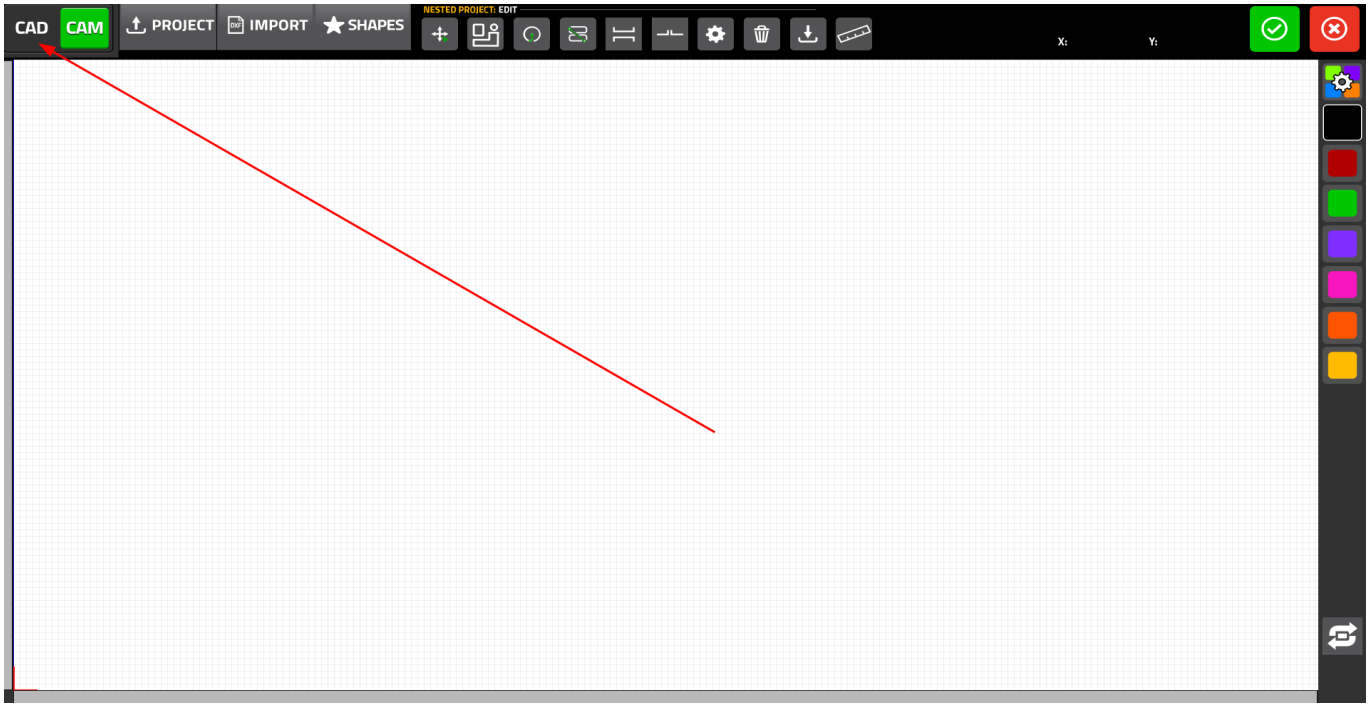
CAD Module

Fix Me! *August 24 2026 - Article under construction, images missing.*

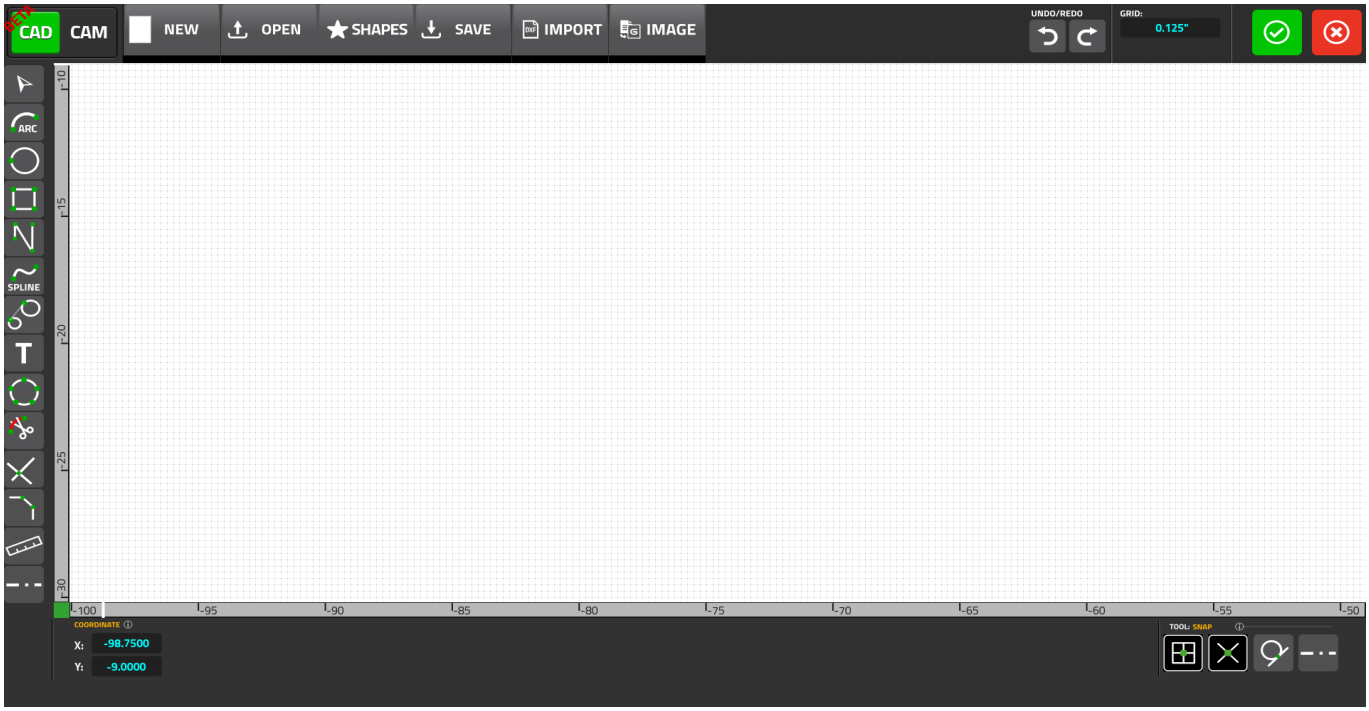
The CAD module allows the user to edit and design parts before bringing them into the CAM Module.

The CAD functionality allows the operator to draw custom shapes and figures, which could then be imported into the CAM module.

To open the CAD window, press the “CAD” button in the CAM Widget:



The window that appears on the screen looks the following way:



Top Panel

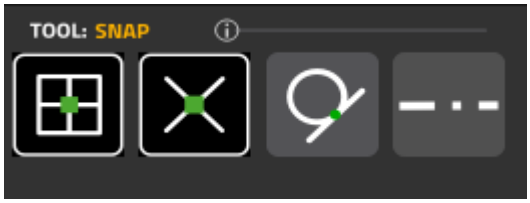
The following buttons are available:

- New - clear the screen to begin a new design
- Open - open a previously saved CAD project
- Shapes - Shapes library with some common designs (brackets, circles, etc)
- Save - save the current CAD design for later
- DXF Import - import a DXF file
- Image (alpha release) - trace an image. Feature is currently in alpha and may not work as expected.

- Undo/redo
- Grid selector
- Import to CAM/close window

The Grid element allows the user to select the grid size (0.0625”, 0.125”, 0.25”, 0.5”, 1”, or Flex).

The Snap element allows the user to enable/disable cursor snapping. Four options are available:

