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Brief Review of Q CAD Used in Mathematics

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Abstract – QCAD is a 2D design and drawing computer-aided design (CAD) software tool. It is available for Linux,Apple macOS,Unix and Microsoft windows. The QCAD user interface is built on the Qt framework.

This paper presents the results of a comparative review of several groups of open - source software(0SS) for modeling & analysis of constructions. Software products of two categories are considered.

1) Geometrically/graphically modeling/drawing of the structures- 2D, 3D CAD.

2) Calculation and analysis of building structures under different load conditions and geometry including geotechnical problems.

3) Drawing Methods in QCAD: Cartesian Coordinate System; Useful drawings and objects. Draw Objects; Drawing Methods.

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INTRODUCTION

The use of Computer-Aided Drafting introduces technical drawings such as building plans, interiors, and so on. mechanical parts and diagrams are all available today. The mechanical parts, point filtering, and diagrams, accounting purpose, economics, statistics are applied to different areas such as marketing and experimental sciences, and so on. Many people used trigonometry, operations research, etc. Computer-Aided drafting is studied to simple concepts you prepared and understanding. Present in general your fresh thoughts. No concepts or skills are built in this article on a basic line by line. and understand the many examples and detailed step-by-step instructions in this It is an application for Computer-Aided Design(drafting)(CAD) in two dimensions, useful many fields. Algorithms as structure and marked the geometry in CAD concepts used in mathematics, such as the center of the rectangle, etc. used in geometry, topology, analysis, and approximation theory. For example, co-ordinates, soft blends, line segment, tangent, curved continuity are also combined and a non-rational locus satisfies a specified geometric tolerance for development of topologically true rational approaches.

EXPLAIN SNAPPING TOOLS IN SHORT

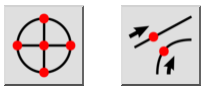
- Free(SF)
- Grid(SG)
- End(SE)
- On Entity(ST)
- Perpendicular(SU)
- Tangential(SB)
- Center(SC)
- Reference(SR)
- Middle(SM)
- Middle Manual(SN)
- Distance(SD)
- Distance Manual(SH)
- Intersection(SI)
- Intersection Manual(SY)
- Auto(SA)
- Coordinate(SX)
- Polar Coordinate(SO)
- X/Y from Points(.X)

- Y/X from Points(.Y)
- Center of Selection(SK)
- Restrictions Off(EN)
- Restrict Orthogonally(EO)
- Restrict Horizontally(EH)
- Restrict Vertically(EV)
- Restrict Angle or Length(EL)
- Set Relative Zero(RZ)
- Lock Relative Zero(RL)

SOME OF THEM EXPLAINED HERE

- **Distance**

Toolbar/Icon:



Menu: Snap>Distance

Shortcut: S,D

Commands: snapdistance | sd

Description

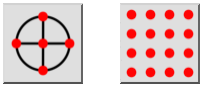
Snaps to points with a given distance to the end point of lines or arcs.

Usage

1. In the settings toolbar, enter the distance.
2. Move the mouse to the entity that measures the distance.
3. To set a coordinate, click the left mouse button while the required position/view is shown.

- **Grid**

Toolbar/Icon:



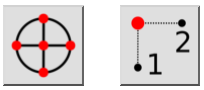
Menu: Snap>Grid

Shortcut: S,G

Commands: snapgrid | sg

- **X/Y from Points**

Toolbar/Icon:



Menu: Snap>X/YfromPoints

Shortcut: .,X

Commands: .x

Description

This snap tool can be used to extract one coordinate (e.g. X) from the location of one existing entity and the other coordinate (e.g. Y) from the location of..another..object.

This can for example be used to locate the center of a rectangle.

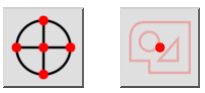
This concept is also referred to as 'combining coordinates', 'point filtering', or 'coordinate filtering'.

Usage

- When specifying a position, start this tool.
- Click a position in the drawing to lock the X coordinate at position.
- Click a position in the drawing to lock the Y coordinate at that position.

