

[How to](#)

How to

This section describes how to perform some tasks in addition to *User Manual* and *User Guides*. The content is the following:

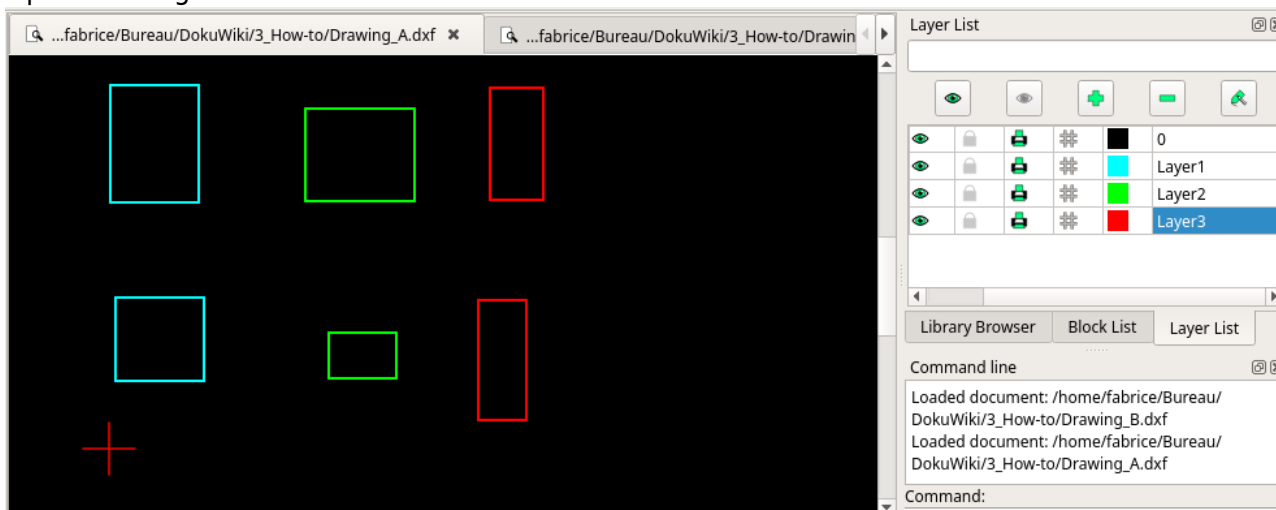
- [How to copy entities from drawing A to drawing B?](#)
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How to copy entities from drawing A to drawing B?

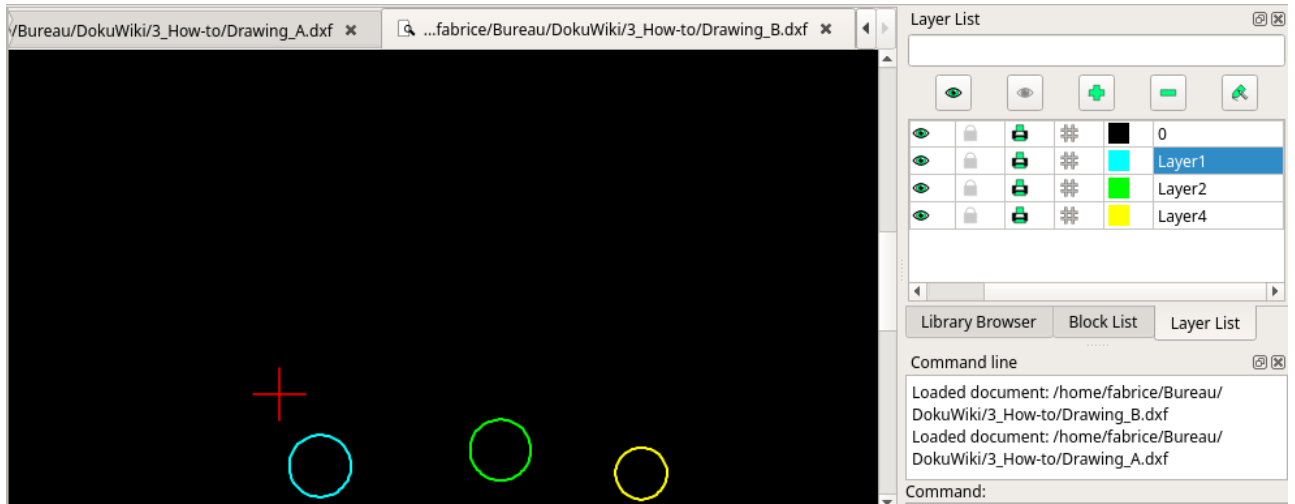
Quite often we need to copy some part of a drawing A and paste it in a drawing B. It is not difficult provided the user knows that **both drawings must be opened in the same LibreCAD session**.