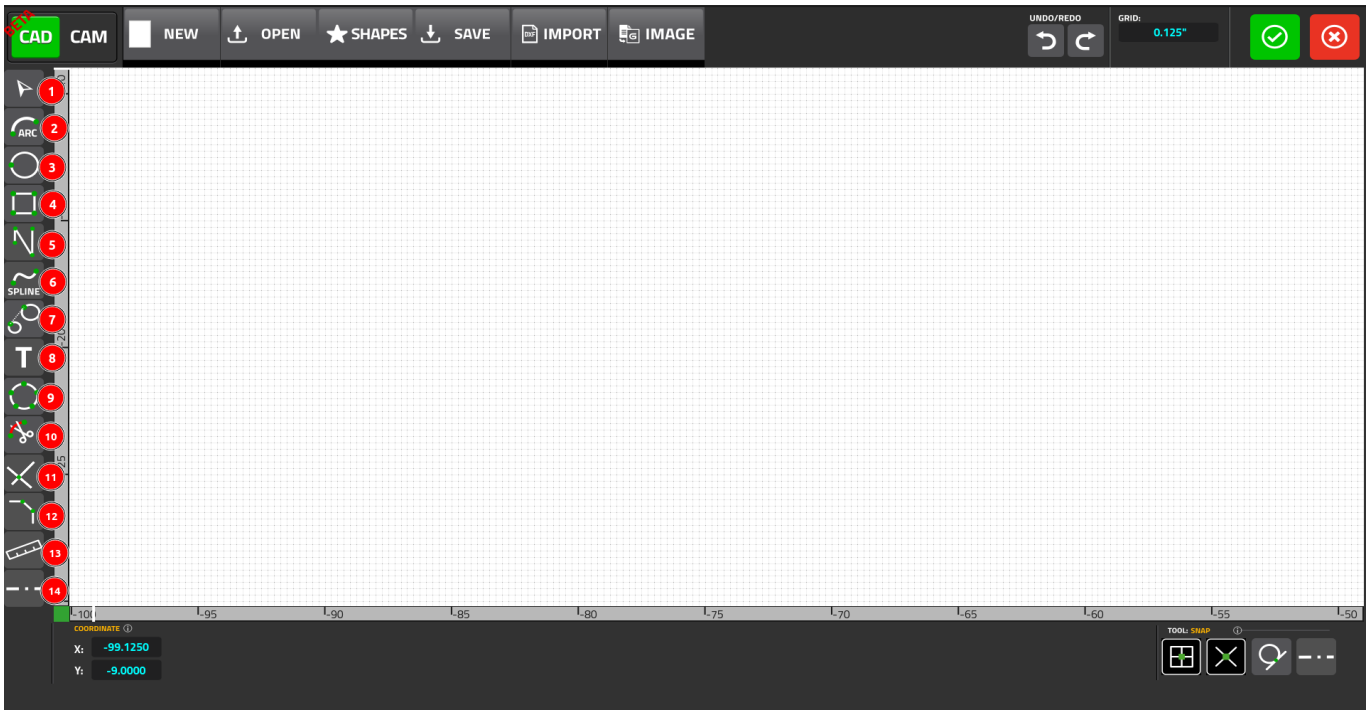


- Snap to grid. This will pull the cursor to the nearest intersection of the grid
- Snap to point. This will pull the cursor to the nearest point on an existing shape or element.
- Snap to tangent

- Snap to X-line (X-lines are guide lines, and are discussed further in this article)

Left Panel

The left panel of the window consists of the following elements:

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1. Select Element
2. Arc
3. Circle
4. Rectangle
5. Free line
6. Spline
7. Tangent line
8. Text
9. Subdivide circle
10. Cut
11. Find Intersections
12. Chamfer
13. Ruler
14. X-lines
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1. Select Element

Pressing the Select Element button brings up a ribbon at the bottom of the screen. The buttons contained within this ribbon allow the user to make operations related to the selection of existing shapes. The following options are available (from left to right):

- Select – this allows the user to click on existing shapes/elements to select them (press SHIFT to select multiple elements), or to drag the mouse around select shapes to select all highlighted

elements
- Move – this allows the user to move the selected part, either by dragging and dropping it with the mouse, or by entering the desired distances in the Translate X & Y fields.
- Delete
- Mirror Horizontally / Mirror Vertically
- Copy
- Rotate – after selecting the shape that need to be rotated and clicking this button, select a reference point around which the shape will be rotated. At this point, the user can either rotate the part by using the mouse or by entering the rotation angle value at the bottom of the screen
- Scale – this allows the user to scale the part by a factor specified in the Scale Selection field.
- Offset – allows to create an offset copy (or copies) of a line or an arc. Enter the number of copies in the Count field and the distance from the original contour in the Offset field. Then hover over the side of the contour that you'd like to offset the newly added line(s)/arc(s) from.
- Lengthen – allows to lengthen an arc or a line by a set value. Enter the value that the contour should be extended by, and hover over the contour on the side that will be extended. To shorten a contour, enter a negative "Lengthen" value.