- **Center of Selection**

Toolbar/Icon:



Menu: Snap>CenterOfSelection

Shortcut: S,K

Commands: snapselectioncenter | sk

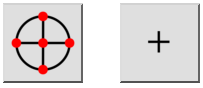
Description

Snaps to the center of the current selection. This is the point at the center between the leftmost and the rightmost edge in the X direction and the lowermost

and the uppermost edge in the Y direction of the selection

- **Free**

Toolbar/Icon:



Menu: Snap>Free

Shortcut: S,F

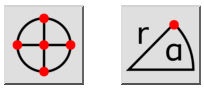
Commands: snapfree | sf

Description

Allows free positioning using the mouse. Note that this is almost never the recommended way of setting coordinates in a CAD system except for drawing freehand lines.

- **Polar Coordinate**

Toolbar/Icon:



Menu: Snap>PolarCoordinate

Shortcut: S,O

Commands: snappolar | so

Description

Defines a point by entering an absolute or relative polar coordinate.

Usage

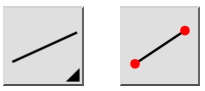
- Start this tool when you have to specify a point and want to do so by entering a polar coordinate.
- Enter the polar coordinate (radius and angle) in the options tool bar and choose if it is an absolute coordinate or a relative one (relative to the relative zero point).
- Click the OK button or press Enter to confirm the input and set the coordinate:

HOW TO DRAW OBJECTS USING COMMAND LINE

QCad may either be controlled through the menus, toolbars, hotkeys or the command line. The control line offers a powerful method to interact with a CAD system. Experienced users frequently utilise their left hand instructions while using the right hand mouse. But you may wish to turn it off via the menu if you don't (yet) use a command-line View - Line of command

- **Line from 2 Points**

Toolbar/Icon:



Menu: Draw>Line>Linefrom2Points

Shortcut: L,I

Commands: line | ln | li | l

Description

This tool lets you draw a sequence of one or more straight lines.

Usage

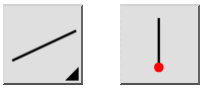
1. Choose the desired line type in the options toolbar.
2. Define the first line segment's start point. You can either use the mouse or the terminal to enter a coordinate.
3. Define the first line segment's start point. You can either use the mouse or the terminal to enter a coordinate:

You can undo a single line segment by selecting it and pressing the "Undo" button.:

4. If you want to restrict the angle or length of line segments, You can do so by selecting the appropriate option from the drop-down menu. "Restrict Angle or Length" button, and entering a length and / or angle.

- **Vertical Line**

Toolbar/Icon:



Menu: Draw>Line>VerticalLine

Shortcut: L,V

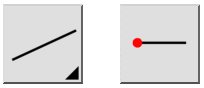
Commands: linevertical | l

Descripion:

Use this tool to create vertical lines. This tool is used like the tool for lines at a given angle, except that the angle is fixed to be vertical.

- **Horizontal Line**

Toolbar/Icon:



Menu: Draw>Line>HorizontalLine

Shortcut: L,H

Commands: linehorizontal | lh

Description

Use this tool to create horizontal lines. This tool is used like the tool for lines at any given angle, except that the angle is fixed to be horizontal.

LIST THE MODIFICATION TOOLS

- Move / Copy(MV)
- Rotate(RO)
- Scale(SZ)
- Mirror(MI)
- Flip Horizontal(FH)
- Flip Vertical(FV)
- Move and Rotate(MR)
- Rotate Two(R2)
- Align Reference Points(AE)
- Align(MA)
- Offset (with Distance)(OF)
- Offset (through Point)(OH)
- Trim(RM)
- Trim Both(TM)
- Lengthen / Shorten(LE)
- Stretch(SS)
- Clip to Rectangle(CLR)
- Chamfer / Bevel(CH)
- Round(RN)
- Divide(DI)
- Split Entities(MS)
- Break out Segment(D2)
- Break out Manual(B2)
- Auto Trim(AX)
- Break out Gap(D3)
- Reverse(RV)
- Edit Text(MT)
- Edit Hatch (MH)
- Explode (XP)