In the example below, each layer has a specified color in order to better visualize what happens. In drawing A, there are 3 layers (layer1, layer2, layer3) showing rectangles and in drawing B there are also 3 layers (layer1, layer2, layer4) showing circles. Each layer has its own color for the sake of clarity. Let us say that we want to copy the green and red rectangles from drawing A to drawing B. The steps are :

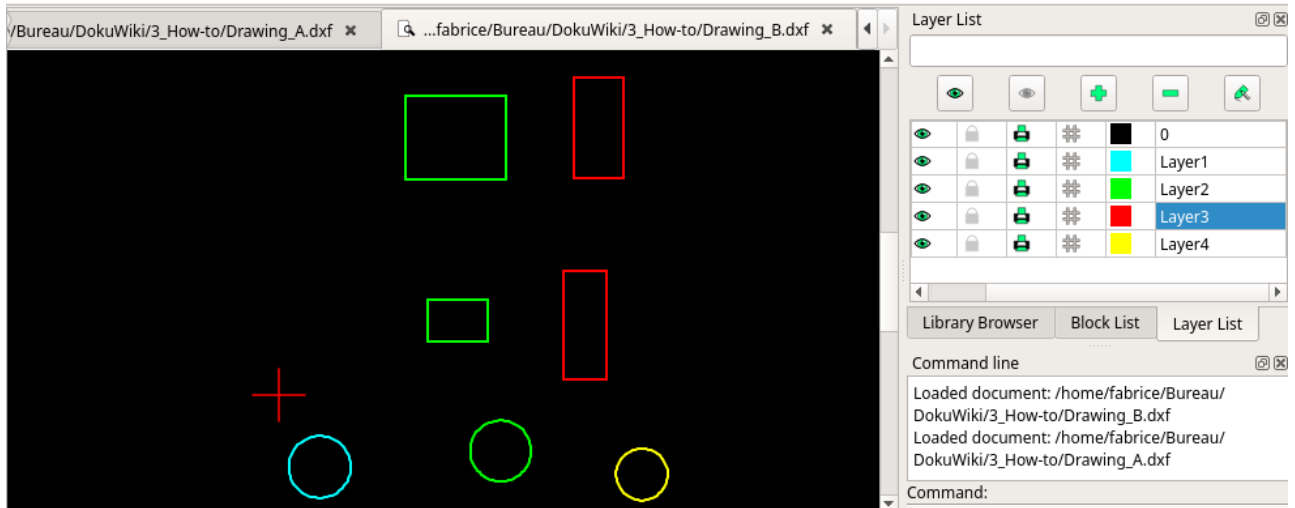
1. Launch LibreCAD.
2. Open drawing A.



3. Open drawing B **without launching a second time LibreCAD**.



- 4. Go to drawing A window, select the green and red rectangles, click on *Copy* icon or press CTRL+C and define a reference point.
- 5. Go to drawing B window, click on *Paste* icon or press CTRL+V and define a reference point.
That's it !

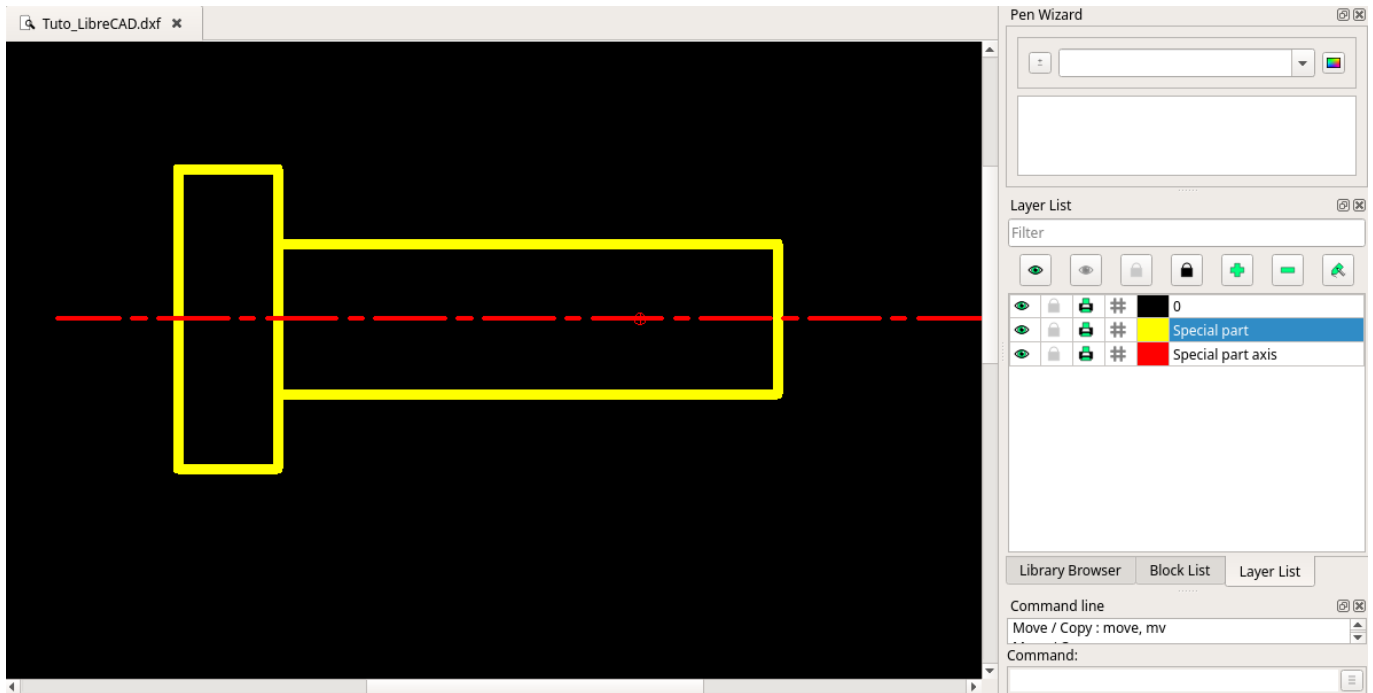


Note that LibreCAD moved the green rectangles of layer 2 in drawing A to the same layer2 in drawing B : so the green rectangles and circles now have the same layer.

How to create standard blocks to be re-used ?

When drafting, one often needs to re-use /several times the same part/: it can be a door for architectural design or a bolt for mechanical design. Even though one can copy and paste the part, it is not efficient. A more efficient way is to use a **library of blocks** which is at hand when you draft in LibreCAD.

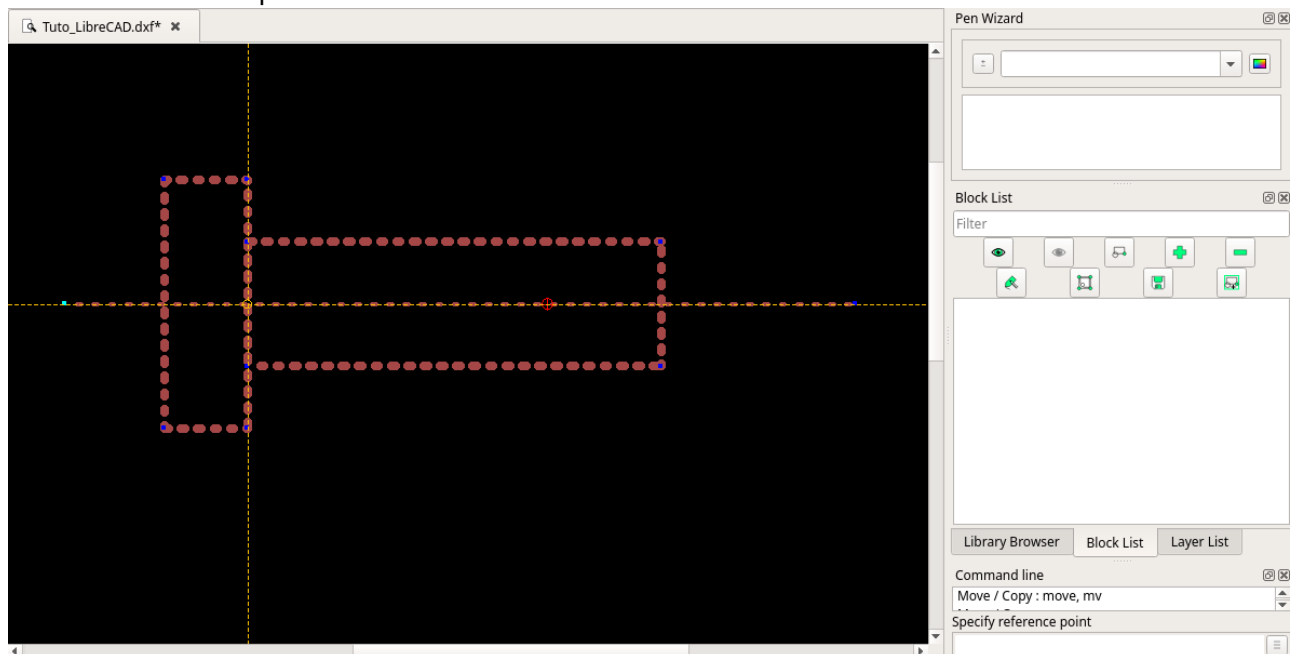
Let's say, we want to create a special part as shown below. Note that there are 2 layers, one for the part (yellow) and one for its axis (red).