2. Arc

An arc can be drawn by first selecting two points as the arc's ends, and then specifying the arc radius (by either dragging the mouse to create the necessary radius, or by entering a value in the Radius field at the bottom).

3. Circle

A circle can be drawn by first selecting its center (placed via a mouse click or via entering the coordinates in the Center X & Y fields), and then specifying its radius (by either dragging the mouse to create a circle of necessary radius, or by entering a value in the Radius field at the bottom).

4. Rectangle

A rectangle can be drawn by first selecting one of its corners (placed via a mouse click or via entering the coordinates in the X & Y fields), and then specifying its width and height (by either dragging the mouse to create a rectangle of necessary dimensions, or by entering the width and height in the W & H fields at the bottom).

5. Free line

This allows the user to draw lines on the canvas. An arbitrary number of segments can be drawn, with the end of one segment being the start of the next. To finish the drawing process, press the right mouse button.

6. Spline

This spline tool allows the user to create splines by placing nodes that OptimumCNC will automatically

connect. To finish the drawing process, press the right mouse button. Below is an example of a shape drawn using the spline functionality:

7. Tangent line

This option allows the user to create a tangent line to two circles/arcs by clicking upon the two circles . Below is a figure of two circles that have had two tangent lines created to them (one on either side):

8. Text

This option allows the operator to add text in a stencil font. Note: During original software setup, the fonts can be added to opt/optimumCNC/fonts folder. The Text toolbar (which appears at the bottom of the CAD window) looks the following way:

The following elements are available:

- Font selection. This allows the user to select from a number of fonts.
- Character spacing. Shown below is the difference between Character Spacing of 1 and 2 (note that results may differ depending on specific font and font size used):
- Font size
- Font positioning (Straight, Curve Down, Curve Up):
- Angle and Radius. These are applicable to the Curve Down and Curve Up options from the previous point only. The angle indicates the rotation of the text on the curve (default value = 0). The Radius indicates the radius of the arc which the text is curved around.

See example below for a text with a Curve Down option selected, and with values of Angle at 90 degrees and a Radius of 12:

After the text has been entered and positioned correctly, press Enter to add it to the CAD canvas.

9. Subdivide circle

This option allows the user to enter the number of subdivisions that the circle will be equally divided into (in the Split # field), and then click on a circle to divide it into segments. In the image below, the circle on the left is the original, while the circle on the right has been subdivided into six segments:

10. Cut

Selecting this option allows the user to delete individual segments and points on the canvas. Note that selecting elements (via the Select Element button described above) and pressing the DELETE key also works.

11. Find Intersections

Selecting this option will instruct the software to find intersections for any selected segments. To find

intersections, click the Find Intersections button, and then click on a segment that you'd like to find intersections with. All of the segment's intersections with other elements and shapes will be marked with new points.

12. Chamfer

This allows the user to create a chamfer, a fillet, or an "inside fillet" (i.e. a cutout with a specified radius). The toolbar is shown below:

The values of the elements can be selected in their respective fields. To create a modification to the part, 1) select the necessary function (for instance, Fillet), 2) enter a value for the radius or chamfer, and 3) click the corner that needs to be modified.

See image below for an example of a chamfer, a fillet, and a cutout:

13. Ruler

The ruler allows the user to quickly measure distances between different points in the drawing. After selecting the Ruler in the left hand side toolbar, click anywhere in the drawing and drag the mouse to see the ruler visualizer pop up:

14. X-lines

The x-line feature allows the user to create guidelines inside the CAD project that will not be imported into the CAM module with the rest of the drawing, but that can still serve as references within the CAD drawing. In the screenshot below, the lines drawn in black are "real" lines that will be brought in once the green checkmark in the top right corner is clicked, while the dashed blue lines are the x-lines (guidelines):

To better utilize the x-lines, enable the Snap to X-Line option in the bottom right corner of the CAD module: A useful method for automatically finding the centerlines of circles is to select the circle in question and press the "x" key on your keyboard. This will automatically add two centerlines that allow to easily locate the center of the circle:

Segment Edit

The user is able to edit the segments/points after adding them into the CAD project.

For more information, see the following video: