date

M. Jacobellis

Electrical Systems Design Group Procedure
C-A-EDG-001
Original Issue Date: 07/01/02
Revision A

Note: Access to the 'dwg-server' is required to view links in this procedure.

COLLIDER – ACCELERATOR DEPARTMENT
ELECTRICAL DESIGN ROOM WORK PROCEDURES

I. INDEX

COLLIDER ACCELERATOR DEPARTMENT
BROOKHAVEN NATIONAL LABORATORY
Upton, New York 11972

C-A-EDG-001 – Sheet 1 of 20
Revision 00 Revision Date April 9, 2007
QA Category A-3

DESIGN ROOM WORK PROCEDURE
Electrical Systems Design Group

Supervisor Approval: _____ Date: _____

Revision Record

1.0 REVISION RECORD

00	INITIAL RELEASE	4-9-07

2.0 PROJECT WORK FLOW

Electrical Design Group Supervisor is responsible for the following

2.1 A project comes into the Design Group Supervisors office

2.2 Information regarding the project is discussed with the cognizant engineer/group leader or designee to obtain account number, details of the project, timeframe, category of drawingnumber index etc.

2.3 Document Control assigns a project number as follows:

2.3.1 A project number with syntax consisting of a "2 digit year–sequential number" - Example: 06-030 would be for a PCB Package consisting of a data base; including a schematic, drilling, assembly, and a B.O.M. – bill of material. They are linked and can be forward and backward annotated utilizing the Mentor software. A printed circuit board Intake Document would be required for this type of project.

2.3.2 A project number with syntax consisting of a "E-2 digit year–sequential number", E = electrical - Example: E-06-037 would be for an informative schematic not associated with a PCB package.

2.4 The information is entered on the Work Status Spread Sheet (see Figure 1 below).

2.5 For PCB projects, link the project number to the appropriate job title description.

2.6 Project priorities are established by the Design Room Supervisor. When multiple projects are in the queue, project priorities will be established by the Chief E.E.

2.7 Place the schedule in the Work Status Spread Sheet.

2.8 Assign a project to Electrical Design Group staff member.

2.9 Staff member discusses project with cognizant engineer/group leader or designee to begin the project. Additional information is added as the project progresses toward completion or if a modification to the schedule is necessary.

	A	B	C	D	E	F	G	H	I	J	L
1									COMPLETE		
2				ELECTRICAL PC BOARD WORK REQUESTS					IN PROGRESS		
3		WORK SCHEDULE							IN QUE		
4		JOB			SUBMITTED	ENGINEER	DESIGNER	STARTED	CHKNG/APP	COMPLETE	PROJECT#
5											
6		C-AD									
7											
8	C-AD	DUAL OUTPUT POWER SUPPLY INTERFACE			10/19/2006	TIAN	RICH				06-023
9	C-AD	ACCOUNT NO. 18071				STATUS:	10/31/06 parts complete				
10	C-AD	CAPACITOR OVERCURRENT CIRCUIT			5/19/2006	PABLO	DAN				06-010
11						STATUS:					
12											
13		AGS									
14											
15											
16	AGS	PPA PWR SUPPLY REGULATOR			3/30/2006	DANOWSKI	RICH				06-008
17	AGS					STATUS:	PARTS IN DATA FOLDER				
18	AGS	RF CHOPPER CONTROLLER			3/17/2005	GOULD					05-006
19	AGS				MENTOR	STATUS:	3/30/05 PARTS CHECKED/OMAR DOING SCHEMATIC				D22E512
20											
21		BOOSTER									
22											
23	BSTR	BMRPS WATER MAT DETECTOR			11/15/2005	ROSAS					CAD0643
24											
25	BSTR	SHARK21160 DSP BOARD			6/29/2004	DELONG			33.5 HRS DAN		04-048
26	BSTR	SHT2	REQUEST		MENTOR	STATUS:	1/6/04 JOE STARTING SCHEMATIC				D36-1603

Figure 1. Work Status Spread Sheet

3.0 DRAWING NUMBER LOG SHEETS

3.1 The Document Control Center assigns drawing numbers for the electrical design group. Numbers are assigned from the Drawing Log Sheets, which are located in eng-arch on 'dwg-server'. Figure 2 shows the Low Level RF log sheet.

ca101.xls [Read-Only]

	A	B	C	D	E	F	G	H	I	J
2	DRAWING NUMBER LOG SHEET									
3	C-A		LOW LEVEL RF							
5	DRAWING NO		TITLE						DATE	DRAWN BY
6										
7	CA101									
8										
9	CA1010001		LLRF DIGITAL CONTROLLER PMC DAUGHTER CARD PROTOTYPE S						May-05	M. JACOBELLIS
10	CA1010002		LLRF DIGITAL CONTROLLER PMC DAUGHTER CARD PROTOTYPE D						May-05	M. JACOBELLIS
11	CA1010003		LLRF DIGITAL CONTROLLER PMC DAUGHTER CARD PROTOTYPE A						May-05	M. JACOBELLIS
12	CA1010004		QUAD DSP XMC MODULE SCHEMATIC						Jun-05	R. PRZYBYLINSKI
13	CA1010005		QUAD DSP XMC MODULE DRILLING						Jun-05	R. PRZYBYLINSKI
14	CA1010006		QUAD DSP XMC MODULE ASSEMBLY						Jun-05	R. PRZYBYLINSKI
15	CA1010007		LLRF DIGITAL CONTROLLER PMC DAUGHTER CARD PROTOTYPE F						May-05	M. JACOBELLIS
16	CA1010008		RF UPGRADE, PMC TEST CARRIER SCHEMATIC						Oct-06	R. PRZYBYLINSKI
17	CA1010009		RF UPGRADE, PMC TEST CARRIER DRILLING						Oct-06	R. PRZYBYLINSKI
18	CA1010010		RF UPGRADE, PMC TEST CARRIER ASSEMBLY						Oct-06	R. PRZYBYLINSKI
19	CA1010011		RF UPGRADE HIGH SPEED ADC PMC SCHEMATIC						Jan-07	M. JACOBELLIS
20	CA1010012		RF UPGRADE HIGH SPEED ADC PMC DRILLING						Jan-07	M. JACOBELLIS
21	CA1010013		RF UPGRADE HIGH SPEED ADC PMC ASSEMBLY						Jan-07	M. JACOBELLIS
22	CA1010014		RF UPGRADE HIGH SPEED ADC PMC PROGRAMMED MODULE ASS						Jan-07	M. JACOBELLIS
23	CA1010015		RF UPGRADE HIGH SPEED DAC PMC SCHEMATIC						Jan-07	M. JACOBELLIS
24	CA1010016		RF UPGRADE HIGH SPEED DAC PMC DRILLING						Jan-07	M. JACOBELLIS
25	CA1010017		RF UPGRADE HIGH SPEED DAC PMC ASSEMBLY						Jan-07	M. JACOBELLIS
26	CA1010018		RF UPGRADE HIGH SPEED DAC PMC PROGRAMMED MODULE ASS						Jan-07	M. JACOBELLIS
27	CA1010019									

Sheet1 / Sheet2 / Sheet3

Figure 2 - Log Sheet for Low Level RF

The drawing numbers and titles can now be added to the appropriate drawings being generated.