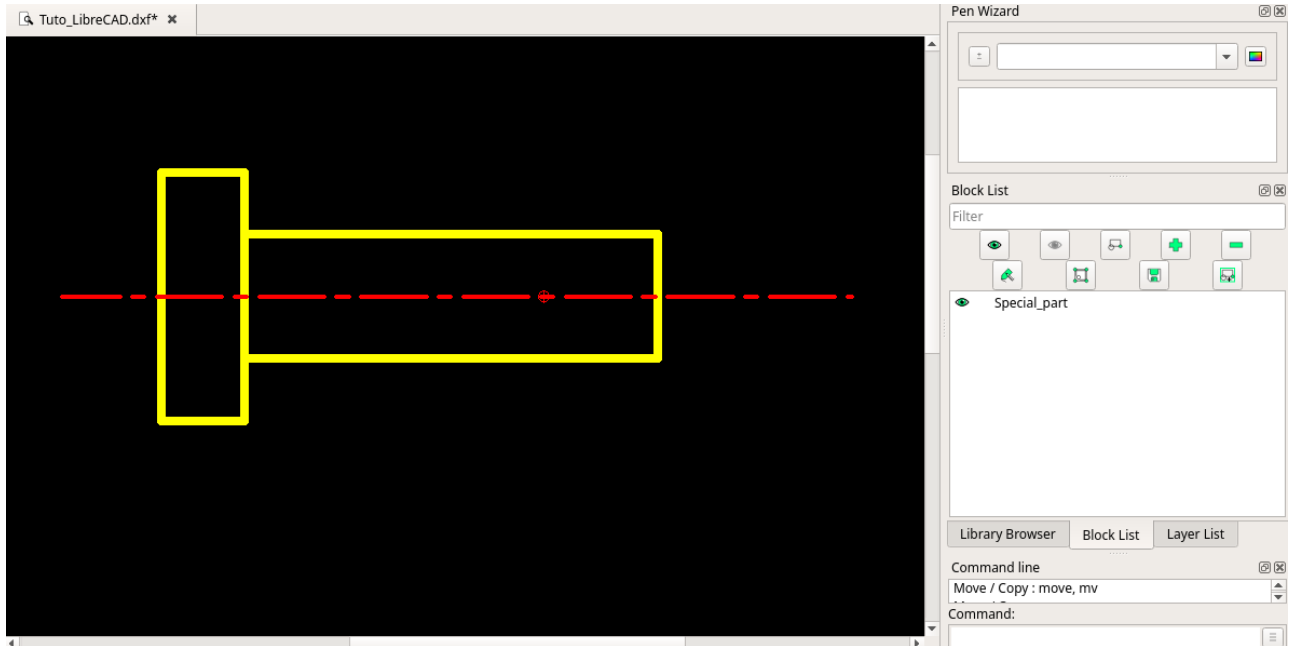
Step 1: create a block

You can refer to the [bloc section in the user manual](#) for supplementary information.

1. Select all the features needed for the block (everything in our example).
2. In the *Block list* dock menu, click on *Create a block* button
3. Select a reference point for insertion as shown.



4. Name this block *Special_part*. You should get the following:



Now, we can insert this block as many times as we want in the current drawing at different locations with different orientations and scales.

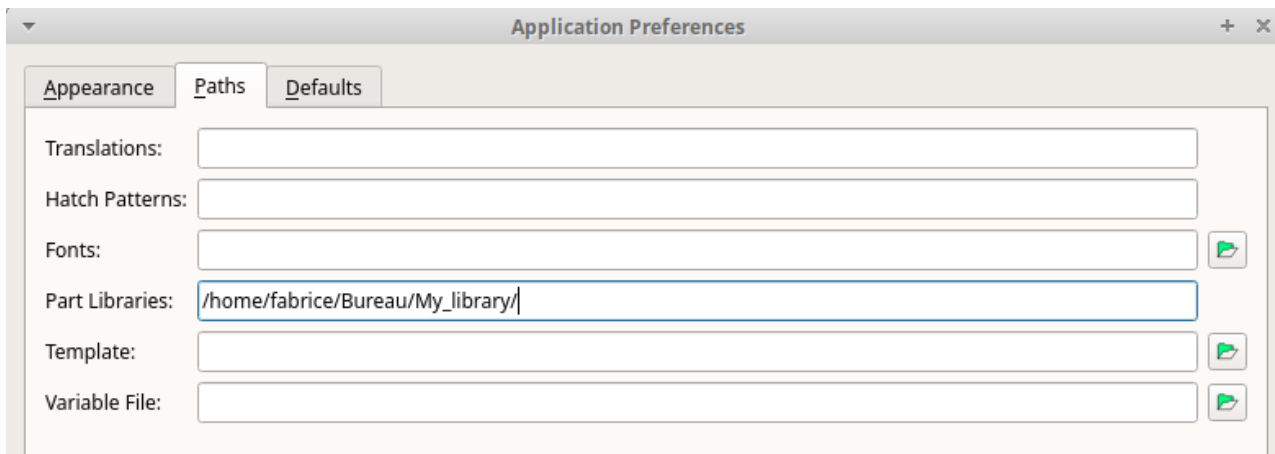
If we want to use this block in another drawing, there are 2 options:

1. Save the block as a DXF file: in the *Block list* dock menu, select the block and click on *Save the active block to a file*. Then, in you next drawing you can import if through the menu *File/Import/Block*.
2. Make this block readily usable by adding it in **your LOCAL library of blocks**.