WHAT IS STRETCH AND MIRROR

- **Stretch**

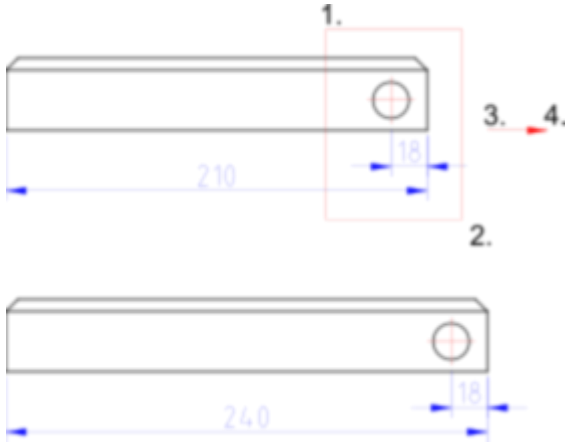
Toolbar/Icon:

Menu: Modify > Stretch

Shortcut: S, S **Commands:** stretch | ss

Description

This tool extends contours and entities of dimension. It is also possible that all endpoints are moved inside a certain rectangular or polygonal zone.

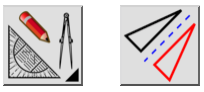


Usage

- Choose a rectangular (rectangular symbol) or polygonal area in the options toolbar (polygon symbol).
- Set the first corner of the stretch area.
- Set the second corner of the stretch area.
- Set additional stretching regions for polygonal sections, and when completed, right-click the final corner.
- Set a moving mouse reference point or provide a command line coordinate.
- Set the target point.

• **Mirror**

Toolbar/Icon:



Menu: Modify>Mirror

Shortcut: M,I

Commands: mirror | mi

Usage

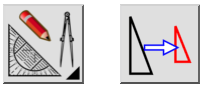
- Select the entities you want to mirror.
- Launch this tool.
- Use the mouse to select a beginning point on the mirror axis, or type a coordinate into the command line.
- Specify a second point on the mirror axis.
- The mirror dialogue box appears. Choose "Delete Original" to mirror the entities without keeping the originals, and "Keep Original" to replicate them.
- The new entities are placed on the same layer as the originals and have the same properties. To use the layer and current characteristics, uncheck "Utilize current layer and attributes."
- Click "OK" to mirror the entities.

EXPLAIN SCALE IN SHORT

QCad invites The user may first design and then scale a printed page picture or enable the user to modify the scope of a print preview relative to the printer page.

• **Scale**

Toolbar/Icon:



Menu: Modify>Scale

Shortcut: S,Z

Commands: scale | sz

Description

Scales entities by a given factor towards a given center.

Usage

- Select the entities you want to scale.
- Launch this tool.
- Set scaling centre using the mouse or input a command-line coordination.
- The scale dialog is displayed where you can enter the scale factor.

If you want to scale with two different factors in X and Y direction, uncheck the button for proportional scaling:

You can then enter two different scaling factors for X and Y.

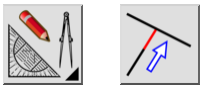
If you want to scale the selection by mouse, check the mouse cursor button:

- Click "OK".
- You must now provide a reference point and a destination point for the scaling operation if you previously selected to scale the selection by mouse.

DISCUSS TRIM AND COPY

- **Trim**

Toolbar/Icon:



Menu: Modify>Trim

Shortcuts: R,M|X,T

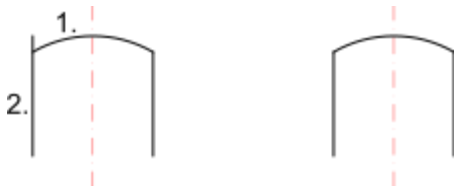
Commands: trim | extend | rm | xt

Description

Trims or extends a line, arc or ellipse to another entity.