4.0 DRAWING NUMBERING SYSTEMS WHEN USING AUTOCAD SOFTWARE

4.1 AGS, BOOSTER and TANDEM drawings are numbered as follows:

4.1.1 On the Drawing Format:
Example: D36-E1634 (this is how the number appears in the drawing title block)

D36	Tune Meter System (from D36 Booster Ring Drawing No. Index)
E	Electrical
1634	Sequential Drawing Number

4.1.2 On the Electronic File:

Syntax is as follows (lower case): d36-e1634.dwg
Multi sheet documents are numbered as follows: d16-e1634s1.dwg (sheet 1)
d36-e1634s2.dwg (sheet 2 of drawing D16-E1634)

These documents are filed in D36 library of the Document Management Program (Adept).

4.2 RHIC and NSRL drawings are numbered as follows:

4.2.1 On Drawing Format:

Example: 94028120 (this is how it appears in the drawing title block)

9402 V115 – RTDL INTERFACE PROGRAM (from RHIC Drawing No index)
8120 Sequential drawing number

4.2.2 On the Electronic File:

Syntax is as follows: 94028120.dwg
Multi sheet document is numbered as follows: 94028120-1.dwg
94028120-2.dwg (this is the second sheet of drawing 94028120)

These documents are filed in 9402 library of the Document Management Program (Adept).

5.0 DRAWING NUMBERING SYSTEMS WHEN USING MENTOR SOFTWARE

5.1 AGS, BOOSTER and TANDEM drawings are numbered as follows:

5.1.1 On the Drawing Format:

Example: D09-E2980 (this is how the number appears in the drawing title block)

D09 Function Generator (from D09 AGS Controls Drawing No. Index)
E Electrical
2980 Sequential Drawing Number

5.1.2 Electronic File:

Syntax is as follows (lower case): d09e2980.sbk
Multi sheet schematics all reside within one sbk file.
Multi sheet drilling and assemblies all reside within one .pcb file.

These documents are filed on the network under Expedition 'elec-server' in the corresponding project folder. Example of project folder name: 04-049-d09-e2980_Theisen.

5.2 RHIC AND NSRL DRAWINGS ARE NUMBERED AS FOLLOWS:

5.2.1 On Drawing Format:

Example: 75018203 (this is how it appears in the drawing title block)

7501 NIM INTERFACE MOTHER BOARD (from RHIC RF System Drawing No index)
8203 Sequential drawing number

5.2.2 Electronic File:

Syntax is as follows: 75018203.sbk
Multi sheet schematics all reside within one sbk file.
Multi sheet drilling and assemblies all reside within one .pcb file.

These documents are filed on the network under Expedition 'elec-server' in the corresponding project folder. Example of project folder name: 05-007-75018203_Butler.

5.3 C-AD MULTIUSE DRAWINGS ARE NUMBERED AS FOLLOWS:

5.3.1 On Drawing Format:

Example: CA1010001 (this is how it appears in the drawing title block)

CA101 LOW LEVEL RF (from C-A Drawing No index)
0001 Sequential drawing number

5.3.2 Electronic File:

Syntax is as follows: ca1010001.sbk
Multi sheet schematics all reside within one sbk file.
Multi sheet drilling and assemblies all reside within one .pcb file.

These documents are filed on the network under Expedition 'elec-server' in the corresponding project folder. Example of project folder name: 05_012_CA1010001_Smith

6.0 "RD" DRAWING NUMBERS AND SIGNATURES

6.1 Drawing Numbers

- "RD" Numbers are for drawings that are in the prototype stage of design.
- RD Drawings do not need a formal checking procedure.
- RD Drawings cannot be used for final installation of components.
- Drawings MUST be checked and approved and "RD" removed before base lining.

6.2 Signatures

RD Drawings need the following signatures only.
Signatures are written in the title block area only. NOT in the revision block

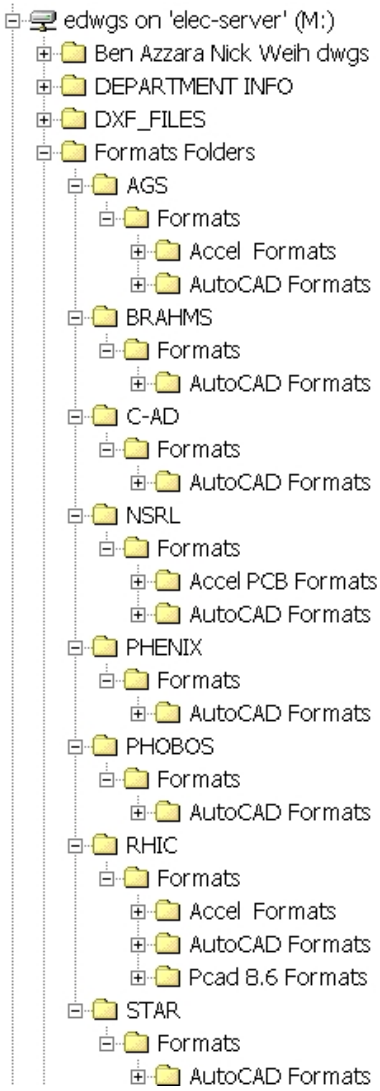
- DRAWN BY
- DESIGN APPROVAL
- ENGINEER APPROVAL

7.0 DRAWING FORMAT LOCATIONS

7.1 Listed below are the format names and the directory where they can be located.
The latest formats will also be available for NEW drawing by using Adept when creating a new drawing.

All the formats are located in the working directories in edwgs on ‘elec-server’ in the folder called Formats Folder. The formats include parts lists, which can be shut off or left on. The parts list is on the layer called parts list.

Important: Only start a new drawing with the latest format located in the directories listed below. When making a revision, be sure to replace old formats with the latest format.

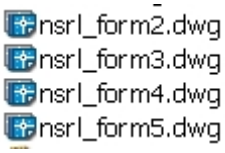


8.0 FORMAT NAMES AND LOCATIONS FOR AUTOCAD

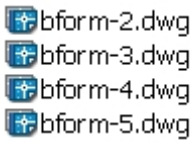
All formats are located in Format Folders on edwgs on ‘elec-server’ (M:)

8.1 AGS FORMAT NAMES:

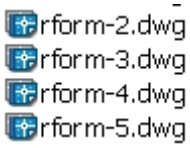
8.2 NSRL FORMATS:



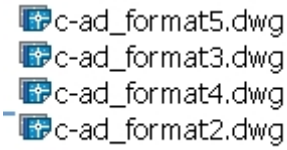
8.3 BRAHMS FORMATS:



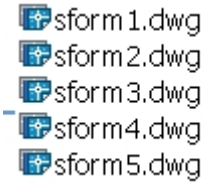
8.4 RHIC FORMATS:
These formats are also used for Phobos and Phenix



8.5 CAD FORMATS:



8.6 STAR FORMATS:



8.7 MENTOR FORMATS reside within the project folders.

9.0 HOW TO UPDATE AN AUTOCAD DRAWING TO A NEW FORMAT

9.1 Check existing drawing for 0,0 origin.

9.2 Open the AutoCad drawing that needs to be updated. Print out a copy of drawing at the current revision. This is so that you can put the correct signatures, dates, drawing title, parts list etc on the new
format. This information will be lost when you delete the old format.

9.3 Delete the entire exploded or incorrect format leaving the parts list intact if there is one, this will save you time in having to retype the parts list.

9.4 After you have deleted the old exploded format, purge the drawing until it is entirely clean. “Save” The drawing each purge until it is entirely clean. The new format will take on attributes from the old format and may not appear as it should if you do not do this.

9.5 Move the old Parts List off to the side – far enough so that, when you insert the new format, it does not overlap the new parts list.

9.6 From the AutoCad menu, pick “insert block”, then pick the browse button on the dialog box to pick the format you require. The dialog box should look like figure 1 with your required format name in the name box.

9.7 Click check box next to explode. Insert the new format at placement point 0,0.

9.8 Add attributes using your printout of the drawing from Step 8.1 above.

9.9 You can bring your parts list text from the old P/L into the new P/L by copying or moving the text. You may have to adjust the text a little due to changes in the P/L format from previous P/L formats. Delete the old P/L that was moved to the side.

9.10 Save the drawing after you have updated the drawing by using the same filename.

See the following example for inserting a format in AutoCad shown below.

Example: From the AutoCad menu click
Insert
Block
Browse for the format required, highlight and click on the file.
For first page formats, click the explode box.
Click ok
The dialog box should look like Figure 1 below.

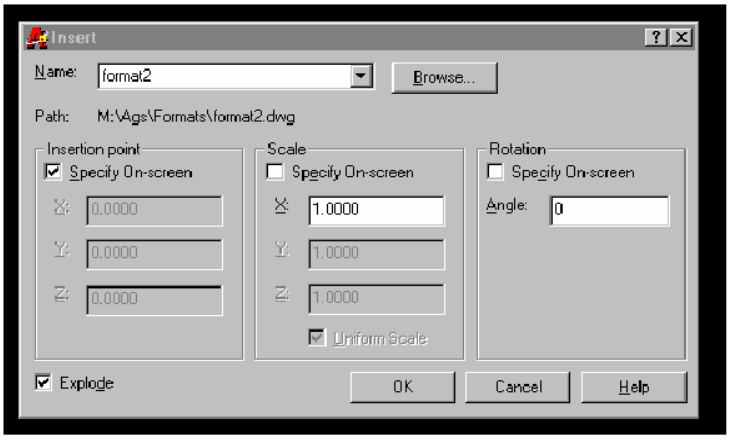


Figure 1

Enter 0,0 in the status bar as shown in Figure 2 below.
When you insert the new format, you will use this as the insertion point and it will place the new format in the same location.

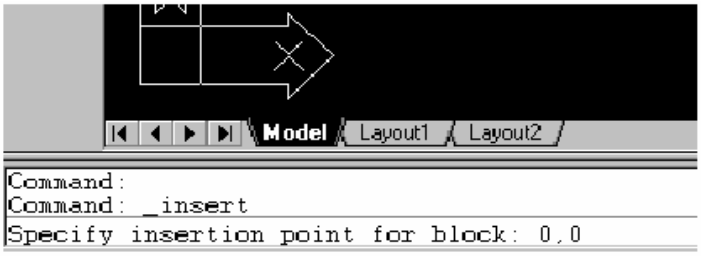


Figure 2

For Continuation Sheets
Leave the “explode box” unchecked
Click OK
Enter 0,0 in the status bar as shown in Figure 3 below.

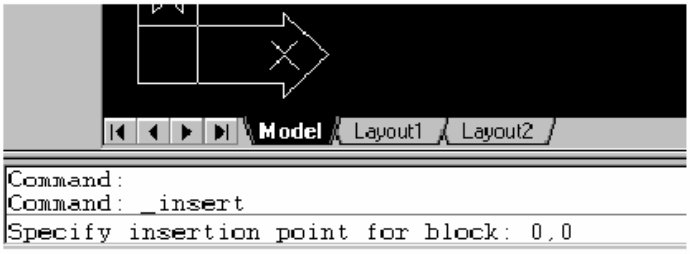


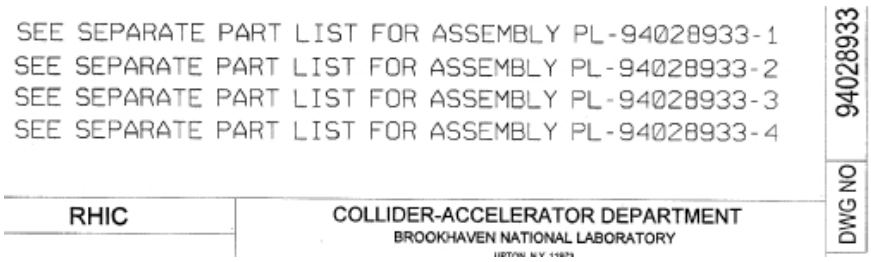
Figure 3

10.0 CONTENT OF DRAWINGS AND DRAWING TITLE

- Parts List – Non PCB Assembly Drawings ONLY
- Used On Drawing Number(S)
- Notes: In upper left corner of format
- Signatures (see 12.0)
  - Signatures will be signed on full size plotted original.

