## EEE 193A Suggested Milestones

<table>
<thead>
<tr>
<th>No.</th>
<th>Activity Name</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
<th>14</th>
<th>15</th>
<th>16</th>
<th>17</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>form team and establish</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>task project</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>preliminary designs</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>team work evaluation structure and</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>operational evaluation structure</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>model parts acquisition</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>test options/alternative designs</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>design analysis</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>design development</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>final project repeat</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>finalize detailed project</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>design review</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>material acquisition</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>design verification testing</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>system integration E</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>final project repeat</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>final project repeat deadline</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

## EEE 193B Suggested Milestones

<table>
<thead>
<tr>
<th>No.</th>
<th>Activity Name</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
<th>14</th>
<th>15</th>
<th>16</th>
<th>17</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>system EE &amp; Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>project review and plan</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>PCB</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>parts acquisition</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>PCB setup &amp; test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>design review</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>final project repeat</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>deadline to turn in PCBs to shop</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>system integration &amp; test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>project demonstration</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>final project report</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>final project repeat deadline</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Notes
- LogXYS: Start Finish Planned Actual
Table of Contents

Part 1: EEE 193A Engineering Design .........................................................................................................5
Preface ...........................................................................................................................................................5
Introduction to EEE 193A Senior Design Project...........................................................................................5
Course Description ........................................................................................................................................6
Course Schedule ...........................................................................................................................................7
Course Suggested Milestones.......................................................................................................................8
  193A Milestone Chart ...................................................................................................................................8
  193B Milestone Chart ...................................................................................................................................9
Course Assignments .....................................................................................................................................10
  H1: Resume and Possible Projects List Assignment ......................................................................................10
  H2: Library Project Research Report ...........................................................................................................10
  H3: Project Goals and Personal Goals Assignment .......................................................................................10
  H4: Functional Description and Concept Diagram .......................................................................................10
  H5: Specifications ..........................................................................................................................................10
  H6: Project Task Schedule Assignment ........................................................................................................11
  H7: Theory of Operation, Block Diagram, and Design Calculations ..........................................................11
  H8: Design Review Outline ..........................................................................................................................11
  H9: Schematics, Component Layout, and Package Layout ........................................................................11
  H10: Project Cost Analysis ...........................................................................................................................11
  H11: Test Procedures .....................................................................................................................................11
  H12: Final Project Report Outline ..............................................................................................................11
  L1: Power Supply Design Lab Assignment ................................................................................................12
  L2: Project Status Reports ..........................................................................................................................12
  L3: Design Verification Demonstration 1 .......................................................................................................12
  L4: Project Presentations ..............................................................................................................................12
  L5: Design Verification Demonstration 2 .......................................................................................................13
  L6: Design Verification Demonstration 3 .......................................................................................................13
  L7: Design Verification Demonstration 4 .......................................................................................................13
  L8: Design Reviews (see Design Review Section) .......................................................................................13
  L9: Design Verification Demonstration 5 (3 or more Modules) .................................................................13
  L10: Design Verification Demonstration 6 (System) ..................................................................................13
Design Reviews ..............................................................................................................................................14
Final Project Report Requirements ................................................................................................................15
Background information on Project planning: ...............................................................................................16
  Project Planning Steps ................................................................................................................................16
  Time Line Chart (Gantt Chart): ..................................................................................................................16
  Typical Problems with Project Plans ........................................................................................................17
  Reference Sources for Project Planning .....................................................................................................17
PCB Guidelines - Shop Document .................................................................................................................18
Design References ......................................................................................................................................20
Computer Aided Design Information ............................................................................................................23
  PSpice on the VAX ......................................................................................................................................23
  A Blow-by-Blow introduction to OrCAD .....................................................................................................28
  OrCAD PCB Notes .....................................................................................................................................31
  Addendum to the OrCAD-PCB Manual ........................................................................................................35
  OrCAD NOTES ..........................................................................................................................................38
Inserting OrCAD Schematics into word processor documents ....................................................................53
  Wintek Notes ..............................................................................................................................................54
68HC711 Notes ..............................................................................................................................................58
An Improved Rectifier Circuit Analysis ........................................................................................................61
Example EEE193A Assignments ..................................................................................................................79
Preface .................................................................................................................................................145
Introduction to EEE 193B Senior Design Project.................................................................145
Course Description ...................................................................................................................146
Course Schedule ..................................................................................................................147
Course Suggested Milestones .............................................................................................148
Project Planning ..................................................................................................................149
  Project Plan Assignment Requirements ........................................................................149
  Project Planning Steps ....................................................................................................149
  Time Line Chart (Gantt Chart) .......................................................................................150
  Typical Problems with Project Plans ............................................................................151
  Reference Sources for Project Planning ......................................................................151
Progress Reports ..............................................................................................................152
  Progress Report Assignment Requirements ............................................................152
Design Reviews ................................................................................................................153
Final Project Report Requirements ....................................................................................154
Ethics Case Study - Mike Cochrane .....................................................................................155
INTELLECTUAL PROPERTY- John O’Bannion ...............................................................156
Outlines from some Guest Speakers .................................................................................158
Project Plan Examples .....................................................................................................164
Progress Report Examples ...............................................................................................174
Design Review Outline Examples .......................................................................................183
Final Project Report Outlines Examples ............................................................................186
Part 1: EEE 193A Engineering Design

Preface

Who is this manual for?

Electrical Engineering Senior Project students in the first semester of their project.

Manual Goals and Objectives

- To provide dissemination of information on the project process.
- To help clarify some of the issues discussed in the course.
- To provide a reference document to refer to during the course.

Introduction to EEE 193A Senior Design Project.

Welcome to the Design Phase of your Senior Project. In this phase of your project you will get the opportunity to experience the designing your project as an engineer. This will encompass the following activities and responsibilities:

1. Responsible for reporting to your manager (your instructor).
2. Responsible for your time and how it is used to achieve your goals and project completion.
3. Project Plan for the semester
4. Topic Reports, written reports exploring topics covered in Lecture.
5. Project Status reports, oral reports
6. Design Reviews
7. Project Design Demonstrations
8. Final Project Report

Using a project notebook is highly recommended throughout your design and project process.
Course Description

EEE 193A Electronic Engineering Design:
This course concentrates on the planning and design of engineering devices, systems and necessary software. Emphasis is placed on design philosophies, problem definition, project planning and budgeting, written and oral communication skills, working with others as part of a team, development of specifications and effective utilization of available resources.

Prerequisites:
Successful completion of writing proficiency examination, General Education Oral Communication requirement, EEE118, EEE161, EEE174 and EEE180

Course Content:

Each topic will be discussed and resource materials will be distributed as needed. Homework relevant to your project will be assigned. Students will develop their projects during the semester and present them during oral design reviews. A written project report will be submitted at the end of the semester.

Course Objective:

This course introduces the fundamentals of electrical and electronic design. Areas addressed will include project planning and engineering design skills. Emphasis is placed on design, written and oral communication skills, development of specifications and effective utilization of available resources.

Your objectives in this course are:

1) To choose an appropriate project that meets the project requirements.
2) To design and document your project.
3) To communicate (in an oral and written manner) the project design.

Course Requirements:

Attendance is required for each class session. One missed session can be excused given valid reasons ahead of the absence. Two unexcused absences and the student will be dropped from the course.

General Notes:

• Students who do not complete the Final Project Report, do not pass the class regardless of other points and grades.
• Late assignments lose 50% of possible points per week late.
• Design reviews and final project reports will not be accepted past the due dates.

Evaluation:

Grades will be based upon:
20% Homework 20% Oral design reviews 20% Labs
10% Attendance 30% Written final project report
## EEE 193A Suggested Milestones

<table>
<thead>
<tr>
<th>No.</th>
<th>Activity Name</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
<th>14</th>
<th>15</th>
<th>16</th>
<th>17</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Form Team and Establish</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Select Project</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Preliminary Designs</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>Team Work Breakdown Structure and Operational Breakdown Structure</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>Initial Parts Acquisition</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>Test Options (Alternative) Designs</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>Design Analysis</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>Design Refinement</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>Final Project Report</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>Finalize Detail Project</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>Design Review</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>Material Acquisition</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>Design Verification Testing</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>System Integration &amp;</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>Final Project Report</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>Final Project Report Deadline</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Legend
- **Planned Task Dates**
- **Actual Task Dates**
- **Deadlines**

**Start** and **Finish** dates are indicated for each milestone.
193B Milestone Chart

EEE 193B  Suggested Milestones

<table>
<thead>
<tr>
<th>No.</th>
<th>Activity Name</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
<th>14</th>
<th>15</th>
<th>16</th>
<th>17</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>System BB &amp; test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Project review and Plan</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>PCB</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>Parts Acquisition</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>PCB Stuff &amp; Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>Design Reviews</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>Final Project Report</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>[Deadline to Turn in PCBs to Shop]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>System Integration &amp; Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>Project Demonstration</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>Final Project</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>[Final Project Report Deadline]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Legend**

- **Planned**: ○  △
- **Actual**: ○  △

Legend:

- ○: Planned
- △: Actual

Legend:

- ○: Planned
- △: Actual
Course Assignments

H1: Resume and Possible Projects List Assignment

1. Resume.
   Career Objective
   Education Experience
   Professional Experience (Work Experience)
   Project Experience
   Special interests (Hobbies)

2. Possible Projects List.
   List and describe three or more possible senior projects you are considering.
   Describe each project in 2 or 3 paragraphs.

H2: Library Project Research Report

1. Research three or more articles (references) related to your most likely project.

2. Summarize each of the reference articles and discuss the relevance to your project.

H3: Project Goals and Personal Goals Assignment

1. List Personal Goals.
   What do you expect to get out of the project?
   What do you plan to learn from it?
   How will you be challenged by the project?

2. List Project Goals.
   What is the project to do?
   What is it to accomplish?

H4: Functional Description and Concept Diagram.

1. Functional Description.
   What is this project?
   What does it do?
   How is it used?

2. Concept Diagram.
   Show what the device looks like and show it in operation in its intended operating environment.

H5: Specifications

1. List the number and types of inputs
2. List the number and types of outputs
3. Accuracy
4. Power Requirements
5. Operating Environment
6. Other project dependent Specifications (Project Features)
H6: Project Task Schedule Assignment

1. **Introduction.** Summarize Project.

2. **Task Names and Descriptions of tasks.** Identify tasks that will require more than 2 weeks and break them into sub-tasks of no greater than 2 weeks.

3. **Bar Chart Schedule,** Layout the schedule on a chart.

H7: Theory of Operation, Block Diagram, and Design Calculations

1. **Theory of Operation:** Narrative that explains the Block Diagram, Design Calculations, and the Theory of the device.

2. **Block Diagram:** A drawing equating Blocks to device functions and Lines to signal that flow to and from Blocks.

3. **Design Calculations:** Mathematical modeling of the project. Design Calculations focus on Equations that define the project.

H8: Design Review Outline

1. **Outline:** Review Design Review section. Write an outline to cover what will be presented in the Design Review.

H9: Schematics, Component Layout, and Package Layout

1. **Schematics:** Multiple drawings including a schematic organization diagram and project schematics.

2. **Component Layout:** Drawing(s) showing component position on circuit boards.

3. **Package Layout:** Drawing(s) showing panel layouts, circuit board position, and Major components in the package.

H10: Project Cost Analysis

1. **Material Costs:** Parts - Component cost direct associated with the project. Show Calculations!

2. **Resource Costs:** Indirect project costs ("Over Head"). Show Calculations!

3. **Labor Costs:** Sum Hours spent per Task times the Labor rate. Show Calculations!

4. **Total Cost:** Sum Material, Resource, and Labor Costs. Show Calculations!

H11: Test Procedures

1. **Test Flow Diagram:** A drawing showing tests to be run from component, module, sub-system, to system level tests.

2. **List and describe each test.**

3. **List Required Equipment for each Test.**

H12: Final Project Report Outline

1. **Outline:** Final Project Report requirements section. Write an outline to cover what will written in the Final Project Report.
L1: Power Supply Design Lab Assignment

1. Determine Power Supply Requirements.
   List and describe Assumptions make in determining Power Supply requirements.
   Show Calculations made in determining Power Supply requirements.

   Show Calculations in the steps taken in the design.
   Be able to answer the question, how and why did you choose that component?

3. Draw the Power Supply Design Schematic.

4. Verify your design with Circuit Simulation (PSpice)
   - Simulate the design under no load.
   - Simulate the design under fully loaded conditions (load determined by PS requirements).

5. Conclusions. Discuss how well your design will work (or why it won’t), and what you have learned from this design.

L2: Project Status Reports

1. Be prepared to discuss current status of the project.

L3: Design Verification Demonstration 1

1. Demonstrate functional Design to Lab Instructor

2. Write up a report on Design results:
   - Introduction: Covering what was done and why.
   - Discussion: What happened? Was it successful?
   - Project Impact: How will this information change or influence the project?
   - Schematic(s), Drawings, & Simulations
   - Conclusions: What was learned? What are the next steps for the project? Will this design be used, modified, or scraped, and why?

L4: Project Presentations

1. Present your project to the Lab class.
   - Show your Concept Diagram (use overheads).
   - Cover your Functional Description.
   - Discuss your Project Goals and your Personal Goals for the Project.
L5: Design Verification Demonstration 2
Choose an appropriate area or section of your project design and verify its operation and feasibility. Follow the same report procedure as Design Verification Demonstration 1.

L6: Design Verification Demonstration 3 (2 or more Modules)
Choose an appropriate area or section of your project design and verify its operation and feasibility. Follow the same report procedure as Design Verification Demonstration 1.

L7: Design Verification Demonstration 4 (3 or more Modules)
Choose an appropriate area or section of your project design and verify its operation and feasibility. At this point you should try combining design modules. Follow the same report procedure as Design Verification Demonstration 1.

L8: Design Reviews (see Design Review Section)

1. Present your project to the Lab class. Covering the following information:
   • Project Introduction, your name and the name of the project.
   • Project Goals and Personal Goals.
   • Functional Description and Concept Diagram.
   • Theory of Operation.
   • Block Diagram.
   • Major Design Calculation(s).
   • Discussion circuitry used (schematic).
   • Project Status (show chart and discuss).
   • Solutions to Problems encountered.
   • Current problems or concerns.
   • Questions from the class.

L9: Design Verification Demonstration 5 (3 or more Modules)
Choose appropriate areas or sections (modules) of your project design and verify their operation and feasibility. Combine modules as much as possible. This will help for subsystem and system level testing. Follow the same report procedure as Design Verification Demonstration 1.

L10: Design Verification Demonstration 6 (System Verification)
Combine as many modules as possible to demonstrate system functionality and feasibility. Follow the same report procedure as Design Verification Demonstration 1.
**Design Reviews**

**PURPOSE:**
- To present your project and receive feedback on it.
- To review the projects of your peers and provide feedback to them.

In order to meet these objectives your attendance and participation in all design review sessions, yours and others, are mandatory.

**CONTENT:** Design Reviews should include:
- Project Introduction, your name and the name of the project.
- Project Goals and Personal Goals.
- Functional Description and Concept Diagram.
- Theory of Operation.
- Block Diagram.
- Major Design Calculation(s).
- Discussion circuitry used (schematic).
- Project Status (show chart and discuss).
- Solutions to Problems encountered.
- Current problems or concerns.
- Questions from the class.

**Design Review Outlines:** should include the items listed above in the Content section expanded to cover your project issues.

**EVALUATION CRITERIA:**
Design Reviews will be evaluated on the completeness of addressing the above issues and the following issues:
- Understanding of the theory behind the project.
- Verbal and visual presentation of the project.
- Discussion of potential problems and the solutions.
- The completeness and accuracy of the Design Calculations, Schematics, and Block Diagrams.
- Level of Project Planning and Project Completion.

As a project reviewer you would use the comment sheet shown below and you should provide feedback to the presenter in the following areas:
- Did the presenter give you a clear understanding of the project?
- What problems do you see with the project? Can you offer any suggestions.
- What would improve the speaker's presentation?
- Other comments.
Final Project Report Requirements

The final report will serve as complete documentation of your proposed senior project. It must be complete in each section and reflect all aspects of your project. Do not just put your homework assignments together to form your report. Those were preliminary sections, the final report must address all details regarding your project.

Reports must be word processed, with the exception given to block diagrams, schematics, and design calculations.

The final report is due during the lecture period of the week before finals and will not be accepted late.

The report must adhere to the following format and will be evaluated according to the points listed for each section.

1. **Title page** (5 points)
   - Project name.
   - Student's Name (and Partners).
   - Course Number & Section & Instructor's Name.
   - Date.

2. **Table of contents** (5 points)
   - List of Sections and Figures with page numbers.

3. **Project description** (10 points)
   - Executive Summary (Project summary & Resume).
   - Project Goals and Personal Goals...Team Work Breakdown.
   - Concept diagram (showing how the device is used).
   - Functional Description.
     (What is the project? Where is it intended to be used? What are it's features?)

4. **Project specifications** (5 points)
   - List of Quantitative Project Requirements.

5. **Operator's manual** (10 points)
   - Introduction (What is this device and what does it do?).
   - Project Diagram (a "Picture" of the project) showing inputs, outputs, and controls.
   - List and describe inputs and outputs, controls, and their functions.
   - Operating Instructions, Installation instructions, and Troubleshooting instructions.

6. **Project Technical Detail** (Maintenance manual) (30 points)
   - Block Diagram, Software Diagram (if applicable).
   - Theory of Operation (including design discussion), simulations, and Design Calculations.
   - Schematic Organization Diagram & Schematics.
   - Parts Lists, and Wire List (list of connections).
   - Circuit Board Component Layout, and Package Layout.
   - Calibration section (How is the device to be calibrated to stay within specs?).

7. **Test Section** (10 points)
   - Test Flow Diagram.
   - Test Procedures. To include: Test Descriptions. Equipment required.
   - Configuration Diagrams (showing equipment setup for the tests).

8. **Schedule and Project Cost Analysis** (15 points)
   - Tasks and Descriptions.
   - Timeline (Bar chart) Diagram.
   - Project Costs (Material Costs, Resource Costs, Labor Costs, and Total Cost).

9. **Project Review** (5 points)
   - Critical Tasks.
   - Current Project Status.
   - Situations Encountered (problems / solutions).
   - What has been learned from this project and process.

10. **Reference and Appendix Sections** (5 points)
    - People, Books, Articles, Data Books, Actual Data Sheets, and any additional information.

**NOTE:** The final project report will constitute 30% of the total course grade.
Background information on Project planning:

Project Planning Steps

1. Review Project requirements
2. Identify and define the end of the project. (Specs)
3. Review Block Diagram, Break into sections based on Block Diagram.
4. Identify the steps required to complete the Project.
5. Visualize the finished project and work backwards.
6. Use 193A Schedule and Suggested Milestones to help identify steps. Suggested Milestones, see Course Schedule and Chart
7. Compare each step against the Project Requirements to verify necessary steps.
8. Break the steps into smaller steps, until you have steps (tasks) of no greater than 2 weeks.
9. Review each of the tasks and look for ways to overlap or to complete more than one task.
10. Layout a Time Line Chart (Gantt Chart).

Time Line Chart (Gantt Chart):

Tasks on the Y axis, starting with the first task on top to the last task on bottom. Time line on the X axis, in week numbers and dates of each week for the project duration. Refer to the Time Chart Elements diagram. For a completed Time Chart refer to the Chart of Suggested Milestones and the examples.

Time Line Chart Elements
Typical Problems with Project Plans.

1. Having tasks greater than one week in duration (Tasks ≤ 2 week).
2. Missing Project name and Project Engineer Name.
3. Insufficient number of tasks, 30 weeks at 2 weeks per task = 15 tasks right? wrong! ~ 30 to 40 tasks total, 2 ~ 4 tasks in parallel per 2 weeks.
5. Not Starting PCB layout before finishing Bread Boarding.
7. Taking a vacation while the shop is working on the PCBs.
8. Unrealistic amount of work in a short time, for example all software to be coded in one week.
10. Not using descriptive task names and tasks descriptions.
11. Don’t use: Task 1, Task 2, or CKT1, CKT2, or Test 1, Test 2, etc. Use task names like: Osc. BB, RS232 BB, Power Amp. PCB, Controller PCB, etc.
12. Don’t combine task name and description to form a long name or a short description. Separate Task names and Task Descriptions.

The more foresight and planning put into the project now will yield greater success of the project completion.

Reference Sources for Project Planning

Books (see Design Reference section): Planning Big with Mac Project. What every Engineer should know about Project Management. Managing the Engineering and Construction of Small Projects.

Software: Microsoft Project, Harvard Project, Primavera
Mac Project Trial Kit $25, Video, Manual, and SW - 1- 800 628-2100
Mac Schedule, Fast Track Schedule
Drawing SW:
Auto CAD, Auto Sketch, Generic CAD, MacDraw, Canvas, MacDraft,
PCB Guidelines & Instructor Approval

Name: _______________________________________________________________

Project Title: __________________________________________________________

Tech Shop - Initial/Date: _____________________ / __________________________

Instructor’s Signature/Date: ___________________ / __________________________

CHECKLIST

______ Be sure to mark your PCB with your NAME.

______ Indicate BOTH Solder AND Component side

______ Mark the corners of the board as shown in the example.

______ Indicate location of pin #1 when needed.

______ Scaling on either horiz. OR vert. axis, as in example.

______ Finished board size NOT TO EXCEED 7 1/4” X 9 1/4”.

______ Artwork must be solid, opaque black, and on mylar film.

______ Traces .020” wide.

** NOTE: All artwork must be approved and turned in NO LATER THAN 4 P.M. EVERY THURSDAY.

** DON’T WASTE SPACE, AND KEEP YOUR CIRCUITS AS COMPACT AS POSSIBLE. INEFFICIENT LAYOUTS WILL BE RETURNED TO THE STUDENT TO BE DONE OVER.

** DON’T WAIT UNTIL THE LAST MINUTE TO SUBMIT ARTWORK, THIS WILL INCREASE THE TIME REQUIRED TO FINISH YOUR PROJECT.

_____________________ FOR SHOP USE __________________________

FLAT RATE = 6.00
DS BOARDS ________ SQ. IN. X .06 = _______________
SS BOARDS ________ SQ. IN. X .06 = _______________
FILM _________ SH. X 1.50 = _______________

+ __________________________

TOTAL

REVISED 1/92
1. NAME

Course No. and Sec.
Rev Date

2. COMPONENT
OR
SOLDER SIDE

3. CORNER'S MARKS
(WE CUT BOARD TO
THESE MARKS)

4. SCALE ON X
OR Y AXIS.
Design References

Books:


**CMOS Cookbook**, Lancaster D., Howard Sams, 1977


**Electronics Engineers' Handbook**, Fink and Christiansen, 2nd edition, McGraw-Hill, 1982


**Getting Started As A Consulting Engineer**, Sunar, Professional Publications, 1986


**Handbook of Electronics calculations**, Kaufman and Seidman, McGraw-Hill, 1979


**How to Build and Use Electronic Devices Without Frustration, Panic, Mountains of Money or an Engineering Degree**, Hoeing, Stuart A., 2nd edition, Little Brown 1980


**IC Timer Cookbook**, Jung, W. G., Howard Sams, 1983


**Interfacing Microcomputers to the Real World**, Sargent and Shoemaker, Addison-Wesley, 1981

**Managing the Engineering and Construction of Small Projects**, Westney, Marcel Dekker, 1985


New Dress for Success, Molloy, Warner Books, 1988
Operational Amplifiers, Tobey, et. al., McGraw-Hill, 1971
Planning Big with Mac Project, Halcomb, McGraw-Hill, 1986
Power Supplies, Shepard, Reston (Prentice-Hall), 1984
Sweaty Palms, The Neglected Art of Being Interviewed, Medley, Ten Speed Press, 1984
Swim with the Sharks, MacKay, Ballantine Books, 1988
Switching Regulators and Power Supplies, Gottlieb, Tab (McGraw-Hill), 1976
The Design and Drafting of Printed Circuits, Lindsey, Darryl, McGraw-Hill, 1982
TTL Cookbook, Lancaster D., Howard Sams, 1974
What Every Engineer Should Know About Project Management, Ruskin and Estes, Marcel Dekker, 1982

PERIODICALS:

Byte
P.O.Box 567
Martinsville, NJ 08836-9956

Electronic Design News (EDN)
Computer Center
P.O. Box 5262
Denver, CO 80217

Electronics Magazine
McGraw-Hill, Inc.
1221 Avenue of the Americas
New York, NY 10020

Electronics Systems Design (ESD)
P.O. Box 8
Winchester, MA 01890

Machine Design
Penton Publishing Inc.
1100 Superior Ave.
Cleveland, OH 44114

NASA Tech Briefs
41E. 42nd Street, Suite 921
New York, NY 10017-5391

Computer Design
P.O. Box 2389
Tulsa, Oklahoma 74101

Electronics Design
Hayden Publishing Co., Inc.
P.O. Box 1497
Riverton, NJ 08077-7097

Electronics Products
645 Stewart Ave
Garden City, NY 11530
(516) 227-1300

INK
4 Park Street, Suite 20
Vernon, CT 06066
(203) 875-2199

Modern Electronics
Ziff-Davis Publishing Co.
1 Park Avenue
New York, NY 10016

Radio Electronics
Box 55115
Boulder, CO 80321-5115
Electronic Component Information

Electronic Engineers Master Catalog (EEM)  
Electronic Buyers Guide (EBG)  
Electronic Design's Gold Book  
IC Master

Electronic Suppliers:

Local Vendors

Chuck Hurley's Electronics .......................................................... 2557 Albatross Way ................................................................. 927-5891
Hamilton-Avnet Electronics .......................................................... 4103 Northgate Blvd ................................................................. 920-3150
Heathkit Electronics ................................................................. 1860 Fulton Avenue ................................................................. 486-1575
HSC Electronic Supply ................................................................. 338-2545
Mar Vacs ................................................................. 2537 Del Paso Blvd ................................................................. 922-6531
Metro Electronics ................................................................. 1831 J st ................................................................. 922-6531
Newark Electronics ................................................................. 721-1633
Popkey Distributing ................................................................. 2110 - 5th ................................................................. 448-8848
Radio Shack ................................................................. (check Phone Book for nearest store)
Sacramento Electronic Supply Co. ..................................................................... 1219 S Street ................................................................. 441-4821
Valley Electronics ................................................................. 4115 Franklin Blvd ................................................................. 451-1186
Zack Electronics ................................................................. 10360 Rockingham ................................................................. 369-1786

Mail Order Vendors

Digi-Key .................................................................................................................. 1-(800) 344-4539
JAMECO ........................................................................................................... (415) 592-8097
Mouser Electronics .................................................................................... 11433 Woodside Ave, Santee, CA 92071 ........................................ (619) 449-2222

Manufacturers

Motorola ........................................................................................................... 1-(800) 521-6274
University Support ......................................................................................... (602) 952-3855

Professional Societies:

Institute of Electrical and Electronic Engineers, IEEE
National Society of Professional Engineers, NSPE
California Society of Professional Engineers, CSPE
Association for Computing Machinery, ACM
American Society of Electrical Engineers, ASEE
PSpice on the VAX

VAX PSpice Professional version is now operational. This version will do both analog and digital simulation and includes: Behavioral Modeling, Digital Simulation, Monte Carlo Analysis, Probe, and Parts programs.

PSpice 4.05 is now operational on the Titan system. To access it you must first SETUP_PSPICE and then SETUPDEV. SETUP_PSPICE activates PSpice, you can put this command in your com file. SETUPDEV defines your input and output devices to PSpice.

I suggest you type in the following commands to activate PSpice:
- Sign on to Titan.
- Type in SETUP_PSPICE (return).
- Type in SETUPDEV (return).
- Select 1 (return). This specifies a non graphics terminal.
- Select 2 (return). PSpice will print to a file.
- Select 1 (return). PSpice will print to a file in text format.
- Select 4 (return). This creates and saves the SETUPDEV file.
- Select 0 (return). This exits from the SETUPDEV program.

You can now run PSpice in a similar manner on the VAX as you do on the MSDOS systems. Please see me if you require further assistance.

VAX PSpice SETUPDEV

TITAN -> SETUPDEV

SETUPDEV - PSPICE Device file creation program. Version 4.05 - January 1991

The PSPICE system requires a device file in order to function properly. Since one was not found in either your current directory or your path, this program will create one in your current directory. It will be called pspice.dev.

A device file is required to contain a display specification. Below is a list of all displays supported for your computer.

1. TEXT            2. VT125           3. VT240           4. VT241
9. GPX            10. XWIN

Select a display type> 1

It is not mandatory to have a hard-copy specification in the device file, but without it, the PSPICE system cannot produce any hard-copy output. The PSPICE system will support up to ten hard-copy devices. You may add a hard-copy specification here, or you may abort this section at any time.

A hard-copy specification consists of a port name and a printer type. A port name is where the PSPICE system will route the hard-copy data. The name should be something the operating system will recognize, such as a filename or a serial port with a printer. A special case is "FILE" which means the PSPICE system will prompt you for a file name before it writes any data. Below is a list of possible ports for your computer.
0. Abort
1. LPA0:
2. FILE
3. Filename

Select a port type> 2

The printer name is the PSPICE defined name specifying the type of printer that the hard copy data is intended for. Generally the name will correspond with the manufacturer's name for the printer. Below is a list of supported printers. For more detailed information on the printer types, see the "Device Specification" section in the "Probe" chapter of the PSPICE manual.

0. Abort
1. TEXT            2. TEXT132         3. IBMGraph        4. Okidata
13. IBMC1r         14. IBMC1r132       15. IBMC1rSlw       16. IBMC1rSlw132
17. CItoh          18. CItoh132        19. CItohSlw        20. CItohSlw132
25. DECLA50        26. DECLA100        27. DECLA100-8      28. DEC75
29. DEC100         30. DEC150          31. TOSHIBA         32. TOSHIBA132
33. NEC            34. NEC132          35. NECCIr          36. NECCIr132
37. HI             38. HP              39. HP6             40. HP8
41. HPLJ           42. HPLJ100         43. HPLJ150         44. HPLJ300
45. HPQJ           46. HPQJ192         47. MLJ             48. MLJ100
49. MLJ150         50. MLJ300          51. HPPJ90          52. HPPJ180
53. HPPJ_T90       54. HPPJ_T180      55. PS              56. MPS

Select a printer type> 1

SETUPDEV - PSPICE Device file creation program. Version 4.05 - January 1991
Device file : pspice.dev
---------------------------
Display = TEXT
Hard-Copy = FILE,TEXT

0. Exit
1. Change Display
2. Add Hard-Copy Device
3. Delete Hard-Copy Device
4. Save Device File

Selection> 4

SETUPDEV - PSPICE Device file creation program. Version 4.05 - January 1991
Device file : pspice.dev
---------------------------
Display = TEXT
Hard-Copy = FILE,TEXT

Creating device file "pspice.dev"
Saved.

0. Exit
1. Change Display
2. Add Hard-Copy Device
3. Delete Hard-Copy Device
4. Save Device File

Selection> 0
VAX PSPICE EXAMPLE RUN

TITAN ->TYPE FWAVE.CIR
* Fullwave Rectifier Circuit
VIN1 1 3 SIN(0 25V 60Hz)
D1 1 2 DOD
D2 0 1 DOD
D3 0 3 DOD
D4 3 2 DOD
CF 2 0 10000uF
RL 2 0 10
.MODEL DOD D
.TRAN 1MS 100MS 50MS .5MS
.PROBE
.end

TITAN ->PSPICE FWAVE.CIR
Simulating circuit: * Fullwave Rectifier Circuit
In file FWAVE.CIR Writing results to FWAVE.OUT

Transient Analysis
Transient Analysis finished

Time step = 281.3E-06 Time = .1 End = .1

TITAN ->DIR FWAVE.*
FWAVE.CIR;1 FWAVE.JOU;1 FWAVE.OUT;1
Total of 3 files.

TITAN ->PROBE
Checking data file...

Loading data file...

Probe
Graphics Post-Processor for PSpice
Version 4.05 - January 1991
(C) Copyright 1985 - 1991 by MicroSim Corporation
Unauthorized copying of this program is prohibited

Circuit: * Fullwave Rectifier Circuit
Date/Time run: 09/13/91 22:45:03 Temperature: 27.0

TITAN ->BYE
VAX PROBE OUTPUT

Exit  Add_trace  Remove_trace  X_axis  Y_axis  Plot_control  Display_control
Macros  Hard_copy  Cursor

Loading data file...

Exit  Add_trace  Remove_trace  X_axis  Y_axis  Plot_control  Display_control
Macros  Hard_copy  Cursor
A Blow-by-Blow introduction to OrCAD

These notes refer to the AT&T computers in Room 3001. We will create a very simple voltage divider using two resistors.

Turn on the computer. After it boots up, a screen offers a choice of programs. Hit any key. Then type SDT <

[Important note: in this memo, < means to hit the <--Enter key after you type something, or in response to a screen or menu command of some other sort.]

Prompt is C:\APP\ORCAD\SDTIII> Respond by typing DRAFT <
a red OrCAD logo appears. Hit any key. A copyright notice appears. Hit any key.

load file? appears. The schematic design technique III (SDTIII) program hasn't been started yet, so there is no file to load. Hit ESC. Then hit < With arrow keys move highlighted area to "Get" and <

Screen says "Get?" < A menu asks "Which library?". With arrow keys move to DEVICE.LIB. A menu appears starting with "12 Header". Use up-down arrow keys to highlight RESISTOR < A picture of a resistor appears with the label R? RESISTOR and a white box around it. Don't worry about the ? after the R. We will deal with this matter much later in these notes. Several choices are listed at the top of the screen. Hit P (for "Place"). Resistor turns blue. Hit ESC. Resistor turns red; label turns white; arrow appears on screen.

You need a second resistor. Hit < Menu appears. With up-down arrows, select "get" < You already know RESISTOR is an available choice, so just type RESISTOR < A resistor appears on top of the first one, so you must move the one you just added. Use the up-down-left-right arrows. In this example, the new resistor is placed directly below the first one using the down arrow and leaving a gap of two or three dots in the grid on the screen. Now hit P then ESC.

Now add the power and ground connections. To get the power (VCC) terminal, hit < A menu appears including the word "Place". Note we do not use "Get" for finding the power connection. Hit P for "Place" or use the down arrow to highlight "Place" and then hit < By either method a menu appears starting with "Wire". Scroll down the list to find "Power". Hit letter P for "Place". A circle marked VCC appears. Use the up-down-left-right arrows to move the symbol above the upper resistor leaving a gap of a few grid spaces. Hit P for place, then ESC.

Now get ground. Hit < then G (for "Get"). Get? shows on screen. Hit < A library menu appears. Scroll down to DEVICE.LIB < A list of items in the library appears. Scroll down and eventually find several items starting with GND. For no special reason, we will use GND POWER. With the up-down arrows, scroll to highlight GND POWER < The GND POWER symbol appears more-or-less on top of the VCC symbol. Use the arrows to move the ground symbol below the bottom resistor, leaving a gap of a few grid squares. Hit P for place, then ESC.

Next we must add the connecting wires. Hit < The menu that appears includes the option "Place". Hit the letter P. The word WIRE is highlighted. Hit < Move the cursor to the bottom of the VCC symbol. Hit the letter B (for "Begin"). With the down arrow move the cursor to join the bottom of the VCC symbol to the top of the upper resistor. A white line is drawn. Hit E (for "End"). The line turns red. Now move the cursor to the bottom of the upper resistor. < then P. "Wire" is highlighted again; this is our choice, so hit < then B. With the down arrow, join the bottom of the upper resistor to the top of the lower one. Hit E (for "End"). Using exactly the same keystroke sequences, join the bottom of the lower resistor to the GND POWER symbol.

Looking ahead to the final PCB layout, remember you introduced two modules: the two resistors, and each one is a separate "module" in the PCB program's terminology.

Now we enter an editing and clean-up phase. First we must save the circuit in a schematic drawing file. Such files should always use the extension .SCH We will call this file VOLTDIV.SCH because it is a voltage divider.
Hit the letter Q (for "Quit"). The Quit menu appears. We are entering a new file, so scroll to "Write to File". Then < Message at top of screen says "Write to File?". Type VOLTDIV.SCH < Menu reappears. This is a good time for coffee or martinis, so with down arrow highlight "Abandon Edits" and hit <. The prompt C:\APP\ORCAD\SDTIII> appears. For now, we could go into the C:\APP\ORCAD\PCB> directory, but let's turn off the computer.

Restarting: The VOLTDIV.SCH file is now on drive C:. It will be useful to have copies of two documents in a different directory, so on power-up select PCB instead of SDT. This directory has lists of the "style" of different parts, i.e., their mechanical sizes and names of parts with these sizes. You must include the "style" in the .SCH file so when the etched board is created, the PCB program knows what the configuration of your modules will be. To get copies of these documents, do the following: type PRINT LIB.DOC then < If this is the first printing after power-up, the computer will ask for the name of the printing device. Type LPT1 then < Now enter the instruction PRINT JEDEC.DOC and then < You already told the computer to print on LPT1, so the question of which printer will not be asked again.

These lists will help you identify the "style" to enter into your VOLTDIV.SCH program. We assume the resistors are physically small, so RC07 (allows 0.5 inch spacing) is a good choice.

[Remark: the Sac State library has a couple books that will help you get ideas on physical spacing of leads for components. TK 7870 H4 Henney "Electronic Components Handbook" and TK 7870 Harper "Handbook of Electronic Engineering" are useful. Dr. Partridge has asked the library to buy a copy of a JEDEC (Joint Electron Device Engineering Council) standard on the mechanical sizes of electronic components.

Go back now to the SDTIII program. Type

```
CD C:\APP\ORCAD\SDTIII
```

and hit <. Now type DRAFT/C A menu appears. At "Command?" type KF (DON'T HIT <) For Selection prompt, type 12 < This takes you to NETLIST Module Value Combine. The Edit command [We will get to this later] allows you to enter the "Style", such as RC07, in your choice of one of eight fields. For NETLIST Module Value Combine, we will plan to use Field #1 in the edit menu later, so type 1 at the blinking cursor opposite line 12, then hit < At the selection prompt, type Q (for "Quit"). The main configuration menu comes back. At the Command? prompt, type Q (for "Quit") and then Y at Update Configuration prompt. This takes you back to the SDTIII> prompt.

Now get back to your VOLTDIV.SCH file. At the SDTIII> prompt, type DRAFT then < Hit any key to get rid of OrCAD logo and any key again to get rid of copyright notice. "load file? appears at top of screen. Type VOLTDIV.SCH then <

Move cursor to make it touch the upper resistor. < A menu appears. Select EDIT. A menu appears horizontally at top of screen. Type E (for "Edit"). Use down arrow to highlight Part Value. Hit < A small menu entitled "Value" appears with "Name" highlighted. You want to "name" the resistor value, so hit < The words at the top of the screen become "Value?RESISTOR". Type a space and then the value. Let's use 10K, so after the space type 10K and then < The value of the resistor is shown after the word RESISTOR on the schematic. With the down arrow, move the highlighted area of "1st Part Field" then < "Name" is highlighted. You reserved this 1st field to tell the "style" of the part [or module], so "Name" is what you want to insert. Therefore, hit < In response to "1st Part Field?" type RC07 then < Next hit the ESC key. Now move the cursor to the lower resistor. Repeat the same sequence, and just for example assign 820 to the value, but the style is RC07 same as before. After you type RC07, hit <. Then hit ESC key twice. Now the information for the schematic is all there except the part numbers. These will be assigned automatically, thus taking care of the question mark displayed at R?.

Hit < Move the highlighted area to Quit and hit <. Move highlighted area to Update File and hit < Everything you can do just now is done, so move the highlighted area to Abandon Edits and hit <

Now use a routine to assign numbers to the components. The command at the SDTIII prompt is ANNOTATE VOLTDIV.SCH /M /O. Type in this command and hit <
Just to see what happened, reload your VOLTDIV.SCH file using the DRAFT command. You will see the resistors are numbered. The one you created first (you chose to put it on top but it was the first wherever you put it) is R1 and the next is R2. Your curiosity is satisfied, so leave the program with the Abandon Edits routine.

Now the NETLIST must be created. This is used in the PCB program to tell PCB what connects to what. [This important command is buried on Page 25 of the OrCAD/PCB II Version 2.02 Addendum.] At the SDTIII prompt, type

```
NETLIST VOLTDIV.SCH VOLTDIV.NET ORCADPCB /S
```

and then hit <

Copy the VOLTDIV.NET file from the SDTIII directory to the PCB directory for subsequent use:

```
COPY \APP\ORCAD\SDTIII\VOLTDIV.NET \APP\ORCAD\PCB\NET\VOLTDIV.NET <
```

[On some computers, the command is `\APP\ORCAD\PCB\NETLIST\VOLTDIV.NET <`]

Now go to the PCB directory:

```
CD \APP\ORCAD\PCB <
```

Refer to Page 41 in the OrCAD PCB II User's Guide. We must create a box which contains the layout of the voltage divider network. At the PCB> prompt, key in PCB then < The OrCAD logo appears. Hit any key. The question "Load Board?" appears. Hit < twice. This brings up the main menu. Select "Quit" [yes, this is correct, quit is the choice!] then hit < Another menu appears. Move highlight to "Initialize". Hit < The menu has "Use Netlist" as a choice, and we will soon use the Netlist VOLTDIV.NET, so hit < We are dealing with just two resistors, so 2 inches wide and 3 inches high should be ample room for the finished etched board. Put the cursor at X = 2, Y = 1 using the up-down-left-right arrows. Hit B for Begin. Move the cursor to X = 4, Y = 4, making 2 inches wide and 3 inches high. A white box is created on the screen, though not all of it shows at once. Now hit E (for "End").

You are asked "Read Net File?" Answer by typing VOLTDIV.NET and hit <

Move the cursor to a suitable place for R1, say at coordinates X= 2.5, Y = 2. Hit P (for "Place"). "Module" appears highlighted. R1 is a module, so press < Move highlighted area to "Get" and press < twice. Question "Get?" appears. Type R1 and hit <. The resistor appears sideways. A vertical placement would be better, so hit R (for "Rotate"). Then hit P (for "Place"), then ESC. Choose "Get" from list at top of screen by hitting G (for "Get") twice. Answer "Get?" by typing R2 and < This resistor appears sideways also, so hit R (for "Rotate"). R2 will appear more-or-less on top of R1, so use the up-down-left-right arrows to move it below R1, say at X = 2.5, Y = 3. Hit P (for "Place"). Then hit ESC until main menu appears (the menu starting with the word "Again"). Select "Routing" and hit < Select Auto by hitting the A key. An Auto Route menu appears. Move highlight to "All" and hit <. A connecting track appears joining R1 and R2. Hit ESC twice.

On main menu, select Quit. Choose option "Update". The layout has just been created, so program asks "Save Board?" Type in a name for the board's file. We will call it VOLTDIV.BRD. Now we leave OrCAD by the "Abandon Program" instruction.

Note there are several items missing: namely the pads to connect resistor R1 to VCC and R2 to GND POWER. Also you need a pad between R1 and R2 to get the output from the voltage divider. Some editing must be done to create these pads.

The above material should get you started, but the fine details must be learned from the OrCAD manuals.

To print, type `Plotall filename.sch /p <`
OrCAD PCB Notes

For

EEE 193A - Engineering Design

Lab Instructor: Prof. Dahlquist
Roadmap to Successfully Plotting OrCAD PCB Layouts

These notes assume that you have already read "A Blow-by-Blow Introduction to OrCAD". As with this previous document, the following refers to the AT&T computers in ECS 3001.

In the OrCAD Draft program, you must assign a style to each of the components in your circuit. See "A Blow-by-Blow..." for instructions on how to do this. Next, a netlist must be created. The netlist tells OrCAD PCB what components connect to what pads. A netlist is generated by entering at the SDTIII prompt:

```
NETLIST <schematic file name> <net file name> ORCADPCB /S
```

where <schematic file name> is the name of your schematic file and <net file name> is the name of the new net file which is going to be created by NETLIST.

The net file is the file that will be loaded into OrCAD PCB. If you want to perform auto-routing, you will have to boot the AT&T computer off of your own boot disk. Otherwise, a bunch of drivers get loaded into memory upon boot-up, and there is not enough memory left for auto-routing to be performed. To make a boot disk, refer to the procedures at the bottom of the page.

The next step involves configuring PCB to plot your layout. Before beginning this step, however, copy the new net file to the PCB directory and then change to the PCB directory. The file can be copied by entering:

```
COPY <net file name> \APP\ORCAD\PCB
```

where <net file name> is the name of the net file created by NETLIST. Now change to the PCB directory by entering PCB at the DOS prompt or by entering:

```
CD \APP\ORCAD\PCB
```

Now you should be ready to enter PCB. First, however, you are going to change the program's configuration to make sure that your layout will plot correctly.

Making a Boot Disk

To make a boot disk, initialize a new disk with the /S option. For example, assuming that the disk to be formatted is in drive A:, enter at the DOS prompt:

```
FORMAT A: /s
```

This will format the disk and install the system files (hidden from normal viewing).

It is also possible to install system information on a disk that already has files on it. This can be done with the SYS command. For example, assuming that the disk to be made bootable is in drive A:, enter at the DOS prompt:

```
SYS A:
```

This will transfer the system files to the appropriate place on the disk.
At the DOS prompt, enter:

```
PCB /C
```

This will bring up the configuration menu for PCB. The configuration needs to be changed for plotting because OrCAD considers the lower-left corner of the sheet to be at coordinates (0,0), while the plotter considers the middle of the sheet to be at coordinates (0,0). The configuration set-up which I am about to describe is for plotting on an 8.5” x 11” sheet of paper in landscape mode (page is wider than it is long). To change the configuration, the following sequence of commands should be entered, where <CR> signifies pressing the return, or enter, key on the keyboard and menu selections are signified as (Menu choice).

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Command?</td>
<td>PC&lt;CR&gt;</td>
<td>Select the pen carousel set-up menu.</td>
</tr>
<tr>
<td>Command-&gt;</td>
<td>X&lt;CR&gt;</td>
<td>Change X paper offset.</td>
</tr>
<tr>
<td>X Paper Offset -&gt;</td>
<td>-5.5&lt;CR&gt;</td>
<td>Set to 1/2 of 11 inches for landscape plotting.</td>
</tr>
<tr>
<td>Y Paper Offset -&gt;</td>
<td>-4.25&lt;CR&gt;</td>
<td>Set to 1/2 of 8.5 inches for landscape plotting.</td>
</tr>
<tr>
<td>Command-&gt;</td>
<td>Q&lt;CR&gt;</td>
<td>Quit the pen carousel set-up menu.</td>
</tr>
</tbody>
</table>

At this point, you can quit the configuration menu by either quitting PCB altogether, or by running the layout program. To run the program, enter R<CR>. This has the same result as if you had just entered PCB<CR> from the DOS prompt. The next step is to load a net file.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Opening screen)</td>
<td>&lt;CR&gt;</td>
<td>Clear OrCAD's opening screen.</td>
</tr>
<tr>
<td>Load Board?</td>
<td>&lt;CR&gt;&lt;CR&gt;</td>
<td>Don't load a board, we want a net file.</td>
</tr>
<tr>
<td>(Quit)</td>
<td></td>
<td>Choose to quit from the menu.</td>
</tr>
<tr>
<td>(Initialize)</td>
<td></td>
<td>Choose to initialize the system.</td>
</tr>
<tr>
<td>(Use Netlist)</td>
<td></td>
<td>We are going to load a net file. This choice results in the program asking you to select an area to place the net file components. Use the mouse to delineate an area on the screen.</td>
</tr>
<tr>
<td>Read Net File?</td>
<td>filename&lt;CR&gt;</td>
<td>Enter the name of the net file.</td>
</tr>
</tbody>
</table>

This will result in a rat's nest of components being placed on your screen in the area you selected. You will have to rearrange these according to your board layout design. Be sure and leave plenty of space between your layout and the edges of the page. When you have placed the components where you want them, you are ready to auto-route.
At this point, you may choose (All) or (Optimize All) to perform the routing. After selecting one of these options, sit back and enjoy the show. If there is some particular trace layout that you don't like, you can always change it individually. I will leave it as an exercise for the reader to figure out how to do that.

Assuming that the board layout is complete, you are now ready to plot. The plotting process consists of writing a plot file to the disk and then taking that file to one of the systems that is connected to the plotter. The following instructions assume that you are in OrCAD PCB with your board layout loaded.

That should do it. Don't forget to save any changes to your PCB layout file by choosing (Update File) from the Quit menu. You can now copy the plot file to a disk and take it to one of the machines connected to a plotter. From there, set up the plotter, and enter

COPY <plot filename> COM1

This should get you a plot of your layout. If it didn't work correctly, then check to see that the PCB configuration is correct. If it is, then make sure there is plenty of free space around your board layout in PCB, go through the above steps again, and then re-plot it to disk.
ADDENDUM TO THE ORCAD-PCB MANUAL
by Del McCracken

This pamphlet is meant to serve as a help in designing PC-Boards using the OrCAD-PCB software package. It is not meant to be a substitute for the OrCAD-PCB Manual, but rather a supplement. There are a few items that are very vague, or even missing in the manual that I will attempt to address here. I hope that this pamphlet will be of some help to you.

Getting Started:

There are three steps involved in designing a PC-board using the OrCAD system:

The first step is to draw out the circuit using OrCAD SDTIII. We will assume that you are already comfortable using SDTIII, and that you have already drawn your circuit using it.

The next step will be to output a netlist from SDT to PCB. This involves labeling the components with their appropriate modules, outputting a netlist, then editing this netlist.

The last step in this process will be to import the netlist to PCB, lay out the board, rout the board, then plot the board.

This pamphlet will deal with the last two of these steps, while focusing on the second step.

Obtaining the Netlist

Labeling Components:

There are two different ways of labeling different component packages:

The first method is to manually go into SDTIII and edit each individual component on your schematic. You simply enter the module name in field 1. This is probably the easiest if your schematic is not too large, and doesn't contain too many of the same component.

If your circuit is large, and contains the same components in multiple places then you will probably want to use the Fieldstuff program. In order to use the Fieldstuff, you must first create a stuff file. A stuff file consists of two columns. The first column contains the part value, and the second column contains the module name e.g.:

'LM741'  '8dip300'

The two columns are separated by a tab. Stuff files have the DOS extension "stf". Several examples of these stuff files are present in the SDTIII directory.

In order to use fieldstuff, you need only to type:

`fldstuff 1 schematic stufffile`

where `schematic` is the name of your schematic, and `stufffile` is the name of the stuff file that you created. Now you need to make sure all your components have their field 1 set to the desired module.

Configuration:

The next thing we need to do is to configure SDT so that the Netlist program knows where to look for the module name:

Go to the SDTIII configuration menu (draft /c). Now go to Key Field Configuration. We want to go to item 12 (Netlist Module Value Combine). Set this equal to 1.

We are now ready to make our netlist. At the DOS prompt type:

`netlist schematic netfile orcadpcb /s`

Where again `schematic` is your schematic file, and `netfile` is the desired name of your netlist.
You now have your netlist, however you're not quite finished. You need to import this netlist into a line editor such as PE, EDLIN or WP if you use the textfile option. You now need to carefully inspect it to make sure it accurately represents your circuit.

Here are a few things to look out for:

* **Are all the power supplies for ICs tied in where they belong?** Often times, a circuit will contain several power supply nodes such as Vdd, Vcc, V+, etc. that should be tied together for a common voltage. Another problem is when you want a CMOS chip to run between 0, and +5 volts (TTL logic). In this case, you need to tie Vss for this chip to ground. This is done by manually editing the netlist, and changing the node number on the appropriate pin from Vss to gnd.

* **Do all connected pins go somewhere?** You need to make sure that each pin that has a connection has a node number next to it rather than a question mark (a question mark denotes an unconnected pin).

* **Are all nodes correct?** The SDTIII software is very finicky about how you make connections between wires. Often a wire will look like it connects to another on your schematic, but the netlist will not acknowledge the connection. This can often be fixed by using the clean-up option in SDTIII. Otherwise the node numbers can just be edited to match.

* **Does each component contain the module name in the second column of the netlist?** If not, then you can edit it in directly into the netlist, or you can go back to SDT, and put it in field 1, then re-run the netlist.

Now you should have a completed netlist that accurately describes how you want your circuit to look. Remember, your PC Board will have the exact same connections as your netlist, so any mistakes in the netlist will be difficult to fix on the PC Board. It is easier to fix a mistake now than later.

### Importing Netlist

Now it is time to get down to the PC Board. First we need to configure the PCB program:

Go to the PCB configuration menu (PCB /c). At this menu, go to design conditions. Set them to the following:

```plaintext
:::Design Conditions Configuration:::
1 - Track Width-------------------------------------- .025
2 - Pad Diameter------------------------------------- .075
3 - VIA Diameter------------------------------------- .075
4 - Drill Diameter - Pad------------------------------- .040
5 - Drill Diameter - VIA-------------------------------- .040
6 - Text Horizontal Dimension------------------------ .050
7 - Text Vertical Dimension-------------------------- .050
8 - Solder Mask Guard-------------------------------- .020
9 - Isolation Track to Track------------------------- .025
10 - Isolation Track to VIA---------------------------- .025
11 - Isolation VIA to VIA----------------------------- .025
12 - Number of Backup Files-------------------------- 2
13 - Number of Layers-------------------------------- 2
14 - Routing Grid------------------------------------- .050
15 - Working Layer A-------------------------------- 1
16 - Working Layer B-------------------------------- 2
17 - Strategy Pass 1-------------------------------- Normal
18 - Strategy Pass 2-------------------------------- Flexible
19 - Net Pattern-------------------------------------- Tree
20 - Cursor Style------------------------------------- Short Cross
```
Now we are ready to import our netlist to PCB. First copy your netlist into the PCB directory. Then enter the PCB program. When it asks you to for the filename, enter the name that you want to use for your PC-board (usually ending with .PCB). Now you want to go to the quit menu. Hit initialize, then use netlist. You will be prompted to create a window for your board. Make sure the window is large enough for all your components to fit. When you are prompted for your netlist file, enter it. Now all your components will be placed on the board. You need only put down an edge, move the components around on your board, then route it.

**Here are a few helpful hints for designing your PC Board:**

Route all power supply tracks first. This will make them the most direct routes on the board.

Since we cannot do plate-through holes on campus, some problems arise. The main problem is in soldering IC sockets on the component side. There is no way of getting a soldering iron under the socket to do the soldering. In order to avoid this problem, the best thing to do is to edit the modules in the library to put pads only on the solder side. This way, no connections will have to be made on the component side. This is not a problem with capacitors and resistors, only with sockets.

**Plotting the Board**

When you go to plot the board there are three things you should note:

The first thing to note is that Orcad plotting program has its origin in the lower left hand corner, while the HP plotter has its origin in the center of the paper. For this reason it is necessary to set the X and Y paper offset. This option is available from the configuration menu (PCB /c) under the heading of Pen Carousel. These X and Y paper offsets will be negative values equal to half the width, and length of the paper respectively. This is covered better in the PCB Manual.

The second note to make is that P1, and P2 must be set on the HP plotter before the plot can be made. This is done from the control panel on the plotter. It is covered in the manual for the plotter.

The third note is that unlike the Wintek Smartworks software, holes are not placed in the center of the pads. This is not a big problem. It is very easy to center a drill bit on the pads, even without a center mark. Just make sure that the bit is very sharp.
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Logging onto OrCAD SDT</td>
<td>39</td>
</tr>
<tr>
<td>Logging onto OrCAD PCB</td>
<td>39</td>
</tr>
<tr>
<td>Printing from OrCAD SDT</td>
<td>39</td>
</tr>
<tr>
<td>Plotting from OrCAD PCB</td>
<td>40</td>
</tr>
<tr>
<td>Loading Libraries</td>
<td>44</td>
</tr>
<tr>
<td>Creating a Drill Tool Template</td>
<td>45</td>
</tr>
<tr>
<td>Annotating your Schematic Design</td>
<td>46</td>
</tr>
<tr>
<td>Creating a Custom Library of Your Schematic Design</td>
<td>47</td>
</tr>
<tr>
<td>Creating a Netlist</td>
<td>48</td>
</tr>
<tr>
<td>Creating a Bill of Materials</td>
<td>48</td>
</tr>
<tr>
<td>Creating a Wirelist</td>
<td>49</td>
</tr>
<tr>
<td>Stuffing Fields</td>
<td>49</td>
</tr>
<tr>
<td>Netlist Example</td>
<td>50</td>
</tr>
<tr>
<td>Wirelist Example</td>
<td>51</td>
</tr>
<tr>
<td>Stuff Field Example</td>
<td>51</td>
</tr>
<tr>
<td>Bill of Materials Example</td>
<td>52</td>
</tr>
<tr>
<td>Drill Tool Example</td>
<td>52</td>
</tr>
</tbody>
</table>
Logging onto OrCAD SDT

The following instructions shows how to log onto OrCAD SDT.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>C:\ORCAD\SDTIII&gt;</td>
<td>DRAFT &lt;CR&gt;</td>
<td>OrCAD logo should appear.</td>
</tr>
<tr>
<td></td>
<td>OrCAD logo</td>
<td>Clear OrCAD logo.</td>
</tr>
<tr>
<td></td>
<td>load file?</td>
<td>A:\Design.SCH</td>
</tr>
</tbody>
</table>

You are now in OrCAD SDT, have fun.

Logging onto OrCAD PCB

The following instructions shows how to log onto OrCAD PCB.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>C:\ORCAD\PCB&gt;</td>
<td>PCB &lt;CR&gt;</td>
<td>OrCAD logo should appear.</td>
</tr>
<tr>
<td></td>
<td>OrCAD logo</td>
<td>Clear OrCAD logo.</td>
</tr>
<tr>
<td></td>
<td>load board?</td>
<td>A:\Design.BRD</td>
</tr>
</tbody>
</table>

You are now in OrCAD PCB, have fun.

Printing from OrCAD SDT

The utility PLOTALL allows you to print your schematic design to the printer. The following instruction shows how to print your schematic design.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>C:\ORCAD\SDTIII&gt;</td>
<td>PLOTALL</td>
<td>This will print your schematic design.</td>
</tr>
<tr>
<td>A:\Design.SCH</td>
<td>/P &lt;CR&gt;</td>
<td></td>
</tr>
</tbody>
</table>

Printing from OrCAD PCB

The utility PRINTPCB allows you to get a quick plot of your printed circuit board. Instead of waiting several minutes for the plotter to generate a plot, you can uses the printer, which takes one fourth the time. You should have a formatted disk in drive A: before starting the following instructions.

Use the following names when saving your plot files:
Plot File Names

Layer1.PRN
Layer2.PRN
Silkscrn.PRN

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Quit)</td>
<td></td>
<td>Chose menu option to quit.</td>
</tr>
<tr>
<td>(Plot)</td>
<td></td>
<td>Choose menu option to access plotting menu.</td>
</tr>
<tr>
<td>(Item to Plot)</td>
<td></td>
<td>Select a particular item to plot.</td>
</tr>
<tr>
<td>(Layer) (Layer 2)</td>
<td></td>
<td>For example select layer 2.</td>
</tr>
<tr>
<td>(All)</td>
<td></td>
<td>Plot all traces and pads.</td>
</tr>
<tr>
<td>&lt;ESC&gt;&lt;ESC&gt;</td>
<td></td>
<td>Return to Plot menu.</td>
</tr>
<tr>
<td>&lt;ESC&gt;&lt;ESC&gt;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(Destination)</td>
<td></td>
<td>Specify to save printer database file.</td>
</tr>
<tr>
<td>(Hard Copy)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Write Drill Tool File? <CR>

Save Printer Data Base?

A:Layer2.PRN Enter name of database file.

<ESC><ESC> Return to Quit menu.

(Abandon program) (Yes) Exit OrCAD PCB.

C:\ORCAD\PCB> PRINTPCB This will send your database file to the printer.

A:Layer2.PRN

Plotting from OrCAD PCB

At this point your printed circuit board (PCB) design should be finished and ready for plotting. From this plot the tech shop will design a photo-plot to develop your final printed circuit board. The Hewlett Packard drum plotter will be used to plot the PCB design.

On larger (C-size and above) Hewlett Packard drum plotters, the sheet origin (0,0) is at the center of the picture, while OrCAD's origin is at the top left. Offsets need to be introduced into the plot and sheet coordinates. The plot coordinates need to be manually set in OrCAD PCB configurations menu and the sheet coordinates need to be manually set in OrCAD PCB. The following table shows the plot and sheet coordinates for A, B, and C size plots.
Plot And Sheet Coordinates:

<table>
<thead>
<tr>
<th>Plot Size</th>
<th>Plot Dimension</th>
<th>Plot Offset</th>
<th>Sheet Offset</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>8.5&quot; X 11&quot;</td>
<td>-3.75&quot; X -5&quot;</td>
<td>4.75&quot; X 6&quot;</td>
</tr>
<tr>
<td>B</td>
<td>11&quot; X 17&quot;</td>
<td>-5&quot; X -8&quot;</td>
<td>6&quot; X 9&quot;</td>
</tr>
<tr>
<td>C</td>
<td>17&quot; X 22&quot;</td>
<td>-8&quot; X -10.5&quot;</td>
<td>9&quot; X 11.5&quot;</td>
</tr>
</tbody>
</table>

The plot coordinates must be set first in OrCAD PCB configurations menu. To change the plot coordinates, the following sequence of commands should be entered, where <CR> represents carriage return (the enter key) on your keyboard. Use the Plot Offsets from the table above to offset the plot coordinates according to the desired plot size.

**Prompt**    **Enter**    **Explanation**

C:\\ORCAD\\PCB> PCB/C <CR> Invoke PCB configuration menu.

Command? PC Select the pen carousel configuration menu.

Command? X Change X plot offset.

X Paper Offset-> "Plot Offset" Set to X Plot Offset from table above.

Command? Y Change Y plot offset.

Y Paper Offset-> "Plot Offset" Set to Y Plot Offset from table above.

Command? Q Quit the pen carousel configuration menu.

Command? U Update configuration file, save new settings.

At this point you need to run OrCAD PCB and load your printed circuit board. Then you will move the top left of your board to the Sheet Offset from the table on the previous page.

**Prompt**    **Enter**    **Explanation**

Command? R Run OrCAD PCB

OrCAD Logo <CR> <CR> Clear OrCAD Logo

Load Board? A:Design.BRD Load your printed circuit board
<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Block) (Move)</td>
<td>Transfer your board to the Sheet Offset coordinates.</td>
<td></td>
</tr>
<tr>
<td>(Begin)</td>
<td>Begin at the bottom right corner of your board.</td>
<td></td>
</tr>
<tr>
<td>(End)</td>
<td>End at the top left corner of the board making sure the entire board is selected.</td>
<td></td>
</tr>
<tr>
<td>(Edge &amp; All)</td>
<td>Select the entire board and move top left corner or cursor to the Sheet Offsets.</td>
<td></td>
</tr>
<tr>
<td>(Place)</td>
<td>When the cursor has reached the Sheet Offset place the board down.</td>
<td></td>
</tr>
</tbody>
</table>

At this point the plot and sheet offsets have been set and you are now ready to plot your printed circuit board. The plotting process consists of writing a plot file to a disk and then taking that file to the computer that is connected to the Hewlett Packard drum plotter.

The first plot will be a test plot and your second plot will be your final plot. The test plot is saved with all traces and pads sketched and is printed on plain paper. This will take less time to plot and will allow you to view your printed circuit board to see if there are any flaws. The final plot is saved with all traces and pads filled and is printed on mylar paper for photo-plot generation. This will take about twice the time as did the test plot. You should have a formatted disk in drive A: before starting the following instructions.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Quit)</td>
<td>Chose menu option to quit.</td>
<td></td>
</tr>
<tr>
<td>(Plot)</td>
<td>Choose menu option to access plotting menu.</td>
<td></td>
</tr>
<tr>
<td>(Scale) (2)</td>
<td>Select a scaling factor of 2 for a 2 to 1 plot. This generates a higher resolution photo-plot.</td>
<td></td>
</tr>
<tr>
<td>(Window) (Rotate)</td>
<td>Rotate the window so that the plot is landscaped.</td>
<td></td>
</tr>
<tr>
<td>(Window) (Size)</td>
<td>Select the Plot Size from the table above.</td>
<td></td>
</tr>
<tr>
<td>(Zoom) (Out)</td>
<td>This will allow you to center window easier.</td>
<td></td>
</tr>
</tbody>
</table>
You are now ready to plot your printed circuit board on the Hewlett Packard drum plotter. The following instructions will show you how to setup and use the plotter.

1) First, you must get a plain piece of paper and a plotter pen from the Tech Shop to plot your test plot (final plot will take a piece of mylar and a special plotter pen).

2) Next, turn on the Hewlett Packard drum plotter (switch is located on the back right side of the plotter).

3) Then pull out the pen carousel platter and load the pen in slot 1 making sure you line slot 1 with the arrow when putting the pen platter back.

4) Then load the paper in length-wise making sure the black load arm (located on the left side of the drum) is up. Also make sure the left side of the paper touches the two white tabs (located on the front left side of the drum and back left side of the drum).
5) Then slide the paper guide onto the right side of the paper (located on the right side of the drum) making sure the white stripe lines up with the right edge of the paper.

6) Finally, flip the black load arm down and stand back. The plotter will go through a setup procedure rolling the paper back and fourth checking for all edges of the paper. The plotter is now ready for plotting.

7) Now, put your disk in drive A and type `COPY A:Layer2.PLT COM1` this will copy your plot file to the plotter. Stand back and watch it go. When the plot is finished, lift up the black load arm and pull your paper out.

### Loading Libraries

The libraries in OrCAD SDT are not updated, so you may find your particular electronic component missing. If this happens, you will have to create your own library part. To create a library part refer to the section on "Creating Library Parts." Make sure PCBDEV.LIB is loaded above DEVICE.LIB. The following instructions show how to load libraries into OrCAD SDT.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>C:\ORCAD\SDTIII&gt;</td>
<td>DRAFT/C &lt;CR&gt;</td>
<td>Invoke SDT configuration.</td>
</tr>
<tr>
<td>Command?</td>
<td>LF</td>
<td>Invoke Library Files.</td>
</tr>
<tr>
<td>Command?</td>
<td>A</td>
<td>Add library name.</td>
</tr>
<tr>
<td>Selection-&gt;</td>
<td>1 &lt;CR&gt;</td>
<td>Add library in location 1.</td>
</tr>
<tr>
<td>Enter New Name</td>
<td>Library.LIB &lt;CR&gt;</td>
<td>Library name will appear in location 1.</td>
</tr>
</tbody>
</table>

If you want to load another library follow the three previous steps.

You should never have more then ten libraries loaded, otherwise you will run out of memory. The following instructions show how to delete libraries in OrCAD SDT.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Command?</td>
<td>R</td>
<td>Remove library name.</td>
</tr>
<tr>
<td>Selection-&gt;</td>
<td>Number of Library.LIB &lt;CR&gt;</td>
<td>This will delete the library.</td>
</tr>
</tbody>
</table>

The following instructions show how to exit and save changes.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Command?</td>
<td>Q</td>
<td>Quite.</td>
</tr>
<tr>
<td>Command?</td>
<td>U</td>
<td>Update library changes.</td>
</tr>
<tr>
<td>Command?</td>
<td>R</td>
<td>Run Draft.</td>
</tr>
</tbody>
</table>

You should now be in OrCAD SDT.
Creating a Drill Tool Template

The utility PRINTPCB allows you to get a drill tool template and a drill tool table of your printed circuit board. The drill tool template shows where to drill the hole and the drill tool table shows what size drill to use. An example of a drill tool template can be found in the section "Drill Tool Template Example" and an example of a drill tool table can be found in the section "Drill Tool Table Example." This information is useful when drilling your printed circuit board. You should have a formatted disk in drive A: before starting the following instructions.

Uses the following table to match the drill tool template to the drill tool table:

Drill Symbol Table

1-0
2-+
3-X
4-*
5-Y
6-Z
7-T
8-H

Note: If there are more than eight drill hole sizes on a board, PRINTPCB will represent those above the eight drill hole size with the eight symbol and display this message: "Warning: More than 8 drill sizes found, forcing to #8."

Use the following names when saving your plot files:

Plot File Names

Layer1.PRN
Layer2.PRN
Drill.TOL
Annotating your Schematic Design

When you run the utility ANNOTATE on your schematic design, it puts a reference designator value in the Reference Field of each library part. For example when you get a resistor from the DEVICE.LIB library, it comes up with a Reference Field of "R?". Now after you run ANNOTATE the Reference Field will look like this "R1". If you add new parts to your schematic, you must run ANNOTATE again. The following instruction shows how to annotate your schematic design.
At this point you are ready to create a netlist of your schematic design. To create a netlist refer to the section on "Creating a Netlist."

---

**Creating a Custom Library of Your Schematic Design**

The utilities LIBARCH and COMPOSER create a custom library from your schematic design. The custom library contains all the library parts in your schematic, this way the custom library is the only library that must be loaded to enter DRAFT. The utility LIBARCH creates an ASCII source file that is readable by a text editor, but not by OrCAD SDT. To solve this problem the utility COMPOSER is used to create an OrCAD file that is readable by OrCAD SDT. Creating a custom library can be done anytime throughout the design of your schematic design (write over the existing custom library when asked). The following instructions show how to create a custom library of your schematic design.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>C:\ORCAD\SDTIII&gt;</td>
<td>ANNOTATE</td>
<td>This will annotate the schematic Design.SCH.</td>
</tr>
<tr>
<td>A:Design.SCH</td>
<td></td>
<td></td>
</tr>
<tr>
<td>A:Design.ANN</td>
<td></td>
<td></td>
</tr>
<tr>
<td>/M &lt;CR&gt;</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

At this point follow the instructions "Loading Libraries" to load your custom library into SDT.
Creating a Netlist

Once your schematic design is finished the next step is to design the printed circuit board (PCB). To design a PC board you must first create a netlist of your schematic design. This is done by using the utility NETLIST in OrCAD SDT, but before you can use this utility you must make sure you have annotated your schematic (refer to the section on "Annotating your Schematic Design") and stuffed Part Field 8 in each library part of your schematic (refer to the "Section on Stuffing Fields.") After your schematic design is annotated and Part Field 8 is stuffed then you are ready to create a netlist. The following instructions shows how to create a netlist.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>C:\ORCAD\SDTIII&gt;</td>
<td>NETLIST</td>
<td>This will crate a netlist from Design.SCH and store it in Design.NET.</td>
</tr>
<tr>
<td>A:Design.SCH</td>
<td></td>
<td></td>
</tr>
<tr>
<td>A:Design.NET</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ORCADPCB /S</td>
<td>&lt;CR&gt;</td>
<td></td>
</tr>
</tbody>
</table>

At this point you are ready to design your printed circuit board. Refer to the section "Tutorial on PCB" to initialize your netlist and get you started on designing your printed circuit board.

Creating a Bill of Materials

The utility PARTLIST creates a Bill of Materials (BOM) of all the library parts in your schematic design. An example of a BOM can be found in the section "Bill of Material Example." The following instructions shows how to create a Bill of Material.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>C:\ORCAD\SDTIII&gt;</td>
<td>PARTLIST</td>
<td>This will create a Bill of Materials and store it in Design.BOM.</td>
</tr>
<tr>
<td>A:Design.SCH</td>
<td></td>
<td></td>
</tr>
<tr>
<td>A:Design.BOM</td>
<td></td>
<td></td>
</tr>
<tr>
<td>&lt;CR&gt;</td>
<td></td>
<td></td>
</tr>
<tr>
<td>C:\ORCAD\SDTIII&gt;</td>
<td>COPY A:Design.BOM</td>
<td>This will print your Design.BOM.</td>
</tr>
<tr>
<td>PRN &lt;CR&gt;</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Creating a Wirelist

The utility WIRELIST creates a list of all nodes, references, pin numbers, pin names, pin types, and part values of your schematic design. An example of a wirelist can be found in the section "Wirelist Example." The following instructions show how to create a wirelist of your schematic design.

<table>
<thead>
<tr>
<th>Prompt</th>
<th>Enter</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>C:\ORCAD\SDTIII&gt;</td>
<td>NETLIST</td>
<td>This will create a wirelist of your schematic design</td>
</tr>
<tr>
<td></td>
<td>A:Design.SCH</td>
<td>and store it in Design.WIR.</td>
</tr>
<tr>
<td></td>
<td>A:Design.WIR</td>
<td></td>
</tr>
<tr>
<td></td>
<td>/S WIRELIST&lt;CR&gt;</td>
<td></td>
</tr>
</tbody>
</table>

C:\ORCAD\SDTIII> COPY A:Design.WIR PRN <CR> This will print your Design.WIR.

Stuffing Fields

When you run the utility FLDSTUFF on your schematic design, it puts a PCB Module in Field 8 of each library part. For example lets say you have a 1K 1/4W resistor and you want to stuff the appropriate module RC07. By running FLDSTUFF it will place the module RC07 in Field 8 of the 1K 1/4W resistor. But before you can run FLDSTUFF you must first print out a Bill of Materials (BOM), usually know as a Partslist, and create a stuff file. To create a BOM refer to the section on "Creating a Bill of Materials." After you have created a BOM, you are now ready to create a stuff file.

To create a stuff file you must use a text editor and create a file called Design.STF. First you look on the BOM and find the value under the heading Part. Type the part value with quotation marks around it. Secondly refer to the section on "Table of Reference Material" and look up the module type in "PCB Module List." This is done by looking on the BOM and finding the value under the heading Reference. Next look the reference value up in the "PCB module List" and type it in with quotation marks around it. An example of a BOM and Stuff file can be found in the sections "Bill of Materials Example" and "Stuff File Example." Finally save the stuff file as an ASCII file with the name Design.STF.

Now that you have created a stuff file you are ready to run the utility FLDSTUFF. The following instruction shows how to stuff filed 8 of each library part.
Prompt    Enter    Explanation
C:\ORCAD\SDTIII> **FLDSTUFF**
A:Design.SCH
8 A:Design.STF
<CR>

This will stuff field 8 of each library part.

At this point you are ready to create a netlist of your schematic design. To create a netlist refer to the section on "Creating a Netlist."

---

**Netlist Example**

```plaintext
( | OrCAD PCB NetList Format
12V-5V DC-DC Converter          Revised:      Jan 1, 1992
1992                             Revision: 1.0
California State University Sacramento

Time Stamp -   1-JAN-1992   12:00:00 }
( A8F5F035 CK05 C1 0.1UF
  ( 1 N00002 )
  ( 2 GND )
)
( A8F5F034 CK05 C2 1000UF
  ( 1 N00002 )
  ( 2 GND )
)
( A8F5F037 FUSE F1 1A
  ( 1 N00001 )
  ( 2 N00002 )
)
( A8F5F036 DO7 D1 1N4747
  ( 1 GND )
  ( 2 N00002 )
)
( A8F5F038 3SIP100 JP1 3 PIN
  ( 1 N00003 )
  ( 2 GND )
  ( 3 N00001 )
)
( A8F5F033 TO220 U1 LM340T-5.0
  ( 1 N00002 )
  ( 2 GND )
  ( 3 N00003 )
)
)
```
Wirelist Example

Wire List

12V-5V DC-DC Converter

Revised: Jan 1, 1992
Revision: 1.0
California State University Sacramento

<<< Component List >>>

0.1UF C1 CK05
1000UF C2 CK05
1A F1 FUSE
1N4747 D1 DO7
3 PIN JP1 3SIP100
LM340T-5.0 U1 TO220

<<< Wire List >>>

<table>
<thead>
<tr>
<th>NODE</th>
<th>REFERENCE</th>
<th>PIN #</th>
<th>PIN NAME</th>
<th>PIN TYPE</th>
<th>PART VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>[00001]</td>
<td>N00001</td>
<td>F1</td>
<td>1</td>
<td>1</td>
<td>Passive</td>
</tr>
<tr>
<td></td>
<td></td>
<td>JP1</td>
<td>3</td>
<td>3</td>
<td>Passive</td>
</tr>
<tr>
<td>[00002]</td>
<td>N00002</td>
<td>F1</td>
<td>2</td>
<td>2</td>
<td>Passive</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C1</td>
<td>1</td>
<td>1</td>
<td>Passive</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C2</td>
<td>1</td>
<td>1</td>
<td>Passive</td>
</tr>
<tr>
<td></td>
<td></td>
<td>U1</td>
<td>1</td>
<td>IN</td>
<td>Input</td>
</tr>
<tr>
<td>[00003]</td>
<td>N00003</td>
<td>JP1</td>
<td>1</td>
<td>1</td>
<td>Passive</td>
</tr>
<tr>
<td></td>
<td></td>
<td>U1</td>
<td>3</td>
<td>OUT</td>
<td>Output</td>
</tr>
<tr>
<td>[00004]</td>
<td>GND</td>
<td>D1</td>
<td>1</td>
<td>ANODE</td>
<td>Passive</td>
</tr>
<tr>
<td></td>
<td></td>
<td>JP1</td>
<td>2</td>
<td>2</td>
<td>Passive</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C1</td>
<td>2</td>
<td>2</td>
<td>Passive</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C2</td>
<td>2</td>
<td>2</td>
<td>Passive</td>
</tr>
<tr>
<td></td>
<td></td>
<td>U1</td>
<td>2</td>
<td>GND</td>
<td>Passive</td>
</tr>
</tbody>
</table>

Stuff Field Example

'0.1uF' 'CK05'
'1000uF' 'CK05'
'1N4747' 'DO7'
'1A' 'FUSE'
'3 PIN' '3SIP100'
'LM340T-5.0' 'TO220'
## Bill of Materials Example

<table>
<thead>
<tr>
<th>Item</th>
<th>Quantity</th>
<th>Reference</th>
<th>Part</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>C1</td>
<td>0.1μF</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>C2</td>
<td>1000μF</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>D1</td>
<td>1N4747</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>F1</td>
<td>1A</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>JP1</td>
<td>3 PIN</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>U1</td>
<td>LM340T-5.0</td>
</tr>
</tbody>
</table>

## Drill Tool Example

```
{{ Time Stamp - 1-JAN-1992  12:00:00 }}
```

<table>
<thead>
<tr>
<th>Tool</th>
<th>Diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Symbol 1</td>
<td>0.040&quot;</td>
</tr>
<tr>
<td>Symbol 2</td>
<td>0.045&quot;</td>
</tr>
<tr>
<td>Symbol 3</td>
<td>0.054&quot;</td>
</tr>
</tbody>
</table>
Inserting OrCAD Schematics into word processor documents

A number of people have asked how can OrCAD drawings be incorporated into word processor documents. There are ways of doing this and what follows is a procedure worked out by Rex Lam and Vince Patron.

After experimenting, with a few files, Vince Patron and Rex Lam have found away to import OrCAD schematics into Microsoft Word for Windows.

The procedure is as follows:

1) After the schematic has successfully been saved in 'draft', exit to DOS.

2) Type 'draft /c' at the DOS prompt to configure the printer.

3) Type 'PL' to change the plotter configuration.

4) Choose the HP.DRV driver.

5) Update the configuration and quit to DOS

6) Use the file 'plotall' to create hard copy to file by typing the following:

   plotall  filename1  filename2

   This command will create a file called filename2.000. Do not be alarmed if binary characters appear on the screen during the file write.

7) At this point, you can enter windows by typing 'win'.

8) Once inside Word for Windows, choose Insert, followed by Picture.

9) Include the path with the filename and extension (i.e. c:\orcad\filename2.000)

10) Once the picture has been imported, you may increase the size, change fonts within the schematic, etc.
An Improved Rectifier Circuit Analysis
Functional Description and Concept Diagram
Specifications

DIGITAL POWER METER SPECIFICATIONS

I. INPUTS:
° Voltage: 0-200V\textsubscript{rms}, 45-65Hz
° Current: 0-150A\textsubscript{rms}, 45-65Hz

II. CONTROLS:
° Three position slide switch for V, A, or VA function
° Two position slide switch for ON and OFF

III. OUTPUT:
° One 3-Digit, “ high LCD display.
  a) V or A function: displays 00.0 to 199.9 V or A
  b) VA function: displays 0.00 to 19.99 KVA
° Analog Output: 0-200mV (1mV per V or A; 10mV per KVA)

IV ACCURACY:
° Display: ±1% of full scale on all three functions.
° Analog Output: within ±1 digit of the display.

V POWER REQUIREMENTS:
° One 9V alkaline battery, <10mA drain (≈50hrs operation)
° 60mA from 12VAC transformer (includes charging battery)

VI OPERATING ENVIRONMENT:
° Temperature: 0°C to +60°C
° Humidity: ≤90% (no condensation)

VII PHYSICAL SIZE:
° Weight: less than one pound.
° Size: 7"x4"x2" (LxWxH)
Theory of Operation, Block Diagram, and Design Calculations
Personal Guitar Amplifier:
 Theory of Operation,
 Block Diagram, 
 and 
 Design Calculations

course: EEE–193A
instructor: Dennis Dahlquist, P.E.
date: Oct. 13, 1992
Introduction

The "personal guitar amplifier" is essentially what the title implies—a small, portable, amplifier for amplifying the signal from an electric guitar to a level audible to the user. The audible sound is produced by a pair of small headphones, thereby drastically reducing the power requirements for this product. In addition to functioning simply as an audio amplifier, extra features are included—some for the purpose of modifying the guitar signal to produce different sounds, while another provides a means of mixing a signal from an external program (i.e., radio, tape-recorder, etc.) with the instrument signal. This latter feature allows the musician to practice with background music accompaniment.

Theory of Operation

Input Stage

The basic theory of operation of the device is as follows. Referring to Fig. 1, the signal from the guitar is fed into an input buffer. The purpose of this stage is to minimize any loading effects on the guitar signal, and also to preamplify the signal to a more workable level. Typical output impedances of electric guitars are in the kilo-ohm range (5kΩ to 10kΩ). Thus, it is important to have a high input impedance. A single operational amplifier configured in the non-inverting mode should provide a suitably high input impedance, while at the same time offering a high degree of flexibility for gain control and frequency filtering.

The exact level of gain will need to be more precisely determined at a future date. This parameter will be based on what voltage levels will be most useful at the next stage—that being the distortion and filtering stage. Preliminary lab work has provided values for some output voltages measured on two types of commonly used guitar transducers (called "pickups" in the industry). These are tabulated below [Table 1].

Table 1: Some open circuit voltage measurements of guitar output signals.

<table>
<thead>
<tr>
<th>Pickup type</th>
<th>Vmax(p-p)</th>
<th>V(t ≈ 2 sec.) &quot;strum&quot;</th>
</tr>
</thead>
<tbody>
<tr>
<td>Double coil &quot;humbucker&quot;</td>
<td>1.67V</td>
<td>0.400V</td>
</tr>
<tr>
<td>Single coil &quot;stratocaster&quot;</td>
<td>0.470</td>
<td>0.172</td>
</tr>
<tr>
<td>combination (2 pickups)</td>
<td>1.0</td>
<td>0.172</td>
</tr>
</tbody>
</table>

Distortion and Filtering stages

Following the input buffer, the signal takes one of two paths: one path (the "clean" signal path) goes directly to the equalization stage; the other path first passes the signal through a distortion stage. The selected path will be chosen by the user by toggling an external SPDT switch to one position or the other.

First we will discuss the "EQ" stage. This block will use three variable gain active bandpass filters designed to function complementarily as a 3-band graphic equalizer. The order of the filters will most likely be second order due to the fact that higher order filters usually require multiple op-amps—and thus more power. Since the power budget is a high concern for this product, the filters will be limited to single op-amp designs.

---

1 Actually, the input can be from any instrument with the right signal and impedance levels.
It is not known exactly what center frequencies and bandwidths will be implemented on the filters. Tentative plans are to design for frequency control from about 20Hz to 5 kHz. So a choice of center frequencies might be 200 Hz, 800 Hz, and 3.2 kHz, each with a 3-db bandwidth of two octaves.

The distortion effect is implemented essentially with an op-amp configured for extremely high gain. This causes virtually all but the lowest dynamic level signals to force the op-amp into saturation mode. Diodes will probably be used to limit the peak output voltage amplitude to a value much lower than the supply levels. Essentially, then, what this substage does is transform any "smooth" input signal into a square-wave output with very little dynamic range. Now, in order to remove some of the offensive sound qualities of the square-wave, a low-pass filter follows the square-wave producing op-amp to remove some of the higher order harmonics. This filtering gives a "warmer" sound that more closely resembles that of a vacuum tube amplifier driven to saturation that guitarists have grown to cherish. Figure 2 shows a time domain and frequency domain representation of how the signal is processed as it passes through each substage.

**Summing Amplifier and Power Amplifier**

Following the filtering network, the signal goes into a summing amplifier, where it is mixed with a signal from another, optional, external source—such as a personal tape player or radio. Following this stage, the mixed signal is power amplified by a class AB amplifier for output to the headphones. Another channel is fed to an output jack that can be tapped into for recording, P.A. mixing, or driving a stage amplifier.

Some preliminary calculations have suggested what the output power delivered by the class AB amplifier should be. First of all, we consider the maximum sound level anyone might wish to have available. The equation concerning sound level is,

\[
\beta = (10 \text{ db}) \log \left( \frac{I}{I_0} \right)
\]  

(1)

where \( \beta \) is sound level in decibels and \( I_0 \) is the Bell standard of \( 10^{-12} \). The value of 120db is given as the "threshold of pain" for human hearing. To design on the conservative side, we will use this value, thereby insuring that enough power will be available. Substituting this value for \( \beta \) in equation (1),

\[
120 = (10 \text{ db}) \log \left( \frac{1}{10^{-12}} \right)
\]  

(2)

which yields \( I = 1 \text{ W/m}^2 \). This is the power flux required to cause the sensation of extreme loudness. Considering the headphone speakers to be isotropic radiators (again, worst case), we can calculate the amount of sonic power at each speaker. Assume the sound energy radiates uniformly in all directions. Then, the power generated at the speaker will need to be at a level such that by the time it reaches the eardrum, the power flux density is 1 W/m\(^2\). Therefore

---

\[ P = 4\pi d^2 \left[ \frac{m^2}{W} \right] \times 1 \left[ \frac{W}{m^2} \right] \]  

(3)

where \( d \) is the distance from the speaker to the eardrum. Substituting the value of 2.5 cm. we get \( P = 7.85 \text{ mW} \). However this is the power for only one speaker; the headphones have two speakers. Thus the total sonic power required is

\[ P_{\text{sonic}} = 15.7 \text{ mW} \]  

(4)

Remembering that this is merely the sonic power we must consider how efficiently the speakers convert electrical power into sonic power. Inevitably, some power will be lost in the form of heat.

Some crude laboratory measurements imply an efficiency of roughly 50%. Therefore, the power stage should be designed to deliver roughly 30mW. This should be more than enough power since all calculations were carried out with generous tolerances and assumptions.

**Controls**

A certain amount of control will be available to the user. Output sound level, distortion level, and frequency control will all be easily controlled by external knobs, slider pots, and switches.
FIGURE PPO-100 SYSTEM BLOCK DIAGRAM
THEORY OF OPERATION

BASIC PRINCIPLES OF PULSE OXIMETRY

Oximetry is based on the relationship of a colored substance, the length of the light path through it, it and the light absorption as expressed by the Lambert-Beer Law:

\[ I = I_0 e^{-(Ec \cdot d)} \]

where 
- \( I \) = the amount of transmitted light
- \( I_0 \) = the impinging light
- \( c \) = the concentration of a substance in solution
- \( d \) = the extinction coefficient, a constant at a given wavelength of light for each substance

The Lambert-Beer Law also may be expressed as a logarithmic equation:

\[ E = \frac{(1/cd) \ln(I_0/I)}{\ln(I_0/I)} \]

where \( \ln(I_0/I) \) is referred to as the optical density (D).

Therefore, if the depth or length of the light path and the intensity of the incident light are held constant, the concentration of a substance in solution is a logarithmic function of the intensity if the transmitted light.

It has been shown [24], that oximetry which employs wavelengths of light near the isobestic point (point where characteristic absorption curves cross) of oxyhemoglobin and deoxyhemoglobin are most effective. Early oximeters employed as many as eight different wavelengths of light for analysis, but today's commercial pulse oximeters universally employ red (600-660nm) and infrared (940-960nm) bands. When red light is passed through oxyhemoglobin, absorption is low and conversely, transmission is high. When infrared (IR) light is passed through oxyhemoglobin the absorption coefficient is much higher, more light is absorbed, and less light is transmitted. Similar types of observations can be made with respect to deoxyhemoglobin noting that its transmission of IR light is greater than that of red light.

Oxyhemoglobin is hemoglobin saturated with oxygen; deoxyhemoglobin is hemoglobin that has given up oxygen and is carrying CO2 and/or other gases. Hemoglobin is carried in the red blood cells (RBC's) and gives blood its characteristic color. Oxyhemoglobin, found in arterial blood, is bright red; deoxyhemoglobin, found in venous blood, has a bluer color.

THEORY OF PULSE OXIMETRY: Pulse oximetry utilizes microprocessor technology to carry the principle of oximetry one step further. The microprocessors timing capabilities enable oximetric calculations to be performed on pulsatile blood flow by making differential measurements comparing transmission or reflection of light through pulsatile and nonpulsatile media. The arterial vascular bed is pulsatile expanding during systole with the incoming blood from the heart. The length of the light path through the tissue is thus increased. The vascular bed, on the other hand, is not pulsatile.

The amplitude of the transducer waveform is determined by the SaO2, the wavelength of the incident light, and the amount of blood entering the arterial bed during systole. Differential measurements comparing the amount of oxyhemoglobin in the arterial bed and deoxyhemoglobin in the venous bed are taken during systole and diastole (the period between heart beats). The microprocessor compares the measurements and also corrects the transmissions from each measured wavelength by canceling measurements for area thickness and the presence of ambient light.

The ratio of light transmittances at the two wavelengths results in a beat-to-beat calculation of the functional SaO2 which is the ratio of oxyhemoglobin to the total hemoglobin available for binding oxygen. Hemoglobin which is made dysfunctional by becoming bound to CO2 or some other gas, is not measured.
THE BLOCK DIAGRAM: Looking at the Block Diagram in Figure 4, we see that it has been broken down into nine blocks or sections. Block 1. is the Optical Sensor or Transducer which consists of a red and infrared LED and a silicone photodiode. As was previously mentioned, the wavelength of the red LED is typically around 660 nm and that of the infrared LED around 940 nm [24]. The photodiode used has a broad spectral response that overlaps the emission spectra of the red and infrared LED's.

In Block 2. the Signal Conditioning takes place. Here the time multiplexed output current of the photodiode, corresponding to the red and infrared transmissions, is first converted to a proportional analog voltage using a low noise operational amplifier configured as a current-to-voltage converter. The resulting output voltage is then decomposed into two separate channels using two sample and hold circuits which are triggered synchronously with the same pulses driving the LED's. The two channels are then passed through a high-pass filter with a cutoff frequency of approximately 15 Hz to separate the AC pulses from the DC signal of each photoplethysmogram.

Block 3. is the LED Driver circuitry. The LED's are illuminated separately so the photodiode output represents a signal first from one LED and then the other. This allows sensor processing to determine the intensity of each wavelength without interference from the other LED. The selection of the LED driving current determines the effective penetration depth of the incident light. The LED's should be driven to produce their highest level of incident light without surpassing the manufacturers maximum power dissipation specifications.

The Microcontroller makes up Block 4. and is responsible for performing the A/D conversion of the of the photoplethysmogram, manipulating the data for output, and data storage in RAM. The algorithm for all of the mathematical operations to be conducted on the data resides in the EPROM as does the timing information for driving the LED's, and the algorithm for outputting stored data to the LCD display. The microcontroller unit also contains an 8-bit parallel port for communicating data to the PC.

The Display Driver, Block 5., provides amplification of the ASCII character output by the microcontroller to the LCD Display of Block 6. The information that may be displayed includes %SaO2, heart rate, high and low limits of both of these parameters, and error messages. This will require at least a six character alphanumeric display to accommodate the range of output values.

The User Controls shown in Block 7. serve as an interface between the user and the control unit. This is the point where the mode of operation is determined, selected, and the operations stopped or reset. This section also shows the presence of several LED indicator lights to provide feedback to the user as to the operating status of the device; test, calibrate, run, or down loading data to the PC.

Block 8. is the Isolated Power Supply. It is isolated to provide for the safety of the user as required by all medical devices according to AMMI standards. The power supply is either a series of batteries or battery eliminator as outlined in Power Supply Design section of the device documentation, the battery requirements and implementation have yet to be developed and will follow. The power supply provides operating voltage levels for all of the components in the device.

Finally, in Block 9. we see the PC. It is here that any additional manipulation or statistical analysis of the data takes place and a hardcopy may be obtained. Also, we may choose to store the data onto a floppy for future analysis and comparison.
<table>
<thead>
<tr>
<th>ACTIVITY</th>
<th>WEEK</th>
<th>FEB</th>
<th>MARCH</th>
<th>APRIL</th>
<th>MAY</th>
<th>SUMMER</th>
<th>SEPT</th>
<th>OCT</th>
<th>NOV</th>
<th>DEC</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROJECT DEFINITION &amp; LITERATURE SEARCH</td>
<td>1/27</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>INITIALIZE NELLCOR CONTACTS</td>
<td></td>
<td>2/7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>POWER SUPPLY RESEARCH &amp; DESIGN</td>
<td></td>
<td>2/10</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PROJECT &amp; PERSONAL GOALS - FUNC. DESCRIPTION</td>
<td></td>
<td>2/17</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ORDER MOTOROLA P+MICROCONTROLLER UNIT</td>
<td></td>
<td>2/24</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>NELLOR TOUR AND CONSULTATION</td>
<td></td>
<td>3/2</td>
<td></td>
<td></td>
<td>8/30</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DESIGN SPECIFICATIONS</td>
<td></td>
<td>3/9</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>THEORY OF OPERATION AND BLOCK DIAGRAM</td>
<td></td>
<td>3/16</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>OXISENSOR DRIVER BREADBOARD</td>
<td></td>
<td>3/23</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>OXISENSOR PRE-AMP BREADBOARD</td>
<td></td>
<td>3/30</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BUDGET PLANNING</td>
<td></td>
<td>4/6</td>
<td></td>
<td></td>
<td>9/6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>POWER SUPPLY REASSESSMENT AND DESIGN</td>
<td></td>
<td>4/13</td>
<td></td>
<td></td>
<td>9/13</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>POWER SUPPLY *2 BREADBOARD</td>
<td></td>
<td>4/20</td>
<td></td>
<td></td>
<td>9/20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PREAMPLIFIER PCB LAYOUT</td>
<td></td>
<td>4/27</td>
<td></td>
<td>4/14</td>
<td>10/7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DEMULTIPLEXER/DE-MODULATOR BREADBOARD</td>
<td></td>
<td>5/4</td>
<td></td>
<td>4/24</td>
<td>10/14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MICROCONTROLLER LITERATURE REVIEW</td>
<td></td>
<td>5/11</td>
<td></td>
<td>4/30</td>
<td>10/20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PACKAGING DESIGN</td>
<td></td>
<td></td>
<td></td>
<td>5/18</td>
<td>10/26</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LED FLASHER CIRCUIT BREADBOARD</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>POWER SUPPLY CAD &amp; PCB</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SYSTEM BREADBOARD</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>C PROGRAMMING AND SOFTWARE</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PROJECT REVIEW AND PLANNING</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PCB LAYOUTS</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>COMPLETE PARTS ACQUISITION</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TEST MICROCONTROLLER INTERRUPT SYSTEM</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LCD DISPLAY BREADBOARD</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TEST MICROCONTROLLER A/D</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PCB STUFF AND TEST</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DEADLINE TO TURN IN PCB TO SHOP</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>WIRE WRAP AND FINAL TESTING</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FINAL PROJECT REPORT DUE</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**NOTE1**: POWER SUPPLY *2 DESIGN ABORTED  
**NOTE2**: HAVE NOT RECEIVED uC AS OF 5/11/92
Test Procedures
FIGURE . PORTABLE PULSE OXIMETER PPO-100 TEST FLOW DIAGRAM
TEST DESCRIPTION AND PROCEDURES

POWER SUPPLY EVALUATION

Transformer secondary winding evaluation:
When designing a power supply the transformer plays a very important role. It is usually required to step down the A.C. (alternating current) voltage. Therefore the voltage is important to know in order to complete the necessary design. To test this voltage a signal must be supplied at the primary winding and an oscilloscope is used to measure the output at the secondary winding.

| Transformer | function generator | sine wave | D.U.T. | oscilloscope |

**Equipment list:** HP 3312A Function generator, Standard oscilloscope

Rectifier output evaluation:
The rectifier is used to change the sine wave from a wave that goes both positive and negative to a wave that is exclusively positive. This is the first step in creating a pure D.C. (direct current) signal. In order to successfully test this component we will need the signal from the secondary windings and an oscilloscope.

| signal from secondary winding | full wave rectifier | D.U.T. | oscilloscope |

**Equipment list:** HP 3312A Function generator, Standard oscilloscope

Voltage regulator output evaluation:
The output of the voltage regulator is very important because this is the voltage that will run the rest of the circuit. The output value must be is a specified range of voltages and must be constant. The test is basically the same as the test for the full wave rectifier.

| signal from filtering capacitor | Voltage regulator | D.U.T. | oscilloscope |

**Equipment list:** HP 3312A Function Generator, Standard oscilloscope

Filtering capacitor ripple reduction:
This capacitor will filter the rectified wave from the secondary windings and change it to a d.c. signal. This is important because the voltage regulator depends on the purity of this particular signal. The test procedure is quite simple. First one must generate a sine wave using a function generator then that signal must be...
rectified. The rectified signal is then fed through the filtering capacitor and viewed on an oscilloscope.

![Diagram of signal flow](image)

**Equipment list:** HP 3312A Function Generator, Standard oscilloscope

### 8-BIT ANALOG TO DIGITAL CONVERTER

**Measure analog signal:**
In order for the A/D converter to work properly an adequate analog signal must be supplied. This signal will come from an outside source, such as a probe or sensor. The test will consist of a reference voltage from a power supply and an oscilloscope to measure the output.

![Diagram of signal flow](image)

**Equipment list:** Power supply, Standard HP oscilloscope

**Digital conversion evaluation:**
The A/D converter will convert an analog signal to a digital code. This code must consist of 8 bits of either 1's(+5v) or 0's(~0v). This can be tested by using a power supply and digital designer (digi designer).

![Diagram of signal flow](image)

**Equipment list:** Power supply, Logic Analyzer HP 1650A

### 8-BIT SHIFT REGISTER

**Register evaluation:**
This test will make sure that the register is doing its job. If the shift register is working properly then the bits that are converted by the A/D converter will be stored in the registers. With every clock the bits will be shifted out into the tone generator. Using a power supply and a logic analyzer (HP 1650A) the bits in the shift register can be determined.
Register output evaluation:
The data must be shifted from the registers into a tone generator. With every clock one bit of data will be shifted to the left of the register train. This can be tested the same way as above. The LEDS on the digital designer will correspond to the bits in the registers. The LEDS will shift as the bits in the registers shift.

Equipment list: Power supply, HP 1650A logic analyzer, Digital designer

FUNCTION GENERATOR EVALUATION

Input signal evaluation:
The input signal will be a d.c. signal directly from the power supply. This can be tested by repeating the voltage regulator output evaluation described in the power supply section.

Output signal evaluation:
The output signal can be measured using a standard oscilloscope. Using the voltage supplied by the power supply the function generator will be able to generate a steady pulse train that will be specified in the design. This pulse train will be used as the clock for both the A/D converter and the shift register.


**Equipment list:** Power supply, Standard HP oscilloscope

Frequency measurement:
The frequency is very important because it will determine how fast the data will be converted into a digital signal and shifted out of the registers and into the tone generator. Luckily the test for this procedure is exactly like the test for the output signal evaluation discussed above.

**Equipment list:** Power supply, Standard HP oscilloscope

**ZERO CROSSING DETECTOR**

Input signal evaluation:
The input signal evaluation is conducted because we want to know if there is an actual signal being transmitted from the remote station. Therefore the signal is tested at the output of the receiver before entering the zero crossing detector.

Input signal evaluation continued:

**Equipment:** HP 3312A Function generator, Standard HP oscilloscope

Output signal evaluation:
The output signal is going to be a digital signal consisting of pulses. This circuit is a simple A/D converter. The pulses can be measured using an oscilloscope and simulating an input signal.

**Equipment list:** HP 3312A Function generator, Standard HP oscilloscope

Evaluation of the pulse period:
The period is very important because that is how the micro-controller will distinguish between the different signals. Fortunately, the same test for the output signal evaluation applies to this procedure as well.

**Equipment list:** HP 3312A Function generator, Standard HP oscilloscope

**DATA GENERATION, CONVERSION AND STORAGE**

This evaluation will test the entire subsystem which consists of the 8-bit A/D converter, shift registers, and the function generator. This will ensure that all modules are working as well as the components themselves.

**Equipment list:** Standard HP oscilloscope, HP 1650A logic analyzer
FIGURE: PPO-100 PACKAGE LAYOUT
9V Power Supply Design

California State Univ., Sacramento
Course: EEE-193A, section 1
Instructor: Dahlquist
1. Power Supply Requirements

The first step of designing any power supply is to determine (or estimate as closely as possible) the voltages and currents necessary to power the load circuitry. For this particular project, a number of operational amplifiers (op-amps) will be used [Fig. 1, p. 5]. These are summarized in Table 1 along with the required voltage and an estimate of the current each will draw. These current values were obtained from the data sheets of the various devices.

The primary power source for the final product will be one or two 9V batteries. Therefore, a reasonable choice for the output voltage of an AC/DC power supply to be used with this product is also 9V. For the sake of simplicity, and parts availability, a full-wave bridge rectifier power supply will be designed as an alternative power source for the personal amplifier [Fig. 2, p. 6]. In the design, emphasis will be placed on compactness, weight, and cost, without making any undue sacrifices on performance.

Table 1: List of probable components and estimated power requirements.

<table>
<thead>
<tr>
<th>Function</th>
<th>Device</th>
<th>$V_{CC}$ (V)</th>
<th>$I_{CC}$ (mA)</th>
<th>$P_{CC}$ (mW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Input Buffer</td>
<td>Texas Instr. NE5532</td>
<td>9</td>
<td>4</td>
<td>36</td>
</tr>
<tr>
<td>Overdrive</td>
<td>RCA 4136 x 2</td>
<td>9</td>
<td>7</td>
<td>63</td>
</tr>
<tr>
<td>Filtering</td>
<td>Analog Dev. OP – 421 or equiv.</td>
<td>9</td>
<td>2</td>
<td>18</td>
</tr>
<tr>
<td>Summing Amp.</td>
<td>RCA 4136 x 1</td>
<td>9</td>
<td>4</td>
<td>36</td>
</tr>
<tr>
<td>Output Stages</td>
<td>Nat. Semi. LM386 (x2)</td>
<td>9</td>
<td>8</td>
<td>72</td>
</tr>
<tr>
<td>Headphones</td>
<td>40Ω SONY &quot;mini headphones&quot;</td>
<td>1.4 (rms)</td>
<td>25</td>
<td>35 (max)</td>
</tr>
<tr>
<td>Load requirements:</td>
<td></td>
<td>9</td>
<td>50</td>
<td>260</td>
</tr>
</tbody>
</table>

2. Power Supply Design

Voltage Regulator

A voltage regulator which meets the above requirements is the UA78L09AWC by National Semiconductor. This device is capable of delivering 100mA of current at 9V. Some other specifications of interest are listed below.

- $V_{reg}$ .........................9.0V
- $I_{load}$ (max) .............100mA
- Drop out voltage ..........1.7V
- $V_{in, max}$ ..................35V
• Operating Temp. .....0 – 125°C

Transformer

From the above, it is apparent that the voltage regulator requires the input voltage to be no less than 10.7V for stable regulation. As an added measure of security, we will design for 11.5V. This will provide an extra margin for error without forcing the regulator to dissipate intolerable amounts of power.

The next item to consider is the amount of ripple voltage that can be tolerated. Since 10% of the input voltage is commonly used as a "rule-of-thumb" value, it will be used here as well.

\[
V_{\text{ripple}} \equiv V_r = 0.1V_c \\
= 0.1(11.5) = 1.15V
\]

Another item to consider is the voltage drop which occurs across the diodes in the bridge rectifier. For this design, since such relatively small currents are involved, four 1N4148 diodes will be used for rectification. These have been measured to have a typical voltage drop of 0.88V when passing 80mA of current. Therefore, the rectifier voltage drop (\(V_{\text{rect}}\)) will be estimated to be 1.8V.

Now, the rms voltage of the secondary winding of the transformer can be calculated.

\[
V_{\text{sec, rms}} = \frac{V_c + V_r + V_{\text{rect}}}{\sqrt{2}} \\
= \frac{11.5 + 1.15 + 1.8}{\sqrt{2}} \\
= 10.2V
\]

A transformer with a secondary output voltage of 11.2V is available from HSC Electronics, Sacramento, California. The current rating of this transformer is not readily available; however, from inspection it appears almost certainly capable of delivering at least one ampere—far in excess of the needs of this design.

Filter Capacitor

A large filtering capacitor will be used to "smooth-out" the full-wave rectified waveform. The time of conduction for the capacitor to recharge is given by the equation,

\[
\Delta t = \frac{1}{2\pi} \sqrt{\frac{2V_r}{V_c}} \\
= \frac{1}{2\pi \times 120} \sqrt{\frac{2 \times 1.15}{12.65}} \\
= 566\mu s
\]

The time in which the capacitor supplies current is then

\[
T_{\text{cond}} = T_0 - \Delta t \\
= 8.33ms - 566\mu s \\
= 7.77ms
\]
The appropriate value of capacitance can now be calculated from the equation

\[ C = \frac{I_{load} T_{cond}}{V_r} \]

Substituting values, we get

\[ C = \frac{I_{load} T_{cond}}{V_r} = \frac{0.1 \times 7.77 \times 10^{-3}}{1.15} = 675 \mu F \]

The closest value that could be obtained is an 820\( \mu \)F rated at 20V.

\[ V_{sec, rms} = \sqrt{\frac{V_c + V_r + V_{rect}}{2}} = \sqrt{\frac{11.5 + 1.15 + 1.8}{2}} = 10.2V \]

3. Power Supply Schematic

See Fig. 2, p. 0

4. Circuit Simulation

A PSpice simulation of a crude model of the power supply circuit was performed. The circuit was simulated both unloaded and loaded (see output files at end of report). Since such a crude model of the circuit was used—for lack of any voltage regulator models—the data produced by the simulations are not very useful as far as testing the behavior of a loaded supply is concerned.

The graph on the following page shows the simulated no-load transient response of the voltage at the filter capacitor. The simulation showed the ripple voltage to be approximately 0.52 V. Minimum voltage was 11.37 V in the steady-state condition. It should be remembered, however, that the circuit file used to model the real circuit is only a gross approximation, and actual behavior can only be discovered with a real circuit and real components.
5. **Concluding Remarks**

A simple 120VAC to 9Vdc power supply has been designed. The design is based on a full-wave rectified, constant output design. In the design process, some assumptions were made; however, most of the choices made were based on sound reasoning and established methods. A computer simulation of the final design was performed which yielded results that are only partially useful. This is due to the crude model of the actual circuit that was used in simulation. Only by building and testing a real circuit can the true behavior of the circuit be measured.
Preface

Who is this manual for?

Electrical Engineering Senior Project students in the second semester of their project who have completed EEE193A.

Manual Goals and Objectives

- To provide dissemination of information on the project process.
- To help clarify some of the issues discussed in the course.
- To provide a reference document to refer to during the course.

Introduction to EEE 193B Senior Design Project.

Welcome to the Fabrication, Test, & Demonstration Phase of your Senior Project. In this phase of your project you will get the opportunity to experience completing your project as an engineer. This will encompass the following activities and responsibilities:

1. Responsible for reporting to your manager (your instructor).
2. Responsible for your time and how it is used to achieve your goals and project completion.
3. Project Plan for the semester
4. Progress Reports, weekly written reports
5. Project Status reports, weekly oral reports
6. Design Reviews
7. Project Demonstrations
8. Final Project Report

Using a project note book is highly recommended throughout your design and project process.
Course Description

Course Objective:

This course involves developing the hardware realization of the design developed in EEE 193A. The instructor has the role of an advisory technical consultant who facilitates project completion. Students are responsible for direction and completion of their projects. This includes keeping the instructor informed with regular attendance, project plan, progress reports, final project report and project demonstration.

Course Requirements:

(Late assignments lose 50% of possible points per week late)

Attendance is required for each class session. One missed session can be excused given valid reasons ahead of the absence. Two unexcused absences and the student will be dropped from the course.

Missing the Shop Deadlines will cost 10% of course grade.

The maximum grade that can be earn without PCB is a B.

The maximum grade that can be earn without a Breadboard project demonstration at the design review is a C.

Course Assignments:

Project Plan
A photocopy of the 193A project report and a detailed schedule of the work to be completed in 193B is due the second week of class and is worth 10% of the final course grade.

Progress Reports
Progress reports will be turned in each week starting 3rd week. Half of the report is oral and the other half is written (word processed). Progress reports will make up 25% of the final course grade.

Design Reviews
Periodic design reviews will be given on project status. This will comprise of 15% of the final course grade.

Project Demonstrations
Each student will demonstrate their project's operation and will show how the project meets its specifications. This demonstration will be worth 25% of the final course grade.

Final Project Report
The Final Project Report is due on the 15th class session (dead week) and is worth 25% of the final course grade. No late reports will be accepted. The report must be word processed.
EEE 193B  Suggested Milestones

<table>
<thead>
<tr>
<th>No.</th>
<th>Activity Name</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
<th>14</th>
<th>15</th>
<th>16</th>
<th>17</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SUGGESTED MILESTONES</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>System BB &amp; test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Project review and Plan</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>PCB</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>Parts Acquisition</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>PCB Stuff &amp; Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>Design Reviews</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>Final Project Report</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>[Deadline to Turn in PCBs to Shop]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>System Integration &amp; Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>Project Demonstration</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>Final Project</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>[Final Project Report Deadline]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Legend:
- Planned: ○ Start △ Finish
- Actual: ●
Project Planning

Project Plan Assignment Requirements

The project plan will include the following:

1. **EEE193A Final Project Report.** A copy of your final project report you turned in for EEE193A.

2. **Introduction.** What is the current status of the project. Any changes from 193A?

3. **Task Names and Descriptions of tasks.** Identify tasks that will require more than 1 week and break them into sub-tasks of no greater than 1 week.

4. **Critical tasks,** List Critical tasks and why they are critical. Identify tasks you’re unsure of or that appear difficult and discuss why you feel it is critical.

5. **Contingency plans,** for the project and critical tasks. How will you complete the project in spite of these critical tasks? What are your plans?

6. **Bar Chart Schedule,** Layout the schedule on a chart. Keep at least on copy of the chart for yourself.

Project Planning Steps

1. Review Project requirements

2. Identify and define the end of the project. (Specs)

3. Review Block Diagram, Break into sections based on Block Diagram.

4. Identify the steps required to complete the Project.

5. Visualize the finished project and work backwards.

6. Use 193A Schedule and Suggested Milestones to help identify steps. Suggested Milestones, see Course Schedule and Chart

7. Compare each step against the Project Requirements to verify necessary steps.

8. Break the steps into smaller steps, until you have steps (tasks) of no greater than one week.

9. Review each of the tasks and look for ways to overlap or to complete more than one task.

10. Layout a Time Line Chart (Gantt Chart).
Time Line Chart (Gantt Chart)

Tasks on the Y axis, starting with the first task on top to the last task on bottom.

Time line on the X axis, in week numbers and dates of each week for the project duration.

Refer to the Time Chart Elements diagram. For a completed Time Chart refer to the Chart of Suggested Milestones.

Time Line Chart Elements
Typical Problems with Project Plans.

1. Having tasks greater than one week in duration (Tasks \( \leq 1 \) week).
2. Missing Project name and Project Engineer Name.
3. Insufficient number of tasks, 15 weeks 15 tasks right? wrong! \( \sim 30 \) tasks total, 2 \( \sim 4 \) tasks per week.
5. Not Starting PCB layout before finishing BB.
7. Taking a vacation while the shop is working on the PCBs.
8. Unrealistic amount of work in a short time, for example all software to be coded in one week.
10. Not using descriptive task names and tasks descriptions.
11. Don’t use: Task 1, Task 2, or CKT1, CKT2, or Test 1, Test 2, etc. Use task names like: Osc. BB, RS232 BB, Power Amp. PCB, Controller PCB, etc.
12. Don’t combine task name and description to form a long name or a short description. Separate Task names and Task Descriptions.
13. Not Identifying Project Critical Tasks. Should be at least 3~6 tasks. Identify Specific Tasks and state why you feel they are so.
14. Not making realistic contingency plans. It is not very helpful to say, if problem work harder. The whole idea behind planning is to anticipate problems. A general plan maybe to allocate more time if needed, but you must go beyond that.
15. Not Developing Plans. Plans must be developed for Critical tasks and project deviations. How will you deal with project changes in a general and specific sense to complete the project on time?

The more foresight and planning put into the project now will yield greater success of the project completion.

Reference Sources for Project Planning

Books (see Design Reference section): Planning Big with MacProject.
What every Engineer should know about Project Management.
Managing the Engineering and Construction of Small Projects.

Software: Microsoft Project, Harvard Project, Primavera
MacProject Trial Kit $25, Video, Manual, and SW - 1- 800 628-2100
MacSchedule, Fast Track Schedule

Drawing SW:
Auto CAD, Auto Sketch, Generic CAD, MacDraw, Canvas, MacDraft,
Progress Reports

Progress Report Assignment Requirements

Progress reports should include the following:

1. The last two weeks activities:
   (Problem areas identified, Breakthroughs, Solutions)

2. Plans for the next two weeks:
   (How to get back on schedule, Proposals for solving problems)

3. Schedule update:
   • Summary Tasks Status Table.
   
<table>
<thead>
<tr>
<th>Task Name</th>
<th>Brief Status</th>
<th>Completion update</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

   • Chart update.

4. Additional Information:
   Schematic changes/updates, new chips/circuits, drawing changes, etc.
Design Reviews

PURPOSE:
- To present your project and receive feedback on it.
- To review the projects of your peers and provide feedback to them.

In order to meet these objectives your attendance and participation in all design review sessions, yours and others, are mandatory.

CONTENT: Design Reviews should include:
- Project Introduction, your name and the name of the project.
- Project Goals and Personal Goals.
- Functional Description and Concept Diagram.
- Theory of Operation.
- Block Diagram.
- Major Design Calculation(s).
- Discussion circuitry used (schematic).
- Project Status (show chart and discuss).
- Solutions to Problems encountered.
- Current problems or concerns.
- Demonstration of project Proto-Type.
- Questions from the class.

Design Review Outlines: should include the items listed above in the Content section expanded to cover your project issues.

EVALUATION CRITERIA:
Design Reviews will be evaluated on the completeness of addressing the above issues and the following issues:
- Understanding of the theory behind the project.
- Verbal and visual presentation of the project.
- Discussion of potential problems and the solutions.
- The completeness and accuracy of the Design Calculations, Schematics, and Block Diagrams.
- Level of Project Planning and Project Completion.

As a project reviewer you would use the comment sheet shown below and you should provide feedback to the presenter in the following areas:
- Did the presenter give you a clear understanding of the project?
- What problems do you see with the project? Can you offer any suggestions.
- What would improve the speaker's presentation?
- Other comments.
Final Project Report Requirements

The final report will serve as complete documentation of your senior project. It must be complete in each section and reflect all aspects of your project. Reports must be typed, with the exception given to block diagrams, schematics, and design calculations. The final report is due during the lecture period of the week before finals and will not be accepted late.

1. Title page
   - Project name.
   - Student's name.
   - Course Number & Section / Instructor's Name.
   - Date.

2. Table of contents
   - List of Sections and Figures with page numbers.

3. Executive Summary
   - Project summary.
   - Resume.

4. Project description
   - Project and Personal Goals. Team Work Breakdown.
   - Concept diagram (showing how the device is used).
   - Functional Description. (What is the project? Where is it intended to be used? What are it's features?)

5. Project specifications
   - List of Quantitative Project Requirements.

6. Operator's manual
   - Introduction (what is it and what does it do).
   - Project Diagram (a "Picture" of the project) showing inputs, outputs, and controls.
   - List and description of inputs and outputs, controls, and their functions.
   - Installation, Operating, and Troubleshooting instructions.

7. Project Technical Detail (Maintenance manual)
   - Block Diagram, Software Diagram (if applicable),
   - Theory of Operation (including design discussion), simulations, and Design Calculations.
   - Schematic Organization Diagram, Schematics, Parts Lists, and Wire List (list of connections).
   - Printed Circuit Board Layout, Component Layout, and Package Layout.
   - Calibration section (How is the device to be calibrated to stay within specs?).

8. Test Section
   - Test Flow Diagram.
   - Test Procedures. To include: Test Descriptions. Equipment required.
   - Test Reports (test results).

9. Project Review
   - Goals: Review of Personal Goals and Project Goals (Did you achieve your goals?).
   - Schedule Review: (Original vs. Actual schedule). Discussion of situations encountered. Where did deviations occur and why? If you were to redo the schedule what would you change?
   - Budget Review: (How close to your estimates did you come?).
   - Next steps for the project: What would be the next steps for the project? What improvements could be made to the project?
   - What do you feel you have learned: From the project? From the class? If you could do the project over, what would you change in the Process? From your experience with this project what will you do on the next project? What will you do the same? What will you do differently?
   - Rate and discuss group members performance.

10. Reference and Appendix Sections
    - People, Books, Articles, Data Books, Actual Data Sheets, and any additional information.
CASE STUDY

Engineer A is trained as a mechanical engineer with particular emphasis in automotive engineering. For a period of 10 years he has been employed by a major car manufacturer as an automotive engineer with primary emphasis in engine development with specific interest in power plant efficiency based on utilization of various mixtures of petro chemical fuels.

Engineer A spent many hours at his office and laboratory, many of which were after-hours. While in his after-hours studies and research he developed plans for a new type carburetor which according to his calculations would result in a significant improvement in fuel economy for all vehicles. The improvement would result in, for example, a Cadillac limousine obtaining 85 miles per gallon fuel economy. He has not notified his employer of the development project, nor the results of it.

One of the Engineer A's friends is Engineer B, an engineer with a competitor automobile company. Both engineers are in fairly equal positions in the company, except engineer B is in the design section as opposed to power plant section of the company. Engineer A tells Engineer B of his discovery in broad terms. Engineer B appears interested and together they form a company for development and marketing of the new type carburetor. Development proceeds to placing the carburetor on an automobile, which, as the calculations have demonstrated, results in a tremendous increase in fuel economy. Unfortunately, the tests also demonstrate a significant increase in levels of carbon monoxide, which under current scientific applications are not able to be reduced to acceptable limits. In fact, Engineer B believes that in production quantities the carburetor would increase the level of carbon monoxide such as to drastically impact on the ecostructure of the planet earth.

Proceeding with their marketing plan, and yet continuing to remain employed by their respective companies, the product is marketed. The product is marketed with no disclosure with respect to the potentially adverse environmental concerns, but stating that the device is a great improvement in fuel economy and fuel efficiency for internal combustion engines. A patent is obtained by Engineers A and B for the process, which is in the names of Engineers A and B.

QUESTIONS FOR DISCUSSION

1. What are the rights of the respective automobile manufacturers.
2. Should the adverse environmental issues have been discussed in the press releases drafted by Engineers A and B.
3. Should the project be scrapped entirely because of the adverse environmental concerns.

(hd:question)blb 929-5018
<table>
<thead>
<tr>
<th>INTELLECTUAL PROPERTY COMPARISON CHART</th>
</tr>
</thead>
</table>

1. **PATENT** *(Utility, Design, Plant)*

   The functional aspect of any new and useful process, machine, article of manufacture, or composition of matter, or any new and useful improvement thereof.

   Any new external and ornamental design for a article of manufacture where the design is nonfunctional, is part of, and is not removable from the article.

   Any asexually-reproduced plant.

   (e.g., machinery, electronic circuit, computer program, chemical compound, manufacturing process, genetically altered animal; bottle, computer case, Jewelry, tire tread design, building or other structure, item of apparel, furniture, tool; non-naturally occurring plant).

2. **TRADEMARK**

   Any symbol, sign, word, sound, design, device, shape, mark, etc., used as a trademark, service mark, certification mark, or collective mark, and a common name.

3. **COPYRIGHT**

   Any book, poem, speech, recording, computer program, work of art (statue, painting, cartoon, label), musical work, dramatic work, pantomime and choreographic work, photograph, graphic work, motion picture, videotape, map, gameboard, game instructions, etc.

4. **TRADE SECRET**

   Any information which is not generally known and which will give a business advantage or is commercially useful (e.g., formulae, ideas, techniques, know-how, designs, materials, processes).

5. **TRADE DRESS/UNFAIR COMPETITION**

   Any distinctive design, slogan, title, shape, color, trade dress, package, etc. which has acquired "secondary meaning."
legal advice.
## INTELLECTUAL PROPERTY PROTECTION
### SUMMARY CHART

<table>
<thead>
<tr>
<th>Underlying Creation</th>
<th>Mental How To Acquire Offensive Rights</th>
<th>Legal Remedy For Misappropriation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Invention</strong></td>
<td>File a patent application as soon as possible, but within one year of sale or publication.</td>
<td>Patent infringement litigation.</td>
</tr>
<tr>
<td><strong>Trademark</strong></td>
<td>Use the mark as a proper adjective with the &quot;TM&quot; superscript; register the TM as soon as possible, with the state and/or the PTO.</td>
<td>Trademark infringement litigation, either before or after registration.</td>
</tr>
<tr>
<td><strong>Writings, Music, Recordings, Art, Software, Etc.</strong></td>
<td>Put a copyright notice on the work; also advisable to register copyright within three months of publication.</td>
<td>Copyright infringement litigation, after registration.</td>
</tr>
<tr>
<td><strong>Confidential Technical Or Business Information</strong></td>
<td>Keep it secret; keep good records so you can prove you kept it secret.</td>
<td>Trade secret litigation.</td>
</tr>
<tr>
<td><strong>Distinctive Trade Dress, slogans, Etc.</strong></td>
<td>Advertising end frequent use.</td>
<td>Unfair competition litigation.</td>
</tr>
</tbody>
</table>

COPYRIGHT 1990 GERBER, RITCHY & O'BANION 5441 Fair Oaks Blvd., Suite B-1, Carmichael, CA 95608 (916) 971-1010

The information contained herein has been summarized for Instructional purposes only and does not constitute legal advice.
Outlines from some Guest Speakers
Progress Report Examples