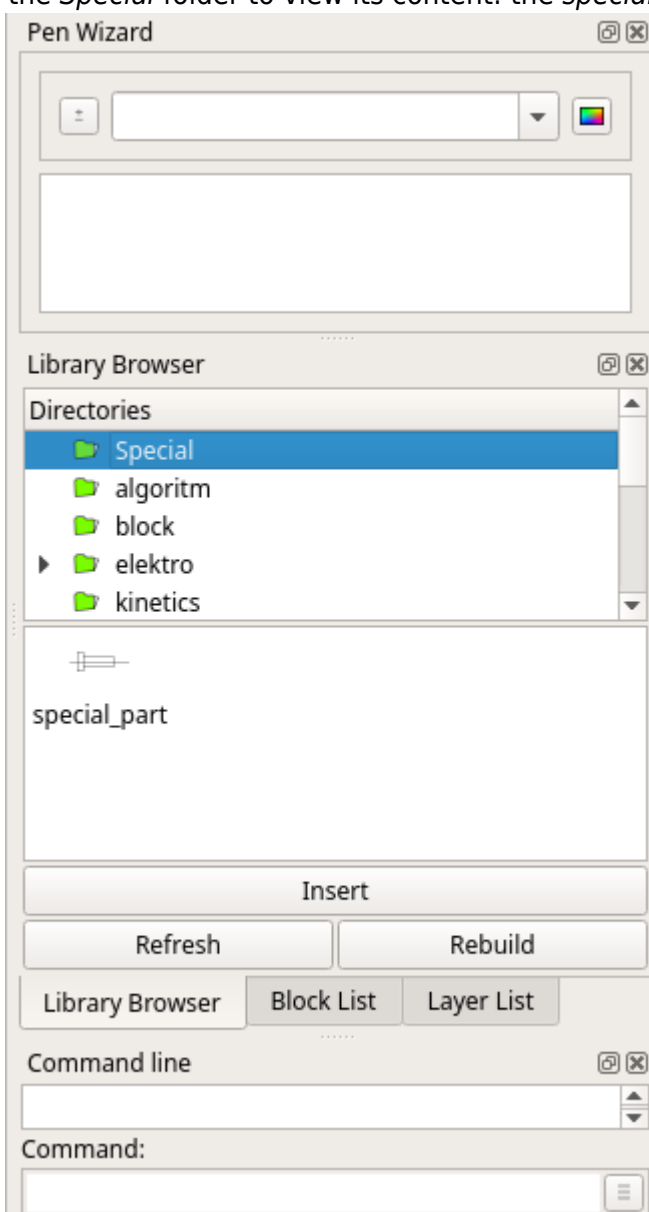
Step 2: create your local library of blocks

Just after creating the block in the current drawing, we need to make this block available for other drawings. The example below in based on a Linux system but the process is the same for MS-Windows system except for the path name usually in the form of `c:\users\fabrice\bureau\`.

1. Save the block as a DXF file: in the *Block list* dock menu, select the block and click on *Save the active block to a file*. The name of the saved file is *special_part.dxf*. Then, in you next drawing you can import if through the menu *File/Import/Block*. You can save the file at any location on your disk.
2. Create your library folder, here `/home/fabrice/Bureau/My_library/`. The folder *My_library* will hold all my library parts.
3. I want to have a sub-folder for the special parts named *Special* so I create it and move the file *special_part.dxf* into the *Special* folder.
4. Copy the path to the top folder of the library so `/home/fabrice/Bureau/My_library/`. **Be careful not to select any sub-folders**.
5. In LibreCAD, go to the menu *Options/Application preferences*. Then select the *Paths* tab and paste the copied path in the field *Part libraries* as shown.



- 6. Activation the new library path: on latest LibreCAD version, you can go to the *Library browser* dock window and click on *Rebuild* button. If this button is not available then you have to quit LibreCAD and relaunch it. Then you should see the *Special* folder in the *Library browser* tree.
- 7. Click on the *Special* folder to view its content: the *special_part.dxf* is available with a miniature



picture.

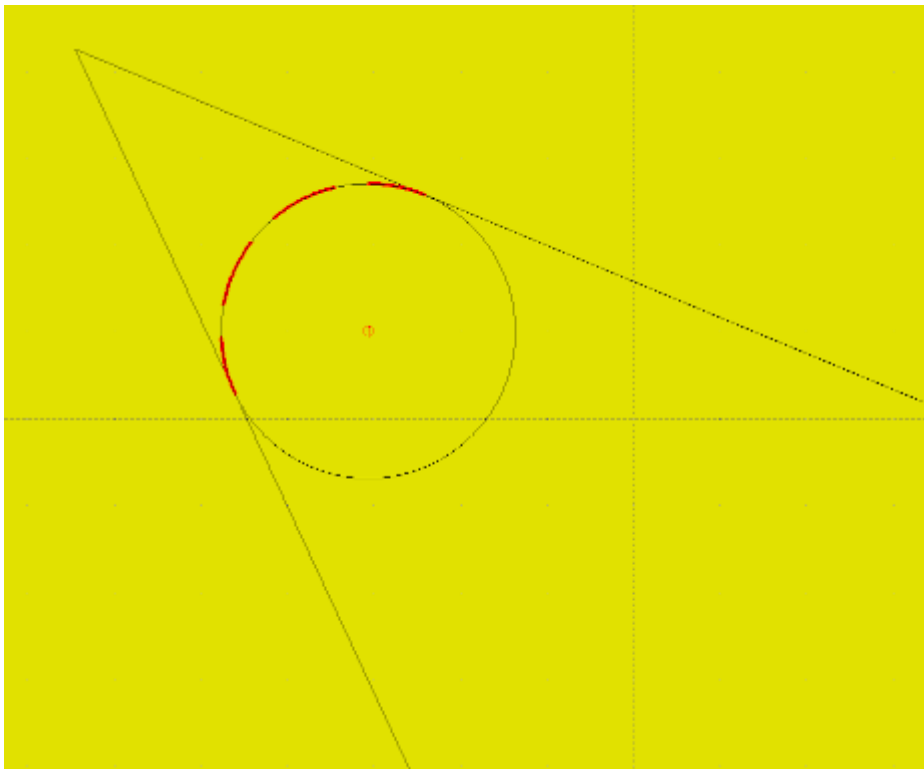
- 8. Finally, you can insert this block in any drawing by selecting this block and clicking on *Insert* button.
- 9. Optionally, you can modify the block: look at the [tool option bar](#), where you can set a [scale factor and rotation angle](#) **before the block is inserted**.

Note that other folders are also visible: these are coming by **standard** with the installation of LibreCAD. Your library will be **added** to the standard library. This way, **your library is not deleted when uninstalling LibreCAD** from your system.

How to add radius circle tangent to two lines ?

1. draw two crossing lines
2. create a construction layer
3. On the construction layer, draw a fillet with the required radius (shown dashed in image so visible)
4. On the drawing layer, draw a circle with the "centre, point" tool, using the centre of the fillet and its radius

The lines and circle will be perfectly tangent, for instance the "trim" will work on both lines.

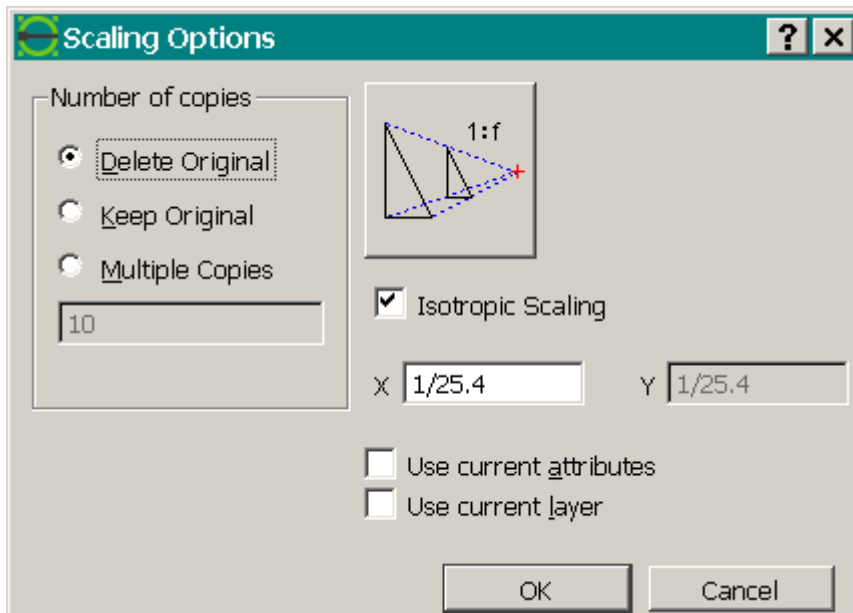


How to convert from mm to inches ?

Let's assume General Scale is set to 1. This is for an intended print scale of 1:1.