10.1 TABULATED PCB ASSEMBLY DRAWINGS

- Tabulated assembly drawings will have a note saying:
  - “See Separate Parts List” 52002004-1, 52002004-2 etc.



10.2 DRAWING TITLE

Top Line: PROJECT NAME (this is the project being worked on)
Middle Line: SYSTEM NAME: (this is the component being worked on)
Bottom Line: DRAWING NAME (this is the specific drawing name)

EXAMPLE 1

Top Line – Project Name:	**Low Power Driver/Beam Feedback Control**
Middle Line – System Name:	**Nim Interface 28 Mhz Daughter Board**
Bottom Line – Drawing Name:	**Schematic Diagram**

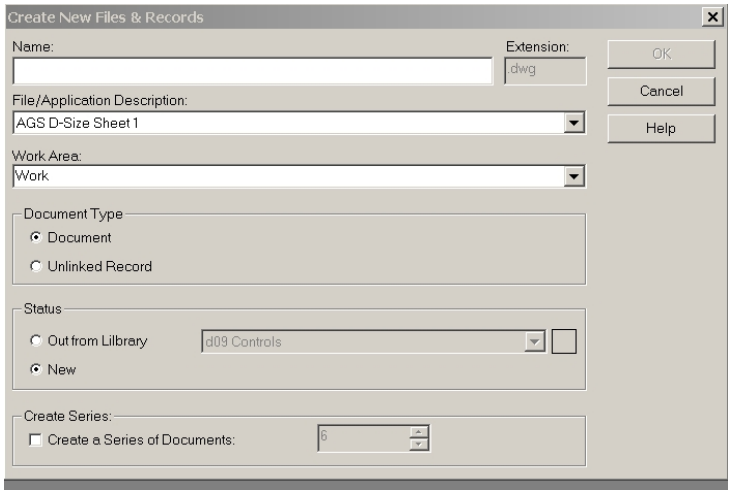
EXAMPLE 2

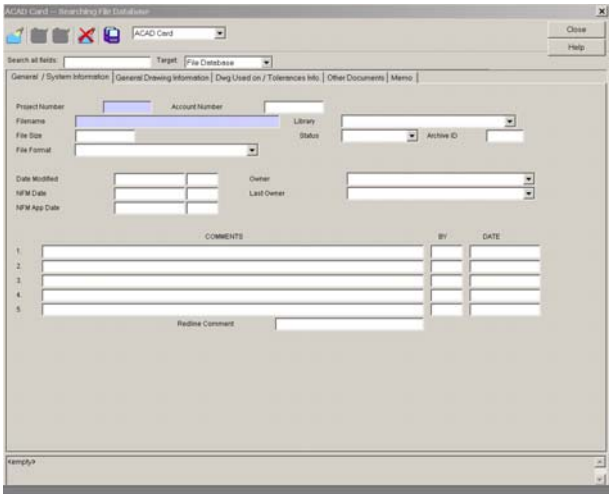
Top Line – Project Name:	**Kicker Power Supply**
Middle Line – System Name:	**Abort Kicker Voltage Ref**
Bottom Line – Drawing Name:	**Printed Wiring Board Assembly**

11.0 DOCUMENT MANAGEMENT

11.1 NON PCB DOCUMENT MANAGEMENT

The Electrical Design Group utilizes a Document Management Software program called Adept, by Synergis, in the creation and management of drawings. Refer to software operation guides for the operation of the program.





11.2 PCB DOCUMENT MANAGEMENT

The Electrical Design Group software for PCB packages is Mentor Graphics. Each PCB designed is assigned a unique Project Number and Project Folder. All files pertaining to the project reside within these folders. When a project is completed and signed off, the security permissions are changed to Read Only. The Project Folder is then moved from 'elec-server' project directories to 'elec-server' Project Released directory.

11.2.1 PCB FILE RELEASE DIRECTORY

Listed below are the project file names and the directory they are located in. All of the project files are located in directories in Expedition on 'elec-server'. The project files include schematics, PCB drilling, PCB assembly, B.O.M. and all files necessary to fabricate the printed wiring board.

12.0 CHECKING PROCEDURE

12.1 THE CHECKING PAGKAGE FOR MECHANICAL DRAWINGS

12.1.1 The checking package for Mechanical Drawings will include the following:
All Assemblies, Sub-Assemblies and Detail drawings
Layouts and Sketches provided by the engineer
Data Sheets of Vendor items
Photo copies of ASTM Number sheets called out on the drawings – if used.
Photo copies of MS Pages called out for hardware – if used.
Checklists:

- Acad Front Panel Detail and Screen checklist.doc
- Acad Front Panel Module Assembly checklist.doc

All .doc files are located on edwgs on 'elec-server' (M:) \Department Info\Procedures*.doc

12.1.2 Drawings are checked for technical accuracy and compliance to standards.

12.1.3 Redlined prints from the cognizant engineer are returned to the Electrical Design Group.

12.1.4 Corrections are made to drawings by the designer / drafter from all marked-up check prints.

12.1.5 Prints are returned to the Electrical Design Group.

12.1.6 Repeat steps 4 and 5 if necessary

12.1.7 After all corrections are made, the engineering process is completed. Full size plots are made and submitted to the checker and cognizant engineer for final check and signatures.

12.2 THE CHECKING PAGKAGE FOR ELECTRICAL DRAWINGS

12.2.1 The checking package for Electrical Drawings will include the following:
All Schematics, Printed Wiring Board Drilling, Printed Wiring Board Assembly, Parts List and Detail Drawings.

Layouts and Sketches provided by the engineer.
Gerber plots of all layers of artwork.
Checklists:
Mentor Standard Schematic checklist.doc
Mentor Standard Drilling checklist.doc
Mentor Standard Assembly checklist.doc

All .doc files are located on edwgs on 'elec-server' (M:) \Department Info\Procedures*.doc

13.0 DRAWING SIGNATURE PROCEDURE

13.1 SIGNATURES BASED ON QA LEVELS

13.1.1 QA LEVEL A3

A. Designer	(name typed in electronic file)
B. Checker	(signature signed on full size original)
C. Design approval	(signature signed by Design Room Supervisor on original)
D. Engineer approval	(signature signed by Engineer on original)
E. Supervisor Approval	(signature signed by Chief Mechanical Engineer or designee on original)

F. The two open boxes are for signatures determined by Chief E.E..
Example: SAFETY APPROVAL

13.1.2 QA LEVEL A2

A. Designer	(name typed in electronic file)
B. Checker	(signature signed on full size original)
C. Design Approval	(signature signed by Design Room Supervisor on original)
D. Engineer Approval	(signature signed by Engineer on original)
E. Supervisor Approval	(signature signed by Chief Mechanical Engineer or designee on original)

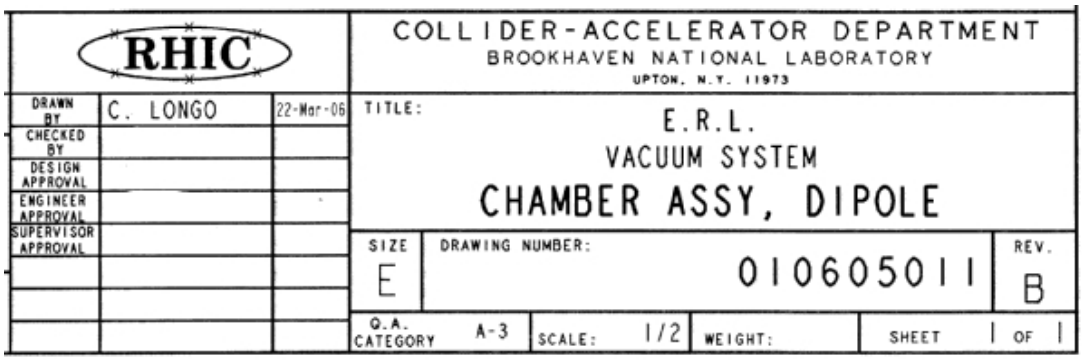
F. Division head / deputy division head (signature signed on original)
G. The Two Open Boxes Are For Signatures Determined By Chief E.E.
Example: SAFETY APPROVAL

13.1.3 QA LEVEL A1

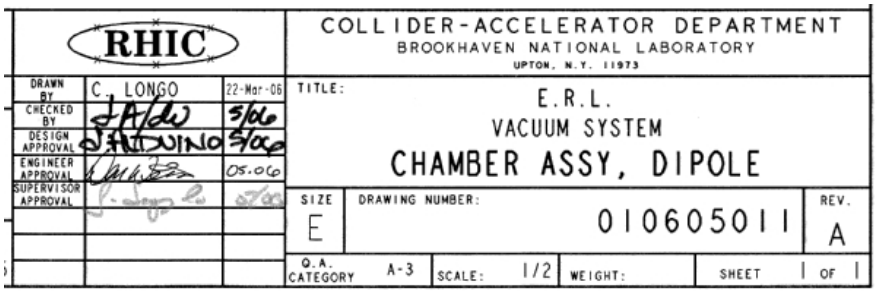
A. Designer	(name typed in electronic file)
B. Checker	(signature signed on full size original)
C. Design Approval	(signature signed by Design Room Supervisor on original)
D. Engineer Approval	(signature signed by Engineer on original)
E. Supervisor Approval	(signature signed by Chief Mechanical Engineer or designee on original)
F. QA Approval	(signature signed by Quality Assurance Rep on original)
G. Safety Approval	(determined by ES&H/QA division head)
H. Div Head/Deputy Div Head	(signature signed on original)
I. Dept Head/Deputy Dept Head	(signature signed on original)

13.2 SIGNATURE ON FORMATS

13.2.1 REV A ELECTRONIC FILE: NO NAMES TYPED IN TITLE BLOCK

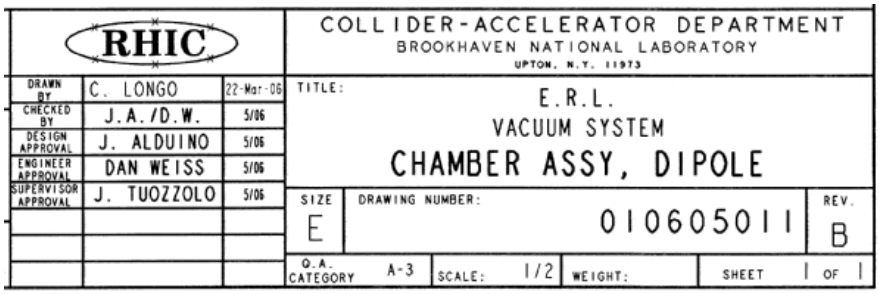


13.2.2 REV A PLOTTED ORIGINAL: SIGNED SIGNATURES



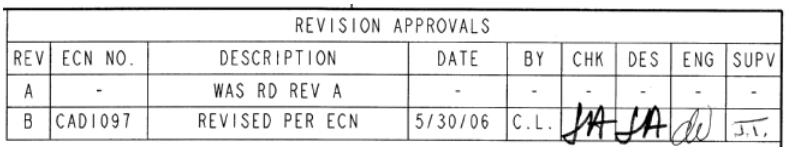
RHIC			COLLIDER-ACCELERATOR DEPARTMENT BROOKHAVEN NATIONAL LABORATORY UPTON, N.Y. 11973			
DRAWN BY	C. LONGO	22-Mar-06	TITLE: E.R.L. VACUUM SYSTEM CHAMBER ASSY, DIPOLE			
CHECKED BY	J.A./D.W.	5/06				
DESIGN APPROVAL	J. ALDUINO	5/06				
ENGINEER APPROVAL		05.06				
SUPERVISOR APPROVAL			SIZE: E	DRAWING NUMBER: 010605011		REV.: A
			Q.A. CATEGORY: A-3	SCALE: 1/2	WEIGHT:	SHEET 1 OF 1

13.2.3 REV B ELECTRONIC FILE: TYPED IN TITLE BLOCK



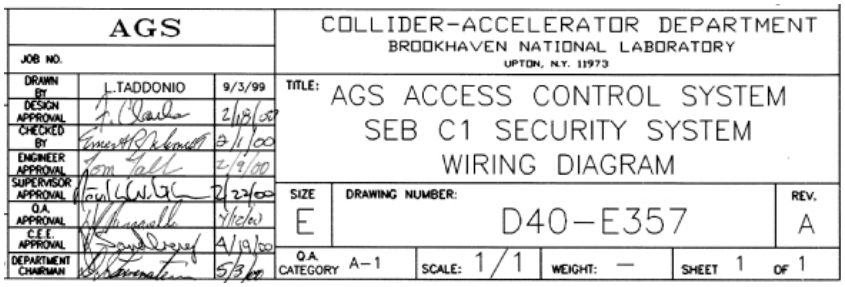
RHIC			COLLIDER-ACCELERATOR DEPARTMENT BROOKHAVEN NATIONAL LABORATORY UPTON, N.Y. 11973			
DRAWN BY	C. LONGO	22-Mar-06	TITLE: E.R.L. VACUUM SYSTEM CHAMBER ASSY, DIPOLE			
CHECKED BY	J.A./D.W.	5/06				
DESIGN APPROVAL	J. ALDUINO	5/06				
ENGINEER APPROVAL	DAN WEISS	5/06				
SUPERVISOR APPROVAL	J. TUOZZOLO	5/06	SIZE: E	DRAWING NUMBER: 010605011		REV.: B
			Q.A. CATEGORY: A-3	SCALE: 1/2	WEIGHT:	SHEET 1 OF 1

13.2.4 REV B PLOTTED ORIGINAL: SIGNED IN REV BOX



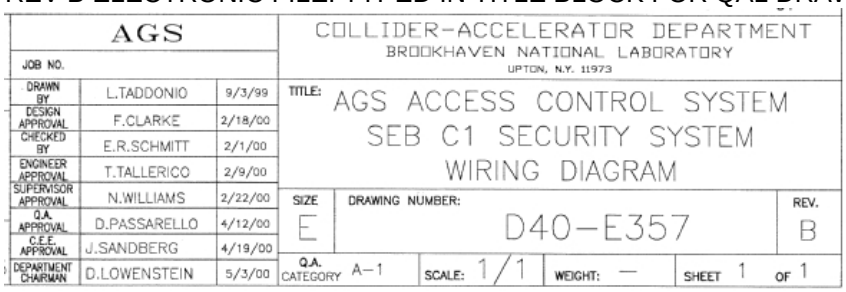
REVISION APPROVALS								
REV	ECN NO.	DESCRIPTION	DATE	BY	CHK	DES	ENG	SUPV
A	-	WAS RD REV A	-	-	-	-	-	-
B	CAD1097	REVISED PER ECN	5/30/06	C.L.	JA	JA	DW	J.T.

13.2.5 REV A PLOTTED ORIGINAL: SIGNED SIGNATURES FOR QA1 DRAWINGS



AGS			COLLIDER-ACCELERATOR DEPARTMENT BROOKHAVEN NATIONAL LABORATORY UPTON, N.Y. 11973			
JOB NO.			TITLE: AGS ACCESS CONTROL SYSTEM SEB C1 SECURITY SYSTEM WIRING DIAGRAM			
DRAWN BY	L.TADDONIO	9/3/99				
DESIGN APPROVAL	F. Clarke	2/18/00				
CHECKED BY	Ernest R. Schmitt	2/1/00				
ENGINEER APPROVAL	Tom Tallerico	2/9/00				
SUPERVISOR APPROVAL	Nan C.W.W.	2/22/00				
Q.A. APPROVAL	D. Passarello	4/12/00	SIZE: E	DRAWING NUMBER: D40-E357		REV.: A
C.E.E. APPROVAL	J. Sandberg	4/19/00				
DEPARTMENT CHAIRMAN	D. Lowenstein	5/3/00	Q.A. CATEGORY: A-1	SCALE: 1/1	WEIGHT: —	SHEET 1 OF 1

13.2.6 REV B ELECTRONIC FILE: TYPED IN TITLE BLOCK FOR QA1 DRAWINGS



AGS			COLLIDER-ACCELERATOR DEPARTMENT BROOKHAVEN NATIONAL LABORATORY UPTON, N.Y. 11973			
JOB NO.			TITLE: AGS ACCESS CONTROL SYSTEM SEB C1 SECURITY SYSTEM WIRING DIAGRAM			
DRAWN BY	L.TADDONIO	9/3/99				
DESIGN APPROVAL	F.CLARKE	2/18/00				
CHECKED BY	E.R.SCHMITT	2/1/00				
ENGINEER APPROVAL	T.TALLERICO	2/9/00				
SUPERVISOR APPROVAL	N.WILLIAMS	2/22/00				
Q.A. APPROVAL	D.PASSARELLO	4/12/00	SIZE: E	DRAWING NUMBER: D40-E357		REV.: B
C.E.E. APPROVAL	J.SANDBERG	4/19/00				
DEPARTMENT CHAIRMAN	D.LOWENSTEIN	5/3/00	Q.A. CATEGORY: A-1	SCALE: 1/1	WEIGHT: —	SHEET 1 OF 1

14.0 PROCESSING DRAWINGS

14.1 PROCESSING "RD" DRAWINGS (NON PCB DRAWINGS)

14.1.1 Complete the drawing(s).

14.1.2 Print a copy (copies) with the words "CHECK PRINT" on it. Submit check print(s) to Electrical Design Group supervisor/checker for initial check. Prints are checked to group standards, design and drafting practices, checklists and templates.

14.1.3 Update prints if necessary.

14.1.4 Submit new "CHECK PRINT" stamped prints to supervisor/checker. Submit to the cognizant engineer/group leader or designee for final check.

14.1.5 Repeat Steps 3 and 4 if necessary and resubmit.

14.1.6 When drawings are ready for signatures, print out full size prints and obtain signatures.

14.1.7 Submit approved drawings with signatures to Document Control Center for scanning. (.tif images)

14.1.8 ADEPT – Locate and electronically enter signatures in the designated areas of the title block on the drawings. "Sign in" the drawings. (AutoCad)

14.1.9 Document Control personnel will send email notification to the designer and system administrator that the drawings have been scanned and .tifs are located on "eng-arch".

14.1.10 System Administrator processes RD drawing into Adept.

14.2 PROCESSING "RD" PCB DRAWINGS

14.2.1 Complete the drawing(s).

14.2.2 Print a copy (copies) with the words "CHECK PRINT" on it. Submit check print(s) to Electrical Design Group supervisor/checker for initial check. Check prints to submit are Schematic(s), Drill Drawing(s), PCB Assembly Drawing(s), PCB individual layers with indication of what layer they are, PLD drawings – if required. Front panel detail and assembly drawings would also be included in check print package – if required. Prints are checked to group standards, design and drafting practices, checklists and templates. Gerber files are checked using CAM350 or PCB software Gerber tool.

14.2.3 Update prints if necessary.

14.2.4 Submit new "CHECK PRINT" stamped prints to supervisor/checker. Submit to the cognizant Engineer/Group leader or designee for final check.

14.2.5 Repeat Steps 3 and 4 if necessary and resubmit.

14.2.6 When drawings are ready for signatures, print out full size prints and obtain signatures.

14.2.7 Submit approved drawings with signatures to Document Control Center for scanning. (.tif images)

14.2.8 ADEPT – Locate and electronically enter signatures in the designated areas of the title block on the drawings. "Sign in" the drawings. (AutoCad)

14.2.9 PCAD and Mentor – Locate and electronically enter signatures in the designated areas of the title Block on the drawings.

14.2.10 Document Control personnel will send email notification to the designer and system administrator that the drawings have been scanned and .tifs are located on "eng-arch".

14.2.11 If done in PCAD: Zip all PCB files together using the Schematic number as the number for the ZIP file. Add the revision as the suffix to the filename in lower case.

14.2.12 System Administrator processes RD drawing into Adept.

14.2.13 For PCAD PCB drawings, System Administrator moves the previous version of the Zip file to the proper area in obsolete on the network server. Then copies the file to the correct area on the network in servers designated erelease or CCB1.

14.2.14 For Mentor drawings, System Administrator will move Project Folder to Project Release Directory.

14.3 PROCESSING DRAWINGS (NON PCB DRAWINGS)

14.3.1 Complete the drawing(s).

14.3.2 Print a copy (copies) with the words “CHECK PRINT” on it. Submit check print(s) to Electrical Design Group supervisor/checker for initial check. Prints are checked to group standards, design and drafting practices, checklists and templates.

14.3.3 Update prints if necessary.

14.3.4 Submit new “CHECK PRINT” stamped prints to supervisor/checker. Submit to the cognizant engineer/group leader or designee for final check.

14.3.5 Repeat Steps 3 and 4 if necessary and resubmit.

14.3.6 When drawings are ready for signatures, print out full size prints and obtain signatures.

14.3.7 Submit approved drawings with signatures to Document Control Center for scanning. (.tif image)

14.3.8 ADEPT – Locate and electronically enter signatures in the designated areas of the title block on the drawings. “Sign in” the drawings. (AutoCad)

14.3.9 Document Control personnel will send email notification to the designer and system administrator that the drawings have been scanned and .tifs are located on “eng-arch”.

14.3.10 System Administrator processes RD drawing into Adept.

14.4 PROCESSING PCB DRAWINGS (NON “RD”)

14.4.1 Complete the drawing(s).

14.4.2 Print a copy (copies) with the words “CHECK PRINT” on it. Submit check print(s) to Electrical Design Group supervisor/checker for initial check. Check prints to submit are Schematic(s), Drill Drawing(s), PCB Assembly Drawing(s), PCB individual layers with indication of what layer they are, PLD drawings – if required. Front panel detail and assembly drawings would also be included in check print package – if required. Prints are checked to group standards, design and drafting practices, checklists and templates. Gerber files are checked using CAM350 or PCB software Gerber tool.

14.4.3 Update prints if necessary.

14.4.4 Submit new “CHECK PRINT” stamped prints to supervisor/checker. Submit to the cognizant engineer/group leader or designee for final check.

14.4.5 Repeat Steps 3 and 4 if necessary and resubmit.

14.4.6 When drawings are ready for signatures, print out full size prints and obtain signatures.

14.4.7 Submit approved drawings with signatures to Document Control Center for scanning. (.tif images)

14.4.8 ADEPT – Locate and electronically enter signatures in the designated areas of the title block on the drawings. “Sign in” the drawings. (AutoCad)

14.4.9 PCAD and Mentor – Locate and electronically enter signatures in the designated areas of the title Block on the drawings.

14.4.10 Document Control personnel will send email notification to the designer and system administrator that the drawings have been scanned and .tifs are located on “eng-arch”.

14.4.11 If done in PCAD: Zip all PCB files together using the Schematic number as the number for the ZIP file. Add the revision as the suffix to the filename in lower case.

14.4.12 System Administrator processes RD drawing into Adept.

14.4.13 For PCAD PCB drawings, System Administrator moves the previous version of the Zip file to the proper area in obsolete on the network server. Then copies the file to the correct area on the network in servers designated erelease or CCB1.

14.4.14 For Mentor drawings, System Administrator will move Project Folder to Project Release Directory.

14.5 PROCESSING TABULATED SCHEMATIC DIAGRAMS IN MENTOR

GOAL: Have multiple schematics and parts lists that can be made from a single PCB drilling. Create non-intelligent documentation that will not compromise the design.

14.5.1 For a tabulated schematic, first rename the schematic in Design Capture. You must be in Design Capture to rename the schematic. Do not attempt to rename the schematic in Windows Explorer. You are going to add –1 to the drawing name. (Example 75018233 would now be 75018233-1). File, Save and Add to the project. You can now remove 75018233 from the project by right clicking under project files on 71058233.sbk and select remove file from project.

14.5.2 Now you have a 75018233–1 schematic open. Change the drawing numbers on the bottom and side of the format from 75018233 to 75018233-1, file and save.

14.5.3 You are now ready to create a –2 schematic in Design Capture. Go to menu bar and click on File, Save As. Enter 71058233-2 where it says File Name.

14.5.4 A dialogue box will appear asking, “Add file 75018233-2.sbk to the current project?” Click on yes. You will now see that 75018233-2 has been added to the workspace.

14.5.5 Go to the workspace and remove 71058233-1 from the project for now.

14.5.6 Now you have a 75018233–2 open. Go to the drawing numbers on the bottom and side of the format and change the 75018233-1 to a 75018233-2 and File Save. You can now edit the schematic to reflect the variant assembly. Incorporate changes required to the schematic.

14.5.7 When replacing discrete components, go to the DNP (Do Not Populate) partition and replace the components from there. Make sure to keep the same Reference Designators needed.

14.5.7.1 Remember the only thing you can change would be component values that are in the same package and remove components from the schematic that are not used in the PCB Assembly. The circuitry has to stay the same as it was in the –1 version. Any change that would change the artwork would require a new board and new drawing numbers. Add whatever text, charts and notes to explain anything that might not be clear. Remember to File Save the drawing.

