

Tutorial 1

Introduction to LibreCAD

INTRODUCTION

CAD – Computer Aided Design tool. A tool to make things better.

The following topics are covered in this tutorial.

- What is CAD?
- How can artisans integrate CAD?
- Which software should we use?
- Real world exercises
- Homework
- Wrap up and recap

CAD & ARTISANS

How can artisans integrate CAD in our daily work? There is no substitute for handmade craft.

But:

- we can use CAD to design without limits,
- we can preview the final results and predict real problems,
- we can get higher precision than using hand tools,
- we can achieve more repeatable results,
- and CAD provides a path to CAM (Computer Aided Manufacturing) such CNC, 3D printer, or a laser.

All of these must be directed by the skills of the artisan. Using only the technical tools will result in failure.

LIBRECAD – WHY?

It is the perfect tool for a new user. It is:

- Free and open source
- Powerful and stable
- Forces a good methodology and encourages learning best practices – it forces you to do things in the right way
- Developed and updated by the community – the user decides the features
- Covers 99% of the needs of the artisan

TUTORIAL 1 – DRILLING TEMPLATE FOR BRIDGES

The requirements are:

- 11.7mm string spacing at the saddle – requested by customer
- Tie block width is 80mm
- Wings of the bridge are 4mm
- Minimum string height is 10mm

- Drill holes are 1mm

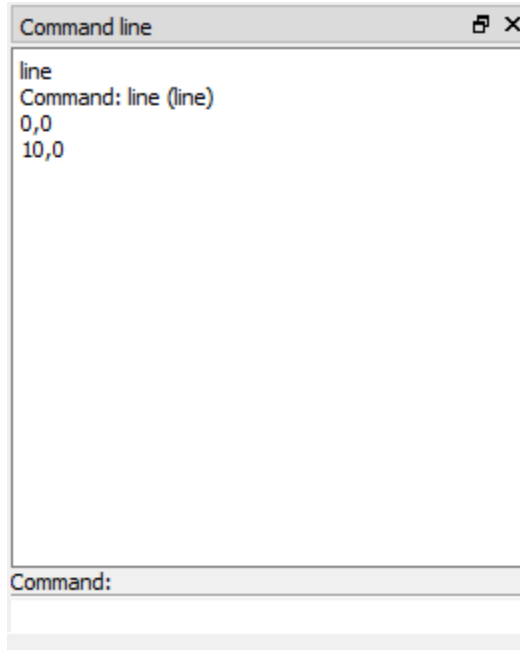
Tutorial Steps

Overview

1. Open LibreCAD
2. If this is the first time using LibreCAD:
 - a. Select default units of mm or inches – mm
 - b. Select language for user interface – English
3. The main working area is the black box
4. Presentation demonstrated opening an existing design using the File->Open menu and then zooming in and out using the wheel on the mouse and moving (panning) the drawing using by pressing the wheel. Then File->Close
5. There are two coordinate systems. Absolute and relative.
6. Move the mouse around and notice that the mouse position is displayed in the lower left corner of the screen in two different columns. The numbers in the left column is the position of the cursor with respect to the origin (shown as a red cross on the screen). The numbers in the right column are with respect to the relative zero position (shown as a small circle with a cross in it). The following picture shows the absolute and relative positions of the cursor.

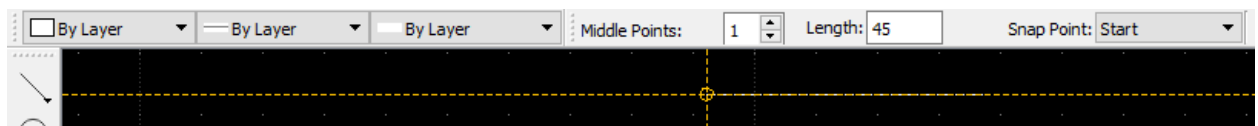


7. Creating items in the drawing can be done using the **Tools** menu, the Toolbar on the left, or by typing commands in **Command line** entry field on the lower right.
8. The **Command line** tool in the lower right has a transcript window in it that is used to display commands you type, below that is a command prompt indicating what is expected for the next input by LibreCAD (currently set to Command:), and below that is the entry field where you can type commands and other information required by the commands. See figure below for an example.