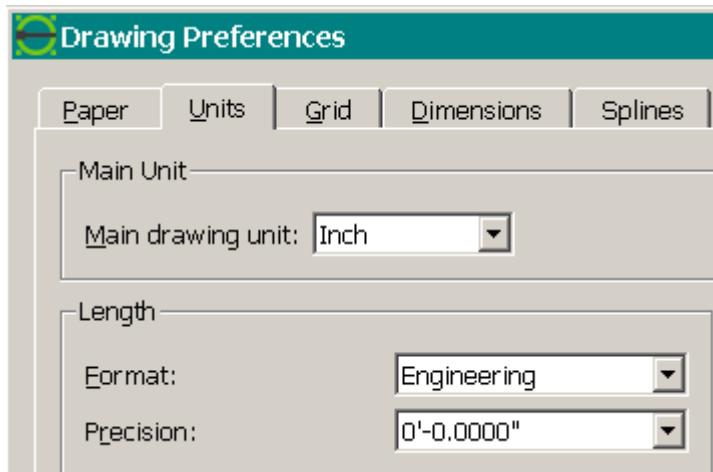
Then to scale from mm to inches:

1. Select all, by command (Tools - Select - Select All), or by pulling up a window around it.
2. Scale (Tools - Modify - Scale)
3. Specify reference point, normally best 0,0. Enter.
4. In the Scaling Options:

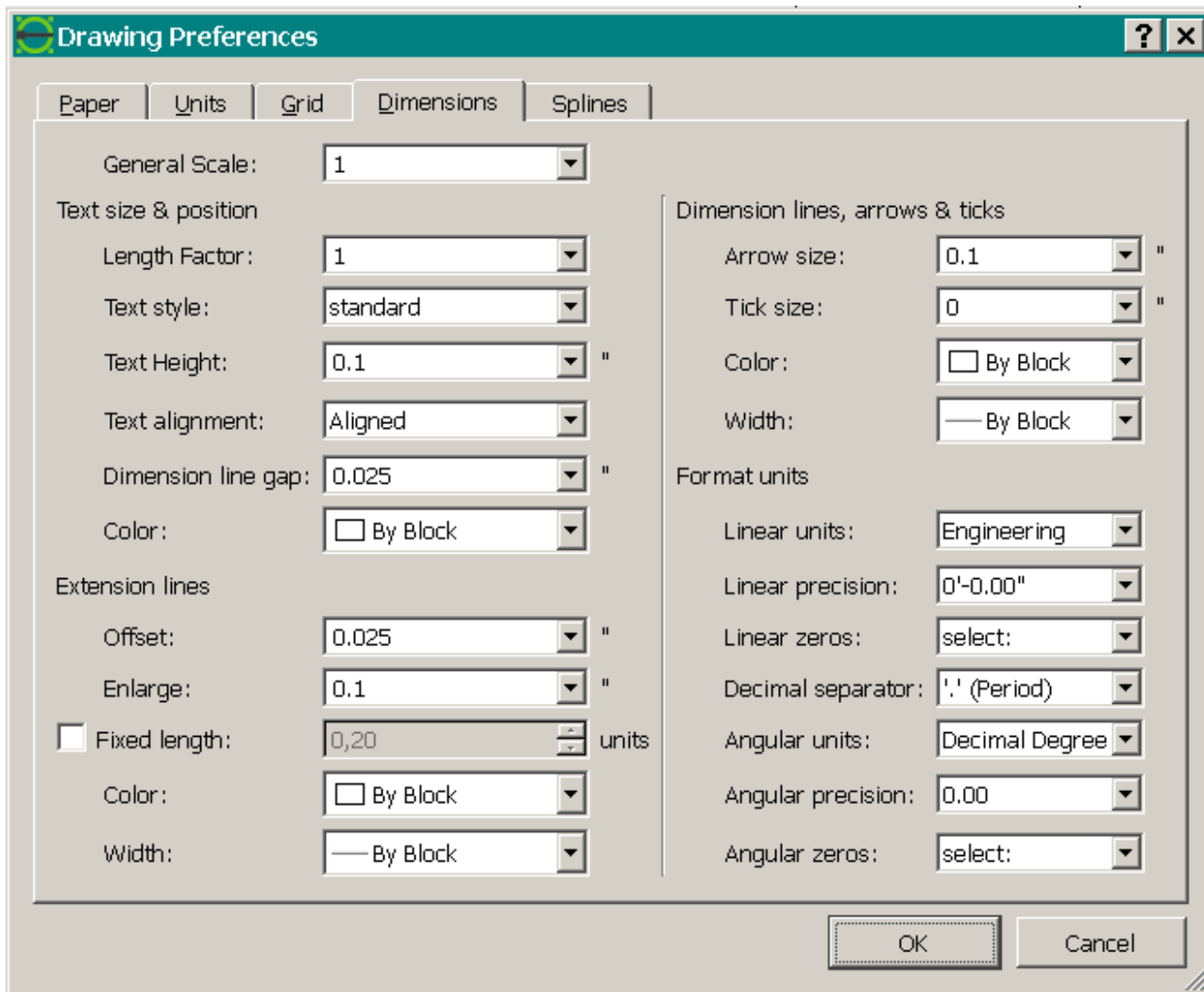


To adapt the Drawing Preferences to inches:

1. Set Main Units:



2. Adapt Dimensions. You could do it with General Scale 1/25.4, but better let it at 1 for possible later print scaling and change the size values individually. You can later use these settings for all your inches drawings.



And save it under a new name of course.

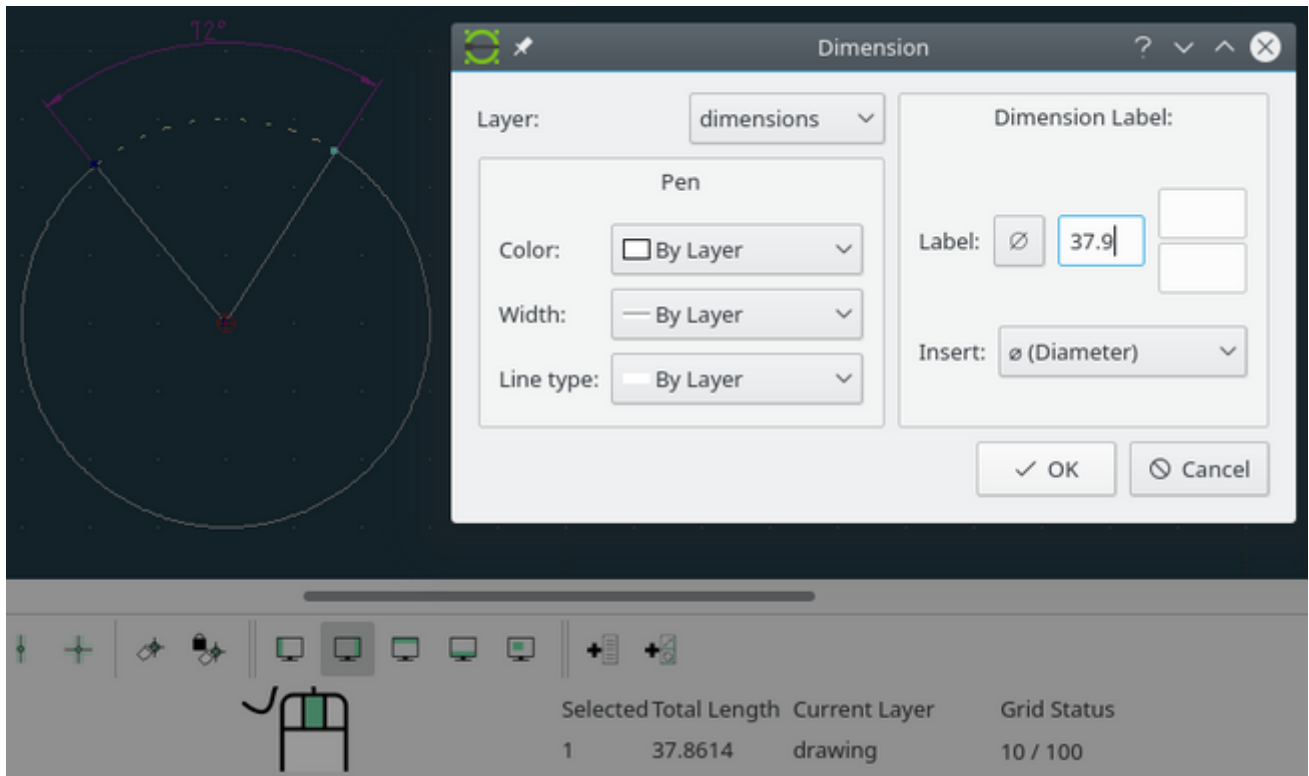
How to 'fake' a circular dimension...

There isn't a circular dimension tool for annotating arcs, but we can use one of the existing dimension tools, and then manually edit the label.

This really only works convincingly on segments of circles, although the approach could be applied to a continuous string of line entities, at a push, maybe with the Dimension: Aligned tool.

Step-by-step example:

1. draw a circle, centred on a grid point :: Circle: Centre, Point tool
2. draw two lines, from centre to perimeter, to define a segment :: Line: 2 Points tool
3. divide the circle at the two lines, to create an arc (two, actually) :: Modify: Divide tool
4. select the arc to find its length in the status bar
5. create an angular dimension annotation between the two lines :: Dimension: Angular tool
6. edit the dimension properties, defining a new label with the length of the arc :: Modify: Properties tool



Hint: in addition to Snap: Centre, you can find the centre of an arc with the Dimension: Radial tool; you can then use two radial dimensions as temporary construction lines from which to create the angular dimension.

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Last update: **2020/09/08 21:49**