Usage

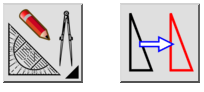
- Choose a boundary to which one or more other entities are to be reduced or expanded.
- Choose the entities you want the limited entity to trim. There are often two ways of cutting the entity. In the example below, you may want to keep the top of the line and cut the bottom portion off. You should click on the trimming object at the top In this instance. Always click that part of the entity which you want to retain.



- Right click twice or press Escape twice to terminate the tool.

- **Move / Copy**

Toolbar/Icon:



Menu: Modify>Move/Copy

Shortcut: M,V

Commands: move | mv

Description

Moves or copies entities.

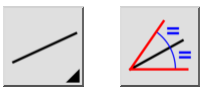
Usage

- Select the entities you want to move or copy.
- Launch this tool.
- Use the mouse to set the reference point, or type a coordinate into the command line.
- Decide on a target spot. Enter a relative coordinate to move the selected entities by a certain amount. Enter @50,0 in the command line to move it 50 drawing units to the right, for example.
- The move dialog is displayed.
- Choose 'Delete Original,' and copy 'Keep Original' in order to transfer the entities. Also, by selecting "Multiple Copies" and entering the number on the next text line, you may make a number of copies at once. Note that "9" creates nine duplicates, keeps the original, so that the chosen entities will eventually have ten instances.
- The new entities are placed on the same layer as the originals and have the same properties. Instead, check "Utilize current layer and attributes" to use the layer and current attributes.

EXPLAIN ONE DRWING METHOD IN SHORT

- **Angle Bisector**

Toolbar/Icon:



Menu: Draw>Line>AngleBisector

Shortcut: L,B

Commands: linebisector | bisector | lb

Description

Use this tool for creating angle bisectors between two line entities.

Usage

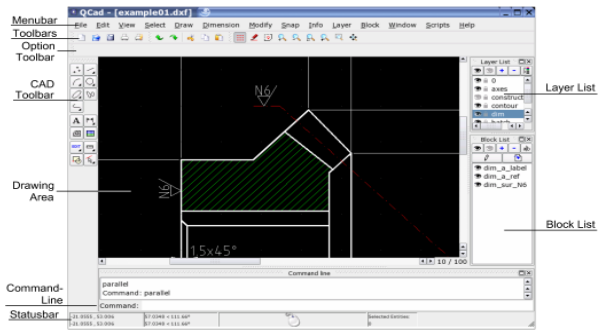
- Choose the desired line type in the options toolbar.
- Enter the bisector(s) length from the junction of the two lines in the toolbar Options. Choose the angle bisectors that you wish to generate in the second text box. The default is '1' however additional bisectors may be created as illustrated below.:

- Click the first line entity which limits the angle.
- Place the mouse pointer on the side of the line on which you wish to make the angle bisector and click the second line entity (s). The angle bisector(s) are shown in the preview before they are generated.

DISCUSS ITEMS

Main Application Window

Some of the major application window components are also frequently accessible in other apps (e.g. the menubar or the toolbars). Additional components are CAD or perhaps QCad specific. The names by which the individual components of the QCad user interface are referred to in this handbook are shown in Figure 8.



Menu and CAD Toolbar

You can access most of QCad's features via its top menu. For tasks linked to CAD, the CAD toolbar on the left may be more handy. The functionality that is presently most useful is always shown in this toolbar. If the starting point of a line is to be set it displays for instance the grabbing capability to set the starting point to an existing endpoint, a grid or to apply a different positioning method.

Layer List and Block List

The layer list and block list may be found to the right of the main window of QCad. They display the current drawing layers and blocks and provide some basic tools, which are unique to the layer or block. The View - Views - Layer List and View - Views - Block List menu may enable you to turn on or off the list.

Status Bar

The status bar at the bottom of the programme window displays different information about QCad's current state. On the left, the coordinate widget shows a cartesian and polar coordinate with an absolute and relative location of the mouse pointer. The middle mouse widget provides information about the left and right mouse buttons' current operation. The right-hand selection widget displays the number of chosen entities.

Using the option View - Views - Statusbar, you can enable or hide the status bar..

Command Widget

You may locate the QCad command line just above the status bar (Professional version only). It is used for entering commands and for warning and error notifications.

The command line can be enabled / disabled using the menu View - Views - Command-Line.

Option Toolbar

Figure 8 shows the toolbar option unfilled. It displays the choices accessible for the tool, depending on the current tool. You may wish to disable this toolbar using the View - Views - Option Toolbar menu if you prefer changing options and tool settings via the command line.

Executing Commands

Every operation in QCad is predicated on the execution of a command. QCad instructions can be run in three different ways.:

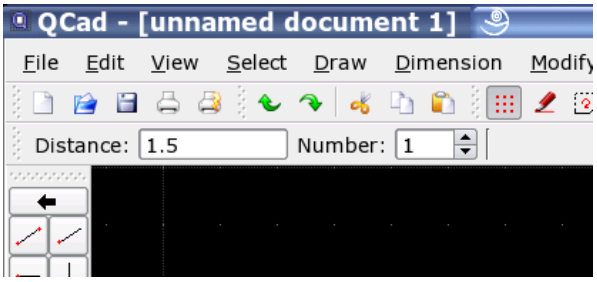
1. Choose a tool icon from one of the toolbars that corresponds to the task at hand.
2. From one of the pull-down menus, select a menu item.
3. Use the command prompt to type a command.

After being launched, most actions require additional user inputs. QCad can seek those inputs in three different ways:

- Dialogs. The text building tool, for example, displays a dialogue for selecting a font and entering the text entity's string.

- Option toolbar is a toolbar that allows you to change the settings of your computer. The tool for drawing parallels, for example, requires the input of the parallel's distance from the original item. The user can type the distance into a text input box in the tool settings toolbar at the top.

- Command-line interface. When drawing a series of lines, for example, you can end the sequence by putting 'close' into the command line.



Modes

In two modes, QCad operates: command mode and regular mode. It accepts instructions via the command line in command mode. All keyboard input flows to the programme interface in regular mode.

Command Mode

The command mode is shown on the left side of the command widget with a blue label and in the command line with a flashing cursor. Each keyboard input that may be printed enters the command line in this mode. To input instructions or co-ordinates, use this mode. For instance, a line may be entered to start the tool on the line. By pressing Z and the A key, the letters 'za' are shown in the command line instead of the command auto-zoom. When in normal mode, the command mode may be entered either by hitting the space key or clicking in the input box on the command line.

Normal Mode

QCad acts like any other standard programme in this mode. All keyboard entries are treated as hotkeys when they have been defined or have absolutely no impact. If you hit Z, then QCad will start auto zooming using the A key in this mode. Press the Escape key to go from Command Mode to Standard Mode. If there is any text in the input box on the command line, double-click Escape.

EXPLAIN TOOLBAR

Toolbars and menus. You may wish to add your own script-based QCAD tools to a customised orr menu, so they may be started in a smooth and consistent manner. These instructions demonstrate how the ECMA Script structure comes with QCAD handling the menus and toolbars.

- Command line. Spacebar. Activate command line. Esc. …

- File Handlng. Ctrl+N. Create a New drawing. Ctrl+O. ...

- Editing Shortcuts. OO/Ctrl+Z. Undo (oops) UU/Ctrl+Shift+Z. ...

- View Shortcuts. RD. Redraw. ZW. ...

- Layer Handling. FR* Freeze all layers. TH* ...

- Snapping. OS. Free positioning. SG. ...

- Construction. PO. Point. LI. ...

- Dimensioning. DA. Aligned Dimension. DH.

CONCLUSION:

The QCAD of this paper is to present the significance of mathematics thoughts in various fields of software applications. An outline is exhibited particularly to extend mathematics.

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