Create a horizontal line which is the bottom of the tie block.

1. To Create the line, select the first item in the left toolbar. This displays a list of line types that can be created. Choose the **Horizontal** menu item to create a horizontal line.
2. The tool is now in a mode that can be used to create a horizontal line.
3. The lower right **Command line** tool now has the prompt **Specify position**.
4. In the top toolbar under the menus there is displayed a set of properties for the line. The **Snap Point** specifies how the position you specify is used to create the line. The setting in the image below has the position entered specified to be the **Start** point of the line. Other options are the **Middle** and **End** points. The **Length** can be used to specify how long a line to draw.



5. Enter 80 into the **Length** entry field which is the length of the tie block. Set the **Snap Point** to **Start**.
6. Use the Absolute origin as the lower left of the tie block. It is a good practice for the absolute origin to be a key point on the object you are creating.
7. Enter 0,0 in the Command line entry field.
8. A horizontal line that is 80 mm long and starts at the absolute origin is now drawn.

Create a vertical line at right end of the tie block.

1. Select the **Vertical** menu item from the Line menu.
2. Set the length to 10mm which is the thickness of the bridge.
3. To create the line we want to have it start at the end of the horizontal line. Doing this using the mouse will result in a gap in the line. To make this easy the Snap options can be used.
4. There is a toolbar below the drawing area that contains a number of different options for how the point entered by the mouse is used. These are (from <https://librecad.readthedocs.io/en/latest/ref/snaps.html>):

- Exclusive Snap Mode – **On**: only one snap mode is allowed. **Off**: multiple snap modes are allowed.
- Free Snap – Allows for the crosshair to move freely while other snap modes are enabled. If this is off, then it is very easy to see the snap for each of the other modes.
- Snap on Grid – Snap to a grid intersection.
- Snap on Endpoints – Snap to the endpoints of a line segment, the quadrants of a circle, a point, or the alignment point of a text or mtext object.
- Snap on Entity – Snap to the path of an entity.
- Snap on Center - Snap to the center of a circle or ellipse. It will also snap to the foci of an ellipse.
- Snap Middle - Snap to the middle of a path. Enabling this mode displays a "Middle points" input. If you change the value to 2 then you can snap to the trisection points of a line segment.
- Snap Distance - If you snap to the endpoint of a line segment then activate "snap distance" and input 50, then it will snap to a point 50 units from the endpoint on the line segment. However, it will also snap to a point that is 50 units from the other endpoint.
- Snap Intersection - Snap to the intersection of two entities. Note this does not currently work for polylines.
- Restrict Horizontal – Restricts the crosshairs to the x-axis (horizontal movement).
- Restrict Vertical – Restricts the crosshairs to the y-axis (vertical movement).
- Restrict Orthogonal – Restricts the crosshairs to the x or y-axis. (either horizontal or vertical movement) -
- Set relative zero position – Manually sets the Relative Zero Point at the selected coordinate.
- Lock relative zero position - Locks the Relative Zero Point to the current coordinate.



5. Select **Free Snap** and **Snap on Endpoints** (they should both be highlighted).
6. Move the cursor near to the end of the horizontal line and click the left mouse button. This will create the vertical line with its start point coincident to the end point of the horizontal line.

Create a vertical line at left end of the tie block.

1. Click the right mouse button to clear the vertical line creation mode.
2. We will use the command line to create the next line.
3. Press **Ctl-G** to remove the display of the grid.
4. We need to draw a line from 0,0 to 0,10.
5. In the **Command line** entry field type: *line*. This sets the line creation mode and the Command line prompt indicates Specify first point.
6. Next enter 0,0 (the start point of the line).
7. Next enter 0,10 (the end point of the line).
8. Click the right mouse button to end the line entry mode.

Create a horizontal line at the top of the tie block

1. Verify that the relative zero position is at the end of the left vertical line.

2. We will create a line using relative positions instead of absolute positions using the @ notation for position.
3. Select **2 Points** menu item from the Line menu.
4. Type @0,0 in the Command line entry field to specify the first position being at the relative zero position.
5. Type @80,0 in the Command line entry field to specify the second position being at 80 x offset and 0 y offset from the relative zero position.
6. Click the right mouse button to end the line entry mode.
7. There are many ways to enter positions as we have seen: absolute position and relative position in the command line entry field and using the mouse with various snap settings.

Create axis and reference lines

Now create axis and reference lines to provide guidance on where the drill holes go.

Select **Snap Middle** so that we can snap to the middle of a line. The top toolbar now has a **Middle Points** field that specifies how many snap points are in the line. The default is that there is one snap point $\frac{1}{2}$ way between the two end points. If you change that to 2 then there are 2 snap points between the end points, to 3 provides 3 snap points. You can experiment with this.

1. Make sure the Middle Points field is set to 1.
2. Select **2 Points** menu item from the Line menu.
3. Click near the middle of the lower line and then the middle of the upper line to draw a vertical axis line. Click the right mouse button to clear the mode or finish with the tool.

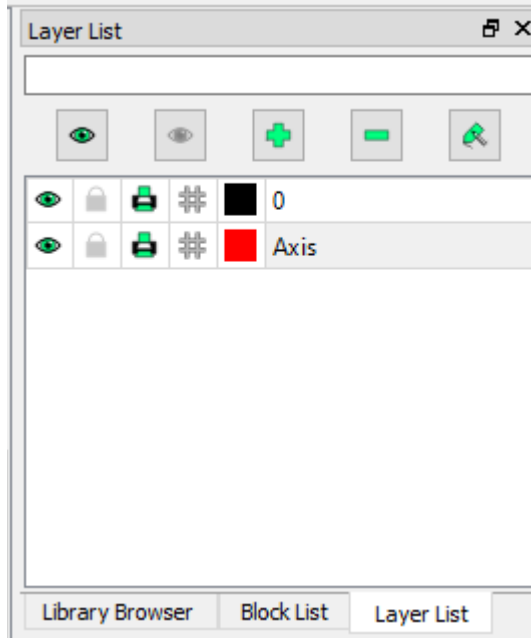
The reference line is 4 mm above the bottom of the tie block or the top of the wings. To create this, we will use the **Modify->Offset** menu item to create a line that is 4 mm above the horizontal line.

1. Select the lower horizontal line (bottom of the tie block)
2. Select the **Tools->Modify->Offset** menu
3. Enter 4 in the Distance entry field that has appeared in the upper toolbar.
4. Move the cursor near the horizontal line on the side of the line that you want the new line to be placed (above the lower line) and press the left mouse button.
5. Press the ESC key to clear the node or finish the command.

An alternate is to type *offset* in the **Command line** entry field, enter 4 in the **Distance** entry field, and then click on the side of the line you want the new line to be placed.

These lines are meant to be guidelines and not part of the object we are creating. Using layers allows us to separate these lines and define what they are meant to represent. On the right-hand side of the drawing there is a tool that is labeled **Layer List** (in the following image). We will create the Axis layer.

1. Select the + in **Layer List** tool.
2. In the form that is displayed:
 - a. Set the **Layer Name** to *Axis*
 - b. The **Color** to **Red**
 - c. The **Width** to **0.13mm (ISO)**
 - d. The **Line Type** to **Dash (small)**.
3. Press **OK** to create the layer



Next the guidelines need to be moved to the Axis layer.

There are several ways to select multiple items.

- The first is to just select each item – the item becomes dashed when selected.
- Pressing the left mouse and dragging it to the lower right creates a blue rectangle. Anything that is completely inside the box will be selected.
- Pressing the left mouse and dragging it to the upper left creates a green rectangle. Anything that is contained or intersects the box will be selected.

1. Use one of these methods to select the axis and reference lines
2. Select **Tools->Modify->Attributes** and change the **Layer** attribute to **Axis** and press **OK**.

Layer Settings can be changed using the icon on the far right of the **+** in the **Layer List** tool.

1. Select the **Axis** layer in the **Layer List** tool.
2. Press the icon on the far right of the **+** in the **Layer List** tool.
3. In the form change the Line Type to Dash(tiny) and select **OK**.
4. All items on the Axis layer now show the new setting.

Create guidelines for each of the drill holes

Use the offset command to create a line on each side of the center axis. This offset is $\frac{1}{2}$ of the string spacing.

1. Type *offset* in the **Command line** entry field.
2. Type $11.7/2$ in the **Command line** entry field. The **Distance** entry in the upper toolbar is now set to 5.85.
3. Move the cursor close to the axis line and create a line on each side.
4. Using the offset command create an offset of 11.7 and 4 more lines.

Create drill holes

1. Select Layer **0** in the **Layer List** tool.

2. Select **Snap Intersection** in the lower tool bar.
3. Select **Tools->Circle->Center,Radius**
4. Enter **1mm** in the **Radius** entry field in the upper toolbar.
5. Click on the 6 intersections where you want the drill holes.

Add Dimensions

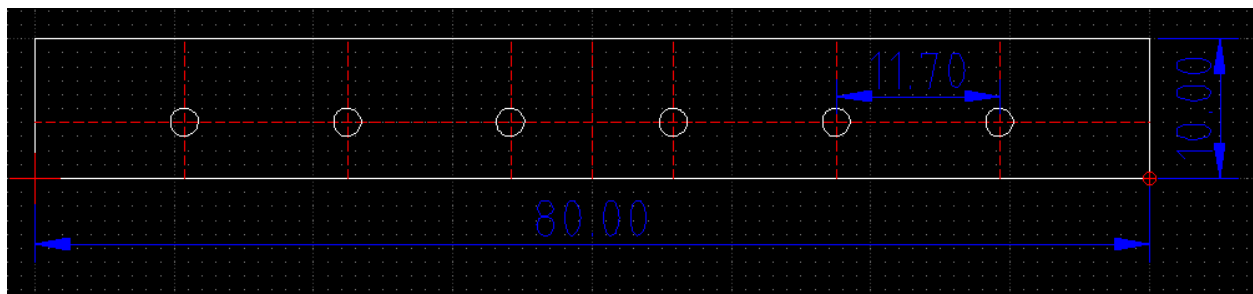
Place all of the dimensions on the drawing in the Dimensions layer.

1. Create a new layer named Dimensions that is Blue, 0.13mm (ISO), and Continuous
2. Select **Tools->Dimension->Horizontal**
3. Select the 2nd intersection from the right and then the 1st intersection from the right
4. Move the dimension to where you want it placed and click (you may need to turn off all snaps but the **Free Snap** to place it in an arbitrary position).
5. Finish the command.
6. To change the precision of the dimension displayed use the **Options->Current Drawing Preferences** menu item.
 - a. Select the **Dimensions** Tab.
 - b. Set the **Linear Precision** to **0.00**.
 - c. Click OK
7. Create a vertical dimension on the right-hand side.
8. Create a horizontal dimension on the bottom.

Completed Design

The following is the completed design.

1. Save the design using the **File->Save** menu item and give it a name and place it wherever you wish.



TUTORIAL 2 – RADIUS TEMPLATE

The requirements are:

- Radius of 5432 mm or 17.82152 ft
- Must cover a body length of 600mm
- Minimum thickness at center of 300mm

Tutorial Steps

Overview

1. Open LibreCAD
2. Create new design

3. Remove the grid – **Ctrl-G** or use **Grid Button** on upper toolbar

Create the bottom line of the template

1. Select **Line->Horizontal** line from the **Tools** toolbar.
2. Set the **Length** to 600mm
3. Set the **Snap Point** to **Middle**
4. Type 0,0 in the **Command line** entry field

Create layers

1. **Axis** – **Color: yellow, Width: 0.13mm ISO, Line type: Continuous**
2. **Dimensions** – **Color: blue, Width: 0.13mm ISO, Line type: Continuous**

Create Axis

The axis is a 30 mm line from the center of the lower line of the template

1. Select the **Axis** layer
2. Create **Line->2 Points** line
3. Enter 0,0 in the **Command line** entry field
4. Enter 0,30 in the **Command line** entry field
5. Finish the line

Create upper curved line of template

The curved part of the template will be an arc that is 5432mm that extends beyond each end of the horizontal line at the bottom of the template.

1. Select layer **0**
2. Select the **Curve->Center, Point, Angles** option from the **Tools** toolbar
3. Specify the center of the curve as 0,30+5432 in the **Command line** entry field
4. Specify the radius of the as 5432 in the **Command line** entry field
5. Select a point to the left of the left end of the lower horizontal line of the template
6. Select a point to the right of the right end of the lower horizontal line of the template
7. End the command

Create right and left vertical lines of template

1. Set the **Snap on Endpoints** in the lower toolbar (remove other snaps besides **Free Snap**)
2. Select the **Line->2 Points** line
3. Select the first point of the left vertical line as the left end of the lower horizontal line
4. Set the **Restrict Vertical** in the lower toolbar
5. Select the second point of the left vertical line to a point above curved line
6. Clear the **Restrict Vertical** in the lower toolbar so that the mouse can move to the other end of the line
7. Type *line* in the **Command line** entry field
8. Set the first point of the right vertical line as the right end of the lower horizontal line by clicking on the end of the line
9. Set the second point of the right vertical line using a relative position by typing @0,60 in the **Command line** entry field
10. End the command (right mouse or escape key)

Trim the vertical and curved lines

1. Select **Tools->Modify->Trim**
2. Select the “knife” or limiting entity by clicking the curved line
3. Select the portion of the right vertical light below the curve (the portion to be kept)

4. Type *trim* in the **Command line** entry field
5. Select the “knife” or limiting entity by clicking the right vertical line
6. Select the portion of the curved line to the left of the right vertical line
7. End the command
8. Select the **Tools->Modify->Trim Two**
9. Select the portion of the curved line to keep
10. Select the portion of the left vertical line to keep
11. End the command

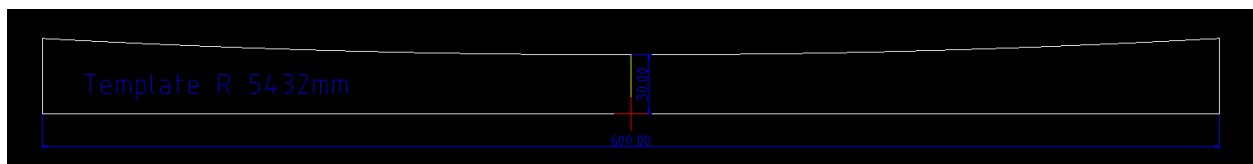
Add Label and Dimensions

1. Select the **Dimensions** layer
2. Select **Tools->Dimensions->Horizontal**
3. Select the lower left corner of the template
4. Select the lower right corner of the template
5. Use the **Tools->Dimensions->Vertical** to create a dimension of the axis line showing the minimum width of the template
6. Create a label by selectin the **Tools->MText** (multi-line text)
7. Enter the text *Template R 2432mm* in the **Text** entry of the popup form
8. Set the height of the text to 10mm by entering *10* in the **Height** entry field of the popup form
9. Click the **OK** button in the popup form
10. Place the text using the cursor in the left portion of the template
11. End the command
12. Set the Linear Precision to **0.00** and the **Text Height** to **5** in the **Options->Current Drawing Preferences** form and press **OK**

Completed Design

The following is the completed design.

1. Save the design using the **File->Save** menu item and give it a name and place it wherever you wish.



HOMEWORK

Make a Headstock template. Use the origin as the center bottom of the headstock. Adapt to your design. Use the main, Axis, and Dimensions layers. Export the file as PDF using the **File->Export->Export as PDF** menu item. The following discussion assumes you have the headstock as your active drawing.

1. To setup the page use **File->Print Preview**. This displays the drawing that page it will be printed upon.
2. The entry field in the upper toolbar shows the scale used and by default it is set so that the drawing fits on the current paper. Set this value to **1:1** (so that the drawing is not scaled). You may have to unset the **fixed** checkbox in order to change the value in the entry field.
3. Select the **fixed** check box. This removes the option to resize the drawing to the page (Fit to Page button is removed).

4. To change the paper size and orientation use the **Options->Current Drawing Preferences** menu item and the **Paper** tab in the form. Select the paper you want to use (for this A4 or Letter) and the orientation to **Portrait** (so that drawing fits well on the paper).
5. Use the **Center to Page** button on the upper toolbar to center the drawing on the page. You can also grab the “paper” and position the paper to contain the drawing.
6. Turn off the Print Preview by selecting the File->Print Preview again (it is a toggle button).

This will now be used when you print or export the drawing.

The PDF file you export can be used to commercial print shop, laser printers, CNC machines, etc...