COMSATS University Islamabad (CUI), Lahore Campus
Department of Electrical & Computer Engineering

CPE214–Computer-Aided Engineering
Lab Manual for Spring 2024

Lab Resource Person
Engr. M. Hassan Aslam

Prepared By
Mr. Muhammad Moin
Mr. Assad Ali

Supervised By
Dr. Khurram Zaidi

Name: _____ **Registration Number:** _____ - _____ -

Program: _____ **Batch:** _____

Semester _____

Revision History

Sr.#	Update	Date	Performed by
1	Lab Manual Preparation	May-2020	Muhammad Moin Mr. Assad Ali
2	Layout Modification	May-2020	Mr. Muhammad Moin
3	Lab Manual Review		Dr. Khurram Zaidi
4	Layout Modification (OBE)		Mr. Muhammad Moin
5	Lab Manual Review (OBE)		Dr. Khurram Zaidi

Preface

This course emphasizes using various electronic graphical media software such to create standardized technical documentation for architectural, electrical, and mechanical applications. It will give students an introduction to the fundamentals of computer-aided engineering design; computer-aided drafting principles and practices: engineering drawing fundamentals using AutoCAD; drawing of electrical machinery and layouts of electronic assemblies; design and layout of circuit boards using software such as Proteus ISIS, NI Multisim.

Text Books

1. Shawna Lockhart, "Tutorial Guide to AutoCAD," First Edition, 2006, Prentice Hall, ISBN: 9780131713833.
2. **Zhou Run Jing, Liu Yan Zhen ,"PROTEUS schematic simulation and PCB design"**

Learning Outcomes

After successful completion of this module, you will be able to:

Lab. CLOs:

1. Design of PCB layouts using software tools for applications in engineering design [PLO3] [C5]

2. Construct properly detailed, formatted, dimensioned drawings and sketch different projection techniques using AutoCAD tool [PLO 5] [P4]

CLOs – PLOs Mapping

PLO / CLO	PLO1	PLO2	PLO3	PLO5	Cognitive Domain	Affective Domain	Psychomotor Domain
Lab CLO1			x		C5		
Lab CLO2				x			P4

CLOs – Lab Experiments Mapping

Lab / CLO	Lab 1	Lab 2	Lab 3	Lab 4	Lab 5	Lab 6	Lab 7	Lab 8	Lab 9	Lab 10	Lab 11	Lab 12
Lab CLO1									x	x	x	x
Lab CLO2	x	x	x	x	x	x	x	x				

Grading Policy

The final marks for lab would comprise of Lab Assessment (25%), Lab Midterm (25%) and Lab Terminal (50%):

Lab Assignments:

    i.   Lab Assignment 1 Marks (Lab marks from Labs 1-3)
    ii.   Lab Assignment 2 Marks (Lab marks from Labs 4-6)
    iii.   Lab Assignment 3 Marks (Lab marks from Labs 7-9)
    iv.   Lab Assignment 4 Marks (Lab marks from Labs 10-12)

Lab Mid Term:

Mid Term = 0.5*(Lab Mid Term) + 0.5*(average of lab evaluation of Lab 1-6)

Lab Terminal:

Lab Terminal = 0.5*(Lab Terminal Exam) +0.375*(average of lab evaluation of Lab 7-12) + 0.125*(average of lab evaluation of Lab 1-6)

Software Resources

AutoCAD, Proteus ISIS, NI Multisim

Lab Instructions

- This lab activity comprises of three parts: Pre-lab, In Lab Tasks and Viva session.
- The students should perform and demonstrate each lab task separately for stepwise evaluation.
- Only those tasks that are completed during the allocated lab time will be credited to the students.
- Students are however encouraged to practice on their own in spare time for enhancing their skills.

Safety Instructions

1. It is the duty of all concerned who use any electrical laboratory to take all reasonable steps to safeguard the health and safety of themselves and all other users and visitors.
2. Be sure that all equipment is properly working before using them for laboratory exercises. Any defective equipment must be reported immediately to the Lab. Instructors or Lab. Technical Staff.
3. Students are allowed to use only the equipment provided in the experiment manual.
4. Observe cleanliness and proper laboratory housekeeping of the equipment and other related accessories.
5. Equipment should not be removed, transferred to any location without permission from the laboratory staffs.
6. Do not eat food, drink beverages, or chew gum in the laboratory.

Table of Contents

LAB # 1: To describe CAD software and different function using AutoCAD.

Objectives
To describe the usage of AutoCAD interface and settings of Boolean buttons

Pre Lab

Menu Bar:Menu bar is the link between you and AutoCAD.

File menu: It deals with saving/remaining/printing etc. of the file.

Edit menu: It helps you to cut/copy/paste/ changes etc.

View menu: Helps you to zoom in/out, view the drawing in 3D etc.

Insert menu: Helps you to embed other drawings in your drawing, change layouts etc.

Format menu: It Deals with the properties of lines, units, drawing limits etc.

Tools menu: It is there to enhance your drawing all kinds of lines and shapes.

Dimension menu: It gives you all the ways of accurately giving the dimensions of shapes in your drawing.

Modify menu: It offers you with several options with which you can change and rectify your drawing according to the requirements.
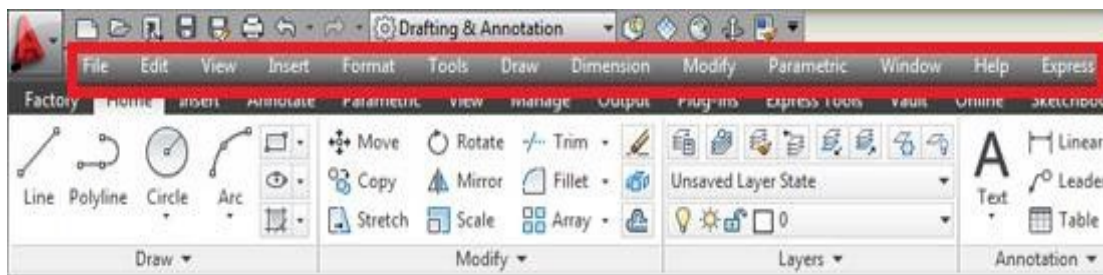
Fig 1.1 Menu bar window

Using AutoCAD toolbars and turning them ON/OFF

There is a toolbar with pictures. Actually, these bars give the most used options from all the minus listed above, so that you can quickly use the commands required. These toolbars can be switched on and off by right clicking on the blank off-white area, selecting the AutoCAD option and then clicking on whatever toolbar you want to activate. All the activated toolbars have a thick to their left side.

Command Line

The command line shows and keeps a history of whatever commands you are using while you are drawing. You can use the menu bar or the toolbar to give commands and you can also give commands by writing the right syntax in the command line for a particular command as shown in figure 1.2.

Fig 1.2 Menu bar window

It gives you number of advantages which will be discussed later. Whenever you want to exit any command, you just have to press the escape button.

Status Bar

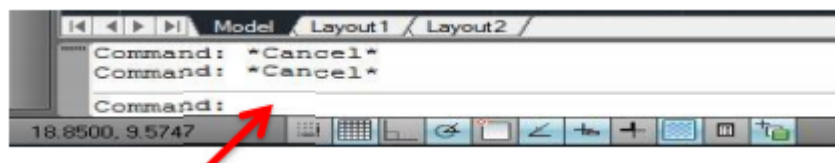This bar is situated at the button of the AutoCAD window as shown in figure 1.3. The buttons on the right side will be discussed later on. On the left side there are 3 values that we can see. These values actually give you the position of the cursor. The first value is the X-coordinate, second is Y-coordinate and the third one is the Z-coordinate.
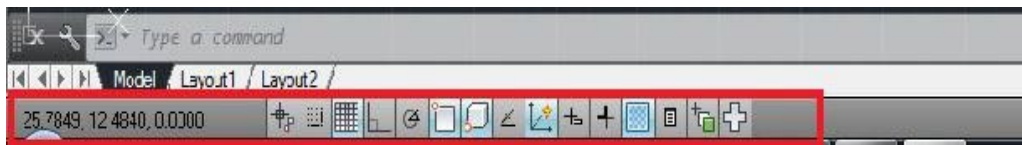
Fig 1.3 Status bar window

Lab Tasks

Setting and Usage of Boolean Buttons

Fig 1.4 Boolean buttons configuration

SNAP

This button when turned on restricts your cursor on the drawing area to move only on the grid points. You can always right click this button, click properties and change snap settings. You can change how much your cursor will move in each step horizontally and vertically on the drawing area.

ORTHO

When this button is turned on you can only make horizontal and vertical lines because cursor will not move other than four directions (i.e. up, down, right, left). When you want to make inclined lines please always turn off this button.

POLAR

Turning on this button will indicate angles while you are making lines, so that u can see easily draw accurate angles between 2 lines or make lines at particular angles with respect to other lines. You can right click on this bottom to change the precision of the angle indicator. e.g. AutoCAD helps you to make angles at an interval of 15.

OSNAP

This option helps you to connect accurately end of lines to centres, midnight,

2

intersections and many other points that you can turn on by right clicking on the button and going to the properties dialog box.

Whenever you end a line, before left clicking and setting its end, while moving on the drawing area, wherever there is a midpoint, an intersection or any other point that has been turned on in the OSNAP settings, as soon as the cursor touches that the point is highlighted with a yellow boundary around that point.

OTRACK

This helps you to move and make the end of any line vertically above/below or horizontally right/left of any other point on the drawing.

LWT

This button when turned on shows the weight of lines on the drawing area. When turned off, all the lines will have equal standard width/weight.

MODEL

When turned on it shows you the black working area and when it is turned off it shows your drawing on white screen.

Lab Task

- Save a file by your name on your desktop and insert it in your flash drive
- Use the command line to give the following commands.

  - ➢ Save
  - ➢ Close
  - ➢ Open (After this command open any AutoCAD template)
- Set the X and Y spacing of snap and grid as 0.5 and see what is happening with your grid points and cursor movements.
- Turn on the model button and go to the paper layout. What is the use of this button in your opinion.

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____

LAB # 2: To construct the drawing model using AutoCAD

Objectives

To demonstrate the settings of drawing limits and units to construct an object using AutoCAD.

Pre Lab

Working Area:

The black background that you see in your AutoCAD window is your working area.

Setting Drawing Limits:

Go to format menu select drawing limits option. On the command line you will be asked to specify the lower left corner of your drawing. The default coordinate is 0, 0 and it is advisable to press enter rather than given any other coordinate because every drawing should start from origin. When you want to make more than one drawing in a single screen only then you change the lower left corner to some other coordinate other than the origin. When you have entered you will be asked to specify the upper right corner. Now you can specify any coordinate e.g. 20, 20 so that your drawing area is 20 by 20 units square. To specify write the coordinate on the command line and press enter.

Setting Units:

This is important because you have to specify the scale of your drawing and keep your drawing dimensionally accurate. Go to the **format menu** and click on **units**. You will come across the window as shown in figure 2.1.

Here you have to specify in what units you would work, what type of lengths you would use (e.g. decimal or fractional). What precision or decimal place you are using to specify each point on your drawing. What type and precision of angles you would use (e.g. degree or radian).
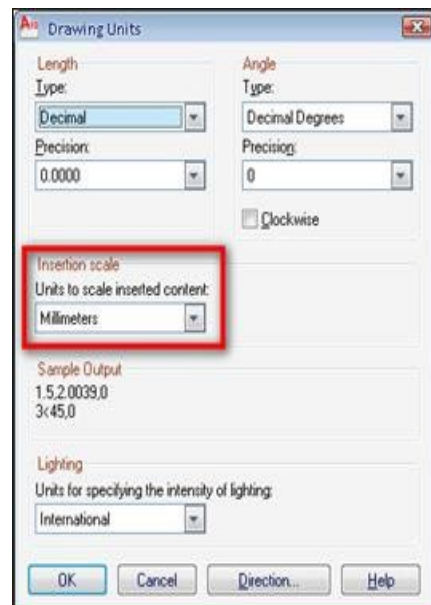
Fig 2.1 Drawing units window

Lab Tasks

Grid, Snap and Zoom Settings:

Once you have set the drawing limits in units you can turn on the grid and snap buttons on the status bar. This will help you to know your working area and turning on snap will help you move the cursor through fixed distance in x and y direction. With this you can accurately draw lines even with the mouse. Make sure that after turning on grid and snap you write zoom in the command line, press enter, write all and press enter again. This thing will help you to remove in screen errors that occur mostly in AutoCAD.

Turning On the required toolbars:

Make sure that the necessary toolbars are turned on such as Draw, modify, dimension and properties and toolbars. How to turn this toolbar ON/OFF is writing before.

Drawing a Line (LINE):

Either you can write "line" in the command line or press the first button in the drawing toolbar to activate the line command. Once you have activate the line command either you can use the mouse to select the starting and finishing points of the line or you can write the coordinates of the starting point in the command press enter, specify coordinate of finishing points, press enter and a line will be made on the working area. Make sure that the coordinate you have specified are inside the drawing limits you have set before the drawing.

Lab Task:

Draw the figure 2.2 in AutoCAD using decimal units and appropriate commands.

Fig 2.2

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____

LAB # 3: To describe and practice the basic commands I using AutoCAD

Objectives

- To describe the concept of different type of lines, Rectangle, Arc, Circle, Spline, Ellipse and Hatch.
- To practice the drawings in accordance to basic command I.

Pre Lab

Basic Commands I

- Line
- Construction line
- Polyline
- Rectangle
- Arc
- Circle
- Spline
- Ellipse
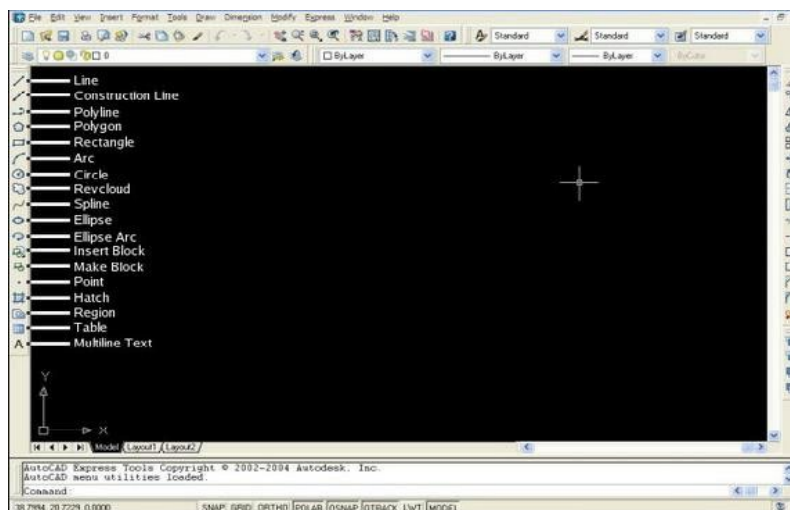- Hatch

Drawing Toolbar

Fig 3.1 AutoCAD home screen window

Lab Tasks

Line (line)
Either you can right "line" in the command line or press the first button in the drawing toolbar to activate line command. Once you have activated the line command either the coordinates of the starting point in command line, press enter, specify coordinates of finishing points, press enter and a line will be made on the working area. Make sure that the coordinates you have specified are inside the drawing limits you have set before starting the drawing.

Construction Line (xline)
This is a continuous line and extending to infinity from both sides. To activate this command, either type "xline" in the command line or just click on the 2nd button on the Draw toolbar. Now you have to specify 2 points in the command line or with the help of the mouse you can click the other point through which you want your construction line pass.

Poly line (pline)
To activate this command either write "pline" in the command line or click on the third button in the toolbar. Specify points either by clicking on the screen or by writing coordinates in the command line and pressing enter. When you want to finish the command press "ESCAPE".

Polygon (polygon)
This command helps you to make regular polygons, circumscribed or inscribed in a circle of specified radius. To make a polygon, type "polygon" in the command line or use the draw toolbar. First of all, you have to specify the number of points of that polygon. Type the number of points in the command line and press enter. Now specify the centre of the polygon/circle by clicking on the drawing area or writing the coordinates of the centre in the command line and pressing enter. Then you have to choose whether the polygon will be inscribed in the circle or circumscribed about that circle. In the command line write "I" for inscribed (All the corners of the polygons will be touching will be touching the circumference of the circle) and "C" for circumscribed (The circle will be inside the polygon) and press enter. Now you have to specify the radius of the circle. Either write it in the command line or use the mouse to set the radius. Using the mouse gives an advantage of changing orientation of the polygon.

Rectangle (rectangle)
Either write "rectangle" in the command line or use the draw toolbar to initiate this command. Specify the first point and then specify the other point opposite to the first point. You can either specify points with the click of your or by entering the coordinates of these two points in the command line.

Arc (arc)

Write "arc" in the command line to initiate this command. You have to specify 3 points to make an arc. You can either do it with your mouse or entering coordinates in the command line.

Spline (spline)
It is a continuous arc in which you keep on specifying the points like you do for and arc and to finish this command press the escape button.

Ellipse (ellipse)
To draw an ellipse, type "ellipse" in the command line or use the draw toolbar. First specify the end points of
the major arc and then you have to specify half the distance of the minor axis.

Hatch (bhatch)
When you want to highlight an area of your drawing you use hatch lines. These are slanting lines that the part of your drawing to be highlighted. To activate this command type "bhatch" in the command line and you will come across the screen shown in figure 3.2. When you have set the required type and pattern of the hatch line, press the "pick points" option.
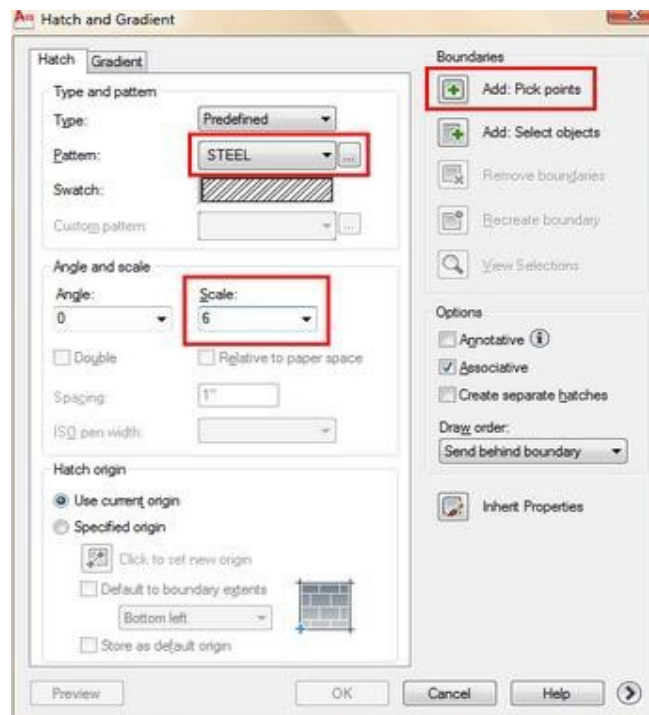
Fig 3.2 Hatch and Gradient setting window

Just click on the enclosed area of your drawing and then press enter, the following window will appear again. Now press ok and you will get the area hatched.

Multiline Text (MTEXT)
Type "mtext" to activate this command which helps you write text besides your drawing. First you have to specify 2 opposite points like you did for a rectangle and after you have done this write the text and change the text properties according to your requirements.
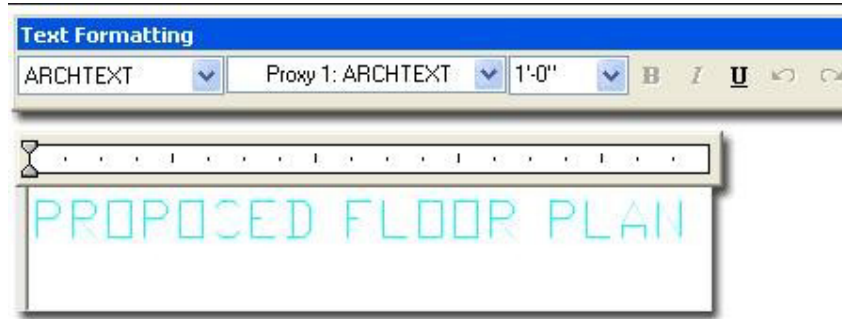
Fig 3.3 Text formatting window

Setting the Line Properties

You can change the type, colour, weight, properties of a line by double clicking it and editing the property box (given bellow) that appears after double clicking.
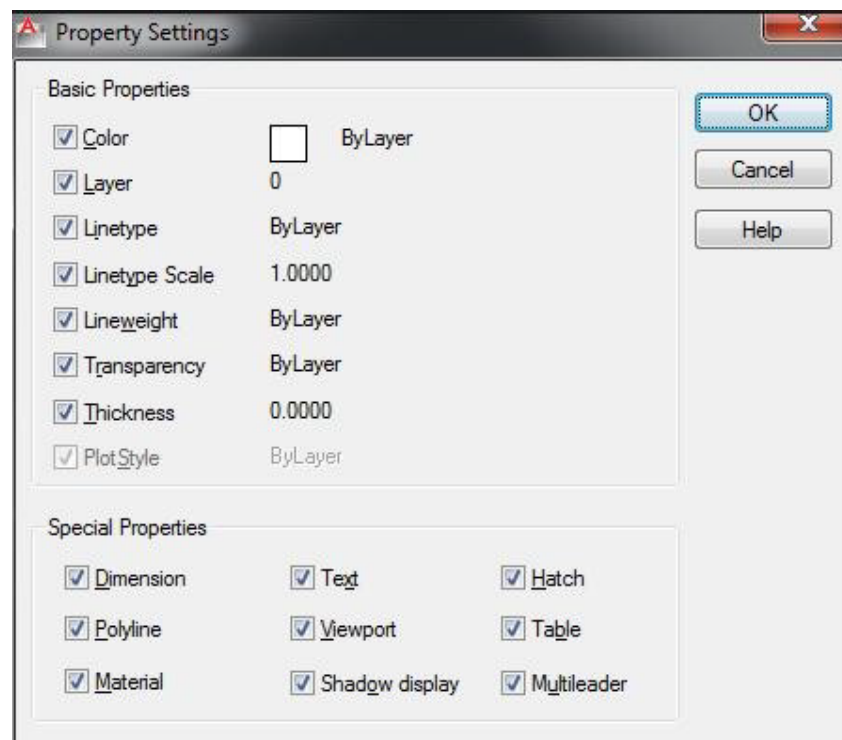

Fig 3.4 Line property settings window

You can change the type, colour, weight, properties of a line by double clicking it and editing the property box (given bellow) that appears after double clicking.

Or you can also select the line with your mouse and then change the properties from the properties toolbar shown below:

Fig 3.5 Layers line property settings

The first drop down menu is colour, second is line type and third is line weight. By layer means that this property of the line, will be the same as the property of the layer, in which the line has been made.

Using different Line Types:
To add a line type, use the above middle drop down menu and click "OTHER". Now in the line-type manage window click on the "LOAD" button. In the Load or re load lane-types window select "DASHED2" and "CENTER2" line types turn by turn. You can now see both of these line types in the line-types manager window. Press OK and after that you will be able to select both these line types in the above drop down menu.

Lab Task 1:

- Draw the first sheet that you have drawn manually, doubling the drawing limits to 48 by 80 and every measurement that you had in the lab exercise.
- Replicate the boundaries you made in the lab exercise and make the following figures in the square boxes.
  - ➢ Circle of the radius 12mm
  - ➢ Square of sides 20mm
  - ➢ A triangle with angles 30,60 and 90
  - ➢ A triangle with angles 90,45 and 45
  - ➢ A square with its 4 corners at the mid points of the sides of any square box.
- Write your name, roll number, drawing number, drawing title (BASIC SHAPES), date and your section in these six rectangles formed below
- Make centre lines (centre 2) in the circle you have made and change the line type of the square to dash2
- Save your drawings as "your name" _ "roll number" _A3.

Lab Task 2:

- Draw an equilateral triangle having side of 4.5m and an angle of 60°. Also draw three circles of 1.6 m diameter each at three corners of the triangle.
- Hatch the circles at the corner of triangle with appropriate pattern.

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____

LAB # 4: To practice the basic commands II and sketch the detailed dimensioned drawings using AutoCAD.

Objectives

- To demonstrate the concept of different dimensioning modes and construct multi view drawings using AutoCAD.
- To practice the properly formatted drawings and modify different dimension styles.

Pre Lab

Dimensioning Toolbar
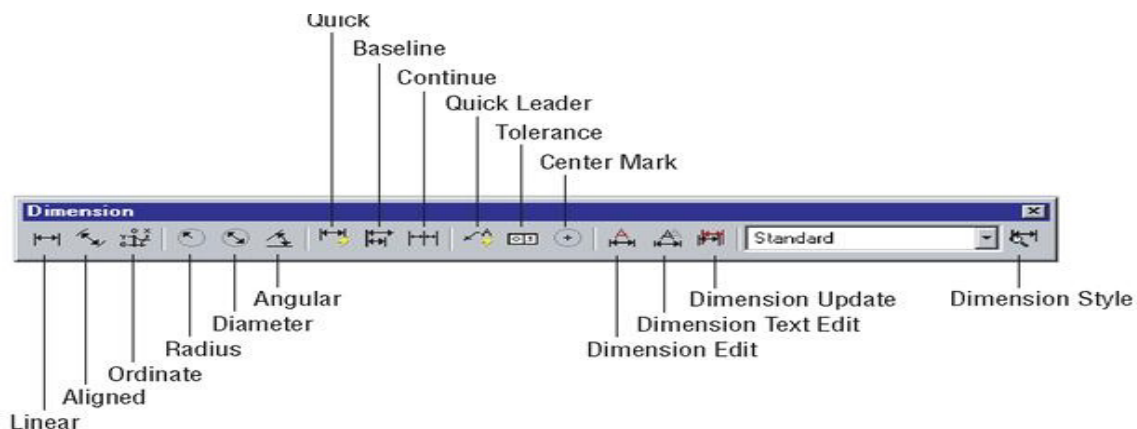
Fig 4.1 Dimension tool bar

While dimensioning please keep the OSNAP button on, So that you are able to specify the start and end points of length you want to dimension accurately.

The command syntax for the following dimensioning commands is specified in the brackets.

Linear dimension (dimlinear)
Type "dimlinear" to dimension horizontal and vertical length. Specify the end point s with your mouse or type their coordinates in the commands line. Now extend the dimension lines with your mouse through appropriate distance from the part you are dimensioning and click again.

Aligned dimension (dimaligned)
This dimension can be activated by typing "dimaligned" in the command line. This dimensioning is used to dimension inclined lines. The procedure is same as the linear dimension command.

Ordinate dimension (dimordinate)
Type "dimordinate" in the command line and press enter to activate this command. This is used to label the x or y coordinates of any point on your drawing. Just specify that point and use your mouse to label the x and y coordinate.

Radius dimension (dimradius)
Just click on the circle and then use your mouse to select the area where you want the radius dimension to be placed.

Diameter dimension (dimdiameter)
Same as you dimension the radius of a circle.

Angular dimension (dimangular)
First specify the first line and then the second line. Move the dimension to respectable distance and click to get the dimension.

Quick dimension (qdim)
This is required when you have to quickly dimension more than one length in a line. You just have keep selecting all the points from the start to the end and then move the dimension lines through a respectable distance.

Quick leader dimension (qleader)
You can use QLEADER if you want to annotate (label or write something about) any part of your drawing. Just activate the command and select the part of your drawing which you want to describe and then click again at the point where you want to write.

Centre mark dimension (dimcenter)
This gives the centre of the circles. Just click in the circle and the centre will be visible.

Lab Tasks

Lab Task 1

Setting of dimension style (dimstyle)

To apply the current dimension style to existing dimensions
1. From the Dimension menu, choose update.
2. Select the dimensions to Update to the current dimension style.
3. Press ENTER.

To restore a dimension style
1. From the Dimension menu, Choose Style.
2. In the Dimension Style Manager, select the dimension style to restore and choose Set Current.
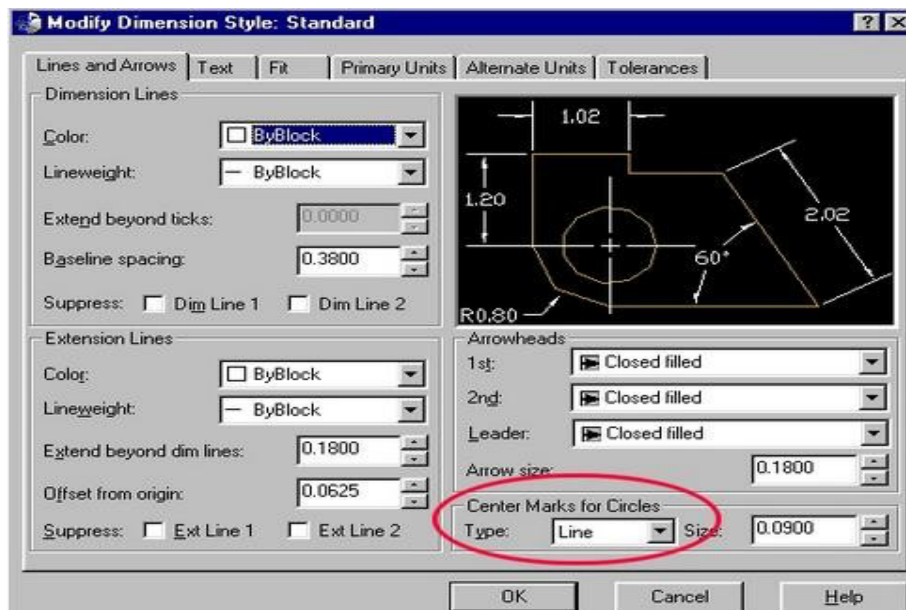3. Choose Close

Fig 4.2 Modify dimension window

15

Lab Task 2

- Draw the diagram mentioned in Fig 4.3 in AutoCAD using appropriate commands.
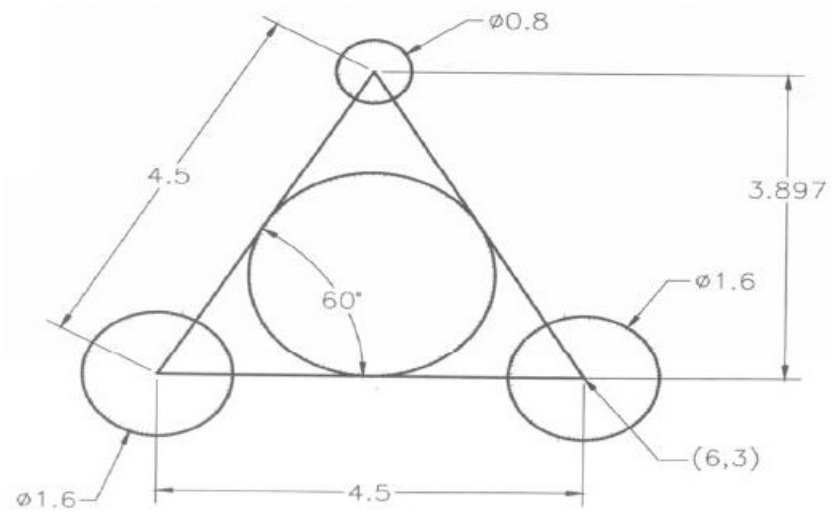- Dimension the entire Figure.

Fig 4.3

Lab Task 3

- Draw the diagram mention in Fig. 4.4 in AutoCAD using appropriate commands.
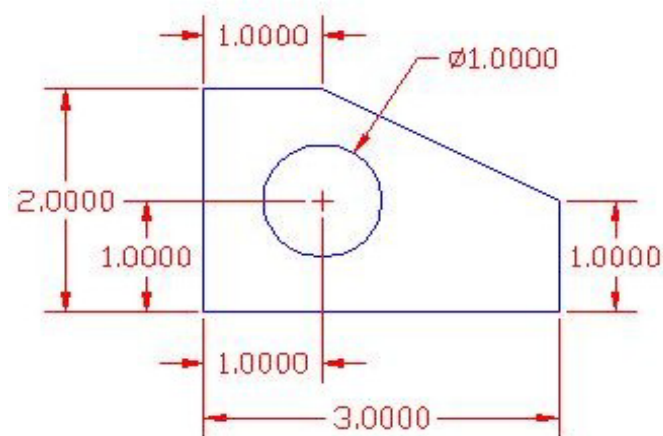- Dimension the entire drawing according to Fig. 4.4.

Fig 4.4

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____

LAB # 5: To describe the modify commands I and practice the drawings using AutoCAD

Objective:

- To construct duplicate objects with the help of Copy, Offset, Mirror and Array commands and rotate the object with the change of angle using AutoCAD.
- To practice the drawings through modify command II using AutoCAD.

Pre Lab

Modify Commands I

- Mirror
- Copy
- Array
- Offset
- Rotate

To mirror object

- From the Modify menu, choose mirror.
- Select the object to mirror.
- Specify the first point of the mirror line.
- Specify the second point.
- Press ENTER to retain the original object, or enter y to delete them.

To copy an object

- From the Modify menu , Choose Copy
- Select the object to copy
- Specify the base point
- Specify the second point of displacement.

To offset an object by specifying a distance.

- From the Modify menu choose Offset
- Specify the offset distance. You can enter a value or use the pointing device
- Select a point on the side where you want to place the new object
- Select another object to offset, or press ENTER to end the command.

To offset an object through a point

- From the Modify menu ,choose Offset
- Enter t (through).
- Select the object to Offset.
- Specify the through point.
- Select another object to Offset or press ENTER to end command.

Lab Tasks

To create a rectangular array
- From the Modify menu, choose array
- In the Array dialog box, choose Rectangular Array.
- Choose select objects

The array dialog box closes and AutoCAD prompts for object selection.

- Select the object to be arrayed and press ENTER.
- In the Rows and column boxes, enter the number of rows and column in the array.
- Specify the horizontal and vertical spacing (offset) between objects by using one of the following methods:
  - ➢ In the row offset and column offset boxes enter the distance between rows and between columns. Adding a plus sign (+) or a minus sign (-) determines direction.
  - ➢ Click the pick both offset button to use the pointing device to specify the diagonals corners of the cell in the array. The cell determines the vertical and horizontal spacing of the rows and column.
  - ➢ Click the pick row offset or pick column offset button to use the pointing device to specify the horizontal and vertical spacing.

The example box displays the result.
- To change the rotation angle of the array, enter the new angle next to angle of array.
- The default angle 0 direction setting can also be changed in units.
- Choose OK to create the array

To create a polar array.
- From the Modify menu, choose array.
- In the array dialog box point, do one of the following:
- Enter an X value and a Y value for the centre point of the polar array.
- Click and pick centre point button. The array dialog box closes and AutoCAD prompts for object selection. Use the pointing device to specify the centre point of polar array.
- Choose select object.

The array dialog box closes and AutoCAD prompts for object selection.
- Select the object to be arrayed.
- In the method box ,select one of the following method:
- Total number of item and angle to fill.
- Total number of items & angle between items.
- Angle to fill & angle between items.
- Enter the number of items (including the original object) , if available.
- Use one of the following methods.

  - ➢ Enter the angle to fill and angle between items, if available. Angle to fill specifies the distance to fill around the circumference of array. Angle between items specifies the distance between each item.

> Click the pick angle to fill button and the pick angle between items button and use the pointing device to specify the angle between items.

The example box displays the result. You can set any of the following options:

- To rotate the object as they are arrayed, select rotate item as copied. The example area displays the result.
- To specify the X and Y base point, choose more , clear the set to object 's default option and enter the values in the X and Y boxes , or the pick base point button and use the pointing device to specify the point.
  Choose OK to create array.

To rotate an object

- From the Modify menu, choose Rotate
- Select the object to rotate.
- Specify the base point for the rotation.
- Do one of the following
- Enter the angle of rotation.
- Drag the object around its base point and specify a point location to which you want to rotate.

To rotate an object using a reference angle

- From the Modify menu choose, rotate.
- Select the object to rotate.
- Specify the base point for the rotation.
- Enter r (reference)

Now define the reference and the new angle by selecting the objects you are aligning.

- Enter in (intersection objet snap), and select the intersection point (2) to begin defining the reference angle.
- Enter end (end point object snap), and select end point of the object you are rotating (3) to complete the definition of the reference angle.
- Enter end again, and select the end point of the object you are aligning to (4).

Lab task:

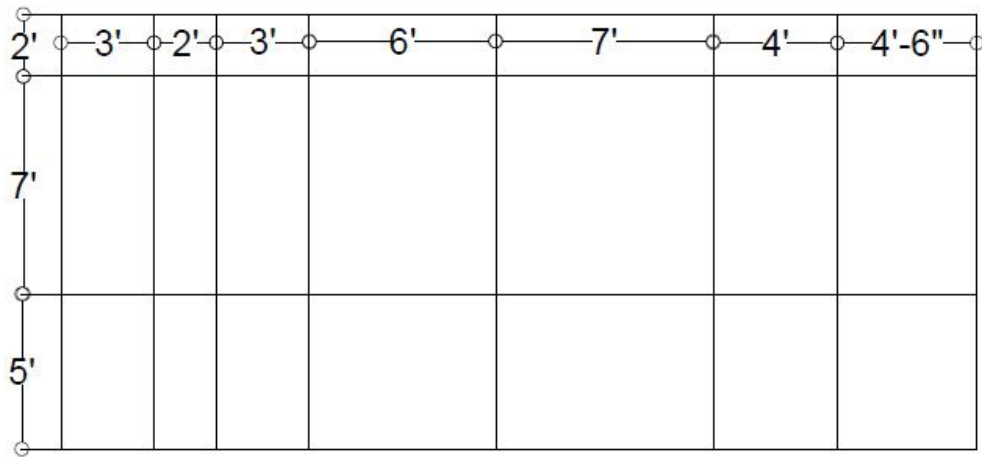1. Draw the diagram in Fig. 5.1 using AutoCAD through appropriate commands.

Fig 5.1

2. Design the drawing mention in Fig. 5.2 in AutoCAD using appropriate commands.
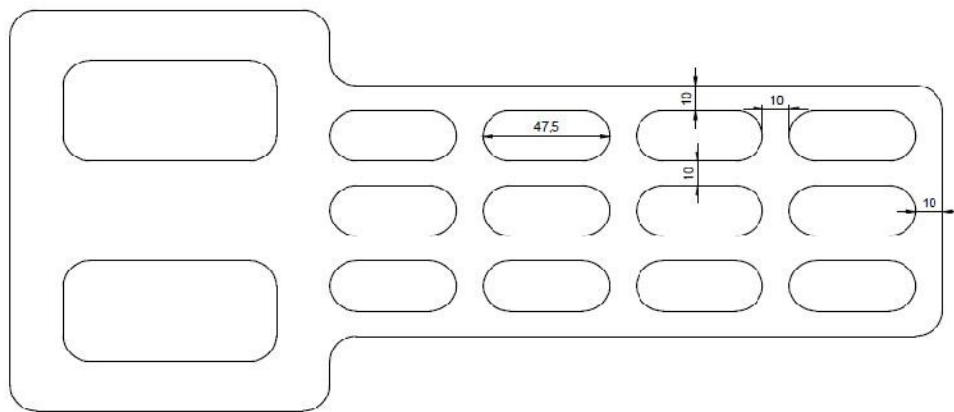

Fig 5.2

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____

LAB # 6: To sketch and practice the structure of Electrical drawing using AutoCAD.

Objectives

To construct and practice the single-line diagram (SLD) of electrical system using AutoCAD.

Pre Lab

An **electrical drawing**, is a type of technical drawing that shows information about power, lighting, and communication for an engineering. Any electrical working drawing consists of "lines, symbols, dimensions, and notations to accurately convey engineering's design to the workers, who install the electrical system on the job. Electrical drafters prepare wiring and layout diagrams used by workers who erect, install, and repair electrical equipment and wiring in communication centres, power plants, electrical distribution systems, and buildings.

One-Line Diagrams:

One-line diagram – a diagram that uses single lines and graphic symbols to indicate the path and components of an electrical circuit. One-line diagrams are used when information about a circuit is required but detail of the actual wire connections and operation of the circuit is not.

Lab Tasks

Fig 6.1

23

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____

LAB # 7: To sketch and practice the Architecture drawing using AutoCAD.

Objectives

To sketch the detailed dimensioned house plan using AutoCAD.

Pre Lab

An **architectural drawing or architect's drawing** is a technical drawing of a building (or building project) that falls within the definition of architecture. Architectural drawings are used by architects and others for a number of purposes: to develop a design idea into a coherent proposal, to communicate ideas and concepts, to convince clients of the merits of a design, to enable a building contractor to construct it, as a record of the completed work, and to make a record of a building that already exists.

The development of the computer had a major impact on the methods used to design and create technical drawings, making manual drawing almost obsolete and opening up new possibilities of form using organic shapes and complex geometry. Today the vast majority of drawings are created using CAD software.

Lab Tasks

Fig 7.1

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____

LAB # 8: To construct and practice the cross-sectional view of Mechanical drawing using AutoCAD.

Objectives

To construct the properly formatted cross sectional view of gear using AutoCAD.

Pre Lab

Mechanical systems drawing is a type of technical drawing that shows information regarding to the different parts/jobs/tools of a particular system. It is a powerful tool that helps analyze complex systems. These drawings are often a set of detailed drawings used for construction projects

Lab Tasks

Fig 8.1

27

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____

LAB #9: An introduction to PCB designing using Proteus

Objectives

In this Lab you will be able to learn

✓ To design the circuit on schematic diagram.
✓ To analyze the circuit from the schematic diagram.
✓ To implement the PCB design of the schematic circuit in ARES

Pre Lab

Introduction

PCB stands for Printed Circuit Board. The naming convention will be clear once steps for the design are understood. On a lower level of project, PCBs are usually designed on a board whose one side is lined with copper. But on the industrial scale or on a professional level, it is preferred to have a double sided PCB. This also complexes the procedure through which PCBs are made.

Proteus ISIS

Open the 'ISIS Professional' from PROTEUS. This is the application where the simulations of the circuits can be tested. But the same file can be further processed to transform it into a layout. Layout is the final design which is needed in order to make the PCB of a circuit. To make the schematic, first we must have its raw design. Below is the schematic, that this document uses to explain the steps to make the PCB.
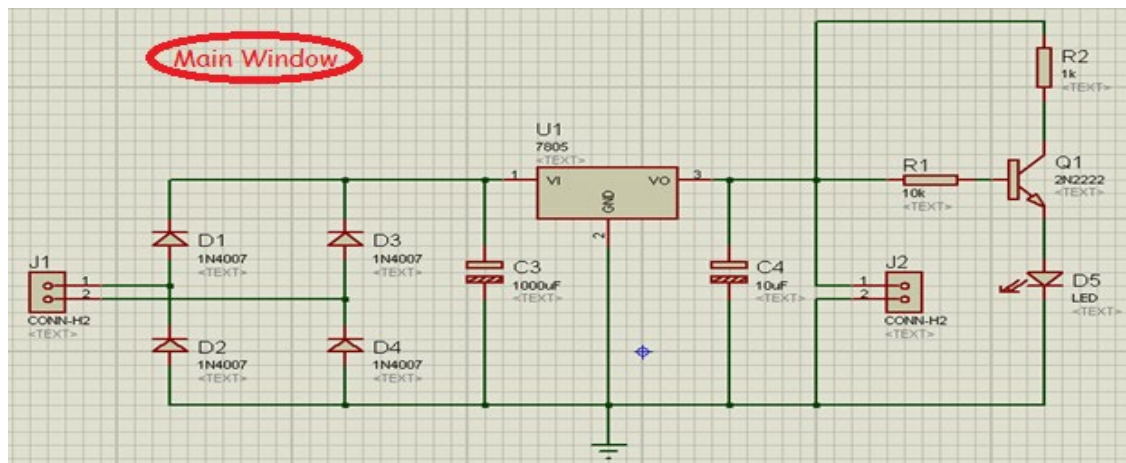
Figure 9.1

Making Schematic

To make the schematic, first add the suitable components in the 'Component Mode' pane

This can be done by selecting the 'Component Mode' icon indicated by the arrow number 1 in Figure-2. The 'PANE' will be empty in the start. To add the components, click on the 'P' indicated by red square in the figure, and then type in the name to add the appropriate component. Fill in all the components
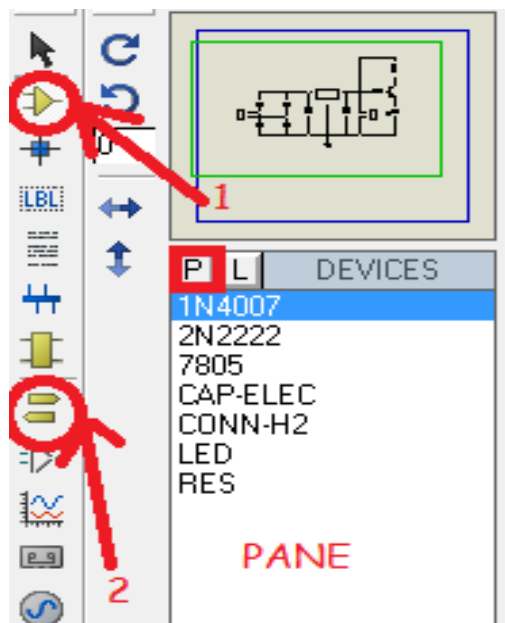
29

Figure 9.2

required to complete the desired schematic. In our case, add the components as shown by figure to proceed further. The icon indicated by arrow number 2 is 'Terminals Mode'. This contains the 'Power' and 'Ground' terminals required in the circuit. The 'Ground' terminal must be connected if it is required to generate a Power Plane. Power Plane will be explained later in the document. Now, to place a component in the main window (right to the pane), just select the component from the pane. Then, click once in the main window. The component will appear instead of the 'pencil' pointer. Move the component to an appropriate location and then place it by clicking once more. To place a wire between the components, simply click on one of the component's end using the 'pencil' pointer. Then, click on other's component end. In this case, click and hold is not necessary.

Adding Footprints:

Adding or editing the footprints is the most crucial and important step in the making of the PCB. But before this, let us discuss how to change the values of the components in the schematic. To do this, double click on the text attached to the component placed in the main window. This will bring another window, so that the change in the component value can be made. To add or edit the footprint, double click on the component instead of the text attached to the component. A window similar to following figure will be opened.
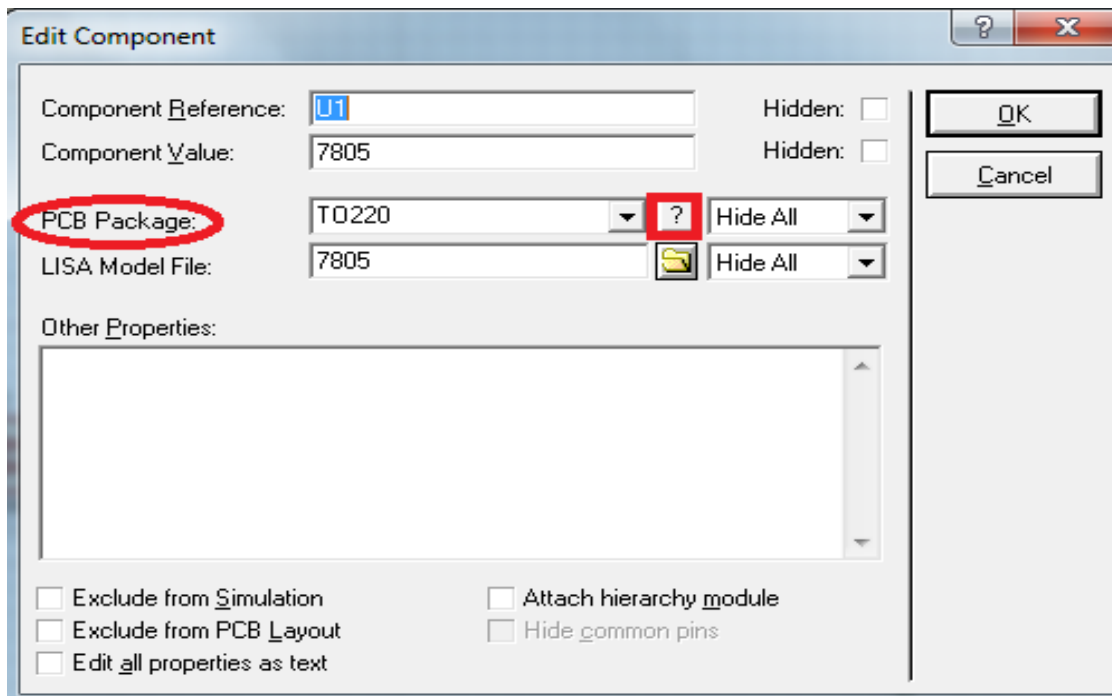

Figure 9.3

If the default footprint is correct, don't bother to change it. But if it is not, click on the question mark indicated by the red square in Figure-3 to change it. The footprint is searched by first erasing the actual footprint, then by choosing the appropriate category, type, and sub-category. The most important issue in dealing with the footprints is its compatibility between the software and the real-world components. To be certain of this, extra work is required before even starting the PCB designing on the software. For this schematic, just add the footprints of the components as follows:

➢ CONN-H2 > TBLOCK-I2. Category: Connectors, Type: Through Hole, Sub-category: *Terminal Blocks*
➢ DIODE > DO41. Category: Discrete Components, Type: Through Hole, Sub-category: *Diode*s
➢ CAP > ELEC-RAD75M. Category: Discrete Components, Type: Through Hole, Subcategory: *Radial Electrolytics*
➢ CAP > ELEC-RAD20M. Category: Discrete Components, Type: Through Hole, Subcategory: *Radial Electrolytics*
➢ 7805 > TO220. Category: Discrete Components, Type: Through Hole, Sub-category: *Transistors*
➢ RESISTORS > RES40. Category: Discrete Components, Type: Through Hole, Sub-category: *Resistors*
➢ 2N2222 > TO92-50. Category: Discrete Components, Type: Through Hole, Sub-category: *Transistors*
➢ LE > SOD93. Category: Discrete Components, Type: Through Hole, Sub-category: *Diodes*

Now, on this stage the simulation of the circuit can also be run. But for the current schematic, the software will not be able to generate any results, as the circuit is incomplete without a transformer which is left only because it doesn't need to be included in the PCB design. All the steps in the 'ISIS' are finished. To proceed with the making of the PCB, we just need to transfer the 'netlist' to 'ARES'. To do this, click on 'Tools'>'Netlist to ARES' or press Alt+A or just press the ARES icon on the right most side in the tab above the main window.

Proteus ARES:

'ARES Professional' will open automatically once the previous step is done. This is the application where the final layout will be made. Once the layout is made, the work on the software will be finished. Proceed with the following steps to make the PCB layout.
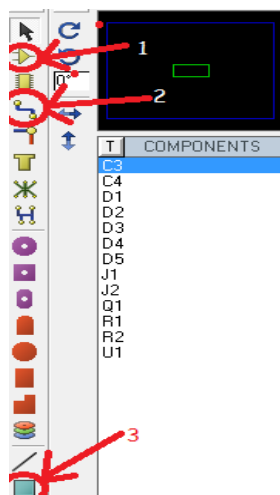
Making Schematic in ARES:

Now, this may seem hectic to make the schematic again but actually it is easier than in the 'ISIS'. The 'PANE' in the 'ARES'. The icon indicated by arrow number 1 is 'Component Mode' that reveals the components pane. Arrow number 2 indicates 'Track Mode' which will be used for routing. Arrow number 3 indicates '2D Graphics Box Mode' which will be used for Power Plane Generation. Now, first select the 'Component Mode' and start placing components in the window. Make the schematic similar to the one in 'ISIS'. Notice that as the components are placed in the main window, they start disappearing from the components pane. Also, notice that when the components that have the interconnections are placed in the main window, they show the green lines and yellow arrows like vectors. The vectors can be annoying a bit. To remove them, go to View > Layers and uncheck the options vectors' or 'force vectors'. Complete the schematic before starting the routing.

Figure 9.4

Routing:

Once the schematic is complete, the next step is to route the track. The kind of routing discussed in this document is manual routing. However, automatic routing can be done. For this go to Tools > Auto Router under the option 'Auto Placer'. 'Auto Placer' makes the schematic and places the components by itself on the main window. But the arrangement of the components done by 'Auto Placer' might not be satisfying, so may be the case with the 'Auto Routing'. But 'Manual Routing' has certain advantages to 'Auto Routing'. The routes and their thickness are chosen as desired. 'Auto Routing' does not cause much trouble for smaller circuit but for larger circuits it is not recommended. To do 'Manual Routing', first of all uncheck Tools > Auto Trace Selection. This prevents from selecting default track for every route. Then select 'Track Mode' as explained by Figure-4. Choose 'T50' from 'Traces PANE' and make routes between the component's pins just like the wiring was done in schematic. Place pencil on one of the component's end, click and release, then take it to another component's end and click there. Click on the second component's end very carefully otherwise the route won't finish. A route of blue color appears. If instead of the first single click, a double click is made, the routes color changes to red. To change it to blue, double click again. The red color is for the 'Top Layer' routing and the blue color is for the 'Bottom Layer' routing. And since the PCB in progress is single bottom layer PCB, all the tracks must be of blue color. T50 is the width of the track. The 50 associated is the width in 'mills'. 'Mills' is a unit frequently used by PCB making softwares. 1000 mills equal 1 inch and 1 inch equals 2.54 cm or 25.4 mm. Using T50 track means we are actually using 1.27 mm wide track. The desired width can be converted from 'mm' to 'mills'. Using T50 layer, make all the routes. Leave the routes which were supposed to be ground and the routes with the BJT terminals. Also, make sure that the routes do not intersect each other. If this happens, the copper in actual PCB will make short connections that were not desired in the circuit. Also, if a bypass is required, the symbols of the components can also be used. Take the route under a components symbol and the route will be completed without intersecting any other route. Use T15 track for the BJT components. The T15 track's width is kept in accordance with the BJT pins' width and the distance between these pins. The T50 track is wide enough that it will be merged when connected to BJT's pin. The T15 track has a width of 0.381 mm and the BJT pins' width is 0.56 mm. T20 track could also have been used here. Leave the ground wires to create the Power Plane discussed in the following section. Now, the schematic might resemble like the figure below.
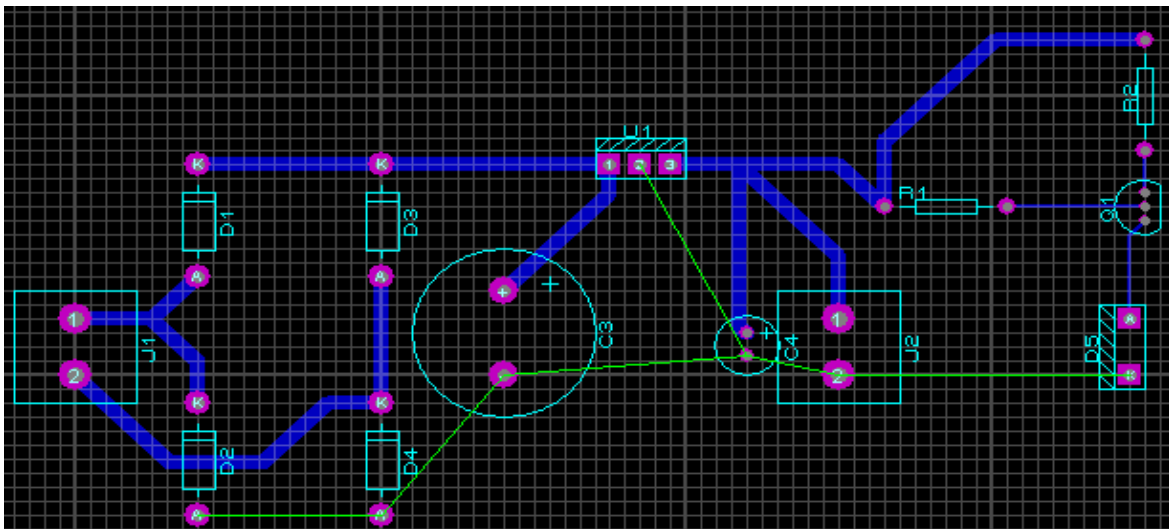

Figure 9.5

Power Plane Generation:

After all the rest of the tracks are complete, the only one left is the ground track. Instead of making a simple track for the ground, it is preferred to make it a Power Plane. The trick is to use all the rest of the copper on the PCB, except the tracks, as ground. This way, the node voltages are very easy to measure which implies the correct circuit. One doesn't need to find the ground this way. To create Power Plane, create a board first. The board is the actual width and length of the PCB that is desirable. Select the '2D Graphics Box Mode' highlighted in Figure-4. There will be a drop down button menu at the bottom left of the ARES window with 'Top Copper' written within it. Expand this button and select Board Edge. Create a Box on the main window which covers all the components. The width and the length in mills can also be read at the bottom of ARES window. This width and length is very important as it decides the actual size of the PCB in hardware. Then go to Tools > Power Plane Generator. A window with 4 options will appear. The first option is 'Net' with drop-down button. Click the button and select the 'GND=Power' net. If there isn't any available, this means that during the schematic, the ground is missed. Now, the easy thing about Proteus is that there is no need of starting all of it from the top. Just go to the ISIS schematic. Add a ground, connect it and select the 'Netlist to ARES' option as discussed at the end of section 2.2 of this document. Now, in 'ARES' again select the 'Power Plane Generator' option. Now, the option 'GND=Power' will be available. Select the option, and then in the 'Layer' option, select 'Bottom Copper'. Let the 'Boundary' be 'Default'. Also, the option given to 'Edge Clearance' by default i.e. '25th' is fine. This is the clearance from the board edge. If the 'OK' button is available, click it otherwise press 'Enter'. The Power Plane is now created. To delete the Power Plane, place the mouse pointer on the inner side of the PCB board drawn and find the location where the mouse pointer icon changes into a 'hand' icon. Then right click once, a list will appear, select delete object or right click twice to delete. The only annoying thing after this will be the track to 'Power Plane' clearance. This can be changed accordingly by selecting Technology > Design Rules. Change the 'Trace-Trace' clearance to change the above width. Now, the schematic will look like Figure-6. To print the layout, go to Output > Print, uncheck all the layers except 'Bottom Copper' and to reassure that the footprints doesn't change, match the type of the paper, on which the print is going to be, from printer options e.g. A4. If printer is not available on the same PC, and Proteus is not installed in the PC with the printer. Take out a print in PDF form by utilizing softwares which allow the prints to be converted into PDF. But, be cautious, and before printing on actual paper, again select A4 or the type of paper, the print is going on to be.

Figure 9.6
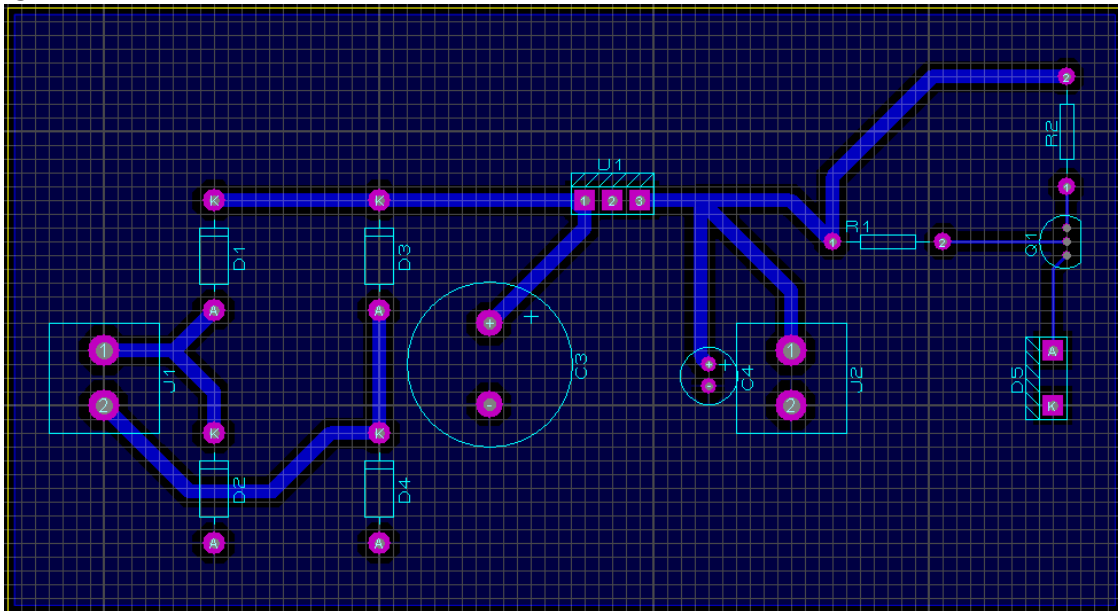
Making PCB:

The print must be printed on a copper board. This can't be done directly. Also, simple A4 paper won't help. The print must be taken on 'Glossy Sheet' commonly known as 'Sticker Sheet' or 'Sticker Paper'. One side of the sticker sheet is rough and may be white. The other side has a thin plastic sheet on it, very thin that is not easily noticeable. The other thin plastic sheet, if present, is to keep this side safe and is noticeable and can also be peeled. It should be peeled before the print. After taking the print on this side, press it against the copper side of the PCB board using Iron. Make sure that all the print is transferred onto the copper before removing the sheet. Because a circuit is printed on a board, that's why it is called Printed Circuit Board commonly called PCB. See Figure-7 for how the print is going to be. After the print is transferred onto the copper, darken the tracks using a Permanent Marker. If inconvenient do not use marker on the Power Plane. The print and the marker are going to act as insulation when the board will be dipped in FeCl3 solution. All the copper where there is no track or power plane must dissolve. The remaining copper are the wires of the circuit, they connect the components and make a whole circuit. The FeCl3 solution must be a 1:1 with water. Generally ¼ kg of powdered FeCl3 is dissolved in ¼ kg of boiling water. The boiling water helps the reaction of copper with FeCl3. 5 to 10 PCBs can be made using this solution. And if lesser copper is needed to be removed, the number of PCBs that can be made increases. After removing unwanted copper, clean the PCB and remove the marker or print using spirit. Then drill the PCB where the components pins are to be soldered. The drilling points will be prominent in the print and thus on PCB. Another terminology to be aware of in the PCB business is jumper. Jumper is simply a wire that can't be traced during the PCB. For it, drill extra holes and use an external wire and solder it to complete the circuit. The PCB will now be complete. Now, connect the circuit with the supply and test it. In this case the circuit is an AC to DC converter and the BJT circuit will allow us to know whenever the circuit is on i.e. LED will blink. So, for the first connector, connect it to an AC transformer that steps down a 220V AC into 12V AC and the output from the second connector will be 5V DC.
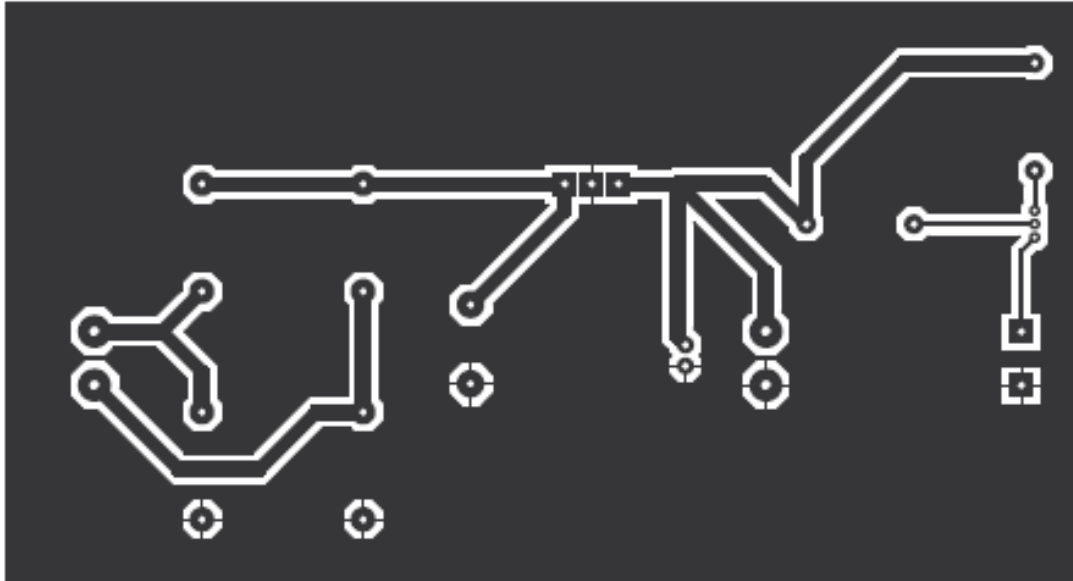

Figure 9.7

In Lab Task

Perform all the steps given above and build a PCD design of the circuit given in Fig. 9-1

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____

LAB # 10 To manipulate the working of LED flasher circuit using 555 timer and implement the PCB design of the schematic circuit in ARES

Objectives

In this Lab you will be able to learn

- ✓ To construct the circuit on schematic diagram
- ✓ To implement the PCB design of the schematic circuit in ARES

Pre-Lab (PCB Designing using Proteus ARES & 3D View)

Components Required

- ✓ Resistance x2 (10kohm).
- ✓ Capacitor 100 μF.
- ✓ 555 timer IC.
- ✓ LED (any color).

Procedure:

- Open Proteus 8 Professional from start menu.
- Create a New Project from File Menu.
- Bring Components that are needed from Component manger by clicking the P sign.
- Make the connections as shown in the given circuit.
- Open the ARES tab in Proteus.
- Bring the components by clicking on component manager.
- Make connections using top copper layer and bottom layer as they are made in the Schematic design. 8. To view 3d view of the circuit board click on the 3D Visualizer.

Circuit Diagram.
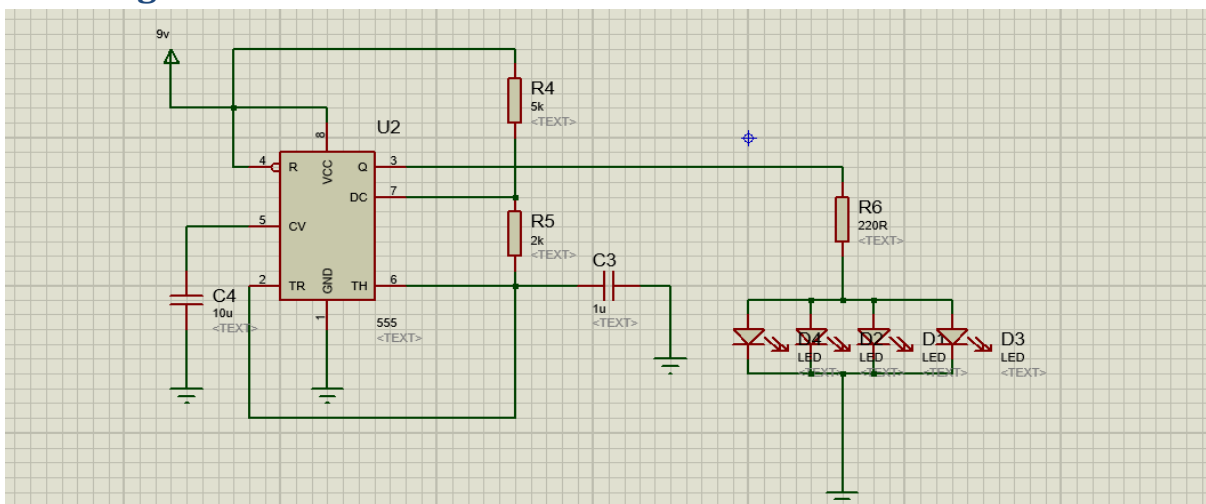
Figure 10.1
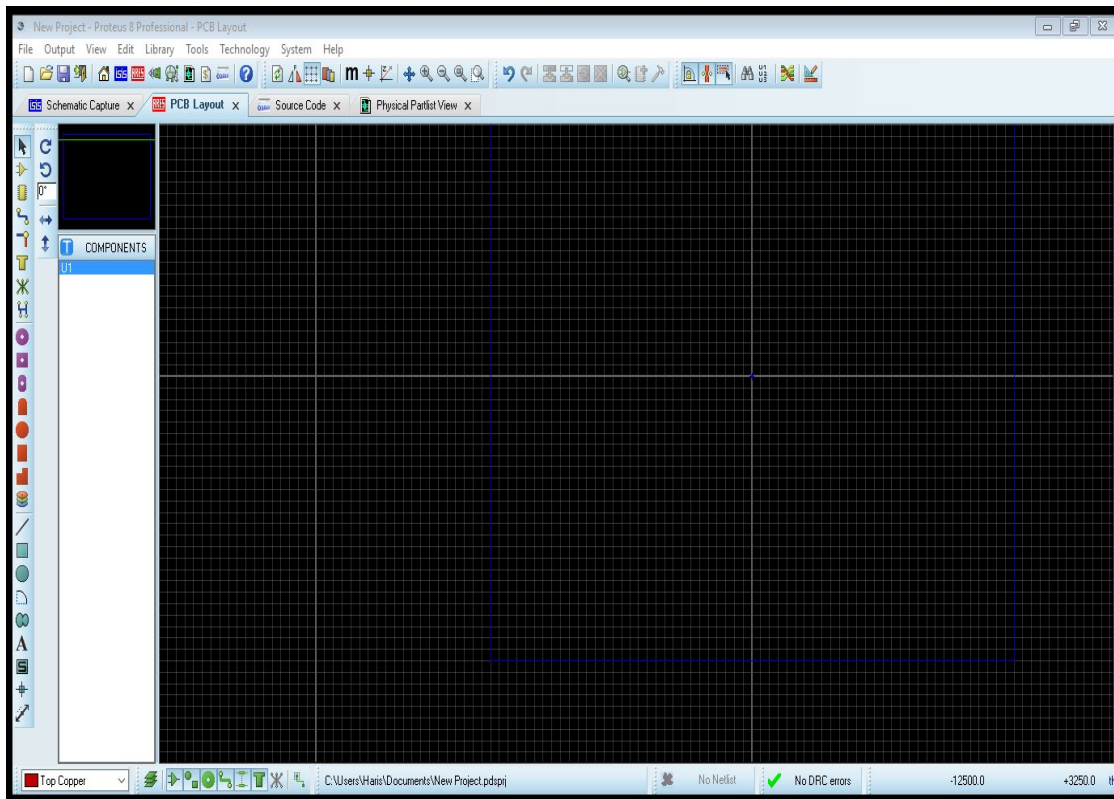
Figure 10.2
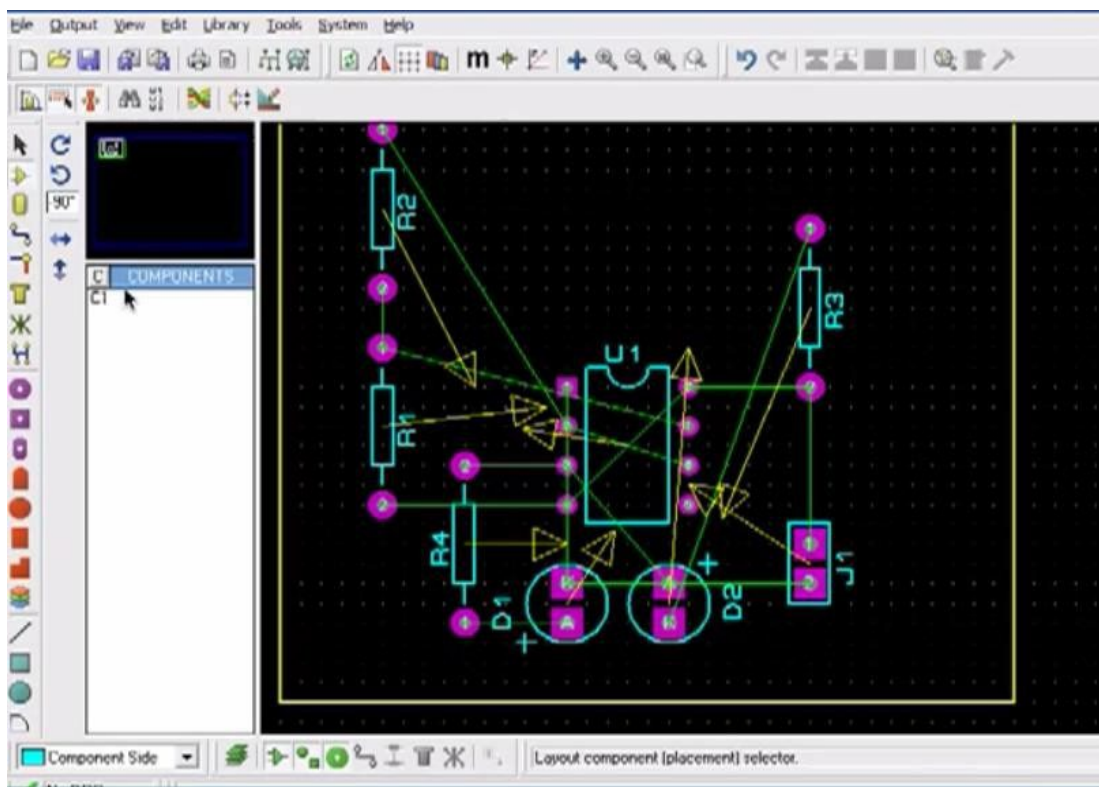
Figure 10.3

In-Lab Task:

An LED Flasher Circuit using 555 timer is shown in Figure 10-4.
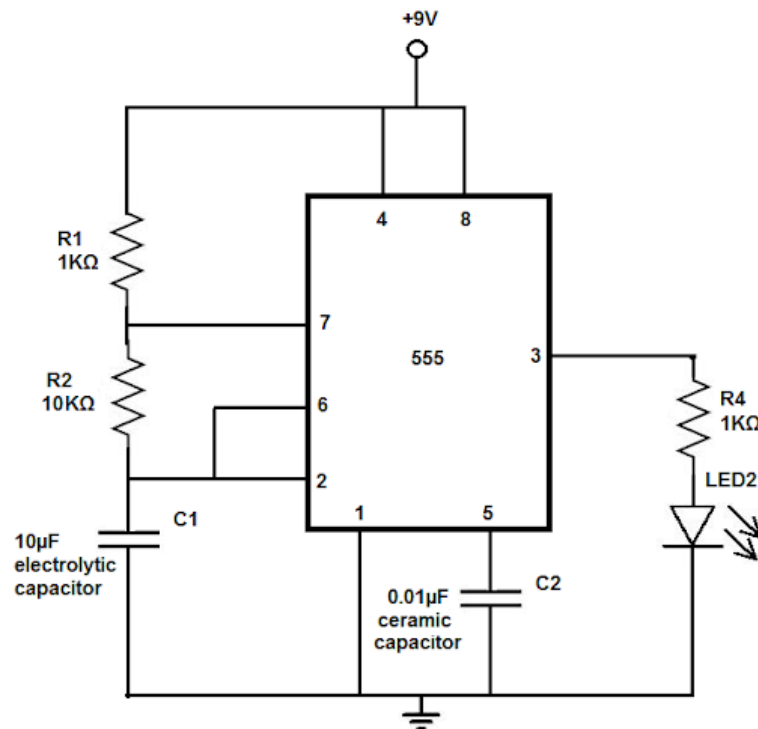
Figure 10.4

Complete the following steps to complete the above design tasks

- ➢ Draw the schematic diagram of LED Flasher Circuit
- ➢ Implement the PCB design of the schematic circuit in ARES
- ➢ Attach a complete PCB design circuit of LED flasher circuit.

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____

Lab # 11 To manipulate the working of a 12 V DC Power Supply using schematic approach and implement the PCB design of the schematic circuit in ARES

Objectives
In this Lab you will be able to learn
- ✓ To construct the 12V DC power supply circuit on schematic diagram
- ✓ To implement the PCB design of the schematic circuit in ARES

Pre-Lab

Introduction
DC Power Supply: Electricity comes in two main forms: alternating current (AC) and direct current (DC). DC current has the flow of electricity going in only one direction (forwards), whereas AC current has the electricity going in two directions (backwards and forwards). DC current is easier for smaller devices to utilize and is the most common method of power supply for anything that runs on battery power as well.

Power supplies: Most electrical power from a wall outlet comes in alternating current form

AC to DC conversion: Alternating current must be converted to direct current (DC) by a rectifier.

Rectifier: The bridge rectifier is one of the most widely used rectifier circuits. It offers a high level of performance when compared to other rectifier circuits. In view of the high level of use of the bridge rectifier circuit, bridge rectifiers are available as blocks containing all the diodes contained in a single "component" for use within the target circuit assembly.
The use of these bridge rectifiers simplifies their use and the assembly of the final equipment as well as reducing the overall component count.
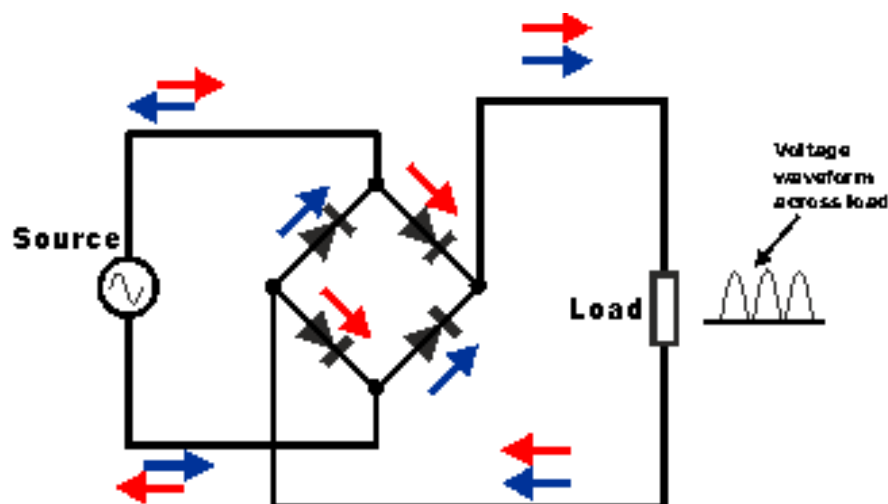
Figure 11.1

Bridge rectifier circuits: A diagram of the basic bridge rectifier circuit is shown below. The circuit has the advantage over the full wave rectifier using a centre tapped transformer that there is no centre tapped transformer requirement and that the both halves of the cycle are used in the winding.
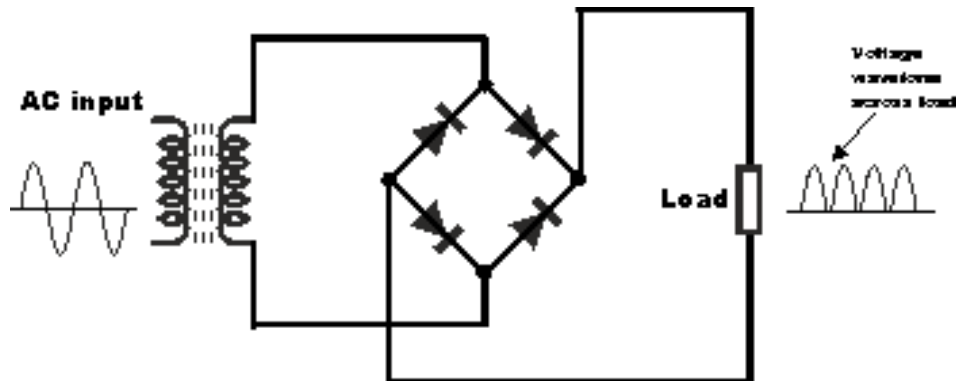

Figure 11.2

The above Figure shows the current flow for the different halves of the cycle and in the different arms of the bridge rectifier circuit.

In-Lab Task

A 12v DC power supply schematic is shown below
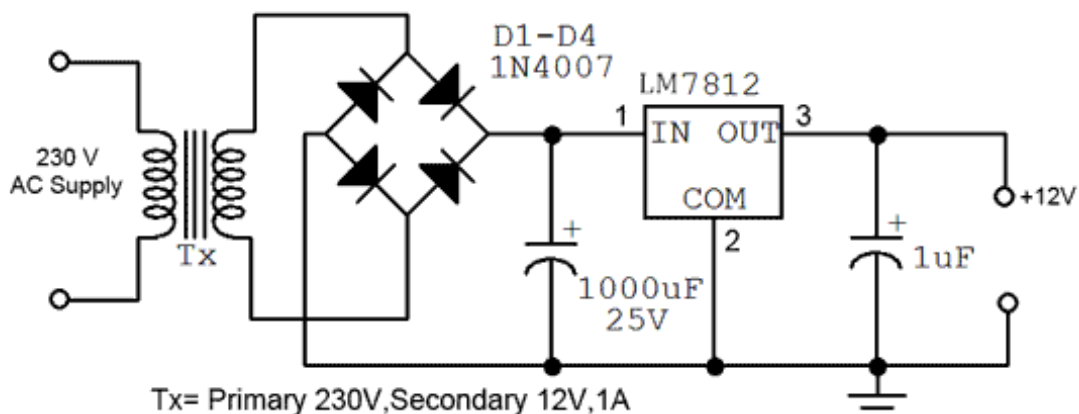
Figure 11.3

Complete the following steps to complete the above design tasks

➢ Draw the schematic diagram of 12V DC power supply in Proteus ISIS

➢ Implement the PCB design of the schematic circuit in ARES

➢ Attach a complete PCB design circuit of `12V DC power supply.

41

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ Date: _____

Lab # 12 NI Multisim as a Schematic Designing and Advance Circuit Simulation Software.

Objective:

To make circuit in NI Multisim.

Pre Lab

NI Multisim is a powerful schematic capture and simulation environment that engineers, students, and professors can use to simulate electronic circuits and prototype Printed Circuit Boards (PCBs). For the introductory example, you will simulate a standard non-inverting operational amplifier circuit (shown in Figure 1). The gain of this non-inverting amplifier is calculated by the expression Gain = 1 + R1/R2. Therefore, if R1 = R2, then the gain is equal to 2, which you will verify when you run interactive simulation in Multisim
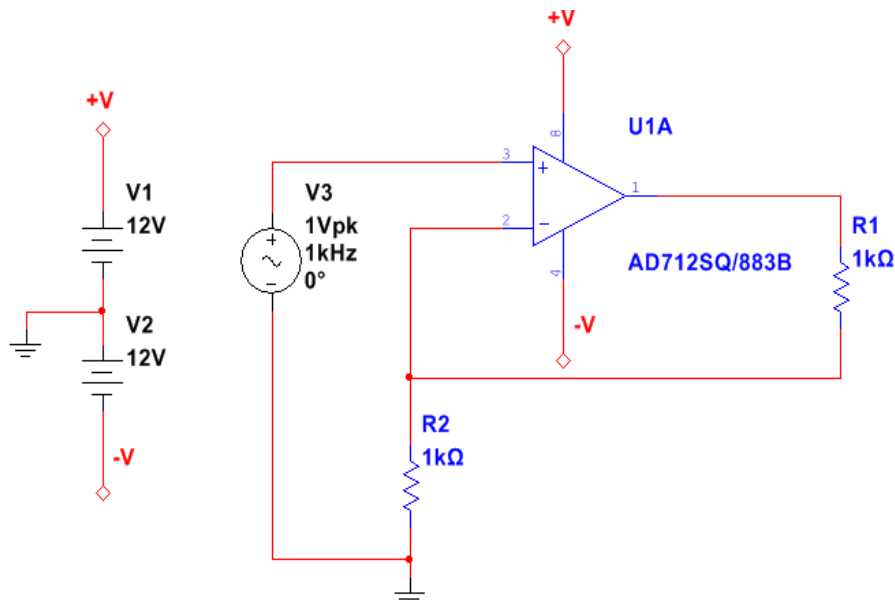
Figure 12.1 Non-inverting amplifier circuit

Part A: Selecting Components

Begin by drawing your schematic in the Multisim environment.

1. Open Multisim by selecting All Programs»National Instruments»Circuit Design Suite 13.0»Multisim 13.0.

2. Select Place»Component. The Select a Component window appears (also known as the Component Browser), as shown in Figure 2.
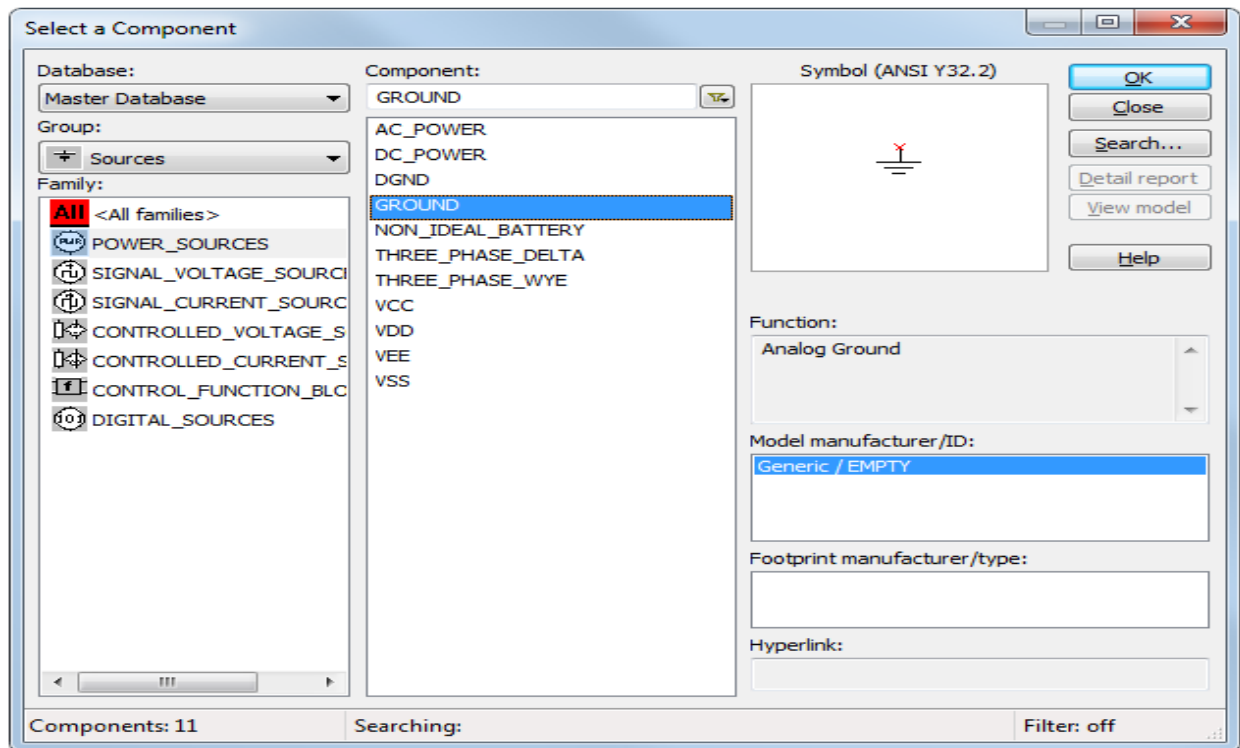
Figure 12.2. Select a Component window.

The Component Browser organizes the database components into three logical levels. The Master Database contains all shipping components in a read-only format. The Corporate Database is where you can save custom components to be shared with colleagues. Finally, the User Database is where custom components are saved that can be used only by the specific designer.

Additional Points

1. The components (or parts) are organized into Groups and Families to intuitively and logically group common parts together and make searching easier and more effective.

2. The Component Browser shows the component name, symbol, functional description, model, and footprint all in a single pop-up.

3. Select the Sources Group and highlight the POWER_SOURCES family.

4. Select the GROUND component (as shown in Figure 2).

5. Click OK. The Select a Component window temporarily closes and the ground symbol is 'ghosted' to the mouse pointer.

6. Move the mouse to the appropriate place on the workspace and left-click once to place the component. After placing the component, the Select a Component window will open again automatically.

7. Go to the Sources Group again and highlight the POWER_SOURCES Family (if not already highlighted from the previous selection).

8. Select the DC_POWER component.

9. Place the DC_POWER component on the schematic.

10. Repeat steps 7, 8 and 9 to place a second DC_POWER component.

44

Additional Points

- Without a power and ground your simulation cannot run.

- If you need multiple components you can repeat the placement steps as shown, or place one component and use copy (Ctrl+C) and paste (Ctrl+V) to place additional components as needed.

- By default, the Select a Component window keeps returning as a pop-up until you have completed placing your components. Close the window to return to the schematic entry window.

Now place the remaining circuit components using the techniques discussed in the previous steps.

11. Select the Analog Group and the OPAMP family.

12. Type AD712 in the Component field.

13. Select the AD712SQ/883B component, as shown in the next figure:
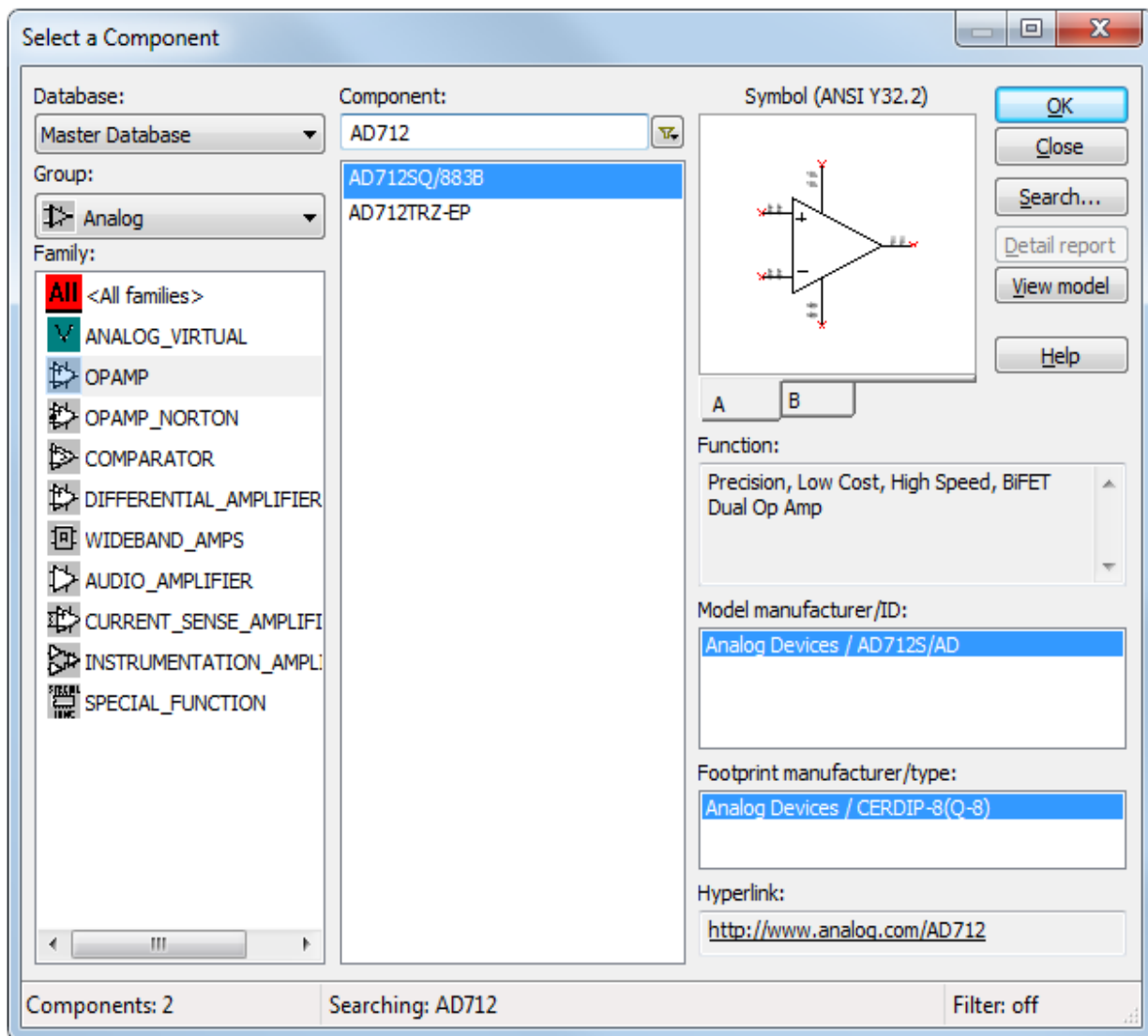
Figure 12.3. Selecting the operational amplifier.

Note that this component is a multisection component, as shown by the A and B tabs.

14. Place section A of the AD712SQ/883B component on the workspace area.

15. Return to the Select a Component window.

16. Select the Basic Group, Resistor Family.

17. Select a 1 kΩ resistor. In the Footprint manufacturer/type field, select IPC-2221A/2222/RES1300-700X250.

18. Place the resistor.

Note: you can rotate a component before placement by using the <Ctrl+R> shortcut on your keyboard when the component is ghosted to the mouse pointer.

19. Repeat steps 16, 17 and 18 to place another 1 kΩ resistor.

20. Select the Sources Group, SIGNAL_VOLTAGE_SOURCES Family, and place the AC_VOLTAGE component. At this point, your schematic should look something like the following figure:
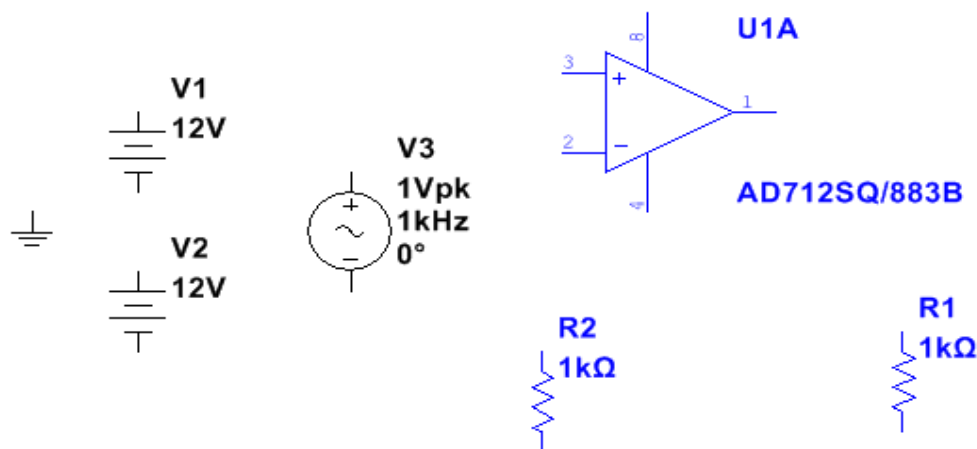
Figure 12.4. Components placed on the workspace area.

Part B: Wiring the Schematic

Multisim is a modeless wiring environment. This means that Multisim determines the functionality of the mouse pointer by the position of the mouse. You do not have to return to the menu to select between the placement, wiring, and editing tools.

1. Begin wiring by moving the mouse pointer close to a pin of a component. The mouse appears as a crosshair rather than the default mouse pointer.

2. Place an initial wire junction by clicking on the pin/terminal of the part (in this case, the output pin of the opamp).

3. Complete the wire by moving the mouse to another terminal or just double-click to anchor the termination point of the wire to a floating location somewhere in the schematic window.

4. Create a copy of the ground symbol using Copy <Ctrl+C> and Paste <Ctrl+V>.

5. Complete the wiring as shown in Figure 5. Do not worry about the labeled numbers on the wires (also called nets).
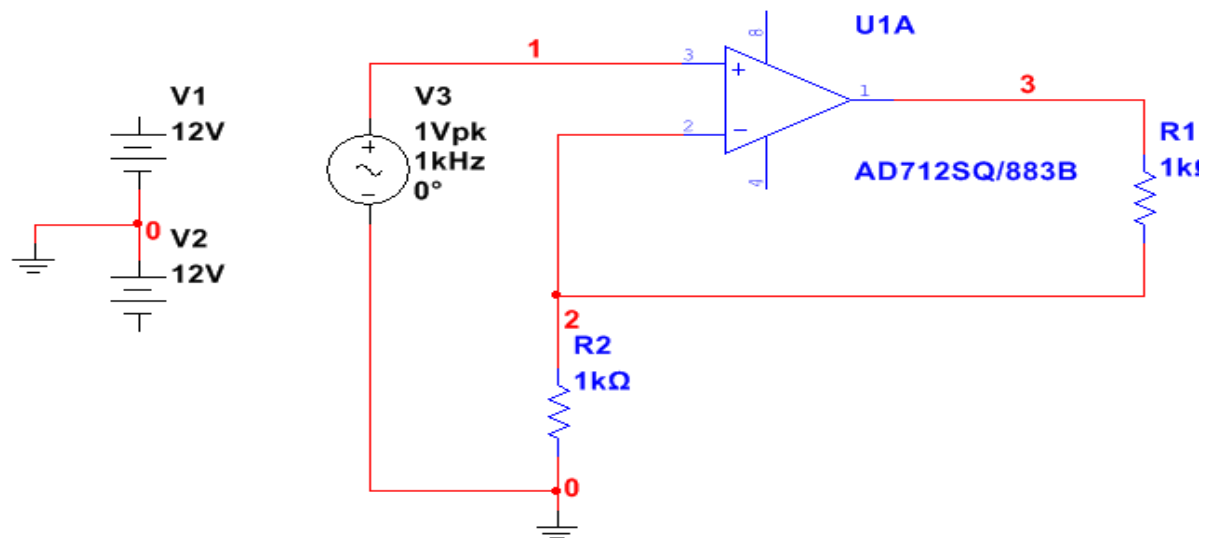
Figure 12.5. Wiring the schematic.

The last key step is to connect the power supply terminals to the positive and negative power rails of the opamp via a virtual connection using On-page connectors.

6. Select Place»Connectors»On-page connector and connect it to the positive terminal of the V1 power supply. The On-page Connector window will open.

7. Enter +V in the Connector name field and click OK.

8. Select another On-page connector and connect it to terminal 8 of the opamp. The On-page Connector window will open again.

9. Select the +V connector in the Available connectors list and click OK. The positive terminal of the V1 DC power supply is now connected to pin 8 of the opamp via a virtual connection.

10. Repeat steps 6 to 9 to connect the negative terminal of V2 to pin 4 of the opamp. Name the On-page connector –V. The schematic should now look like the following figure:
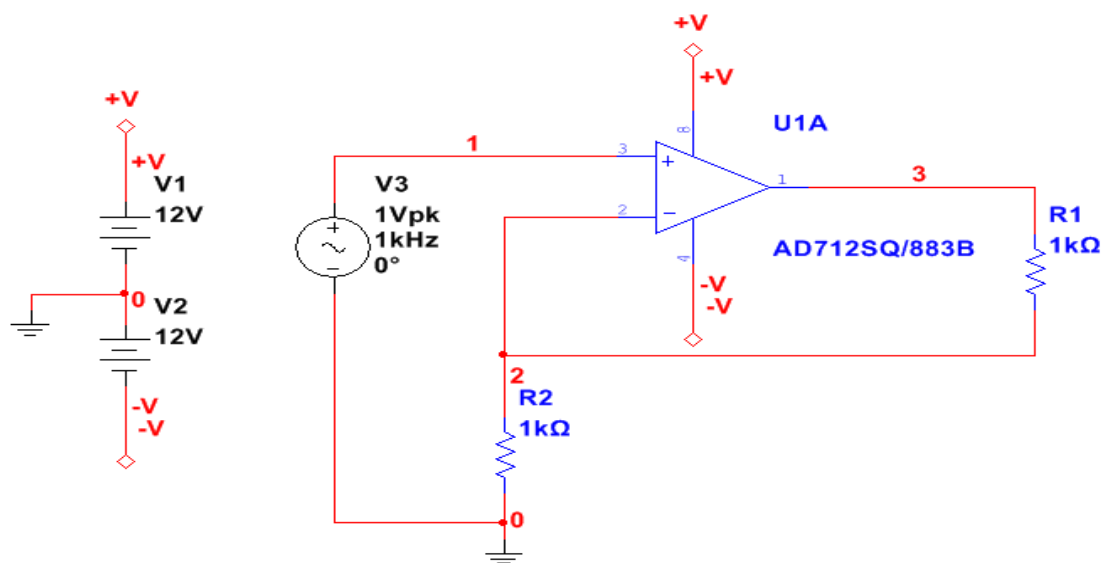
Figure 12.6. Schematic with On-page connectors.

Part C: Simulating the Circuit

You are now ready to run an interactive Multisim simulation; however, you need a way to visualize the data. Multisim provides instruments to visualize the simulated measurements. Instruments can be found on the right menu bar and are indicated by the following icons.

Figure 12.7. Instruments toolbar.

1. Select the Oscilloscope from the menu and place this onto the schematic.

2. Wire the Channel A and Channel B terminals of the Oscilloscope to both the input and output of the amplifier circuit.

3. Place a ground component and connect it to the negative terminals of the Oscilloscope.

4. Right-click the wire connected to Channel B and select Segment color.

5. Select a shade of blue and click the OK button. The schematic should look like Figure 8.
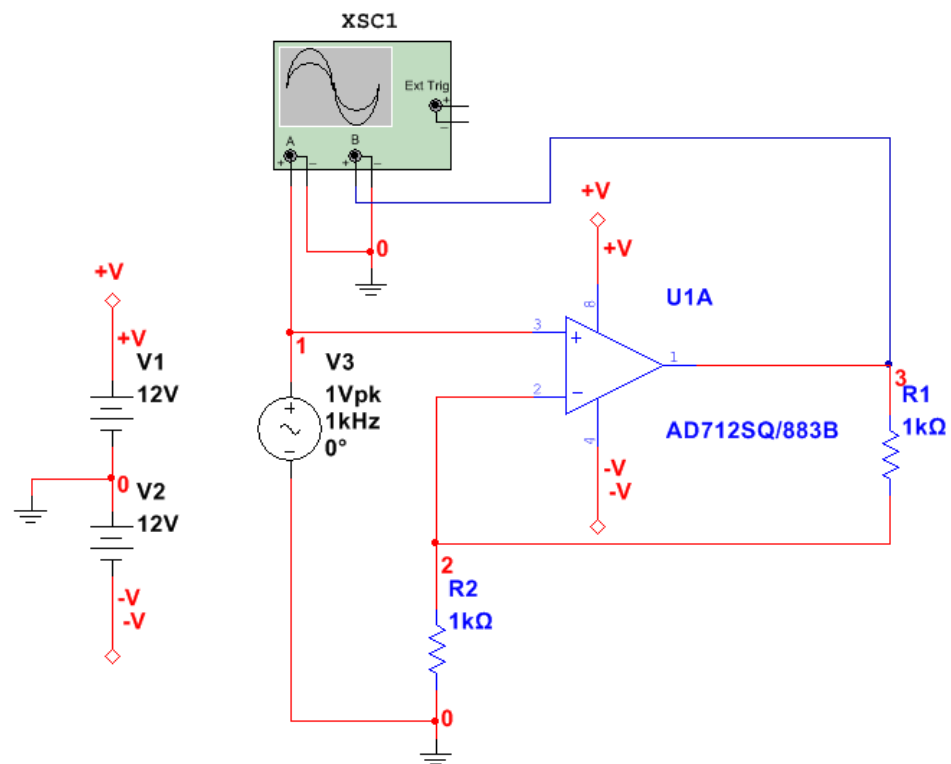
Figure 12.8. Connecting the Oscilloscope to the schematic.

6. Select Simulate»Run to start the simulation.

7. Double-click on the Oscilloscope to open its Front Panel and observe the simulation results (see Figure 9). As expected, the input signal is being amplified by a factor of 2.

8. Stop the simulation by pressing the red stop button in the simulation toolbar.
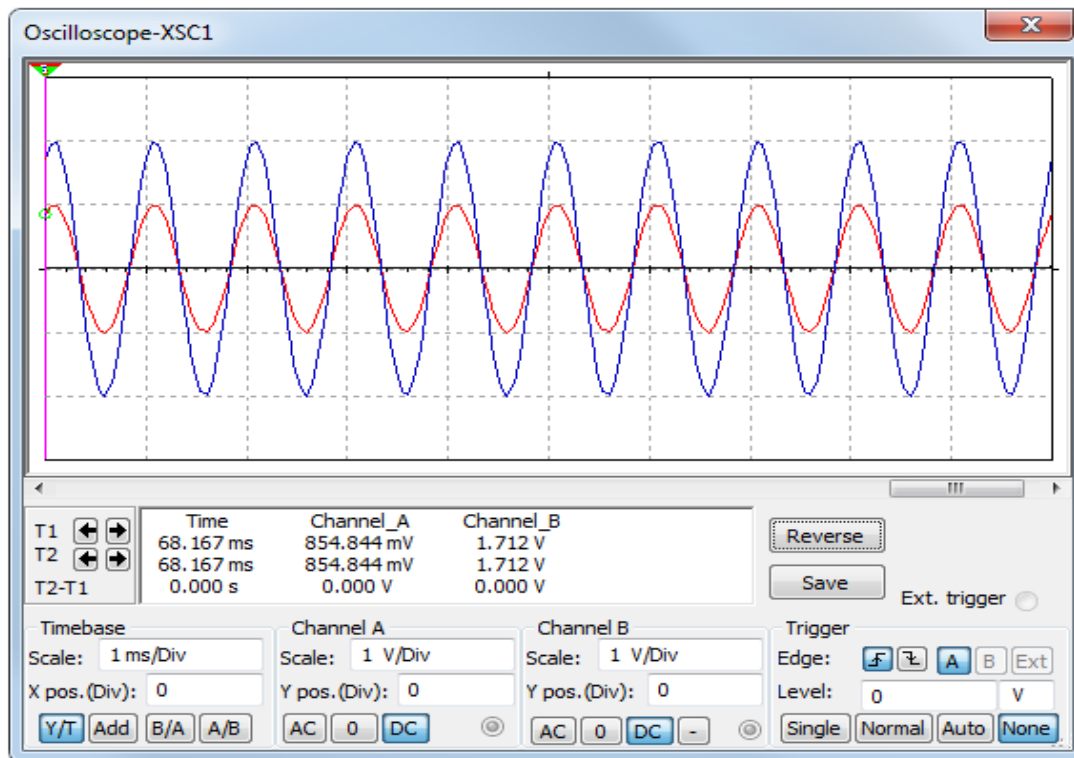
Figure 12.9. Simulation results

Part D: Transferring to PCB Layout

We are now ready to transfer the Multisim design to Ultiboard for PCB layout. In preparation for this we need to take into consideration that sources (power, signal) and ground are virtual components and, therefore, they cannot be transferred to Ultiboard. Also, all components must include footprint information. It is a good practice to replace power sources and ground with connectors.

1. Remove V1, V2, V3 and the Oscilloscope from the schematic. Do not remove the On-page connectors.

2. Open the Component Browser, and place the 282834-4 terminal block from the Connectors Group, TERMINAL_BLOCKS Family. This connector will be used to connect the power supplies (+V, -V).

3. Connect pin 1 of the connector to the +V On-page connector, pin 4 to the –V On-page connector, and pins 2 and 3 to ground, as shown in the next figure:
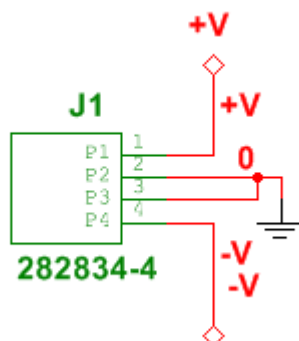
Figure 12.10. Connecting the terminal block.

4. Place another 282834-4 terminal block on the workspace. This connector will be used to connect the input and output signals.

5. Connect pin 1 of the connector to pin 3 (input) of the opamp.

6. Connect pin 2 of the connector to pin 1 (output) of the opamp.

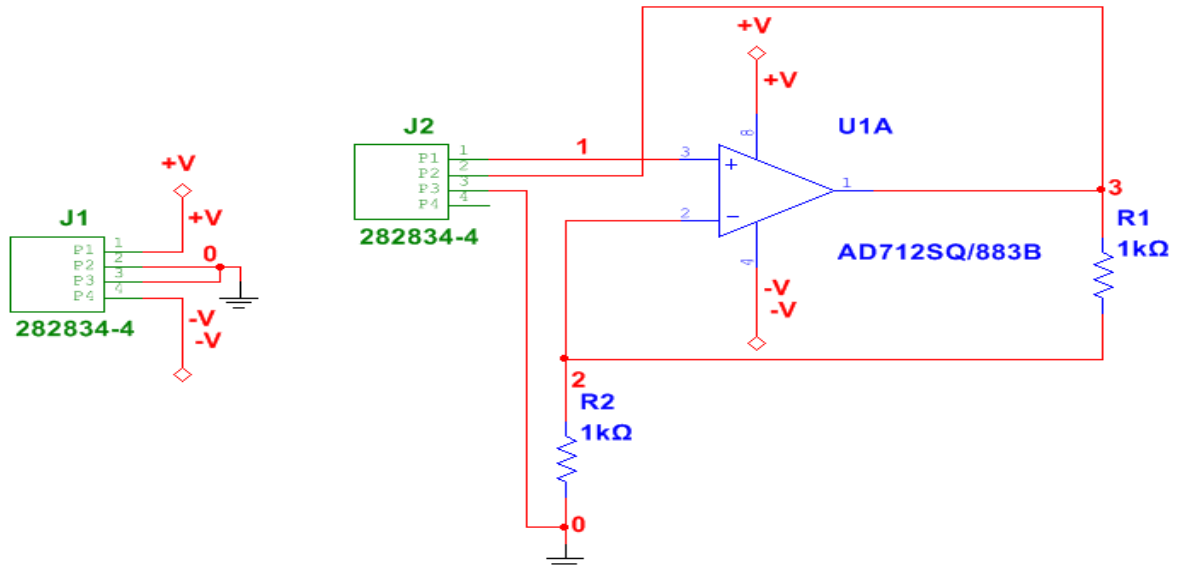7. Connect pin 3 of the connector to ground. The schematic will look like the next figure

Figure 12.11. Schematic with terminal blocks.

8. Select Transfer»Transfer to Ultiboard»Transfer to Ultiboard 13.0 and save the netlist file. Ultiboard will open automatically.

9. Click OK to accept all the actions listed in the Import Netlist window. Ultiboard will create a default board outline. Note that all the parts are placed outside of the board outline and the yellow lines (ratsnests) identifying the connections between pins, as shown in Figure 12.
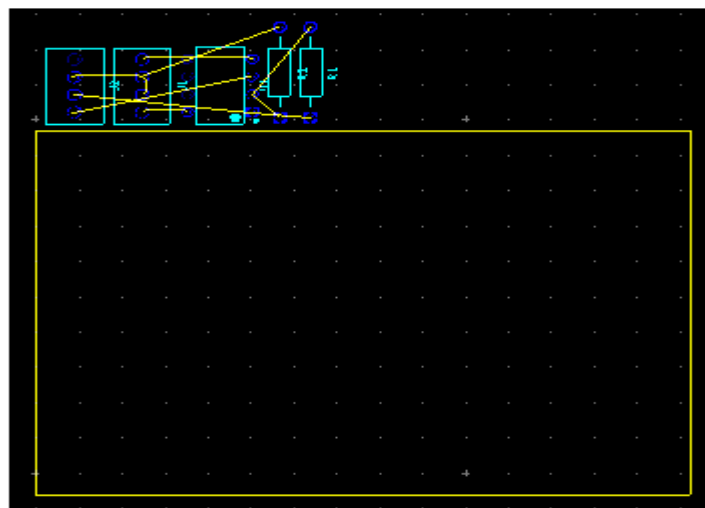
Figure 12.12. Default board outline and parts transferred from Multisim.

For this exercise we will use a 2x2 inch board. Follow these steps to resize the board outline.

10. Locate the Design Toolbox on the left side of the screen.

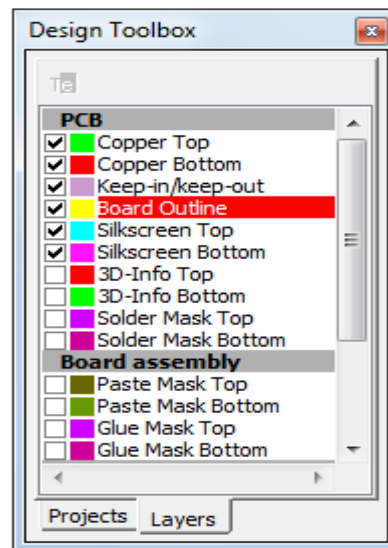11. Select the Layers tab and double-click Board Outline to enable this layer, as shown below.

Figure 12.13. Design Toolbox.

The Layers tab of the Design Toolbox allows you to move between layers of your design and control the appearance of the layers.
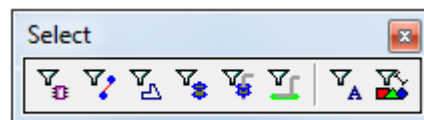
12. Go to the toolbar area and locate the Select toolbar, referring to the following figure.

Figure 12.14. Select toolbar.

The Select toolbar contains the functions used to control selection filters. In other words, these filters control what can be selected by the mouse pointer.

13. Disable all the filters except Enable selecting other objects.

14. Double-click the board outline on the workspace area to open the Rectangle Properties window.

15. Select the Rectangle tab, change Units to inch and enter 2 for Width and 2 for Height.

16. Click OK.

Part E: Routing the Board

Place components inside the board.

1. Go to the Select toolbar and disable all the filters except Enable selecting parts.

2. Drag part J2 and drop it inside the Board Outline. You can rotate parts by using the <Ctrl+R> shortcut.

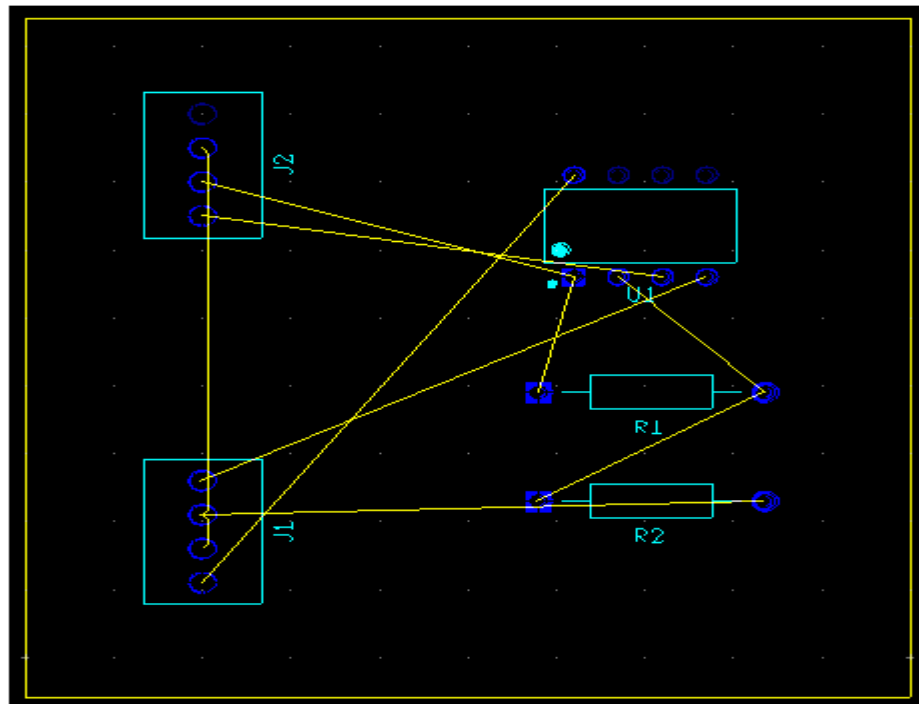3. Place the rest of the parts inside the Board Outline, use Figure 15 as guidance.

Figure 12.15. Parts placement.

For this exercise you will place traces on both the Copper Top and Copper Bottom layers.

4. Double-click the Copper Top layer in the Design Toolbox.

5. Select Place»Line.

6. Locate part U1 (the opamp). Note that pin 1 needs to be connected to R1, as indicated by the ratsnest.

7. Click pin 1 of part U1, draw a line to R1 and click its pin to finish the trace. Press Esc to exit the routing mode. The trace will look like the following figure:
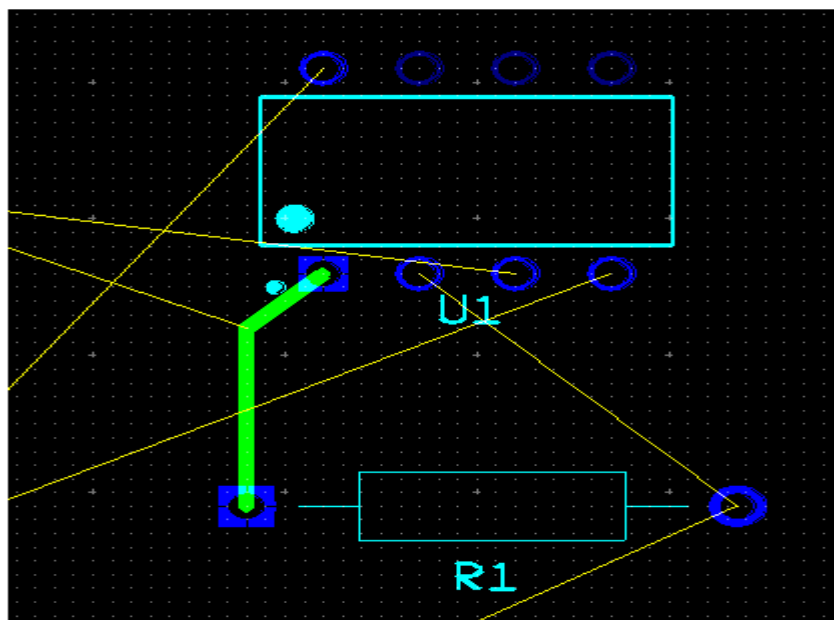

Figure 12.16. Placing a trace.

8. Double-click the the Copper Bottom layer in the Design Toolbox.

9. Select Place»Line.

10. Click pin 2 of part U1, draw a line to R1 and click its pin to finish the trace. Press Esc to exit the routing mode. Note that the color of the trace is red, which is the color configured for the Copper Bottom layer.

11. Finish placing traces for the rest of the connections. Your board should look like Figure 17.
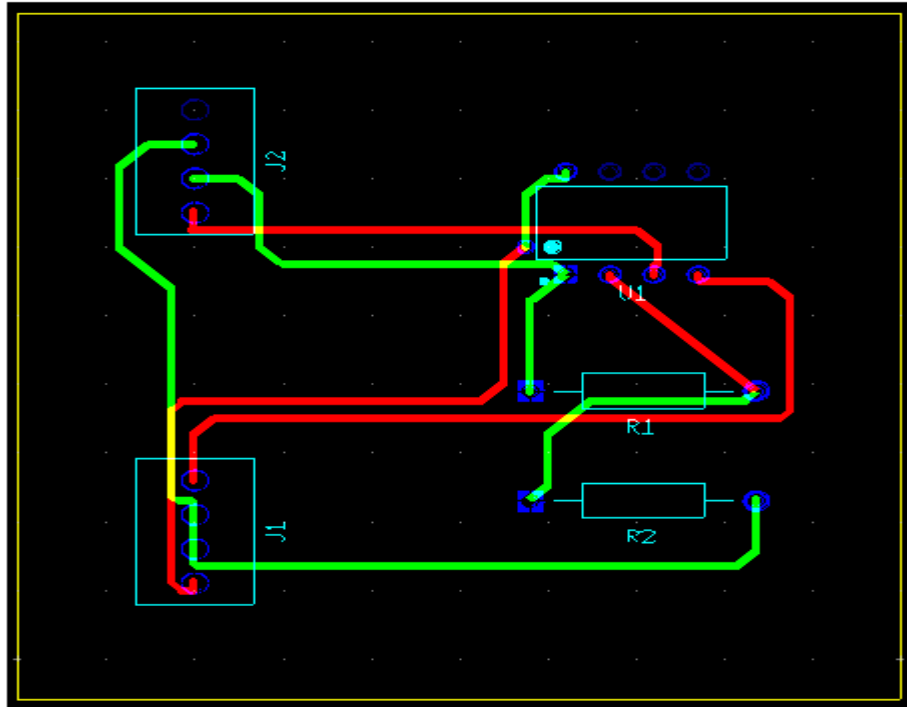

Figure 12.17. Routed board.

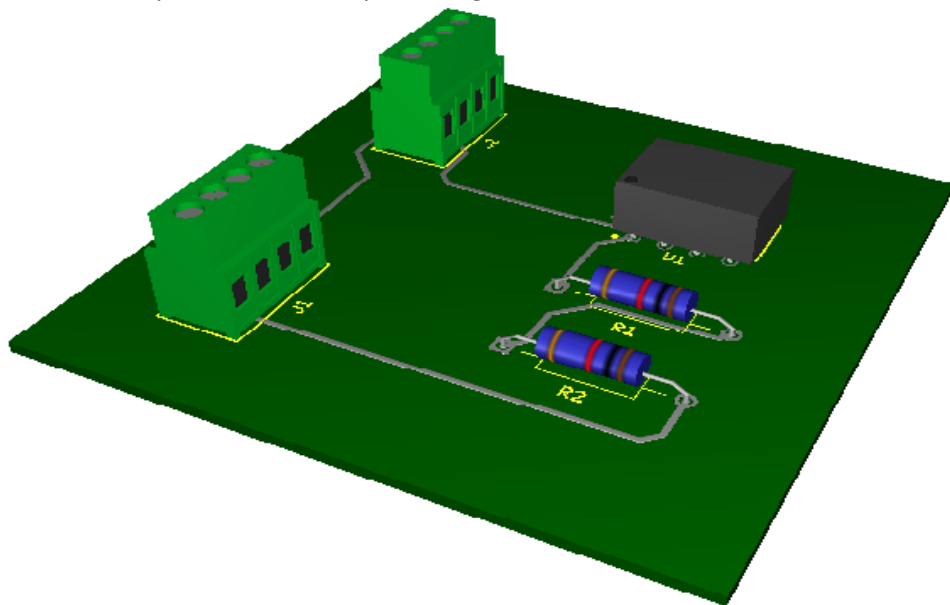Select View»3D Preview to open a 3D view of your design, as show below.

Figure 12.18. 3D Preview.

In Lab Task

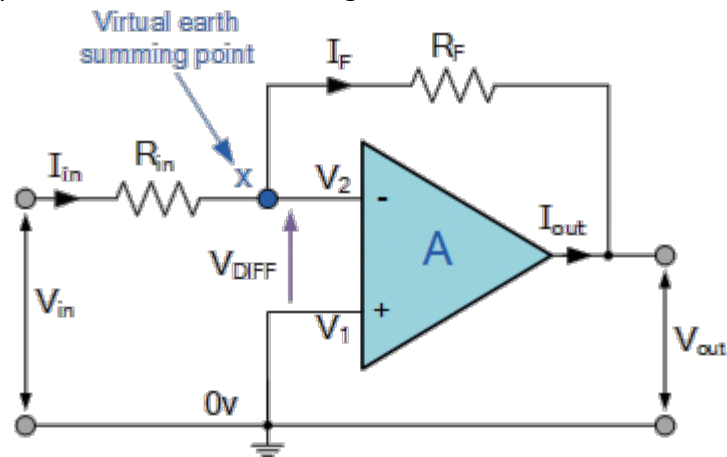An inverting amplifier circuit is shown in Fig. 12.19

Figure 12.19 Inverting Amplifier Circuit

Complete the following steps to complete the above design tasks

➢ Draw the schematic diagram of Inverting Amplifier Circuit

➢ Implement the PCB design of the schematic circuit in ARES

➢ Attach a complete PCB design circuit of Inverting Amplifier Circuit.

Rubric for lab assessment

The student performance for the assigned task during the lab session was:			
Excellent	The student completed all the tasks and showed the results without any help of the instructor.	4	
Good	The student completed all the tasks and showed the results with minimal help of the instructor.	3	
Average	The student partially completed the task and showed results.	2	
Worst	The student did not complete the task.	1	

Instructor Signature: _____ **Date:** _____