14.5.8 Create a parts list. Go to the Design Capture Utilities dialogue box and run the Packager. After that is completed, open the CDB to BOM dialogue box. Put 75018235-2.bom in the

output file and run the CDB to BOM. In the output file you would put in 75018235-2.bom run the CDB to BOM.

14.5.9 Bring the schematic back in sync with the PCB Data base. Remove 75018233-2 from the project file. Next open 75018233-1 and do a file "Save As". A dialogue box will appear an ask, "75018233-1 already exists. Do you want to replace it?" Click yes. Another dialogue box will appear and ask, "Add file 75018233-1 to the current project?" Click on yes. You will see 75018233-1 in the workspace. You should not see any other dash number schematics in the workspace.

14.5.10 The final step is to get the schematic back in sync with the PCB. First make absolutely sure that the –1 schematic is the only schematic in the workspace. Run the packager again and forward annotate into the PCB and everything will be in sync.

Repeat items 2 through 5 for as many variant assemblies that are required.

14.5.11 Go to windows explorer to the schematic folder. Remove the original schematic and its backup file once you have confirmed everything is in sync and working.

14.6 PROCESSING TABULATED ASSEMBLY DRAWINGS WITH SEPARATE PARTS LIST:

14.6.1 Tabulated assembly drawings will indicate the number of the separate parts list associated with that assembly by a dash (-) number.

14.6.2 Only the parts list of the assembly affected by the ECN will have the revision letter advanced.

14.6.3 The assembly drawing revision letter will advance with each ECN indicating how many ECNs have been written to the document.

14.6.4 If a new assembly configuration is added (dash number), the new parts list will be Rev "A" and the assembly revision letter will advance indicating there has been a change to the document.

14.6.5 Associated PC board drill drawing and schematic WILL NOT advance the revision letter unless there is an actual change to the board.

15.0 PROCESSING REVISIONS

15.1 PROCESSING REVISIONS OF RELEASED DRAWINGS

Released drawings can only be updated with an approved ECN

15.1.1 Obtain the ECN which will revise the drawing(s).

15.1.2 Incorporate and complete the drawings(s)

15.1.3 Print out hard copy/copies with the words "CHECK PRINT" on it. Submit check print(s) to Electrical Design Group supervisor / checker for initial check. Prints are checked to group standards, design and drafting practices, checklists and templates.

15.1.4 Update prints if necessary.

15.1.5 Submit new "CHECK PRINT" stamped prints to supervisor/checker. Submit to the cognizant engineer/group leader or designee for final check.

15.1.6 Repeat Steps 3 and 4 if necessary and resubmit.

15.1.7 When drawings are ready for signatures, print out full size prints and obtain signatures. Sign ECN indicating you have incorporated the ECN.

15.1.8 Submit approved drawings with signatures and ECN to Documentation Control Center for scanning. (.tif images)

15.1.9 ADEPT – Locate and electronically enter signatures in the designated areas of the title block on the drawings. “Sign in” the drawings. (AutoCad)

15.1.10 Document Control personnel will send email notification to the designer and system administrator that the drawings have been scanned and .tifs are located on “eng-arch”.

15.1.11 System Administrator processes RD drawing into Adept.

15.2 PROCESSING REVISIONS OF MULTI SHEET MECHANICAL DRAWINGS:

Released drawings can only be updated with an approved ECN

15.2.1 Sheet one of a multi sheet drawing will have the current Rev letter indicating how many revisions have been made to the document. Sheet one will have a “Rev Box” indicating the Rev level of all sheets. Only the sheets being revised will advance Rev. letter. All other sheets indicate which sheets or parts lists are affected by the ECN in the description column.

15.2..2 Multi detail drawings will be revised in the same manner.

REV	REV	REV	REV	REV	REV	REV	REV	REV	REV STATUS OF SHEETS
NO	NO	NO	NO	NO	NO	2	1	SHEET	

15.3 PROCESSING OF REVISIONS OF RELEASED PCB DRAWINGS

Released drawings can only be updated with an approved ECN

15.3.1 Obtain the ECN to revise the drawing(s).

15.3.2 Incorporate the changes on the drawings(s) per the ECN.

15.3.3 Print a copy (copies) with the words “CHECK PRINT” on it. Submit check print(s) to Electrical Design Group supervisor/checker for initial check. Check prints to submit are Schematic(s), Drill Drawing(s), PCB Assembly Drawing(s), PCB individual layers with indication of what layer they are, PLD drawings – if required. Front panel detail and assembly drawings would also be included in check print package – if required. Prints are checked to group standards, design and drafting practices, checklists and templates. Gerber files are checked using CAM350 or PCB software Gerber tool.

15.3.4 Update prints if necessary

15.3.5 Submit new “CHECK PRINT” stamped prints to supervisor/checker. Submit to the cognizant engineer/group leader or designee for final check.

15.3.6 Repeat steps 3 and 4 if necessary and resubmit.

15.3.7 When drawings are ready for signatures, print out full size prints and obtain signatures. Sign ECN indicating you have incorporated the ECN.

15.3.8 Submit approved drawings with signatures and ECN to Documentation Control Center for scanning. (.tif images)

15.3.9 ADEPT – Locate and electronically enter signatures in the designated areas of the title block on the drawings. “Sign in” the drawings. (AutoCad)

15.3.10 PCAD and Mentor – Locate and electronically enter signatures in the designated areas of the title Block on the drawings.

15.3.11 Document Control personnel will send email notification to the designer and system administrator that the drawings have been scanned and .tifs are located on “eng-arch”.

15.3.12 If done in PCAD: Zip all PCB files together using the Schematic number as the number for the ZIP file. Add the revision as the suffix to the filename in lower case.

15.3.13 System Administrator processes RD drawing into Adept.

15.3.14 For PCAD PCB drawings, System Administrator moves the previous version of the Zip file to the proper area in obsolete on the network server. Then copies the file to the correct area on the network in servers designated erelease or CCB1.

15.3.15 For Mentor drawings, System Administrator will move Project Folder to Project Release Directory.

16.0 PRINTING RELEASED DRAWINGS

16.1 Final released drawings with signatures are printed from the “eng-arch” directory

16.2 Copy requests should be given to Document Control Center personnel REQUEST FORM

These are the final approved copies of drawings that will be printed and sent out to engineers, shops, vendors etc.

Drawings will be scanned and filed in the following directories: