LinceCAD

User's Guide and Reference

Version 0.9.8-rc4

User's Guide

Welcome to LinceCAD

LinceCAD is a 2D technical drawing program that works directly with DWG and DXF files, the AutoCAD formats. It opens drawings as they arrive, edits them with the commands and aliases AutoCAD and progeCAD users already know, and saves them back to DWG or DXF, in whatever version the recipient needs.

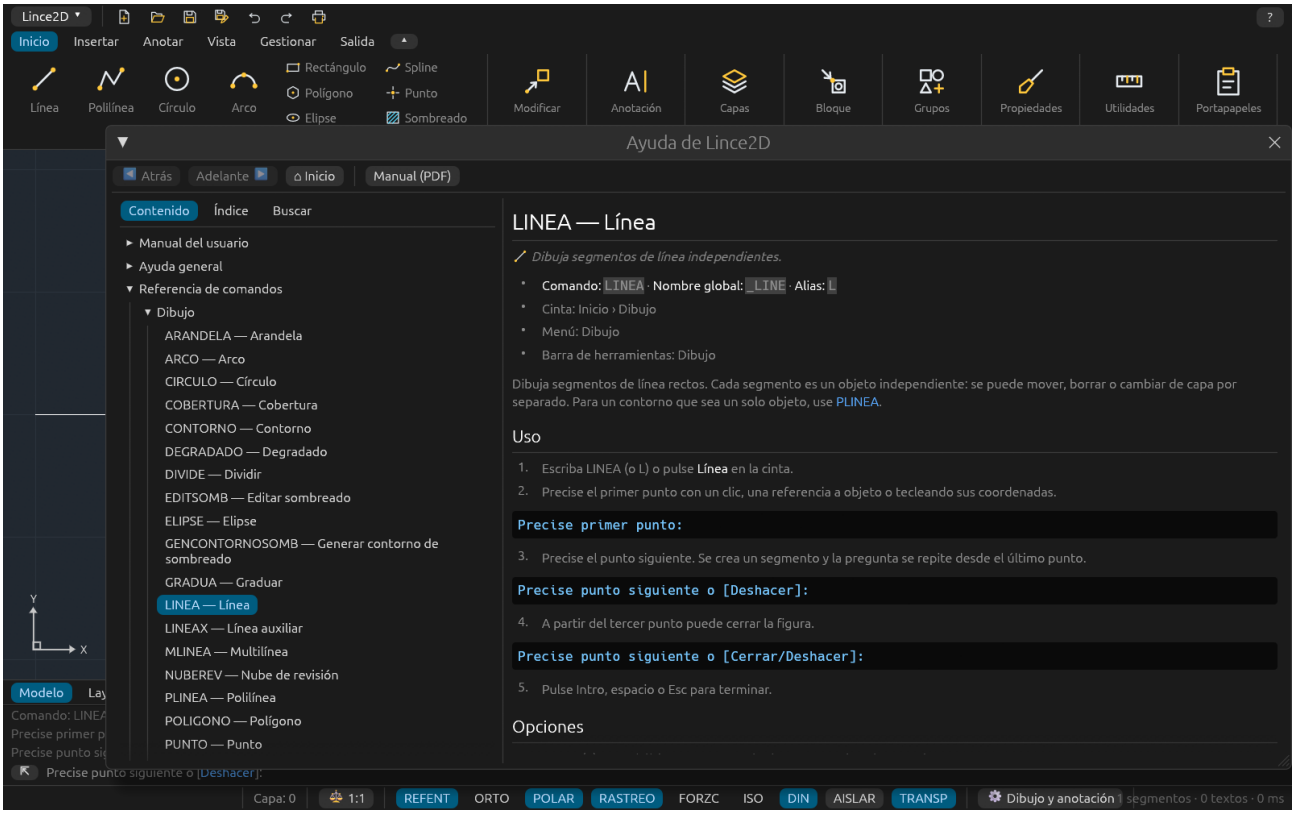
Who it is for

- For people who draw architectural, building services, civil engineering or surveying plans and exchange DWG files with others.
- For AutoCAD users: commands have the same names (in the interface language, or their English name prefixed with _), the command line prompts come in the same order and the usual aliases work (L, C, PL, O...).
- For beginners: the tutorials in this manual go from an empty drawing to a dimensioned plan with blocks, plotted to PDF.

What it includes

- 2D drawing and editing: lines, polylines, arcs, circles, hatches, text, dimensions, leaders and tables.
- Layers, linetypes, lineweights, blocks with attributes, dynamic blocks and external references.
- Layouts with their viewports and plotting to PDF with plot style tables (CTB and STB).
- Raster images, georeferenced images, coordinate systems (EPSG) and import of SHP, KML and GeoJSON.
- Basic AutoLISP and scripts (SCRIPT) to automate tasks.
- Spanish and English interface.

How the help is organized



- **F1** opens the help. With a command running, it opens that command's page directly. The HELP command does the same.
- **Contents**: the tree on the left, in three parts: this *User manual* (installation and step-by-step tutorials), the *General help* (concepts: interface, coordinates, layers, dimensions...) and the *Reference* (one page per command, the system variables and the AutoLISP functions).
- **Index**: terms in alphabetical order; type the first letters to jump to them.
- **Search**: looks for a word in every page.
- **PDF manual**: the Help > Manual (PDF) menu opens this same content as a single document, handy for printing or reading without the program.

Each command page starts with its name, its English global name, its aliases and where it is on the ribbon and in the menus. Prompts are shown exactly as they appear on the command line, for example:

```
Specify first point:
```

Where to start

1. Install LinceCAD.
2. First drawing: draw, save and open again.
3. The tutorials, in order: floor plan, dimensioning, blocks, plotting to PDF and georeferenced drawings.

If you already know AutoCAD, have a look at The interface, the alias table and What's new.

See also

The interface · File formats · Keyboard and shortcuts

Installing LinceCAD

LinceCAD is installed on Windows with a setup program and on Linux with a compressed package. In both cases it is installed for the current user, without administrator rights.

Requirements

- 64-bit Windows (x64) or 64-bit Linux (x86_64) with a graphical desktop.
- A graphics driver with OpenGL, as found on any computer from recent years.
- About 40 MB of disk space for the program and the symbol library.
- AutoCAD or any other program is not needed: LinceCAD reads and writes DWG and DXF files by itself.

Windows

The setup program is in Spanish for now; the steps below give the meaning of each option.

1. Run `LinceCAD-<version>-setup.exe`.
2. Choose whether to install for your user only (recommended, no administrator needed) or for all users of the computer. For your user, the program goes to `%LOCALAPPDATA%\Programs\lince`; for all users, to the program files folder.
3. Tick the optional tasks you want: - **Create a desktop shortcut**. - **Add LinceCAD to "Open with" for .dwg and .dxf files**: the program appears in the Explorer *Open with* menu when you right-click a DWG or DXF file. It does not change the default program: if AutoCAD opens those files on double-click, it still does.
4. Click **Install**. When it finishes you can start LinceCAD from the last page of the wizard or from the Start menu.

The `bibliotecas` folder is installed next to the program, with the 2D symbols used by the Blocks palette and DesignCenter (see Inserting blocks).

Updating

Run the setup program of the new version: it replaces the previous one in the same folder and keeps your preferences.

Uninstalling

Open Settings > Apps > Installed apps, find **LinceCAD** and click **Uninstall**. The program, its shortcuts and, if you chose it, the "Open with" entry are removed. User preferences (`%APPDATA%\lince`) are left in place in case you install it again; delete them by hand if you do not want them.

Microsoft Store

If you installed LinceCAD from Microsoft Store, the Store updates it and uninstalls it (Settings > Apps). Windows keeps the user data of that installation in its own folder, `%LOCALAPPDATA%\Packages\<LinceCAD package>\LocalCache\Roaming\lince`, apart from `%APPDATA%\lince`: the license, trial and preferences are separate from those of a normal installation, and they are deleted when you uninstall it. The program shows that folder and opens it with the "Open folder" buttons.

Linux

The package is `lince-<version>-linux-x86_64.tar.gz`. It can be used without installing:

```
tar xzf lince-<version>-linux-x86_64.tar.gz
cd lince-<version>-linux-x86_64
./lince
```

To install it for your user, with a launcher in the applications menu, run from that folder:

```
install -Dm755 lince ~/.local/bin/lince
install -Dm644 lince.desktop ~/.local/share/applications/lince.desktop
```

```
install -Dm644 lince.png ~/.local/share/icons/hicolor/256x256/apps/lince.png
mkdir -p ~/.local/share/lince && cp -r bibliotecas ayuda ~/.local/share/lince/
```

The last line copies the symbol library and this help to `~/.local/share/lince`, where the program looks for them when they are not next to the executable. To uninstall, delete those files and the `~/.config/lince` folder (preferences).

Language

On first start, LinceCAD uses the system language if it has it (Spanish or English) and Spanish otherwise. To change it, open OPTIONS, Interface tab, and choose the Language; the change applies at once. The number format (decimal comma or point) is chosen on the same tab.

On the command line you always type as in AutoCAD, whatever the language: decimal point and a comma between coordinates (`10.5,20`). Global command names (`_LINE`, `_CIRCLE`...) work in any language.

Languages can be added without reinstalling: a `<code>.ftl` catalog in the `idiomas` folder next to the executable or in the user configuration folder is enough.

See also

First drawing · Options · OPTIONS

First drawing

In this tutorial you create a drawing, draw a rectangle with lines and a circle, save it as DWG, close it and open it again. It takes about five minutes.

1. A new drawing

On start, LinceCAD shows an empty drawing. To start another one at any time, enter NEW, click **New** on the Quick Access toolbar or press Ctrl+N. Each open drawing has its own tab above the drawing area.

New drawings are metric: one drawing unit is one millimetre. Coordinates are typed on the command line, at the bottom, where `Command:` appears.

2. Drawing some lines

1. Enter LINE (or L) and press Enter.

```
Specify first point:
```

2. Type `0,0` and press Enter: this is the coordinate origin.

```
Specify next point or [Undo]:
```

3. Type `100,0` and Enter: the first segment is drawn, 100 mm to the right.
4. Type `@0,60` and Enter. The `@` means a relative coordinate: 60 mm up from the last point.

```
Specify next point or [Close/Undo]:
```

5. Type `@-100,0` and Enter: 100 mm to the left.
6. Type C (Close) and Enter: the line goes back to the first point and the command ends.

If you get a point wrong, type U (Undo) without leaving the command. After the command has finished, Ctrl+Z undoes the whole command.

3. Drawing a circle

1. Enter CIRCLE (or C) and Enter.

```
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
```

2. Type `50,30` and Enter: the centre of the rectangle.

```
Specify radius of circle or [Diameter]:
```

3. Type `20` and Enter.

To see the whole drawing on screen, double-click the mouse wheel or type Z, Enter, E (Extents) and Enter. The wheel zooms in and out; pressed and dragged, it pans the view.

4. Saving as DWG

1. Enter SAVEAS or press Ctrl+Shift+S.
2. In the dialog box, choose the folder, type the name `primero` and click **Save**.

The status bar confirms the file and the version. SAVEAS typed on the command line saves in the DWG 2018 format, the one used by AutoCAD 2018 to 2027. For another version, open the **LinceCAD** menu (top left), hover over **Save As** and choose from the list: DWG 2018, 2013, 2010, 2007, 2004 or 2000, or DXF 2018 or 2000. Choose an older version when the recipient has an old program.

From then on, QSAVE (Ctrl+S) saves to the same file and in the same version, without asking.

5. Closing and opening again

1. Enter CLOSE (or use **Close** in the **LinceCAD** menu). If there are unsaved changes, the program asks whether to save them.
2. Enter OPEN or press Ctrl+O, find `primero.dwg` and click **Open**. You can also drag the file from Explorer onto the program window.

The drawing opens in its own tab, as it was saved. LinceCAD opens DWG files from release R14 up to the current one and DXF files from R12 on; large drawings are read in the background, with a progress bar.

Notes

- Press Esc to cancel a command halfway, and Enter (or Space) on the empty command line to repeat the last one.
- The program saves an automatic copy of the drawing every so often; if it closes unexpectedly, opening the file again warns that there is a more recent copy.

See also

Coordinates and input · File formats · Floor plan with layers · LINE · CIRCLE · SAVEAS

Tutorial: floor plan with layers

In this tutorial you draw the plan of a simple 8 × 6 m dwelling: 25 cm outer walls, a 10 cm partition with a doorway, an entrance door and two windows, each on its own layer. The following tutorials dimension this plan, add furniture and plot it.

The drawing is in millimetres (the unit of new drawings). All coordinates are typed on the command line; press Enter after each value. A coordinate with `@` is relative to the last point: `@8000,0` is 8 m to the right.

1. Creating the layers

1. Start a new drawing with NEW.
2. Enter LAYER (or LA). The Layer Properties Manager opens.
3. Click **New Layer**. A layer appears with its name ready to type: type `Walls` and Enter.
4. Repeat step 3 for `Doors`, `Windows`, `Furniture`, `Dims` and `Text`.
5. In the **Color** column of each layer, drop down the colour and choose the one in the table. On the Walls layer, also change the **Lineweight** to 0.50 mm: walls will plot thicker.
6. Double-click the **Status** column of the Walls layer to make it current (the ? mark appears) and close the dialog.

Layer	Colour	For
Walls	1 (red), lineweight 0.50	walls, partitions and jambs
Doors	3 (green)	door leaves and swings
Windows	4 (cyan)	window frames
Furniture	8 (grey)	furniture (blocks tutorial)
Dims	2 (yellow)	dimensions (dimensioning tutorial)
Text	7 (white)	room names

The same, from the command line with -LAYER (useful in scripts): each line is one answer and the empty one at the end is the Enter that ends the command.

```
-LAYER
N
Walls,Doors,Windows,Furniture,Dims,Text
C
1
Walls
C
3
Doors
C
4
Windows
C
8
Furniture
C
2
Dims
LW
0.5
Walls
S
Walls
```

2. Outer walls with PLINE and OFFSET

1. Enter PLINE (or PL).

```
Specify first point:
```

2. Type `0,0`.

```
Specify next point or [Arc/Halfwidth/Length/Undo/Width]:
```

3. Type `@8000,0`, then `@0,6000` and then `@-8000,0`.
4. Type C (Close): the polyline closes. It is the outer face of the walls, a single object.
5. Double-click the mouse wheel to see the whole drawing.
6. Enter OFFSET (or O).

```
Specify offset distance or [Through/Erase/Layer] <…>:
```

7. Type `250`, the wall thickness.

```
Select object to offset or [Exit/Undo] <Exit>:
```

8. Click the polyline.

```
Specify point on side to offset or [Exit/Multiple/Undo] <Exit>:
```

9. Type `4000,3000` (a point inside the dwelling) or click inside. The inner face appears. Press Enter to finish.

3. The partition

1. Enter LINE (or L), `5000,250`, `5000,5750` and Enter: the line runs from wall to wall.
2. Enter OFFSET, `100`, click the line you have just drawn, type `6000,3000` and press Enter. The partition lies between x = 5000 and x = 5100: the living room on the left, the bedroom on the right.

4. Opening the gaps

Each opening is marked with two jambs (short lines across the wall) and then the wall is trimmed between them.

1. Enter LINE, `1000,0`, `@0,250` and Enter: the first jamb of the entrance door.
2. Enter COPY (or CO) to repeat it at the other openings.

```
Select objects:
```

3. Click the jamb and press Enter.

```
Specify base point:
```

4. Type `1000,0`.

```
Specify second point or <exit>:
```

5. Type, one after the other, `1900,0` (door), `2000,5750` and `3500,5750` (living room window), `5900,5750` and `7400,5750` (bedroom window). Press Enter to finish.
6. For the partition doorway: LINE, `5000,1000`, `@100,0` and Enter; then COPY that line with base point `5000,1000` and second point `5000,1800`.
7. Enter TRIM (or TR).

```
Select object to trim or shift-select to extend or [cuTting edges/Crossing/mOde/Project/eRase]:
```

8. In Quick mode (the AutoCAD 2027 default) you do not select edges: every object acts as a cutting edge.
9. Click each piece of wall between two jambs: at the door, the outer and the inner line; at each window, both; at the partition, its two lines (zoom in with the wheel if they are close together). That is eight clicks. Press Enter to finish. If you trim the wrong piece, finish and press Ctrl+Z.

5. The entrance door

1. Drop down the layer control on the ribbon and choose **Doors**: it becomes the current layer.
2. Draw the open leaf: LINE, `1000,250`, `@0,900` and Enter.
3. Draw the swing with an arc by its centre. Enter ARC (or A).

```
Specify start point of arc or [Center]:
```

4. Type C (Center).

```
Specify center point of arc:
```

5. Type `1000,250` (the hinge).

```
Specify start point of arc:
```

6. Type `1900,250` (the other jamb).

```
Specify end point of arc:
```

7. Type `1000,1150` (the end of the leaf). The arc is drawn counterclockwise, from the jamb to the leaf.

6. The windows

1. Make the **Windows** layer current.
2. Draw the two lines on the wall faces: LINE, `2000,5750`, `@1500,0`, Enter; and LINE, `2000,6000`, `@1500,0`, Enter.
3. Draw the glass with RECTANG (or REC).

```
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
```

4. Type `2000,5850`.

```
Specify other corner point or [Area/Dimensions/Rotation]:
```

5. Type `@1500,50`.
6. Copy the window to the bedroom: COPY, click the three pieces of the window, Enter, base point `2000,5750` and second point `5900,5750`, Enter.

7. Saving

Enter SAVEAS and save the drawing as `plano.dwg`. The next tutorial starts from it.

Notes

- Instead of typing, points can be specified with the mouse and object snaps (F3), such as *Endpoint* or *Intersection*, and with Ortho (F8) for horizontal and vertical lines. See Object snaps and precision.
- The partition doorway is left open: in the blocks tutorial a door from the library is placed in it.

See also

Layers · Coordinates and input · Editing · Dimensioning the plan · PLINE · OFFSET · TRIM · COPY

Tutorial: dimensioning the plan

In this tutorial you dimension the plan from the previous tutorial and label the rooms. You create a dimension style for plotting at 1:50, draw chains of dimensions with DIMLINEAR and DIMCONTINUE and write text with TEXT.

Open `plano.dwg`. As before, press Enter after each typed value.

1. A dimension style for 1:50

The drawing is in millimetres and will be plotted at 1:50. The Standard style has text and arrows of 2.5 units: at 1:50 they would come out 0.05 mm high. The overall scale of the style multiplies all those sizes.

1. Enter DIMSTYLE (or D). The Dimension Style Manager opens.
2. Click **New...**. Create New Dimension Style appears below.
3. In **New Style Name:** type `Plan50`; in **Start With:** leave Standard. Click **Continue**.
4. Modify Dimension Style opens. On the **Fit** tab, set **Overall scale** = `50`.
5. On the **Primary Units** tab, set **Precision (decimal places)** = `0`: measurements will show in whole millimetres (`8000`).
6. Click **OK**.
7. In the manager, with Plan50 selected, click **Set Current** and close the dialog.

With a scale of 50, the 2.5 text is 125 mm high in the drawing and 2.5 mm on paper.

2. Dimension chain along the lower façade

1. Make the **Dims** layer current in the layer control.
2. Enter DIMLINEAR (or DLI).

```
Specify first extension line origin or <select object>:
```

3. Type `0,0` (corner of the wall).

```
Specify second extension line origin:
```

4. Type `1000,0` (first jamb of the door).

```
Specify dimension line location:
```

5. Type `500,-600`: the dimension line sits 60 cm below the wall. It measures 1000.
6. Enter DIMCONTINUE (or DCO). It starts from the last dimension drawn.

```
Specify a second extension line origin or [Undo/Select] <Select>:
```

7. Type `1900,0` and then `8000,0`. Each point adds a dimension next to the previous one: 900 (the door) and 6100. Press Enter to finish.
8. Add the overall dimension outside: DIMLINEAR, `0,0`, `8000,0`, `4000,-1200`.

If a point in the chain goes wrong, type U (Undo) in DIMCONTINUE: it removes the last dimension without leaving the command.

3. Dimensions on the left side and the upper façade

1. Left side: DIMLINEAR, `0,0`, `0,6000`, `-1200,3000`. As the location is to one side, the dimension is vertical: 6000.
2. Upper façade, with the windows: DIMLINEAR, `0,6000`, `2000,6000`, `1000,6600`.
3. DIMCONTINUE, `3500,6000`, `5900,6000`, `7400,6000`, `8000,6000` and Enter. The chain measures 2000, 1500, 2400, 1500 and 600.

4. Interior dimensions

1. DIMLINEAR, `250,2000`, `5000,2000`, `2625,2000`: the clear width of the living room, 4750. The dimension line runs inside the room.
2. DIMCONTINUE, `5100,2000`, `7750,2000` and Enter: the partition (100) and the bedroom (2650).

5. Room names

1. Make the **Text** layer current.
2. Enter TEXT (or DT).

```
Specify start point of text or [Justify/Style]:
```

3. Type J (Justify).

```
Enter an option [Left/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR]:
```

4. Type MC: the text is centred on the point, horizontally and vertically.
5. Specify the point: `2625,1500`.

```
Specify height <…>:
```

6. Type `200` (4 mm on paper at 1:50).

```
Specify rotation angle of text <…>:
```

7. Press Enter to keep 0.

```
Enter text:
```

8. Type `LIVING ROOM` and Enter. The command asks for another line below: press Enter on the empty line to finish.
9. Repeat for the bedroom: TEXT, J, MC, `6425,1500`, `200`, Enter, `BEDROOM`, Enter, Enter.
10. Add a title under the plan: TEXT, `0,-2200`, height `250`, Enter, `GROUND FLOOR - 1:50`, Enter, Enter.

6. Saving

Press Ctrl+S (QSAVE): the drawing is saved to `plano.dwg`, in the same version.

Notes

- Dimensions are drawn with the current style and on the current layer. To change the style of a dimension later, select it and change it in the PROPERTIES palette; when a style is modified, all its dimensions are redrawn.
- DIMBASELINE makes chains from a common origin, each dimension further out than the previous one.
- To change the text of a dimension (for example, to add a prefix), use DIMEDIT.
- TEXT always proposes 2.5 as the height: type the one you want for each text. For multi-line formatted text, use MTEXT.

See also

Dimensions · Text · Inserting blocks · DIMSTYLE · DIMLINEAR · DIMCONTINUE · DIMSCALE

Tutorial: inserting blocks from the library

In this tutorial you furnish the plan from the previous tutorials with symbols from the library installed with LinceCAD: a door in the partition doorway, furniture and a socket with its attribute. You use the Blocks palette (INSERT) and ADCENTER.

Open `plano.dwg`.

The included library

The `bibliotecas` folder, next to the program, holds two drawings with 2D symbols in millimetres, on layer 0 (they take the layer they are inserted on). Block names are in Spanish:

- **Arquitectura.dwg** (architecture): swing doors PUERTA_70, PUERTA_80 and PUERTA_90 (base point at the hinge), PUERTA_DOBLE_160 (double door) and PUERTA_CORREDERA_80 (sliding door); windows VENTANA_60, VENTANA_120 and VENTANA_180 (for a 15 cm wall); INODORO (WC), LAVABO (washbasin), BANERA_170 (bath), PLATO_DUCHA_80 (shower tray), FREGADERO_2 (sink); SILLA (chair), MESA_4_SILLAS (table with four chairs), CAMA_150 and CAMA_90 (beds), SOFA_3 and ARMARIO_120 (wardrobe).
- **Electricidad.dwg** (electrical): ENCHUFE and ENCHUFE_DOBLE (sockets), INTERRUPTOR (switch), CONMUTADOR (two-way switch), PULSADOR (push button), PUNTO_LUZ (ceiling light), APLIQUE (wall light), FLUORESCENTE_120 (linear luminaire), TOMA_TV (TV outlet), TOMA_DATOS (data outlet) and CUADRO_ELECTRICO (distribution board). Most carry the CIR attribute (circuit).

Opened as drawings, each one shows its catalogue: every symbol with its name below.

1. The Blocks palette

1. Enter INSERT (or I), or click **Insert** on the ribbon. The **Blocks** palette opens, with three tabs: **Current Drawing** (the blocks already in the drawing), **Recent** and **Libraries**.
2. Click **Libraries**. The library folder appears with its two drawings and the thumbnails of their blocks. The **Search** box filters by name: type `puerta` to see only the doors.
3. Below are the **Insertion Options**: the **Insertion Point** is specified on screen; the **Scale** is 1 and the **Rotation** 0. Leave them like that: the symbols are drawn at real size.

The palette can stay open while you draw.

2. The partition door

1. Make the **Doors** layer current.
2. In the palette, click **PUERTA_80**. The command line starts inserting and the block follows the cursor.

```
Specify insertion point or [Scale/Rotate]:
```

3. The library door opens upwards; in the partition it has to open towards the living room. Type R (Rotate).

```
Specify rotation angle <0>:
```

4. Type `90`.
5. It asks for the point again: type `5000,1000`, the corner of the doorway on the living room face. The hinge is at the bottom and the leaf opens to the left, with its swing inside the living room.

The block is copied from the library file into the drawing the first time; from then on it also appears on the Current Drawing tab.

3. Living room furniture

1. Make the **Furniture** layer current.
2. Click **MESA_4_SILLAS** and type `2600,3300`: the table is centred on that point, with its four chairs (a SILLA block inside another block).

3. Click **SOFA_3** and type `2800,4750`: the base point is the lower left corner and the back rests against the wall.

To place the same block several times, tick **Repeat Placement**: after each insertion it asks for the point again, until you press Enter or Esc.

4. The bedroom with DesignCenter

DesignCenter browses other drawings (open ones or those in the library folders) and copies blocks, layers, styles and layouts from them.

1. Enter ADCENTER (or ADC). **DesignCenter** opens.
2. In the tree on the left, under the library folder, choose **Arquitectura.dwg**.
3. Click **Blocks** in the list of categories: its thumbnails appear.
4. Double-click **CAMA_150** and type `5700,3500`.
5. Click **ARMARIO_120**, click **Insert** and type `6400,250`: the wardrobe stands against the lower wall.

With the other categories (Layers, Text Styles, Dimension Styles...) the **Add to Drawing** button copies the chosen item into the drawing; if an entry with that name already exists, the drawing's own is kept.

5. A socket with an attribute

1. In the **Blocks** palette, **Libraries** tab, search for `enchufe` and click **ENCHUFE** (from Electricidad.dwg).
2. Type `5600,250`: the socket sits on the inner face of the lower wall.
3. The block has an attribute, and the program asks for its value with the block's own prompt, `Circuito <C2>:`. Type `C3` and Enter (or press Enter to keep C2).

The value can be changed later by double-clicking the block (ATTEDIT) or with EATTEDIT.

6. Saving

Press Ctrl+S. The plan now has walls, doors and windows, dimensions, text and furniture: the next tutorial plots it to PDF.

Notes

- On the Current Drawing tab, **Browse…** inserts another whole DWG or DXF as a block.
- **Add Folder…** (Libraries tab) adds your own folder of drawings with blocks: it appears in the palette and in DesignCenter.
- -INSERT asks everything on the command line (name, point, scale and rotation); it is the one scripts use.
- To keep the blocks you use most at hand, with their layer, scale and rotation, drag them to a tool palette (TOOLPALETTES).
- To create your own blocks, see BLOCK and WBLOCK.

See also

Blocks and attributes · Plotting to PDF · INSERT · ADCENTER · BEDIT

Tutorial: plotting to PDF

In this tutorial you plot the plan from the previous tutorials to a PDF file, at 1:50 on an A4 sheet, in black and white with the `monochrome.ctb` plot style table. Then you see how layouts, with their viewports, are plotted.

Open `plano.dwg` and check that the **Model** tab, below the drawing area, is active.

1. The Plot dialog

Enter PLOT (or PRINT) or press Ctrl+P. The **Plot** dialog opens. LinceCAD plots to vector PDF: lines, text (with its fonts) and hatches come out as on screen, at any magnification.

1. **Printer/plotter:** leave *DWG To PDF.pc3 (lince PDF)*. It is the same name AutoCAD uses, so the settings saved in the drawing work for it too.
2. **Paper size:** choose *ISO full bleed A4 (297.00 x 210.00 mm)*.
3. **Drawing orientation:** select **Landscape**.

2. What to plot: a window

1. In **What to plot:** choose **Window** and click **Window <**. The dialog hides and the command line asks for the corners.

```
Specify first corner:
```

2. Type `-1800,-2400` (bottom left, outside the dimensions and the title).

```
Specify opposite corner:
```

3. Type `9200,7200`. The Plot dialog comes back.

The other options are **Extents** (everything drawn), **Display** (what is on screen) and **Limits**.

3. The scale

1. In **Plot scale:** clear **Fit to paper**.
2. Choose **1:50** from the list. Below it you can read the equivalence: 1 mm of paper = 50 drawing units. As the drawing is in millimetres, 1 m of the plan measures 2 cm on paper.
3. Tick **Center the plot**: the window is placed in the middle of the sheet.

The window measures 11 × 9.6 m: at 1:50 it takes 220 × 192 mm and fits on the landscape A4.

4. The plot style table

The plot style table (CTB) decides the colour, lineweight and screening each drawing colour is plotted with.

1. In **Plot style table (pen assignments):** choose **monochrome.ctb**: everything comes out black, whatever its colour on screen. With **grayscale.ctb** colours become greys.
2. In **Plot options:** leave **Plot object lineweights** ticked (walls, at 0.50 mm, come out thicker) and **Plot with plot styles** (without this box the table is not applied).

Your own tables (`.ctb` files, or `.stb` if the drawing uses named plot styles) are copied to the `%APPDATA%\lince\estilos_trazado` folder or to the drawing folder, and appear in the same list.

5. Preview and PDF

1. Click **Preview…** to see the sheet as it will come out. Close the preview to return to the dialog.
2. If you want the drawing to remember these settings for next time, tick **Apply to Layout**.
3. Click **OK**, choose the folder and type the name `plano.pdf`. The command line confirms the file and its size.

The same dialog saves PNG or SVG instead of PDF: just type the name with that extension. The PNG resolution is chosen in **PNG:**.

6. Layouts and viewports

Besides Model, a drawing can have layouts (paper space): each tab is a sheet with its title block and one or more viewports that show the model at a scale. Drawings coming from AutoCAD usually have them ready.

1. Click the layout tab. You see the sheet, its printable area and the viewports.
2. Double-click inside a viewport to enter the model through it (the status bar button changes from **PAPER** to **MODEL**); double-clicking outside it, or PSPACE, goes back to paper.
3. With a viewport active, the sun-with-viewport icon of the layer control freezes a layer in that viewport only; in the Layer Properties Manager it is the **VP Freeze** column. That way each viewport shows its own content.
4. Whatever you draw with the layout active and outside the viewports goes on paper: a title block, a legend.
5. To plot it, enter PLOT on that tab. **What to plot:** proposes **Layout** and a 1:1 scale: the sheet comes out at its size and each viewport keeps its own scale.

New layouts and viewports are created with LAYOUT and MVIEW; the viewport scale is set in the status bar. For a plan drawn from scratch you can also plot from Model as in steps 1 to 5.

Notes

- PAGESETUP saves named page setups to reuse them or import them from another drawing.
- PREVIEW opens the preview directly, with the current settings.
- To print on paper, choose the printer in **Printer/plotter:**.

See also

Layouts and plotting · Georeferenced drawings · PLOT · PAGESETUP · MSPACE

Tutorial: georeferenced drawings

In this tutorial you set up a drawing in real-world coordinates: you give the drawing a coordinate system, insert a georeferenced orthophoto and overlay the parcels of a SHP file, which LinceCAD converts to the drawing's system if it comes in another one.

You need two sets of files for the same area, like those published by official mapping agencies:

- An orthophoto in GeoTIFF, or in TIFF, JPG or PNG with its *world file* (`.tfw`, `.jgw`, `.pgw` or `.wld`) next to it. Here it is called `ortofoto.tif`.
- A SHP with its companion files (`.shx`, `.dbf` and, preferably, `.prj`). Here it is called `parcelas.shp`.

The example uses ETRS89 / UTM zone 30N (EPSG:25830), the official system in much of mainland Spain; use the one for your area. In UTM, one drawing unit is one metre.

1. The drawing's coordinate system

1. Start a drawing with NEW.
2. Enter GEOGRAPHICLOCATION (or GEO). The Geographic Location dialog opens and, at the same time, the command line asks:

```
Enter EPSG code of the coordinate system or [Remove/List] <…>:
```

3. Type `25830` and Enter (or choose the system from the list in the dialog and click OK). The command line confirms the chosen system.

This is the **target** system: everything imported with its own system is converted to it. Type L (List) to see the available systems: ETRS89, WGS84 and ED50 in zones 28 to 31, REGCAN95, Web Mercator (3857) and geographic coordinates. The AutoCAD code, such as `ETRS89.UTM-30N`, is accepted too.

The drawing is assumed to be, or to be going to be, in coordinates of that system: GEOGRAPHICLOCATION does not move what is already drawn.

2. The orthophoto

1. Create an `Orthophoto` layer and make it current, so that you can turn it off or lock it later.
2. Enter GEOIMAGE.

```
Enter image file name to attach or [Browse] <Browse>:
```

3. Press Enter (Browse) and choose `ortofoto.tif`.
4. The image is placed at its coordinates, with its pixel size and rotation, read from the world file or the GeoTIFF tags. The command line says which georeferencing file it used and which system the image is in.
5. Double-click the mouse wheel to see the whole image.

If the image states its coordinate system and it differs from the drawing's, it is reprojected on insertion. If it does not, it is inserted as it is: its coordinates must already be in the drawing's system.

3. The SHP parcels

1. Create a `Parcels` layer, with a colour that stands out on the photo (for example 2, yellow), and make it current.
2. Enter MAPIMPORT (or SHP2CAD). The **Import SHP, KML or GeoJSON** dialog opens.
3. In **File** click **Browse…** and choose `parcelas.shp`. The dialog shows the **Contents**: how many features and how many fields it has.
4. **Layer from field**: leave none so that everything goes to the current layer, or choose a field of the table (for example, the land use or type field): each feature goes to a layer named after the value of that field, created if it does not exist.
5. **Block for the points**: only matters if the file has points. With (point object), each point is a POINT object; with a block of the drawing, each point is that block, and its attributes take the values of the fields with the same

name.

6. **File coordinate system**: the one stated in the `.prj` appears. If the file has no `.prj`, choose its system here; with (no reprojection) the coordinates are imported as they are.
7. **Drawing coordinate system** shows EPSG:25830. Click **OK**.

The parcels are drawn as closed polylines (polygon holes too) over the orthophoto. If the SHP was in another system, for example zone 29 or geographic coordinates, it has already been converted. The table data of each feature is kept with it, as extended data.

The same can be done on the command line, for example in a script:

```
SHP, KML, KMZ or GeoJSON file:
```

```
Field for the layer (…) or [None] <None>:
```

```
Block name for the points <point object>:
```

4. Checking coordinates

1. Enter GEOCOORD.

```
Specify point or [System] <EPSG:…>:
```

2. Click a parcel corner: the command line shows its drawing coordinates and its WGS84 latitude and longitude. With S (System) you choose another target system, for example `4326` or `23030` (ED50).
3. With the drawing georeferenced, clicking the coordinates on the status bar toggles them with the latitude and longitude of the cursor.

5. Saving and sharing

- Save as DWG: the coordinate system is saved the way AutoCAD saves it, and AutoCAD recognizes it when it opens the drawing (GEOGRAPHICLOCATION).
- The image is not stored inside the DWG: the drawing points to the file. Send the orthophoto and its world file along with the drawing.
- MAPEXPORT writes the selected objects to SHP, KML, KMZ or GeoJSON, with the table data they came with. KML and GeoJSON are in WGS84: a KMZ opens directly in map viewers.

Notes

- Image formats: TIFF and GeoTIFF, JPG, PNG, BMP, GIF and WebP. ECW and JPEG 2000 are not read: convert them to GeoTIFF first.
- If the image has no georeferencing, place it with GEOREFIMAGE, giving two or three points on the image and their real position.
- So that the orthophoto does not cover the parcels, send it to the back with DRAWORDER. IMAGEADJUST changes its brightness and contrast, and TRANSPARENCY lets what is underneath show through its transparent pixels.
- With the ED50 datum, the conversion to ETRS89 uses the official IGN grid (`SPED2ETV2.gsb`) if it is in the program's `rejillas` folder (centimetre accuracy); otherwise, a 7-parameter transformation (? 1.5 m).

See also

Georeferencing · Raster images · GEOGRAPHICLOCATION · GEOIMAGE · MAPIMPORT · GEOCOORD

Appendix: command aliases

An alias is a short way of typing a command on the command line: L instead of LINE, C instead of CIRCLE. LinceCAD uses the usual AutoCAD aliases in each language, so muscle memory works just the same.

- Type the alias and press Enter or Space, as if it were the full name.
- Every command also has an English **global name** prefixed with _ (`_LINE`, `_CIRCLE`), which works in any interface language: it is the one to use in scripts and in AutoLISP.
- As you type, the command line suggests the commands starting with those letters; an exact alias always comes first.

Your own aliases

Aliases can be changed and added with ALIASEDIT: they are saved in `lince.pgp`, in the configuration folder (`%APPDATA%\lince`; on Linux, `~/.config/lince`), with the syntax of AutoCAD's `acad.pgp` (`LL, *LINE`). An AutoCAD acad.pgp can be imported as it is: its external commands are ignored.

- Your own alias wins over a built-in one with the same letters, but not over a command name.
- With _ or . in front, your own aliases are not used.
- They apply at once, without restarting.

Aliases changed in 1.0.0

So that the aliases are those of the AutoCAD 2027 `acad.pgp` (the English ones in English and the Spanish ones in Spanish), these have changed. If you prefer the old ones, add them with ALIASEDIT.

Alias	Before	Now
LO	Layout (PRESENTACION)	-LAYOUT (-PRESENTACION), as in AutoCAD
DX	ATTEXT (EXTRACCATR)	—(DATAEXTRACTION in AutoCAD)
BVS	BVSTATE (PARAMVISIBILIDAD)	—

Also:

- With the English interface the Spanish aliases no longer work (they clashed with AutoCAD's English aliases, such as R, IM or ALI); the Spanish command names still do.
- With a leading _, English aliases work in any interface language (`_DDEDIT`, `_P`), as in AutoCAD.
- The global name of BVPARAM is `_BVPARAM` (`_BVSTATE` is another AutoCAD command); in English it is BVPARAM. DIMEDIT's Spanish name is now ACOEDIC, as in AutoCAD.
- In Spanish, the aliases follow AutoCAD 2027 in Spanish: POL is PLINE, U is JOIN, RR is TRIM, CO is SLICE… (see the Spanish version of this page).

Built-in aliases

The table lists every command with its name, its global name and its aliases in the language of this page.

Alias	Command	Global name
-ATE, ATTE	-ATTEDIT	_-ATTEDIT
-ATT	-ATTDEF	_-ATTDEF
-B	-BLOCK	_-BLOCK
-BE	-BEDIT	_-BEDIT
-CH	CHANGE	_CHANGE
-COL	-COLOR	_-COLOR
-I	-INSERT	_-INSERT
-IM	-IMAGE	_-IMAGE

-LA	-LAYER	_-LAYER
-LW	-LWEIGHT	_-LWEIGHT
-OS	-OSNAP	_-OSNAP
-PRINT	-PLOT	_-PLOT
-PU	-PURGE	_-PURGE
-REN	-RENAME	_-RENAME
-V	-VIEW	_-VIEW
-VP, VP	VPOINT	_VPOINT
-W	-WBLOCK	_-WBLOCK
-XB	-XBIND	_-XBIND
-XR	-XREF	_-XREF
3DO, ORBIT	3DORBIT	_3DORBIT
3DPRINT, 3DP, 3DPLOT, RAPIDPROTOTYPE	STLOUT	_STLOUT
3P	3DPOLY	_3DPOLY
A	ARC	_ARC
AA	AREA	_AREA
ADC, DC, DCENTER, CONTENT	ADCENTER	_ADCENTER
AIA	AIACTIONS	_AIACTIONS
AL	ALIGN	_ALIGN
AP	APPLOAD	_APPLOAD
AR	ARRAY	_ARRAY
ATE, DDATTE	ATTEDIT	_ATTEDIT
ATI	ATTIPEDIT	_ATTIPEDIT
ATT, DDATTDEF	ATTDEF	_ATTDEF
B, ACADBLOCKDIALOG, BMAKE, BMOD	BLOCK	_BLOCK
BC	BCLOSE	_BCLOSE
BE	BEDIT	_BEDIT
BO, BPOLY	BOUNDARY	_BOUNDARY
BR	BREAK	_BREAK
BS	BSAVE	_BSAVE
BTEST	BTESTBLOCK	_BTESTBLOCK
BVS	BVSTATE	_BVSTATE
C	CIRCLE	_CIRCLE
CADASTRE	CADASTREDOWNLOAD	_CADASTREDOWNLOAD
CELLS	TABLECELL	_TABLECELL
CHA	CHAMFER	_CHAMFER
CHK	CHECKSTANDARDS	_CHECKSTANDARDS
CL	CENTERLINE	_CENTERLINE
CLB	CONTOURLABEL	_CONTOURLABEL
CM	CENTERMARK	_CENTERMARK
CN, ISOLINES	CONTOUR	_CONTOUR
CO, CP	COPY	_COPY

CODETABLE	SURVEYCODES	_SURVEYCODES
COL, COLOUR, DDCOLOR	COLOR	_COLOR
CONVERTIRIMAGEN	CONVERTIMAGE	_CONVERTIMAGE
CYL	CYLINDER	_CYLINDER
D, DST, DIMSTY, DDIM	DIMSTYLE	_DIMSTYLE
DAL, DIMALI	DIMALIGNED	_DIMALIGNED
DAN, DIMANG	DIMANGULAR	_DIMANGULAR
DAR	DIMARC	_DIMARC
DBA, DIMBASE	DIMBASELINE	_DIMBASELINE
DBR	DIMBREAK	_DIMBREAK
DCE	DIMCENTER	_DIMCENTER
DCO, DIMCONT	DIMCONTINUE	_DIMCONTINUE
DDA	DIMDISASSOCIATE	_DIMDISASSOCIATE
DDATTEXT	ATTEXT	_ATTEXT
DDI, DIMDIA	DIMDIAMETER	_DIMDIAMETER
DDINSERT	CLASSICINSERT	_CLASSICINSERT
DDPLOTSTAMP	PLOTSTAMP	_PLOTSTAMP
DDPTYPE	PTYPE	_PTYPE
DDVP	DDVPOINT	_DDVPOINT
DED, DIMED	DIMEDIT	_DIMEDIT
DI	DIST	_DIST
DIMTED	DIMTEDIT	_DIMTEDIT
DIV	DIVIDE	_DIVIDE
DJL	DIMJOGLINE	_DIMJOGLINE
DJO, JOG	DIMJOGGED	_DIMJOGGED
DL	DATALINK	_DATALINK
DLI, DIMLIN, DIMHORIZONTAL, DIMROTATED, DIMVERTICAL	DIMLINEAR	_DIMLINEAR
DLU	ACTUALIZARDATOS	_DATALINKUPDATE
DO, DOUGHNUT	DONUT	_DONUT
DOR, DIMORD	DIMORDINATE	_DIMORDINATE
DOV, DIMOVER	DIMOVERRIDE	_DIMOVERRIDE
DR	DRAWORDER	_DRAWORDER
DRA, DIMRAD	DIMRADIUS	_DIMRADIUS
DRE	DIMREASSOCIATE	_DIMREASSOCIATE
DRM	DRAWINGRECOVERY	_DRAWINGRECOVERY
DS, SE, DDRMODES	DSETTINGS	_DSETTINGS
DSP	DIMSPACE	_DIMSPACE
DT, DTEXT	TEXT	_TEXT
DWGCOMPARE	COMPARE	_COMPARE
DX, EXTRACDAT	DATAEXTRACTION	_DATAEXTRACTION
DXFOUT	SAVEAS	_SAVEAS
E, DELETE	ERASE	_ERASE

ED, DDEDIT, TEDIT	TEXTEDIT	_TEXTEDIT
EDITARRAY	ARRAYEDIT	_ARRAYEDIT
EDITTABLE	TABLEDIT	_TABLEDIT
EDS	EDITSURFACE	_EDITSURFACE
EL	ELLIPSE	_ELLIPSE
EX	EXTEND	_EXTEND
EXIT	QUIT	_QUIT
EXP	EXPORT	_EXPORT
EXPORTTABLE	TABLEEXPORT	_TABLEEXPORT
EXT	EXTRUDE	_EXTRUDE
F	FILLET	_FILLET
FI	FILTER	_FILTER
FINDPOINT, ZTP	ZOOMTOPOINT	_ZOOMTOPOINT
FLD	FIELD	_FIELD
FND	FIND	_FIND
FREEPOINT	POINTLIGHT	_POINTLIGHT
FSHOT	FLATSHOT	_FLATSHOT
G	GROUP	_GROUP
GD	GRADIENT	_GRADIENT
GEO, NORTH, NORTHDIR	GEOGRAPHICLOCATION	_GEOGRAPHICLOCATION
H, BH	HATCH	_HATCH
HB	HATCHTOBACK	_HATCHTOBACK
HE	HATCHEDIT	_HATCHEDIT
I, INSERTURL	INSERT	_INSERT
IAD	IMAGEADJUST	_IMAGEADJUST
IAT	IMAGEATTACH	_IMAGEATTACH
ICL	IMAGECLIP	_IMAGECLIP
IED	IMAGEEDIT	_IMAGEEDIT
IFIX	IMAGEPERSP	_IMAGEPERSP
IMPORTARMALLA, IMP	IMPORT	_IMPORT
IMPORTPDF	PDFIMPORT	_PDFIMPORT
IN	INTERSECT	_INTERSECT
IO	INSERTOBJ	_INSERTOBJ
IPIN	IMAGEPAINT	_IMAGEPAINT
IREC	IMAGECROP	_IMAGECROP
ISOD	ISODRAFT	_ISODRAFT
ISOLATE	ISOLATEOBJECTS	_ISOLATEOBJECTS
J	JOIN	_JOIN
L	LINE	_LINE
LA, DDLMODES	LAYER	_LAYER
LABELSTYLE	POINTLABELSTYLE	_POINTLABELSTYLE
LAS, LMAN	LAYERSTATE	_LAYERSTATE

LE	QLEADER	_QLEADER
LEAD	LEADER	_LEADER
LEN	LENGTHEN	_LENGTHEN
LI, LS, SHOWMAT	LIST	_LIST
LINEWORK	PROCESSLINEWORK	_PROCESSLINEWORK
LO	-LAYOUT	_-LAYOUT
LT, LTYPE, DDLTYPE	LINETYPE	_LINETYPE
LW, LINEWEIGHT	LWEIGHT	_LWEIGHT
M	MOVE	_MOVE
MA, PAINTER	MATCHPROP	_MATCHPROP
MAT, RMAT	MATBROWSER	_MATBROWSER
ME	MEASURE	_MEASURE
MEA	MEASUREGEOM	_MEASUREGEOM
MI	MIRROR	_MIRROR
ML	MLINE	_MLINE
MLA	MLEADERALIGN	_MLEADERALIGN
MLABEL	MOVEPOINTLABEL	_MOVEPOINTLABEL
MLC	MLEADERCOLLECT	_MLEADERCOLLECT
MLD	MLEADER	_MLEADER
MLE	MLEADEREDIT	_MLEADEREDIT
MLS	MLEADERSTYLE	_MLEADERSTYLE
MORE	MESHSMOOTHMORE	_MESHSMOOTHMORE
MS	MSPACE	_MSPACE
MV	MVIEW	_MVIEW
O	OFFSET	_OFFSET
OBJSC	OBJECTSCALE	_OBJECTSCALE
OP, PREFERENCES	OPTIONS	_OPTIONS
OPENURL, DXFIN	OPEN	_OPEN
OS, DDOSNAP	OSNAP	_OSNAP
P	PAN	_PAN
PA	PASTESPEC	_PASTESPEC
PARAM	BPARAMETER	_BPARAMETER
PATHARRAY	ARRAYPATH	_ARRAYPATH
PDFA	PDFATTACH	_PDFATTACH
PE	PEDIT	_PEDIT
PL	PLINE	_PLINE
PO	POINT	_POINT
POFF	HIDEPALETTES	_HIDEPALETTES
POL	POLYGON	_POLYGON
PON	SHOWPALETTES	_SHOWPALETTES
PR, CH, MO, PROPS, DDCHPROP, DDMODIFY	PROPERTIES	_PROPERTIES
PRE	PREVIEW	_PREVIEW

PRINT, DWFOUT	PLOT	_PLOT
PROFILE	PROFILEVIEW	_PROFILEVIEW
PS	PSPACE	_PSPACE
PSOLID	POLYSOLID	_POLYSOLID
PU	PURGE	_PURGE
PVUPDATE	PROFILEVIEWUPDATE	_PROFILEVIEWUPDATE
PYR	PYRAMID	_PYRAMID
QC	QUICKCALC	_QUICKCALC
QD	QDIM	_QDIM
QP	QUICKPROPERTIES	_QUICKPROPERTIES
R	REDRAW	_REDRAW
RA	REDRAWALL	_REDRAWALL
RE	REGEN	_REGEN
REA	REGENALL	_REGENALL
REC, RECTANGLE	RECTANG	_RECTANG
RECOGNIZETEXT, OCR	OCRIMAGE	_OCRIMAGE
REFINE	MESHREFINE	_MESHREFINE
REG	REGION	_REGION
REN	RENAME	_RENAME
RENUMPT	RENUMBERPOINTS	_RENUMBERPOINTS
REV	REVOLVE	_REVOLVE
RO	ROTATE	_ROTATE
RR	RENDER	_RENDER
RSB	RESETBLOCK	_RESETBLOCK
S	STRETCH	_STRETCH
SAVEURL	QSAVE	_QSAVE
SC	SCALE	_SCALE
SCR	SCRIPT	_SCRIPT
SET	SETVAR	_SETVAR
SHP2CAD	MAPIMPORT	_MAPIMPORT
SL	SLICE	_SLICE
SLOPEMAP	SURFACEANALYSIS	_SURFACEANALYSIS
SMOOTH	CONVTOMESH	_CONVTOMESH
SN	SNAP	_SNAP
SO	SOLID	_SOLID
SP	SPELL	_SPELL
SPE	SPLINEDIT	_SPLINEDIT
SPL	SPLINE	_SPLINE
SPOTZ	SPOTELEVATION	_SPOTELEVATION
SSM	SHEETSET	_SHEETSET
ST, DDSTYLE	STYLE	_STYLE
STA	STANDARDS	_STANDARDS

STEP	STEPOUT	_STEPOUT
SU	SUBTRACT	_SUBTRACT
T, MT	MTEXT	_MTEXT
TA	TEXTALIGN	_TEXTALIGN
TB	TABLE	_TABLE
TIN, MDT	SURFACE	_SURFACE
TOL	TOLERANCE	_TOLERANCE
TOR	TORUS	_TORUS
TP	TOOLPALETTES	_TOOLPALETTES
TR	TRIM	_TRIM
TS	TABLESTYLE	_TABLESTYLE
UC, DDUCS, DDUCSP	UCSMAN	_UCSMAN
UN, DDUNITS	UNITS	_UNITS
UNHIDE, UNISOLATE	UNISOLATEOBJECTS	_UNISOLATEOBJECTS
UNI	UNION	_UNION
UPDFIELD	UPDATEFIELD	_UPDATEFIELD
V, DDVIEW	VIEW	_VIEW
VIEWPORTS	VPORTS	_VPORTS
VOLUMES, CUTFILL	SURFACEVOLUME	_SURFACEVOLUME
VS	VSCURRENT	_VSCURRENT
W, ACADWBLOCKDIALOG	WBLOCK	_WBLOCK
WE	WEDGE	_WEDGE
WEBMAP	MAPSERVICE	_MAPSERVICE
X	EXPLODE	_EXPLODE
XA	XATTACH	_XATTACH
XB	XBIND	_XBIND
XC	XCLIP	_XCLIP
XL	XLINE	_XLINE
XR	XREF	_XREF
Z	ZOOM	_ZOOM

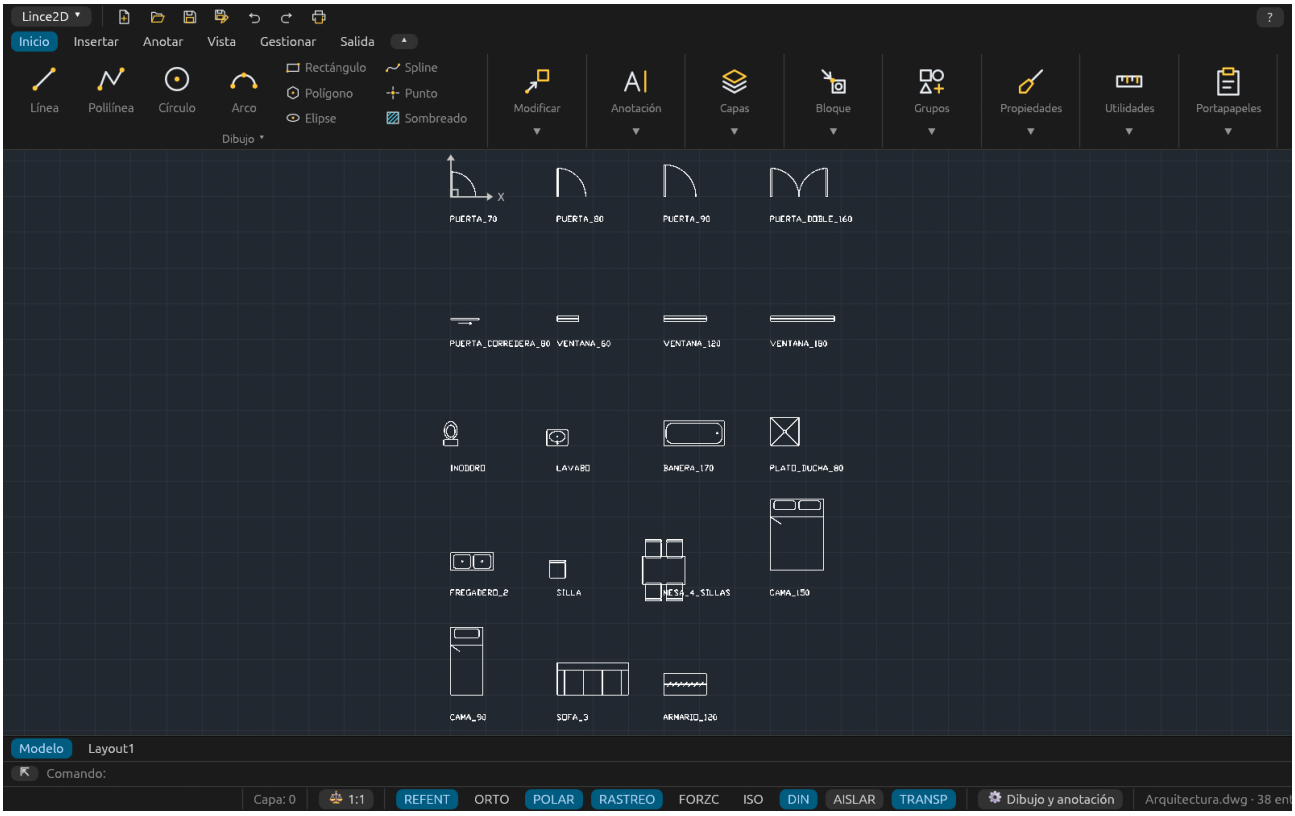
See also

Keyboard and shortcuts · The interface · Welcome

General Help

The user interface

From top to bottom, the LinceCAD window has the ribbon (or the menus and toolbars of the Classic workspace), the drawing area, the layout tabs, the command line and the status bar. Palettes (Properties, layers, blocks...) open when you need them and can float over the drawing or be docked at its sides.



Workspaces

A workspace is a named layout of the interface. Two are built in:

- **Drafting & Annotation (ribbon)**: the ribbon with its tabs, the **LinceCAD ?** application button and the Quick Access toolbar (New, Open, Save, Save As, Undo, Redo and Plot).
- **Classic (menus and toolbars)**: the traditional menu bar (File, Edit, View, Insert, Format, Tools, Draw, Dimension, Modify, Window and Help) and the Standard, Draw, Modify and Layers toolbars.

Each workspace also remembers the ribbon state, how the command line is placed and which palettes are open, on which side and how wide. The **?** gear on the status bar shows the current one and, when clicked, lists them all to switch with one click. WSSAVE saves the current layout with a name and WORKSPACE renames and deletes them. They are kept when the program closes.

Both built-in workspaces give access to the same commands. Commands that are not finished yet are dimmed; hovering over any button shows its description.

The ribbon

Tab	Panels
Home	Draw, Modify, Annotation, Layers, Block, Groups, Properties, Utilities and Clipboard
Insert	Block, Attributes, Reference, Content and Geolocation
Annotate	Text, Dimensions, Leaders, Tables and Annotation Scaling
View	Navigate, Isolate, Palettes, Coordinates and Interface
Manage	Drawing, Applications and Customization
Output	Plot and Export

Every available command is in some panel. Each panel shows its large buttons and two columns of small ones; if it has more, the ? in its title expands the rest. While you edit a block, the **Block Editor** contextual tab appears (see Blocks).

When the window is narrow, the panels that do not fit collapse, from right to left, into a button with their icon, title and ? that drops the panel down below.

Minimizing the ribbon

The ?/? button to the right of the tabs cycles through the ribbon states; right-click it to choose one directly:

- **Show Full Ribbon**: tabs and panels.
- **Minimize to Tabs**: only the tabs; clicking one drops its panels over the drawing, and they fold away when you choose a command or click elsewhere.
- **Minimize to Panel Titles**: the tabs and the titles of their panels; each title drops its panel down.

The command line

The keyboard always goes to the command line, as in AutoCAD: just start typing.

- It shows the last three lines; the text window (F2) has the full history.
- While you type a command name, a list of up to 8 suggestions with their descriptions appears: choose with the arrow keys and complete with Tab or a click.
- Enter or Space sends what you typed (Space does not while you are typing text). Enter on an empty line repeats the last command.
- The options in brackets in the prompt are shown in blue and can be clicked; typing their capital letters also works.
- You can type the command name, its alias (built-in or your own) or the global English name with `_` (`_LINE`) in any language.
- A line that starts with `(` is evaluated as AutoLISP (see AutoLISP).

Floating command line

The **?** button at its left undocks the command line: it becomes a window without a title bar that you can drag anywhere and widen by its edges. While it floats, the **?** button sets its transparency (opacity from 10 to 100 %); it turns opaque when the mouse is over it. **?** docks it again below the drawing.

Palettes

Palette	How to open it
Properties	PROPERTIES, Ctrl+1 or double-click an object
Layer Properties Manager	LAYER or the button next to the layer control
Blocks	INSERT
DesignCenter	ADCENTER

Tool palettes	TOOLPALETTES
External references and images	XREF

Each one has buttons in its header to place it:

- **?** / **?** (Dock left / Dock right): the palette docks on that side and the drawing narrows to make room. Several palettes on the same side share its height. Drag the edge facing the drawing to change the width.
- **?** (Floating): it becomes a floating window again.
- **?** (Auto-hide), docked only: the palette folds into a vertical tab at the edge of the window and slides out when the mouse passes over it.

Customizing

CUI changes the commands of any ribbon panel or Classic toolbar (add, remove and reorder) and assigns your own keyboard shortcuts. Your own aliases are edited with ALIASEDIT and saved in `lince.pgp`, with the syntax of AutoCAD's `acad.pgp` (one can be imported). Everything is saved in the configuration folder (`%APPDATA%\lince`; on Linux, `~/.config/lince`), together with the workspaces.

The status bar

Item	What it does
Coordinates	Cursor position. Click to add latitude and longitude when the drawing has a coordinate system
Layer	Current layer
? 1:1	Annotation scale (CANNOSCALE); click to choose another one from the drawing's list
OSNAP	Object snaps (F3); right-click to choose the modes
ORTHO	Ortho mode (F8)
POLAR	Polar tracking (F10); right-click for the increment angle
OTRACK	Object snap tracking (F11)
SNAP	Snap the cursor to the grid (F9)
ISO	Isometric drafting, with the current plane (F5 switches planes)
DYN	Dynamic input (F12)
ISOLATE	Isolate or hide objects; on while there are hidden objects
TRANSP	Show or hide the transparency of objects and layers (TRANSPARENCYDISPLAY)
?	Current workspace; its menu switches it, saves it and opens the customization

Next come the program messages and, at the far right, what was drawn in the last redraw and how long it took.

Drawing tabs and layout tabs

With two or more drawings open, their tabs appear above the drawing area; Ctrl+Tab and Ctrl+Shift+Tab switch to the next and previous one. Below the drawing are the **Model** tab and one tab per layout (see Layouts and plotting).

Shortcut menu and double-click

Right-clicking in the drawing area opens a menu that depends on what you are doing:

- With a command running: **Enter**, **Cancel** and the options of the prompt.
- With no command: repeat the last command, the usual editing commands if something is selected (Erase, Move, Copy, Scale, Rotate, Mirror) and **Deselect All**, plus Undo, Redo, Zoom Extents, Select All and Properties.

Double-clicking an object opens its editor: a text, TEXTEDIT; a hatch, HATCHEDIT; a block with attributes, ATTEDIT; a block without attributes, BEDIT; a table cell, TABLEDIT. For any other object it opens the Properties palette.

You can also open a drawing by dragging the file onto the window.

See also

Keyboard shortcuts · Command aliases · Options · OPTIONS

Coordinates and input

When a command asks for a point, you can specify it by clicking in the drawing, with an object snap, or by typing its coordinates on the command line. Coordinates are Cartesian, in drawing units, in the current User Coordinate System (UCS), which at first is the World Coordinate System (WCS).

Ways to type a point

Form	Example	Meaning
Absolute	`100,50`	X = 100, Y = 50 from the origin
Relative	`@20,-10`	20 to the right and 10 down from the last point
Relative polar	`@50<30`	50 units from the last point, at 30°
Absolute polar	`80<45`	80 units from the origin, at 45°
Direct distance	`35`	35 units from the last point, in the direction of the cursor
World	`*100,50` or `@*20,0`	In the WCS even if there is a UCS

- Input is always typed as in AutoCAD: **decimal point** and **comma between X and Y** (`10.5,20`), whatever number format is chosen in OPTIONS.
- Angles are typed in degrees and measured from the X axis, counterclockwise.
- Direct distance entry is very handy with Ortho (F8) or polar tracking (F10): point with the mouse and type only the length.
- When the command expects a number (a radius, a height, a factor), what you type is that number, not a point.

Dynamic input (F12)

With **DYN** on (it is on at startup), the command prompt appears next to the cursor and, when a point is requested, two fields with live values:

- for the first point, X and Y;
- for the following ones, the distance and the angle from the last point.

What you type goes to the active field. **Tab** locks that value (the cursor is constrained to it and a padlock appears) and moves to the other field; **Enter** sends the point. Without Tab, what you type is read exactly as on the command line. F12 or the DYN button on the status bar turns it on and off.

Coordinates on the status bar

The left end of the status bar shows the cursor position (or that of the point it snaps to, if there is an object snap or grid snap). It uses the units and precision set with UNITS and the number format from OPTIONS: with a decimal comma, X and Y are separated by a semicolon (`12,5; 30`). If the drawing has a coordinate system (GEOGRAPHICLOCATION), clicking the coordinates adds latitude and longitude.

Drawing units

UNITS sets how lengths and angles are displayed (decimal, feet and inches, degrees-minutes-seconds, bearings...) and their precision. The variables LUNITS, LUPREC, AUNITS, AUPREC and INSUNITS store these settings in the drawing.

Querying coordinates and distances

- ID shows the coordinates of a point.
- DIST measures the distance and angle between two points.
- LIST shows the data of the selected objects, including their coordinates.
- MEASUREGEOM measures distances, radii and areas.

User coordinate systems (UCS)

A UCS is an origin and axes of your own, moved or rotated with respect to the WCS: for example, aligned with a building that does not follow the axes of the plan. LinceCAD has UCSs in the plane, like AutoCAD in 2D:

- UCS defines the current UCS (by origin and X axis, aligned with an object, rotated with Z...), saves it with a name and goes back to the previous one or to the world system. UCSMAN manages named UCSs.
- With a UCS, the coordinates you type and those shown in the status bar are in the UCS; an asterisk in front (`*x,y`) gives them in the world system.
- Ortho, the grid, snap and the crosshairs are aligned with the UCS, and so are RECTANG and the rotation of TEXT. Polar tracking, dynamic input and the angles of other commands stay on the world axes.
- The UCS icon, at the lower left or at the origin, is set up with UCSICON (variable UCSICON).
- PLAN rotates the view so that the UCS X axis is horizontal.
- The UCS is saved in the drawing as in AutoCAD. In AutoLISP, trans converts points between the WCS and the UCS.

For georeferenced drawings, see Geolocation.

See also

Object snaps and precision · LINE · UNITS · UCS · The user interface

Object snaps and precision

Clicking "close enough" is not enough to draw accurately: object snaps take the cursor to exact points on objects (endpoints, centers, intersections...), and Ortho, polar tracking and object snap tracking guide it along exact directions. All these aids work only while a command is asking for a point.

Object snaps (F3)

With **OSNAP** on, moving the cursor near a key point shows a yellow marker with its name, and the click snaps to that point. F3 or the status bar button turns snaps on and off; right-clicking the button opens the list of modes, with **Select All** and **Clear All**. At startup all modes are on except Nearest.

Mode	Type	Point
Endpoint	END	End of lines, arcs, polyline segments
Midpoint	MID	Midpoint of a line, an arc or a segment
Center	CEN	Center of circles, arcs and ellipses
Node	NOD	Point objects
Quadrant	QUA	Points at 0°, 90°, 180° and 270° of circles, arcs and ellipses
Intersection	INT	Crossing of two objects
Extension	EXT	Extension of a line or arc beyond its endpoint
Insertion	INS	Insertion point of blocks, texts and attributes
Perpendicular	PER	Foot of the perpendicular from the last point
Tangent	TAN	Tangent point from the last point
Nearest	NEA	Point on the object nearest to the cursor

Snap override: when the command asks for a point, type the abbreviation (`END`, `MID`, `TAN`...) and press Enter: only that mode applies to the next point, even if OSNAP is off. The Spanish abbreviations (`FIN`, `MED`, `CER`...) and the underscore forms (`_end`) are accepted too. Object snaps also work inside inserted blocks and dimensions.

Object snap tracking (F11)

With **OTRACK** and object snaps on, rest the cursor on an object snap for half a second: the point is acquired and marked with a green cross. Horizontal and vertical tracking paths come out of each acquired point; the cursor snaps to them and to their intersections with other paths or with polar paths. Up to 7 points can be acquired; resting on one again releases it. The Extension mode is acquired the same way, by resting on an endpoint.

Ortho (F8) and polar tracking (F10)

- **ORTHO** forces the next point to be horizontal or vertical from the last one.
- **POLAR** snaps the cursor to directions that are multiples of the increment angle when it passes near them, with a dashed tracking path and a tooltip with the distance and angle. Right-click the button to choose the increment: 90°, 45°, 30°, 22.5°, 18°, 15°, 10° or 5°.

Ortho and polar tracking exclude each other: turning one on turns the other off. With either of them, typing just a distance places the point in the marked direction (see Coordinates and input).

Grid and snap (F9)

The model background shows a grid that adapts to the zoom: its spacing is a power of 10 with at least 25 pixels between lines. With **SNAP** (F9) the cursor jumps to the nodes of that grid. The grid cannot be configured or hidden yet (the GRID toggle and F7 are not available yet). With a rotated UCS, the grid, snap and Ortho follow its axes (see Coordinates and input).

Isometric drafting (F5, Ctrl+E)

To draw isometric views in 2D, turn on isometric drafting with ISODRAFT or the **ISO** button on the status bar. Ortho, the grid, snap and the crosshairs then follow the two axes of the current isometric plane:

Plane	Axes
Left	90° and 150°
Top	30° and 150°
Right	30° and 90°

F5 or Ctrl+E switch to the next plane (so does ISOPLANE), and ELLIPSE offers the Isocircle option to draw circles in the plane. The mode is saved in the drawing (SNAPSTYL and SNAPISOPAIR).

Order of priority

When several aids are on, the point is decided as follows: first the object snap under the cursor, if any; otherwise tracking (polar or object snap tracking); otherwise grid snap and Ortho. Dynamic input can then lock the distance or the angle with Tab.

See also

OSNAP · ORTHO · POLAR · SNAP · ISODRAFT · Coordinates and input · Keyboard shortcuts

Selecting objects

Almost all editing commands work on a set of selected objects. You can select them before starting the command (pre-selection, or noun/verb selection) or when the command asks for them with the prompt:

```
Select objects:
```

Ways to select

Action	Result
Click an object	Selects it (5-pixel pick box). A block is selected by its lines and a text by the rectangle of its letters, as in AutoCAD
Drag from left to right	**Window**: selects what lies entirely inside the rectangle
Drag from right to left	**Crossing**: selects what lies inside or crosses the rectangle
Shift + click or Shift + drag	Removes those objects from the selection
Ctrl+A or AI_SELALL	Selects everything that can be selected
Enter, Space or right-click > Enter	Ends the selection and the command goes on

"Select objects" options

While selecting for a command you can type AutoCAD's options (**?** lists them):

Option	Result
Window	Two corners: what lies entirely inside
Crossing	Two corners: what lies inside or crosses the rectangle
BOX	Two corners: window if the second is to the right, crossing if to the left
AUto	A point: the object it touches or, if none, the first corner of a box. A typed point that touches nothing does the same without the option
Fence	A polyline (Enter to end it): what crosses it
WPolygon	A polygon (Enter to close it): what lies entirely inside
CPolygon	A polygon: what lies inside or crosses it
ALL	Everything that can be selected
Last	The last object created
Previous	What the last command selected
Group	Asks for a group name and selects its members
Remove / **Add**	From then on, what you select is removed / added again
Undo	Undoes the last selection
SIngle	The first selection that finds something ends the prompt
Multiple, **SUbobject**, **Object**	Accepted; they change nothing (each selection is added at once)

If the command's prompt has its own options (for example GROUP: [Name/Description]), the command's win.

While selecting, the object under the cursor is highlighted so you can see what a click would select. Selected objects are drawn highlighted and, with no command running, show their grips (see Editing).

Pre-selection

With no command running, each click or window adds objects to the selection. If you then start an editing command (ERASE, MOVE, COPY...), the command uses that selection and does not ask again. To clear it:

- press Esc (with a command running, the first Esc cancels the command and the second one clears the

selection);
- click an empty area of the drawing;
- or use **Deselect All** on the shortcut menu.

With something selected, Delete erases it and right-clicking offers the most common editing commands.

What cannot be selected

- Objects on **locked** layers: they are visible, but neither clicks nor windows select them (a few layer commands, which only need to know an object's layer, do accept them).
- Objects on layers that are off or frozen, which are not visible.
- Objects hidden with HIDEOBJECTS or left out by ISOLATEOBJECTS.

Groups

A group (GROUP) is a named set of objects that are selected together. With the PICKSTYLE variable set to 1, selecting a member selects the whole group; Ctrl+Shift+A toggles this behavior. GROUPEDIT adds or removes members and UNGROUP dissolves the group.

Selecting by properties

- QSELECT: selects by object type and by the value of a property (layer, color, linetype...).
- SELECTSIMILAR: selects the objects of the same type and properties as the selected ones.
- FILTER: selects with a list of type and property conditions combined with AND, OR, XOR and NOT, which can be saved with a name.
- The PROPERTIES palette shows a drop-down list by type ("All (6)", "Line (2)"...) to view or change only part of the selection.

Table cells

With no command running, a click inside a table selects the cell; Shift+click or dragging from one cell to another selects a range (see Tables).

Notes

- Selection cycling between overlapping objects is not available yet.
- AutoCAD's selection variables (PICKFIRST, PICKADD...) cannot be changed: LinceCAD behaves as with their default values.

See also

Editing · Layers · AI_SELALL · QSELECT · GROUP

Editing: grips, undo, clipboard and properties

Objects are changed with the editing commands (MOVE, COPY, ROTATE, TRIM...) or directly with the mouse, through their grips. This page covers what all of them have in common.

Grips

When you select objects with no command running, their **grips** appear: blue squares at the points that define each object (endpoints and midpoints of segments, center and quadrants of circles and arcs, insertion point of blocks and texts...). The one under the cursor turns red.

1. Click a grip: it becomes the **hot grip** (red) and STRETCH mode starts.

```
Specify stretch point or [Base point/Copy/eXit]:
```

2. Move the cursor and click where the point must go (object snaps and typed coordinates work). You can also drag the grip and drop it in place.
3. Enter or Space cycles to the next mode, in this order:

Mode	What it does
** STRETCH **	Moves the point; if several objects share it, all of them are stretched
** MOVE **	Moves the selected objects
** ROTATE **	Rotates the objects around the grip (or the base point); a number is the angle
** SCALE **	Scales the objects from the grip; a number is the factor
** MIRROR **	Mirrors them about the line from the grip to the point you specify

Every mode has three options:

- **Base point** (B): picks another base point instead of the grip.
- **Copy** (C): keeps the original and creates a copy with each click, until you press Enter.
- **eXit** (X): ends without further changes. Esc cancels as well.

Dimensions, tables (column widths, row heights) and dynamic blocks have their own grips; those of dynamic blocks are drawn in cyan.

Undo and redo

- UNDO (Ctrl+Z) undoes the whole last command; it can be repeated up to 50 times.
- REDO (Ctrl+Y) redoes what was undone.
- A change made with grips, in the Properties palette or in a table cell is also an undo step.
- A whole Block Editor session counts as a single step.

Clipboard

Command	Key	What it does
COPYCLIP	Ctrl+C	Copies the selection
CUTCLIP	Ctrl+X	Copies the selection and erases it
COPYBASE	Ctrl+Shift+C	Copies with the base point you specify
PASTECLIP	Ctrl+V	Pastes; the objects follow the cursor until you click
PASTEBLOCK	Ctrl+Shift+V	Pastes the copied objects as a block
PASTEORIG	—	Pastes at the original coordinates

You can copy from one drawing and paste into another open drawing. The clipboard keys only act when the command line is empty and no command is running; if another text box has the keyboard focus, they copy and paste text as usual.

Properties

The PROPERTIES palette (Ctrl+1, or double-click an object) shows and changes the properties of the selection:

- **General**: color, layer, linetype, linetype scale and lineweight, common to all objects.
- **Geometry** and sections specific to each type: coordinates of a line, radius of a circle, vertices of a polyline, contents of a text, style of a dimension, cells of a table, properties of a dynamic block...
- When the objects have different values, `*VARIES*` is shown. Read-only values are grayed out.
- With nothing selected, the palette shows the **current** properties: the ones new objects will take.

MATCHPROP copies the properties of one object to others.

Other editing aids

- Delete erases the selection (ERASE).
- DRAWORDER changes which objects are drawn on top.
- ISOLATEOBJECTS and HIDEOBJECTS let you work with part of the drawing only, without changing it.

See also

Selecting objects · Layers and properties · Object snaps

Layers and object properties

Layers organize a drawing like stacked transparent sheets: every object is on a layer and, unless the object says otherwise, the layer sets its color, linetype, lineweight and transparency. Turning a layer off, freezing it or locking it affects all its objects at once.

The layer control

The layer control is on the ribbon (Home tab, Layers panel) and on the Layers toolbar of the Classic workspace: the drop-down list with the current layer and, for each layer, the state icons:

Icon	State	Effect
Light bulb	On or off	Objects on a layer that is off are neither shown nor plotted
Sun or snowflake	Thawed or frozen	Like off, and also left out of calculations; the current layer cannot be frozen
Sun with viewport	Frozen in the current viewport	Only in a layout viewport
Padlock	Unlocked or locked	Objects are visible but cannot be selected or modified
Color swatch	Layer color	Click to change it

Choosing a layer in the drop-down list makes it **current**: new objects are created on it. The current layer is also shown on the status bar.

The Layer Properties Manager

LAYER opens the **Layer Properties Manager**, a palette with one row per layer and AutoCAD's columns: status, name, on, freeze, VP freeze, lock, color, linetype, lineweight, transparency, plot, new VP freeze and description. Click a column title to sort, select several layers at once, and search by name with wildcards (`*`, `?`).

- **Filters**: property filters (conditions on name, state, color, linetype...) and group filters (a list of chosen layers), in a tree on the left, with the option to invert them.
- **Layer states** (LAYERSTATE): save the settings of all layers so you can restore them later. They are stored in the drawing as in AutoCAD, which lists and restores them.

Layer commands

Command	What it does
LAYMCUR	Makes an object's layer current
LAYMCH	Moves objects to another object's layer
LAYERP	Undoes the last change to layer settings
LAYISO / LAYUNISO	Leaves only the layers of the selection visible, and undoes it
LAYOFF / LAYON	Turns off an object's layer / turns all layers on
LAYFRZ / LAYTHW	Freezes an object's layer / thaws all layers
LAYLCK / LAYULK	Locks or unlocks an object's layer
LAYMRG	Moves the objects of some layers to another one and deletes the former
LAYDEL	Deletes layers together with all their objects
-LAYER	All of the above from the command line (for scripts)

Color, linetype, lineweight and transparency

Each object has four properties that can take a specific value or one of these two:

- **ByLayer**: the object takes the value of its layer. This is the usual choice, and what new objects have at first.
- **ByBlock**: an object inside a block takes the value of the inserted block.

Objects on layer **0** inside a block are drawn with the layer of the inserted block.

Property	Command	Values
Color	COLOR	ByLayer, ByBlock, the 255 index colors (ACI) or an RGB true color
Linetype	LINETYPE	Continuous and the loaded linetypes; more can be loaded from `.lin` files
Lineweight	LWEIGHT	ByLayer, ByBlock or a lineweight in millimeters
Transparency	PROPERTIES	ByLayer, ByBlock or 0 (opaque) to 90 %

The global linetype scale is LTSCALE; the one for new objects is CELTSCALE. The current values are in CLAYER , CECOLOR and CELTYPE. Lineweights are always shown on screen (the LWDISPLAY toggle is not available yet), and when plotting they depend on the plot style table.

To change the properties of objects already drawn, use the PROPERTIES palette or MATCHPROP.

Transparency

Transparency lets what is underneath an object show through: at 50 %, the object is half visible over the background and over whatever is behind it. Object transparency is changed in the Properties palette; layer transparency, in the Transparency column of the layer manager or with the TRansparency option of -LAYER. New objects take CETRANSPARENCY, which starts as ByLayer.

- The TRANSP button on the status bar (TRANSPARENCYDISPLAY) shows or hides it on screen; it does not change the drawing.
- It is always plotted to PDF, SVG and PNG (there is no option to plot without it).
- It is saved in DWG files from version 2004 (group code 440 on objects and AcCmTransparency data on layers); DWG 2000 keeps the layer transparency but not the object transparency.
- Multiline text with its own colors inside the text loses its transparency.

See also

Editing and properties · Blocks · Layouts and plotting · PURGE

Text, styles and fonts

LinceCAD has AutoCAD's two kinds of text: **single-line text** (TEXT), created with TEXT, and **multiline text** or paragraph text (MTEXT), created with MTEXT. Both use a text style, which sets the font and its shape, and both are drawn with the real SHX or TrueType fonts, on screen and when plotting.

Single-line text

TEXT asks for the start point, the height and the rotation, and then the contents. Each line you type is a separate text object and the next one is placed below it; Enter on an empty line ends the command.

```
Specify start point of text or [Justify/Style]:
```

- **Justify** chooses which of the 15 points of the text is the insertion point (Left, Center, Right, Middle and the nine combinations of top, middle and bottom).
- **Style** changes the text style used (? lists them).

Multiline text

MTEXT asks for the two corners of the paragraph box; the width of the box decides where lines wrap. Before the second corner you can change the height, justification, style, rotation and width:

```
Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width/Columns]:
```

Then the **Text Editor** opens, with style, height, justification, width and color, and bold, italic and underline buttons that apply to the selected text. Each line in the editor is a paragraph. AutoCAD's MTEXT formatting codes are honored (font change, height, color, underline, stacked fractions...).

Editing text

Double-click a text, or use TEXTEDIT, to change its contents: a single-line text on the command line itself and an MTEXT in its editor. The PROPERTIES palette also changes the height, style, justification and rotation.

Text styles

STYLE opens the text style manager: create, rename, delete and set current. Each style sets:

Setting	Effect
Font	An SHX file (with an optional big font for Asian languages) or a TrueType font
Height	0 to ask for it in each text, or a fixed height
Width factor	Narrows (less than 1) or widens (more than 1) the characters
Oblique angle	Slants the characters, between -85° and 85°
Annotative	New texts are written with the paper height and shown at the annotation scale

The current style is in TEXTSTYLE and the default height in TEXTSIZE. Annotative texts and annotation scales are explained in Dimensions and annotation scales.

Fields

A field is a piece of text that is computed automatically: the date, the file name, the drawing author or title, the user, or a property of an object (the length of a line, the area of a boundary, the layer...). FIELD creates a text with a field or puts it in an existing text.

- Fields are shown on a gray background that is not plotted (FIELDDISPLAY turns it off).
- They are updated when the drawing is opened and saved; UPDATEFIELD updates them at any time, including the date.
- They are saved as in AutoCAD: the fields of an AutoCAD drawing are read and updated, and the other way round.

Find, replace and check spelling

- FIND finds a text and replaces it with another in the whole drawing or in the selection: texts, attributes, dimension texts, multileaders and table cells, matching case and whole words or not.
- SPELL checks those same texts with the dictionary of the interface language (Spanish or English, both included with the program) and suggests corrections; your own words are added to a user dictionary.

Fonts

- **SHX**: AutoCAD's stroke fonts (shapes 1.0 and 1.1, Unicode and big fonts).
- **TrueType**: the system fonts (`.ttf`).

LinceCAD does not ship AutoCAD's own fonts: it looks for them, in this order, in the drawing folder, in AutoCAD's font folders if AutoCAD is installed, and in the system fonts (on Linux, those of fontconfig). If a style's font is missing, it uses simplex.shx, txt.shx or Arial instead, the first one it finds, as AutoCAD does with FONTALT. For a drawing to look the same on another computer, copy its SHX fonts next to the DWG.

Notes

- When a DWG is read, texts are placed as AutoCAD left them; texts from a DXF are justified again.
- MTEXT indents, tabs and columns, text background masks and fields inside the MTEXT editor are not available yet.
- The texts of dimensions, leaders and tables also use text styles.

See also

TEXT · MTEXT · STYLE · Dimensions · Tables

Dimensions, leaders and annotation scales

A dimension measures the drawing and writes the measurement. In LinceCAD every dimension is an AutoCAD dimension object (DIMENSION) with its style: it is drawn from its definition points and regenerated when it is moved, copied or its style changes, so AutoCAD recognizes it and can edit it.

Dimension types

Command	Measures
DIMLINEAR	Horizontal or vertical distance between two points
DIMALIGNED	True distance, parallel to the two points
DIMANGULAR	Angle between two lines or defined by a vertex and two points
DIMRADIUS / DIMDIAMETER	Radius or diameter of a circle or an arc
DIMJOGGED	Radius of a large arc, with a jogged dimension line
DIMARC	Length of an arc
DIMORDINATE	X or Y coordinate of a point
DIMCONTINUE	Dimensions chained after the last one
DIMBASELINE	Several dimensions from the same origin
QDIM	A series of dimensions of all the selected geometry

All of them ask for the points one by one, with a preview; object snaps (Endpoint, Center, Intersection...) are almost essential to dimension accurately. The dimension is created on the current layer; its lines, arrows and text go on layer 0 of the dimension's block with ByBlock properties, as in AutoCAD.

TOLERANCE creates geometric tolerance feature control frames (position, flatness, perpendicularity...); measurement tolerances (± and limits) are turned on in the dimension style.

Editing dimensions

- **Grips** move the dimension line, the text or the measured points.
- MOVE, COPY, ROTATE, SCALE, MIRROR and ARRAY regenerate the dimension; each copy gets its own block.
- DIMEDIT changes the text of dimensions or puts it back in its home position.
- DIMSPACE evens out the spacing between parallel dimensions, DIMBREAK breaks them where other objects cross them and DIMJOGLINE adds a jog to a linear dimension.
- The PROPERTIES palette has **Dimension** and **Overrides** sections: changing a dimension variable there overrides it for that dimension only, without touching the style (stored as AutoCAD does).

Associative dimensions

With DIMASSOC set to 2 (the initial value), a dimension whose points lie on the geometry (endpoint, midpoint, center, quadrant, node or polyline vertex) is **associated** with it: when the line, arc or polyline is moved, stretched, scaled or edited with grips, the dimension moves and its measurement changes.

- If the dimension is moved on its own or its geometry is erased, the dimension is disassociated and stays where it is.
- DIMREASSOCIATE associates it again by specifying its points once more; DIMDISASSOCIATE removes the association.
- Associations are saved as in AutoCAD, both ways: associative dimensions of an AutoCAD drawing stay associative in LinceCAD, and the other way round.
- A point at an intersection or a nearest point does not associate the dimension.

Dimension styles

DIMSTYLE opens the dimension style manager, with **Set Current**, **New**, **Modify**, **Rename** and **Delete**. When you modify a style, its settings are on AutoCAD's tabs: Lines, Symbols and Arrows, Text, Fit and Primary Units. Changes to a style apply to all the dimensions that use it.

A new drawing starts, as with AutoCAD's acadiso.dwt, with two styles: Standard (the inch one) and ISO-25, the current one (metric, arrows and text 2.5 high, with the same variables as in AutoCAD 2027). The decimal separator of the measurements is the style's: with a comma, as in Spanish AutoCAD, the dimension reads "56,57". The current style is in DIMSTYLE.

Multileaders

MLEADER draws a multileader: arrowhead, leader line, landing and a text or a block.

- MLEADERSTYLE: creates, modifies and sets current multileader styles.
- MLEADEREDIT: adds leader lines to an existing multileader or removes some.
- MLEADERALIGN: aligns the contents of several multileaders.

Annotation scales

For texts and dimensions to have the intended size on paper there are two ways:

- **Fixed scale**: DIMSCALE multiplies all dimension sizes (with 50, a 2.5 arrowhead is 125 drawing units long, which is 2.5 mm on a 1:50 plot), and texts are written with the height already multiplied.
- **Annotative objects**: they are written with the paper height (2.5) and shown at the size that matches the annotation scale (CANNOSCALE). If the plot changes scale, just change the annotation scale.

The current annotation scale is chosen with the selector on the status bar (or with CANNOSCALE) among the scales of the drawing's list, which is edited with SCALELISTEDIT. What is created annotative:

- texts, multiline texts and dimensions created with an annotative style (**Annotative** box in STYLE and on the **Fit** tab of DIMSTYLE);
- multileaders and tolerances with an annotative style;
- any text or dimension set to **Annotative: Yes** in the PROPERTIES palette.

An annotative object can support several scales and keeps one representation (position and size) for each. OBJECTSCALE adds scales to it or removes them and, with ANNOAUTOSCALE, the new scale is added automatically when the annotation scale changes. In a layout, each viewport shows the annotative objects at its own scale: the same text comes out the same size in a 1:50 viewport and in a 1:100 one.

Everything is saved as in AutoCAD: annotative objects of an AutoCAD drawing look the same in LinceCAD, and the other way round.

Notes

- Annotative blocks cannot be created yet (those that come in drawings do change scale), and annotative multileader styles are the ones that come in drawings.
- A grip moves all the representations of an annotative object, not only the one shown, and a copy of an annotative object supports only the current scale.

See also

Text and styles · Object snaps · Layouts · Tutorial: dimensioning a plan

Blocks, attributes and libraries

A block is a named set of objects that you define once and insert as many times as needed (a door, a socket, a symbol). Each insertion is a single object that can be moved, rotated or scaled; if you change the definition, all insertions change.

Creating and inserting

Command	What it does
BLOCK	Defines a block with a name, a base point and some objects
INSERT	Opens the **Blocks** palette to insert
-INSERT	Inserts by asking on the command line (also a DWG or DXF file)
WBLOCK	Saves a block as a DWG or DXF drawing
PASTEBLOCK	Pastes the copied objects as a new block
EXPLODE	Explodes an insertion into its objects

When you create a block you choose what to do with the original objects: retain them, convert them to an insertion of the block, or delete them. If the name already exists, you are asked whether to redefine it.

The **Blocks** palette shows thumbnails of the blocks in the current drawing, the recently used ones and those of the libraries. Click or drag a block to place it; scale and rotation are typed or picked with the mouse. Inserting a DWG or DXF file brings it in as a block with its layers, styles and nested blocks; definitions that already exist in the drawing are kept.

Tip: draw the contents of a block on layer **0** with **ByBlock** properties; that way each insertion takes the layer, color and linetype it is inserted with (see Layers).

Attributes

Attributes are variable texts inside a block (a reference, a circuit, a price).

- ATTDEF creates an attribute definition: tag, prompt and default value. It is included in the block when the block is created.
- When the block is inserted, the values are requested.
- ATTEDIT (or double-click the block) and EATTEDIT change the values of an insertion.
- BATTMAN changes the order, prompt, default value and modes of a block's attributes, and ATTSYNC updates the existing insertions to match the definition.
- ATTEXT extracts attributes to a CSV file or to a table in the drawing.

Dynamic blocks

Dynamic blocks made with AutoCAD work in LinceCAD: their grips are drawn in cyan and change visibility, flip, lengths, point positions, rotation, arrays and lookup tables. The Properties palette shows their values in the **Custom** section. RESETBLOCK resets them to the values of the definition. LinceCAD cannot create new parameters or actions yet.

Block Editor

BEDIT (or double-click a block without attributes) opens a block definition as if it were a drawing: all commands, object snaps, grips and undo work. The **Block Editor** ribbon tab appears and, at the bottom, a bar with the block name.

- BSAVE saves the definition and redraws all its insertions.
- BSAVEAS saves the edited contents as a new block and leaves the original unchanged.
- BCLOSE closes the editor; if there are unsaved changes, it asks.

The whole editor session is undone in one step. External references and anonymous blocks cannot be edited.

DesignCenter, palettes and libraries

- ADCENTER browses other drawings, open or in a folder, and copies their blocks, layers, linetypes, styles and layouts into the current drawing.
- TOOLPALETTES are palettes with blocks ready to insert with their scale, rotation and layer; fill them by dragging blocks from the Blocks palette or from DesignCenter.
- LinceCAD is installed with two libraries of its own in the `bibliotecas` folder: **Arquitectura.dwg** (doors, windows, furniture and sanitary fittings) and **Electricidad.dwg** (switches, sockets and lights, with CIR and REF attributes). Any DWG with blocks can serve as a library.

Notes

- PURGE deletes unused block definitions.
- The base point of a drawing inserted as a block is INSBASE.

See also

Layers and properties · External references · Tables · Tutorial: blocks

Hatches and fills

A hatch (HATCH) fills a closed boundary with a pattern of lines, a solid fill or a gradient: sections, materials, areas of a plan. LinceCAD draws the hatches of any DWG or DXF with their pattern and islands, and creates new hatches with HATCH and gradients with GRADIENT.

Creating a hatch

1. Enter HATCH or click **Hatch** on the ribbon. The **Hatch and Gradient** dialog box opens.
2. Choose the **Pattern**, the **Scale** and the **Angle**.
3. Specify the boundary with **Add: Pick points** (click inside the area: LinceCAD finds the closed boundary around it and any islands inside) or with **Add: Select objects** (select the closed objects that form it). Both can be combined. If the boundary has small gaps, HPGAPTOL closes them.
4. Click OK.

Islands (boundaries inside the boundary) are treated as in AutoCAD's **Normal** style: hatched and empty areas alternate inwards.

Patterns

Pattern	Look
SOLID	Solid fill in the hatch color
ANSI31	45° hatching, the usual one for sections
ANSI32	Pairs of lines at 45° (steel)
ANSI37	Grid at 45°
LINE	Horizontal lines
NET	Horizontal and vertical grid
CROSS	Crosses
DOTS	Dots
BRICK	Brick
HONEY	Hexagonal honeycomb
HORMIGON	Concrete

They are LinceCAD's own definitions with the usual names, so AutoCAD recognizes the hatch. Hatches that come with a drawing are always drawn with the pattern they store, even if it is not in this list.

The scale multiplies the pattern spacing; the angle rotates it (with ANSI31, 0° gives the 45° hatching).

The Origin option of HATCH chooses the point the pattern passes through: for example, so that a row of bricks starts at the corner of the wall.

Editing a hatch

Double-click the hatch, or use HATCHEDIT, to change its pattern, scale, angle, gradient or origin. On the command line, HATCHEDIT also disassociates it from its boundary or associates it with other objects, separates it into independent hatches (one per area) and recreates its boundary as polylines. The PROPERTIES palette also changes the layer, the color and the other general properties. A hatch is moved, copied, rotated and scaled like any other object, and TRIM removes the piece you pick between the cutting edges.

Associative hatches

The hatches LinceCAD creates are **associative**, as in AutoCAD: they keep the objects of their boundary and, if one of them is edited (with grips, MOVE, STRETCH, PEDIT...), the hatch is rebuilt to follow it. It stops being associative if the hatch is moved on its own, if a boundary object is erased or if it is trimmed. AutoCAD recognizes

the associativity when it opens the drawing.

HATCHGENERATEBOUNDARY creates polylines with the boundary of a hatch that has lost it (without associating them).

Gradients

GRADIENT, or the **Gradient** tab of the HATCH dialog box, fills with a gradient of one color (towards black or white) or of two, in the nine AutoCAD types (linear, cylinder, spherical, hemispherical, curved and their inverted versions), centered or shifted and at any angle. They are saved as in AutoCAD, and the gradients that come in drawings are shown with their colors.

Related commands

- BOUNDARY creates a polyline or a region from the boundary around a point, with the same search as HATCH.
- REGION converts closed curves into regions.
- SOLID draws filled triangles and quadrilaterals, and DONUT, filled rings or circles.
- WIPEOUT covers what lies under a boundary with the background color.
- DRAWORDER decides whether the hatch is below or above other objects.

Notes

- AutoCAD's Outer and Ignore island styles are drawn as Normal.
- User-defined patterns (`.pat` files) are not available yet.
- The hatch is not automatically sent behind its boundary: use DRAWORDER if needed.
- When plotting, hatches come out with their pattern or fill, just as on screen.

See also

HATCH · HATCHEDIT · GRADIENT · Layers and properties · Layouts and plotting

Tables

A table is an object with rows and columns of cells holding text, numbers or formulas: bills of materials, area schedules, legends. LinceCAD creates and edits tables compatible with AutoCAD's (TABLE object), which look and plot the same in both programs.

Creating a table

TABLE opens the **Insert Table** dialog box:

- the **table style**;
- the number of **columns** and **rows** and their size (column width and row height in lines of text);
- what the first and second rows are: title, header or data;
- how it is placed: by its insertion point or with a window that sets the overall size;
- the contents: empty, from a **CSV** file, or pasted text (separated by tabs, commas or semicolons).

Once placed, you start typing in the first cell: Tab and Shift+Tab move to the next and previous cell, and the arrow keys move between cells.

Selecting and editing cells

With no command running:

- a click inside a cell selects it; Shift+click or dragging to another cell selects a range;
- double-clicking a cell, or TABLEDIT, edits its text with the multiline text editor.

With cells selected, the **Table Cell** palette appears: insert and delete rows and columns, merge and unmerge cells, alignment, text height and color, fill color, and quick formulas (Sum, Average, Count). Each button is equivalent to TABLECELL and is undone separately. The PROPERTIES palette also shows Table and Cells sections.

Formulas

A cell that starts with = is a formula. References are column and row, as in a spreadsheet (`A3`, `B2:B9`).

Formula	Result
`=Sum(B2:B9)`	Sum of the range
`=Average(B2:B9)`	Average
`=Count(B2:B9)`	Number of cells with a value
`=Min(B2:B9)`, `=Max(B2:B9)`	Minimum and maximum
`=B2*C2+1`	Arithmetic with `+`, `-`, `*` and `/` between cells and numbers

Formulas are recalculated when a cell changes, and a formula can use the result of another. A decimal comma is accepted, as in Spanish AutoCAD. They are stored as AutoCAD stores them, so AutoCAD keeps calculating them.

Grips

A selected table has grips to move it, to change the width of each column, the overall width and height (at the corners), and the height of each row. COPY, ROTATE, SCALE and EXPLODE work with tables as with any other object.

Table styles

TABLESTYLE creates and modifies table styles: cell margins and, for each kind of cell (title, header and data), text style and height, color, fill and alignment. The current style (CTABLESTYLE) can also be chosen on the ribbon, Annotate tab, Tables panel.

Import and export

- **CSV**: TABLE imports it when the table is created; TABLEEXPORT saves the cells of a table to a CSV file, with formulas as their values.
- ATTEXT creates a table with the attributes of blocks.

Notes

- Formulas do not update their references when rows or columns are inserted or deleted.
- Rows grow with the text but do not shrink by themselves. SCALE does not change text heights.
- In DWG 2000, formulas are saved as values (in DXF 2000 they are kept).
- There are no data links to Excel: use CSV.

See also

TABLE · TABLESTYLE · Text and styles · Blocks and attributes

External references

An external reference (xref) is another DWG or DXF drawing displayed inside the current one without copying it: an architectural plan under the building services, a survey base, a project's common title block. If someone changes the referenced drawing, the change shows when it is reloaded. A **PDF underlay** is the same with a page of a PDF: a plan received as a PDF that serves as a background to draw on.

Attaching a reference

1. Enter XATTACH or click **Attach DWG…** in the XREF palette.
2. Choose the file and specify the insertion point, the scale and the rotation.

With the **Overlay** option the reference is an overlay: it is shown in this drawing, but not in the drawings that attach this one (useful to avoid carrying bases from one drawing to another). The reference's layers appear as `reference|layer`: they can be turned off, frozen or recolored like any other, and with VISRETAIN = 1 those changes are kept when it is reloaded. The reference content is shown somewhat faded (XFADECTL) to tell it apart from your own objects.

The references of a reference (nested ones) are also loaded and drawn; nested overlays are not.

The External References palette

XREF opens the **External References** palette, with the tree of DWG references (and their nested ones), the PDF underlays and the raster images of the drawing. For each reference it shows its status, type, size, date, saved path and the file that was found, and it has buttons to:

Button	What it does
Reload	Reads it again from disk
Unload	Stops showing it without removing it from the drawing
Detach	Removes it from the drawing with all its inserts
Bind	Turns it into a normal block of the drawing (BINDTYPE)
Change Path…	Chooses another file or fixes its path

The same is done on the command line with -XREF, which accepts wildcards in the names. If a reference's file changes while the drawing is open (for example, because someone else saved it), the status bar warns about it with a button to reload it.

Where files are looked for

As in AutoCAD, for each reference the program tries, in this order:

1. the saved path, if it is absolute;
2. the saved path, relative to the drawing's folder;
3. just the file name, in the drawing's folder.

That is why, to take a project to another computer, it is enough to copy the drawing and its references together. How the path of new ones is saved (full, relative or no path) is decided by REFPATHTYPE. For security, network paths (`\\server\...`) are not read, and nothing from a referenced file is ever run: only its geometry is read.

Clipping

XCLIP shows only part of a reference or a block, inside or outside a rectangular, polygonal or polyline boundary. XCLIPFRAME decides whether the boundary is shown.

PDF underlays

- PDFATTACH attaches a page of a PDF; at scale 1 it measures the same as on paper.

- PDFCLIP clips the underlay, PDFADJUST changes its fade or contrast or turns it monochrome, and PDFLAYERS turns the PDF layers off or on.
- LinceCAD turns the page into an image by itself, without external programs. That is why object snaps do not work on what is drawn in the PDF, and with a lot of zoom the pixels show.

Not available yet

- Editing references in place (REFEDIT); the Block Editor does not open external references.
- XCLIP clip depth (3D).
- DWF and DGN underlays.

Alternatives

- To bring the content of another drawing in permanently, insert it as a block with INSERT or -INSERT, or bind it with -XREF: its objects become part of the current drawing.
- To copy only layers, styles or blocks from another drawing, use ADCENTER.

See also

Blocks · Raster images · File formats · XREF · XATTACH · PDFATTACH

Raster images

You can place raster images (photos, orthophotos, scanned drawings, logos) in the drawing, as in AutoCAD: the drawing stores only the image **path**, its position, size and rotation, not the pixels. So the image file must always travel with the drawing.

Formats

PNG, JPEG, BMP, TIFF (including GeoTIFF and tiled TIFF), GIF and WebP. ECW, MrSID and JPEG 2000 cannot be read (the Spanish PNOA orthophotos come as ECW): when a drawing that attaches them is opened you are warned, their frame says **Unsupported format**, and CONVERTIMAGE converts them to GeoTIFF with GDAL (QGIS or OSGeo4W, installed separately) and attaches the result in the same place. The original is not deleted.

Inserting an image

1. Enter IMAGEATTACH or use the Insert tab. Enter opens the dialog box to choose the file.
2. Specify the insertion point (lower-left corner). The **Path** option chooses whether the path is saved **Relative** to the drawing folder or **Full**.
3. Enter the scale factor (with 1, the image takes the size given by its resolution and the drawing units) and the rotation.

To place an image at its real-world coordinates, use GEOIMAGE or GEOREFIMAGE (see Geolocation).

Managing the images of a drawing

- The XREF window lists the images with their status, size and coordinate system, and has buttons to **Reload**, **Detach** and **Change Path…**; those in an unsupported format also have **Convert with GDAL…** (in the row and in its shortcut menu).
- -IMAGE does the same on the command line: `?` lists them; **Detach**, **Path**, **Reload** and **Unload** ask for the image name (`*` for all); **Attach** attaches a new one.
- If an image is not found when the drawing is opened, its frame is shown with the file name; change the path to see it again.

Adjusting the image

Command	What it does
IMAGECLIP	Shows only part of the image, inside (or outside, with Invert) a rectangular or polygonal boundary
IMAGEADJUST	Changes brightness, contrast and fade (0 to 100)
TRANSPARENCY	Lets what is underneath show through the transparent pixels of the file
DRAWORDER	Brings the image below or above other objects

The IMAGEFRAME variable controls image frames: 0, no frame; 1, displayed and plotted; 2, displayed but not plotted (for now LinceCAD plots it too).

An image is moved, copied, rotated and scaled like any other object.

Large images

Very large images (orthophotos of hundreds of megapixels) are loaded in tiles and in the background: a reduced version appears first and the detail arrives as you zoom in. Their memory use is capped, so you can work with several at once. Huge PNG and JPEG files, unlike TIFF, are read whole; it is better to convert them to TIFF.

Plotting

Images come out in PDF, SVG and PNG with their clipping and transparency. To keep files from growing without

limit, each image is plotted at 300 dpi and 16 megapixels at most.

Notes

- Images are saved in DWG and DXF as AutoCAD saves them, and AutoCAD opens them in place.
- Brightness and contrast are close to, but not exactly, AutoCAD's.

See also

Geolocation · External references · IMAGEATTACH · Layouts and plotting

Geolocation and GIS

A georeferenced drawing has its coordinates in a real-world reference system (for example, ETRS89 / UTM 30N): every point of the drawing is a point on the ground. This way orthophotos, maps, GIS files and GPS data can be overlaid without moving them by hand.

Coordinate system of the drawing

GEOGRAPHICLOCATION assigns a coordinate system to the drawing. Type its EPSG code (`25830`, `4326`...) or the Autodesk code (`ETRS89.UTM-30N`), or choose it in the dialog box; the **List** option shows the available systems and **Remove** deletes it. The drawing must already be in coordinates of that system: GEOGRAPHICLOCATION moves nothing, it only states which system the drawing is in. It is saved as in AutoCAD, which recognizes it.

Systems included	EPSG codes
World: WGS 84 geographic and Web Mercator	4326, 3857
WGS 84 / UTM, all north and south zones	32601–32660, 32701–32760
Europe: ETRS89 geographic, ETRS89 / UTM 28N–38N, ETRS89-LAEA Europe	4258, 25828–25838, 3035
Spain: ED50 (geographic and UTM 28N–31N), REGCAN95 (geographic and UTM 27N and 28N)	4230, 23028–23031, 4081–4083
Portugal: PT-TM06	3763
France: RGF93 v1 geographic, Lambert-93, CC42–CC50 conics	4171, 2154, 3942–3950
Netherlands: Amersfoort / RD New	4289, 28992
Belgium: BD72 / Lambert 72, ETRS89 / Lambert 2008	4313, 31370, 3812
Luxembourg: LUREF / Luxembourg TM	4181, 2169
Switzerland: CH1903+ / LV95	4150, 2056
Germany: DHDN / Gauss-Krüger 2–5 (and ETRS89 / UTM 32N and 33N, under Europe)	4314, 31466–31469
Austria: MGI / Austria GK West, Central and East	4312, 31254–31256
Italy: RDN2008 / UTM 32N–34N, Monte Mario / Italy zone 1 and 2	6706, 7791–7793, 4265, 3003, 3004
United Kingdom: OSGB36 / British National Grid	4277, 27700
Ireland: IRENET95 / Irish Transverse Mercator	4173, 2157
Poland: ETRF2000-PL / CS92 (PL-1992) and CS2000 (PL-2000) zones 15–24	9702, 2180, 2176–2179
Sweden: SWEREF99 TM · Finland: TM35FIN	4619, 3006, 3067
Greece: GGRS87 / Greek Grid · Hungary: HD72 / EOV · Romania: Stereo 70	4121, 2100, 4237, 23700, 4179, 3844
Slovenia: D96/TM · Croatia: HTRS96/TM	4765, 3794, 4761, 3765
Lithuania, Latvia and Estonia: LKS94, LKS-92, L-EST97	4669, 3346, 4661, 3059, 4180, 3301
SIRGAS 2000 (Brazil and South America): geographic, UTM 11N–24N and 17S–26S, Brazil Polyconic	4674, 31965–31985, 6210, 6211, 5396, 5880
Mexico: Mexico ITRF2008 geographic, LCC and UTM 11N–16N	6365, 6372, 6366–6371
Argentina: POSGAR 2007 geographic, zones 1–7 and UTM 19S	5340, 5343–5349, 9265
Chile: SIRGAS-Chile 2021 / UTM 18S and 19S	20041, 20048, 20049
Colombia: MAGNA-SIRGAS 2018 / Origen-Nacional; MAGNA-SIRGAS / Far West to East zones (Bogota: 3116)	20046, 9377, 4686, 3114–3118

Peru: Peru96 / UTM 17S–19S · Uruguay: SIRGAS-ROU98 / UTM 21S and 22S	5373, 5839, 5387, 5389, 5381–5383
Venezuela: REGVEN / UTM 18N–20N · Costa Rica: CR05 / CRTM05	4189, 2201–2203, 5365, 5367
Ecuador, Bolivia, Paraguay and the rest: WGS 84 / UTM or SIRGAS 2000 / UTM of their zone	(see above)
Morocco: Merchich / Nord Maroc, Sud Maroc, Sahara Nord and Sahara Sud	4261, 26191, 26192, 26194, 26195
Algeria: Nord Sahara 1959 / UTM 29N–32N, Nord Algérie and Sud Algérie	4307, 30729–30732, 30791, 30792
Tunisia: Carthage / UTM 32N, Nord Tunisie and Sud Tunisie	4223, 22332, 22391, 22392
Egypt: Egypt 1907 / Red, Blue, Purple and Extended Purple Belt · Libya: LGD2006 / Libya TM and UTM 32N–35N	4229, 22991–22994, 4754, 3177, 3199, 3201–3203

The full list (over 300) is in the GEOGRAPHICLOCATION dialog box, grouped by country or region and searchable by EPSG code, name or country, and in the **List** command-line option. The parameters of each projection are those of EPSG (checked against PROJ to the millimeter).

Datum shift accuracy. Between ETRS89, WGS84 and the GRS80-based frames (SIRGAS, MAGNA-SIRGAS, POSGAR, RGF93, ETRF2000-PL, SWEREF99…) there is no datum shift: the transformation is exact (as in PROJ, ETRS89 = WGS84). Older datums use the best EPSG transformation that needs no grid, with its nominal accuracy: about 1 m for Amersfoort, BD72, LUREF, CH1903+, GGRS87, HD72 and Nord Sahara 1959; 1.2 m for Egypt 1907; 1.5 m for MGI; 2 m for OSGB36 (the official OSTN15 grid is not included: it can reach 3–4 m); 3 m for DHDN (the former West Germany one) and Stereo 70; 4 m for Monte Mario; 7 m for Merchich and 14 m for Carthage. For accurate work, draw in the country's modern system (ETRS89, SIRGAS…). ED50 transformations use the official NTv2 grid of the Spanish IGN if it is installed (in the `rejillas` folder next to the program or in AutoCAD's); otherwise a 7-parameter transformation with an error of about 1.5 m, valid only in Spain.

Geographic coordinates

- With a coordinate system assigned, clicking the coordinates on the status bar adds the cursor's latitude and longitude.
- GEOCOORD shows a point in WGS84 (latitude and longitude) or, with the **System** option, in any other system of the catalog.

GIS import and export

MAPIMPORT brings **SHP** (with its DBF), **KML**, **KMZ** and **GeoJSON** files into the drawing:

- points, lines and polygons (holes as closed polylines);
- reprojected to the drawing's system according to the SHP's `.prj` (KML and GeoJSON are always in WGS84);
- the attributes of each feature are stored on the object as extended data;
- the layer can come from a field, and points can be a block whose attributes take the values of the fields with the same name.

MAPEXPORT saves the selection to SHP, KML, KMZ or GeoJSON; the format comes from the file extension. SHP is written in the drawing's system, with its `.prj` and one file per geometry type. KML, KMZ and GeoJSON are written in WGS84 and need the drawing to have a coordinate system.

Georeferenced images

- GEOIMAGE inserts an image at its real coordinates from its *world file* (`.tfw`, `.jgw`, `.pgw`, `.bpw`, `.wld`...) or its GeoTIFF tags. If the image carries its coordinate system (`.prj` or GeoTIFF) and it is not the drawing's, the image is reprojected.
- GEOREFIMAGE places an image without georeferencing using control points: for each one, specify the point on the image and its real position. With two points the image is moved, rotated and scaled exactly; with three it is fitted by least squares.

Large orthophotos are loaded in tiles (see Raster images). ECW, MrSID and JPEG 2000 cannot be read: CONVERTIMAGE turns them into GeoTIFF with GDAL (installed separately), keeping the size and the georeferencing, and attaches the GeoTIFF in the same place in the drawing.

Tips

- Use meters as drawing units (UNITS) when working in UTM.
- With UTM coordinates the numbers are large (X ? 400,000, Y ? 4,500,000): use ZOOMEXTENTS to find the drawing.

Online maps

MAPSERVICE loads maps from WMS, WFS, WMTS and XYZ services as a background or as vectors. The Spanish servers are built in; those of other countries are in Map servers by country.

See also

Raster images · Coordinates · Tutorial: georeferenced plan · Map servers by country

Layouts and plotting

In **model space** you draw at full size (1 unit = 1 meter, or 1 millimeter, depending on the project). In the **paper space** of each **layout** you compose the sheet to be printed: the title block, notes, and one or more **viewports** that show the model at the chosen scale. Plotting turns the model or a layout into a PDF, SVG or PNG file.

Model and layout tabs

Below the drawing area are the **Model** tab and one tab per layout in the drawing. Switching to a layout shows the sheet of paper with its shadow and printable area, and the viewports with their contents.

A button next to the tabs shows whether you are working on paper (**PAPER**) or inside a viewport (**MODEL**); a click switches.

Viewports

- Double-click inside a viewport, or use MSPACE, to enter it: what you draw or edit goes to the model, clipped and at the viewport scale.
- Double-click outside the viewports, or use PSPACE, to return to paper.
- On paper you draw the title block, the texts and the dimensions of the sheet, at paper size (in millimeters or inches).
- Each viewport keeps its scale and its list of frozen layers: the **Freeze in current viewport** icon of the layer control hides a layer in that viewport only, on screen and when plotting. AutoCAD respects it.
- Annotative objects of the model are shown in each viewport at its annotation scale (the viewport's own or the one matching its scale): the same text comes out the same size on paper in a 1:50 viewport and in a 1:100 one.

LAYOUT and the layout tab context menu create, copy, delete, rename and insert layouts from templates (drag the tabs to reorder). MVIEW creates viewports (also non-rectangular) and VPCLIP clips them; inside a viewport zoom changes its view, and its scale is set in the status bar or PROPERTIES. VPMAX maximizes it.

Plotting

PLOT (Ctrl+P) opens the **Plot** dialog box:

Setting	Options
Page setup	One saved with PAGESETUP, the previous one, or none
Printer/plotter	Output to a vector PDF file (saved as "DWG To PDF.pc3", as in AutoCAD)
Paper size	Those of "DWG To PDF.pc3": ISO (A and B), ANSI and ARCH; portrait or landscape
What to plot	Layout, Extents, Display, Limits or Window (two corners in the drawing)
Scale	Fit to paper or a scale: millimeters (or inches) of paper per drawing units
Offset	Plot origin or Center the plot
Plot style table	Color, screening and lineweight of each color when plotting
Options	Plot lineweights, plot with plot styles, plot paper space last, scale lineweights

When you click OK you choose the file: the extension (**.pdf**, **.svg** or **.png**) decides the format. For PNG you choose the resolution in dots per inch. **Apply to Layout** saves the settings in the layout.

The PDF is vector-based: lines, fills and texts (with their real SHX or TrueType fonts) stay sharp at any zoom. With a system printer, the sheet is printed at the printer resolution.

Plot styles (CTB and STB)

The plot style table decides the color, screening and lineweight each object is plotted with. LinceCAD includes

monochrome.ctb (everything in black) and **grayscale.ctb** (grays), and also uses the `.ctb` files (or `.stb`, if the drawing uses named plot styles) in the drawing folder and in `%APPDATA%\lince\estilos_trazado`. You can copy AutoCAD's tables there.

Page setups and preview

- PAGESETUP manages the page setup of each layout and the named page setups, and imports them from other drawings. They are stored in the drawing as in AutoCAD.
- PREVIEW shows the sheet exactly as it will be plotted, with plot styles; the wheel zooms and dragging pans.
- EXPORT plots directly to PDF, SVG or PNG with the same dialog box.

Notes

- Plotting is always wireframe (no hidden lines), which is what a 2D drawing needs.
- Viewport views cannot be rotated yet, and ZOOM does not work inside a viewport yet.

See also

Layers · Dimensions and annotation scales · Raster images · Tutorial: from a plan to PDF

File formats and compatibility

LinceCAD works directly with AutoCAD's formats: it opens and saves **DWG** and **DXF** with no conversion, and what it saves opens in AutoCAD with all its entities. Compatibility is checked with AutoCAD 2027 on dozens of real drawings, auditing every copy.

Opening

OPEN (Ctrl+O), or dragging the file onto the window, opens:

- **DWG** files from R14 to 2018, that is, those written by every AutoCAD release from R14 to 2027;
- text **DXF** files of the same versions.

Each drawing opens in its own tab, in the background and with a progress bar (an 80 MB plan takes a few seconds). If the empty drawing from startup (Drawing1) has not been touched, it is replaced. Several drawings can be open at once, and you can copy and paste between them.

Saving

- QSAVE (Ctrl+S) saves the drawing in its format and version. A DWG older than 2000 (R14) is saved as DWG 2018.
- SAVEAS chooses the format and version. In the application menu and the File menu, **Save As** has a submenu with all the versions; when typed, it proposes DWG 2018.

Save as	Opened by
DWG 2018	AutoCAD 2018 to 2027
DWG 2013	AutoCAD 2013 to 2027
DWG 2010	AutoCAD 2010 to 2027
DWG 2007	AutoCAD 2007 to 2027
DWG 2004	AutoCAD 2004 to 2027
DWG 2000	AutoCAD 2000 to 2027
DXF 2018	AutoCAD 2018 to 2027 and almost any CAD program
DXF 2000	AutoCAD 2000 to 2027 and older programs

When saving to an earlier version, objects that do not exist in it (proxy objects of third-party applications, render sun) are left out, with a warning. Drawings with AutoCAD Architecture objects can only be saved as DWG in their original version. The DWG 2000 format is not compressed, so it takes about twice the space of the same drawing in 2004 or later.

Safe saving

- The drawing is first written to a temporary file; only when it is complete does the original become **.bak** and the new file take its place. If anything fails, the original is left intact.
- Read-only files are not overwritten.
- When you close a drawing, exit, or open another drawing in its place, you are asked whether to save the changes.

Automatic save

Every 10 minutes, drawings with unsaved changes are saved in the background to the user's temporary folder (`%TEMP%\lince\<name>_<n>.sv$`). The file is deleted when the drawing is closed normally. If LinceCAD closes unexpectedly, opening the drawing again warns that there is a more recent automatic save: change its extension to `.dwg` and open it.

Damaged drawings

- AUDIT checks the integrity of the open drawing and fixes what is safe to fix.
- RECOVER opens a damaged file in tolerant mode, recovers whatever can be read and shows a report. Then save the drawing to keep what was repaired.
- If a file cannot be read, the error is shown and the drawing that was open is not lost.

Other formats

Format	How
PDF, SVG, PNG	Output with PLOT or EXPORT (see Layouts and plotting)
SHP, KML, KMZ, GeoJSON	MAPIMPORT and MAPEXPORT
Raster images	IMAGEATTACH
CSV	Tables (TABLE, TABLEEXPORT) and attributes (ATTEXT)
`.scr` scripts	SCRIPT
AutoLISP `.lsp`	APPLOAD

Notes

- Saving to R14 or R12, to DXF 2004 to 2013, or to binary DXF is not in the save dialog yet.
- There are no `.dwt` templates: NEW starts an empty metric drawing (ISO-25).
- LinceCAD does not associate itself with `.dwg` files when installed, so as not to take them from AutoCAD; the installer can add it to "Open with".

See also

What's new · External references · SAVEAS · RECOVER

Options and languages

OPTIONS (alias OP; also on the application menu and the Tools menu) opens the **Options** window with the program preferences. They belong to the user, not to the drawing: they apply to every drawing you open.

Interface tab

Setting	What it does
Language	Language of the whole interface: ribbon, menus, prompts, options, messages and help
Number format	Automatic (by language), Decimal comma (1234,56) or Decimal point (1234.56)
Decimal places	Number of decimal places of the lengths displayed (0 to 8)
Example	How a number will look with the chosen format

Apply tries the change without closing the window; **OK** applies and saves it; **Cancel** leaves everything as it was. Changing the language takes effect at once: no restart is needed.

Open and Save tab

Setting	What it does
QNEW template	Template that QNEW uses for new drawings
Start page	Shows the start page with templates and recent files while nothing has been drawn
Updates	Check for updates automatically: at startup and every 12 hours, in the background. On by default
Update channel	Stable (default) or Beta: the beta channel also gets preview versions

Updates are explained in CHECKFORUPDATES and in License and maintenance.

Automation tab

Lets other programs on the computer (Python scripts, AI assistants over MCP) read and change the drawings: Allow automation (API/MCP), read-only mode, confirmation in the AIACTIONS palette, allowed folders and the local HTTP MCP server. Changes are saved at once. Everything is explained in Automation (API and MCP).

Language and command names

- In English you type the English command names (LINE, CIRCLE...) and their aliases (L, C...).
- In Spanish you type the Spanish AutoCAD names (LINEA, CIRCULO...).
- In any language the global names with an underscore (`_LINE`, `_CIRCLE`) and the Spanish names always work, so `.scr` scripts and AutoLISP routines run with any interface language.
- The first time, LinceCAD starts in the system language if it has a catalog for it; otherwise in Spanish.

Number format

The format decides how numbers are **displayed**: status bar coordinates, measurements from DIST or LIST, properties... With **Automatic (by language)** the language's format is used (comma in Spanish, point in English). With a decimal comma, X and Y are separated by a semicolon (`12,5; 30`).

What you **type** never changes: on the command line it is always decimal point and comma between coordinates (10.5,20), AutoCAD-style.

Drawing units (decimal, feet and inches, angles) are a different thing: they are stored in each drawing and changed with UNITS. The decimal separator of dimensions is set by their style (DIMSTYLE).

Where they are stored

Preferences are stored in `preferencias.toml`, in the user configuration folder: `%APPDATA%\lince` on Windows and `~/.config/lince` on Linux. The same folder holds `lisp.toml` (APPLOAD routines and trusted folders), `idiomas` (added language catalogs) and `estilos_trazado` (your own CTB and STB tables).

Adding a language

Each language is a text file `<code>.ftl` (UTF-8) with one `key = text` line for each text of the interface. Nothing needs to be compiled:

1. Start from `es.ftl`, the complete Spanish catalog (in the `idiomas` folder of the LinceCAD source code), and save it with the code of the new language: `fr.ftl`, `de.ftl`, `pt-BR.ftl`...
2. Translate the texts to the right of `=`; do not change the keys or the `{ $name }` variables. Use AutoCAD's official terms in that language.
3. Put the name of the language, written in that language, in `idioma-nombre`, and its decimal separator (`,` or `.`) in `idioma-separador-decimal`.
4. `cmd-<command>-nombre` and `cmd-<command>-alias` hold the names and aliases to be typed in that language. In the options (`opt-…`), the capital letters are the abbreviation.
5. Copy the file to the `idiomas` folder next to `lince.exe` or to `%APPDATA%\lince\idiomas` and restart LinceCAD: the language appears under Language.

A partial catalog works: whatever is missing is shown in Spanish. A file with the code of an existing language (`es.ftl`, `en.ftl`) replaces only the keys it contains, which is useful to adjust a few texts.

The pages of this help are translated the same way: an `ayuda\<code>` folder next to the program with the pages you want to translate; missing pages are shown in Spanish.

Other preferences

- The workspace (ribbon, Classic or a saved one) is chosen with the ? button on the status bar; the ribbon, toolbars, shortcuts and aliases are adapted with CUI (see The user interface).
- Automatic save runs every 10 minutes (see File formats).

See also

The user interface · Coordinates · Keyboard shortcuts · UNITS

Keyboard and mouse shortcuts

LinceCAD uses AutoCAD's keys and mouse gestures. The keyboard always goes to the command line: you do not need to click it before typing a command.

Function keys

Key	Function
F1	Help
F2	Text window with the full command line history
F3	Object snaps (OSNAP)
F5	Next isometric plane, with isometric drafting (ISODRAFT)
F7	Grid (not available yet: the grid is always shown)
F8	Ortho mode (ORTHO)
F9	Snap the cursor to the grid (SNAP)
F10	Polar tracking (POLAR)
F11	Object snap tracking (OTRACK)
F12	Dynamic input (DYN)

Command shortcuts

Shortcuts assigned to commands in the program's command registry:

Shortcut	Command	Function
Ctrl+1	PROPERTIES	Properties
Ctrl+A	AI_SELALL	Select All
Ctrl+C	COPYCLIP	Copy Clip
Ctrl+N	NEW	New
Ctrl+O	OPEN	Open
Ctrl+P	PLOT	Plot
Ctrl+Q	QUIT	Exit
Ctrl+S	QSAVE	Save
Ctrl+Shift+C	COPYBASE	Copy with Base Point
Ctrl+Shift+S	SAVEAS	Save As
Ctrl+Shift+V	PASTEBLOCK	Paste as Block
Ctrl+V	PASTECLIP	Paste
Ctrl+X	CUTCLIP	Cut
Ctrl+Y	REDO	Redo
Ctrl+Z	UNDO	Undo
Del	ERASE	Erase
F1	HELP	Help
F10	POLAR	POLAR
F3	OSNAP	OSNAP
F7	GRID	GRID
F8	ORTHO	ORTHO

F9	SNAP	SNAP

Other combinations

Keys	Function
Ctrl+Shift+A	Turns whole-group selection on or off (PICKSTYLE)
Ctrl+E	Next isometric plane, like F5
Ctrl+Tab / Ctrl+Shift+Tab	Next or previous open drawing

Editing keys

Key	Function
Enter or Space	Sends what you typed or ends the selection; on an empty line, repeats the last command
Esc	Cancels the running command (or grip editing); a second Esc clears the selection
Delete	Erases the selection, with no command running
Tab	Completes the suggested command name; in dynamic input, locks the value and moves to the other field; in a table, moves to the next cell
Arrow keys	Choose among the autocomplete suggestions; in a table, move between cells
Shift while selecting	Removes objects from the selection instead of adding them

Space does not send while you are typing text (TEXT, MTEXT, names) or inside an AutoLISP expression that starts with `(`: in those cases, only Enter does.

Mouse

Action	Function
Click	Specifies a point or selects an object
Drag right / left	Window / crossing selection
Click a grip	Makes it hot to stretch, move, rotate, scale or mirror
Double-click an object	Opens its editor (text, hatch, attributes, block, cell) or the Properties palette
Right-click	Shortcut menu: Enter, Cancel, command options, repeat, edit the selection...
Wheel	Zoom centered on the cursor
Drag with the middle button	Pan: moves the view
Double-click the middle button	Zoom extents: the whole drawing on screen
Click the blue options of the prompt	Chooses that option

In a layout, double-clicking inside a viewport enters it, and double-clicking outside the viewports returns to paper.

Notes

- Command options are chosen by typing their capital letters (`C` for **Close**, `U` for **Undo**), as in AutoCAD; in Spanish the letters may be different.
- The clipboard keys (Ctrl+C, Ctrl+X, Ctrl+V) only act when the command line is empty; if another text box has the keyboard focus, they copy and paste text.
- With CUI you can add your own shortcuts (Ctrl, Alt or a function key plus a key) for any command; your own shortcut wins over a built-in one with the same keys. Aliases are changed with ALIASEDIT.

See also

The user interface · Object snaps and precision · Selecting objects

What's new

History of the LinceCAD versions, from the most recent to the oldest. The installed version is shown in the window title and in ABOUT.

1.0.0 (coming): bug fixes

- **Aliases as in AutoCAD 2027**; with the English interface the Spanish aliases no longer apply, and _ plus an English alias works in any language. See Command aliases.
- LINE: Undo also removes the segment, not only the point.
- OPEN, SAVEAS (and its versions), RECOVER and NEW follow FILEDIA and, in scripts, SCRIPT, AutoLISP and the API, ask for the file on the command line. SAVEAS offers DWG, DXF and DWT in the dialog.
- ORTHO, OSNAP, SNAP, POLAR, LWDISPLAY and PAN can be typed; SETVAR, scripts and AutoLISP read and change OSMODE, ORTHOMODE, SNAPMODE, AUTOSNAP, POLARANG and GRIDMODE, and `getvar`/`setvar` see every SETVAR variable.
- In evaluation mode the query palettes can be opened (PROPERTIES, LAYER, XREF…).
- MESHREFINE makes the smoothed shape the mesh; ARRAY offers PAth; RECTANG previews in the UCS; texts and help fixed.

0.1.4-beta (November 2026)

- **Complete external references**: XATTACH (attach or overlay), -XREF (bind, detach, unload, reload, path and path type), the XREF palette with the tree of nested references and a status bar warning when a file changes, XCLIP and the BINDTYPE, REFPATHTYPE, VISRETAIN, XCLIPFRAME and XFADECTL variables.
- **PDF underlays**: PDFATTACH, -PDFATTACH, PDFCLIP, PDFADJUST and PDFLAYERS, without external programs.
- **Gradients** (GRADIENT and the Gradient tab of HATCH) and **associative hatches**, with the new options of HATCHEDIT, HATCHGENERATEBOUNDARY, TRIM on hatches and HPGAPTOL.
- XLINE and RAY.
- **Isometric drafting**: ISODRAFT, ISOPLANE, F5 and Ctrl+E, the ISO button and isocircles in ELLIPSE.
- **UCS**: UCS, UCSMAN, UCSICON and PLAN; trans converts between the WCS and the UCS.
- **Annotation scale and annotative objects**: CANNOSCALE with a selector on the status bar, annotative text and dimension styles, OBJECTSCALE, ANNOAUTOSCALE, **Annotative** in PROPERTIES and viewports that each show their own scale.
- **Associative dimensions** (DIMASSOC, DIMREASSOCIATE and DIMDISASSOCIATE) and new dimensions: DIMARC, DIMJOGGED, DIMJOGLINE, QDIM, DIMSPACE, DIMBREAK, three-point DIMANGULAR, TOLERANCE and measurement tolerances in DIMSTYLE.
- **Fields**: FIELD, UPDATEFIELD and FIELDDISPLAY.
- FIND (find and replace) and SPELL, with Spanish and English dictionaries included.
- **Output**: PUBLISH (several sheets to one PDF or one PDF per sheet), ETRANSMIT (ZIP package with the dependent files), layered PDF and PDF/A-2b, SAVEAS to DWG R14 and DXF R12, and proxies kept as AutoCAD does when changing version.
- **Help** (F1): this viewer, with help for every command, extended tooltips and the PDF manual.
- **Interface**: collapsible ribbon (full, tabs only or panel titles; panels that do not fit collapse) with every command; **workspaces** (WORKSPACE, WSSAVE and the ? gear); **dockable palettes** on the left or right with auto-hide; **floating command line** with transparency; CUI (ribbon, toolbars and shortcuts) and ALIASEDIT (aliases in `lince.pgp`, compatible with acad.pgp).
- **Object and layer transparency** (0–90 %): in PROPERTIES, the layer manager and -LAYER, with CETRANSPARENCY, TRANSPARENCYDISPLAY and the TRANSP button; saved in DWG files and plotted.

0.1.3-beta (November 2026)

- **Block Editor**: BEDIT (also by double-clicking a block), BSAVE, BSAVEAS and BCLOSE; dynamic blocks

remain dynamic after being edited.

- **Editing**: complete PEDIT, SPLINEDIT, LENGTHEN, ALIGN, OVERKILL and SCALELISTEDIT.
- **Groups**: GROUP, GROUPEDIT and UNGROUP, with Ctrl+Shift+A.
- ISOLATEOBJECTS, HIDEOBJECTS and UNISOLATEOBJECTS, with the ISOLATE button on the status bar.
- **Multilines**: MLINE and MLEDIT.
- **Tables**: TABLE, cell editing, formulas, TABLESTYLE, CSV import and TABLEEXPORT.

0.1.2-beta

- **Raster images**: IMAGEATTACH, -IMAGE, IMAGECLIP, IMAGEADJUST and TRANSPARENCY, with large tiled orthophotos and plotting to PDF, SVG and PNG.
- **Georeferenced images**: GEOIMAGE (world files and GeoTIFF) and GEOREFIMAGE (control points).
- **Dynamic blocks**: visibility, flip, stretch, rotation, arrays and lookups, with their grips; RESETBLOCK.

0.1.1-beta

- **Blocks** palette with thumbnails and libraries (INSERT), -INSERT, **DesignCenter** (ADCENTER) and **tool palettes** (TOOLPALETTES). Architecture and Electrical libraries included.
- Attributes: BATTMAN, ATTSYNC and ATTEXT.
- **Geolocation**: GEOGRAPHICLOCATION, GEOCOORD, latitude and longitude on the status bar, MAPIMPORT and MAPEXPORT (SHP, KML, KMZ and GeoJSON).
- **AutoLISP**: `.lsp` routines, expressions on the command line, `c:` commands and APPLOAD.

0.1.0-beta: first complete basic 2D clone

Brings together everything from the 0.0.x versions: drawing, editing, precision, layers, text, dimensions, blocks, hatches, layouts, plotting, utilities and several open drawings. The program is now split into independent modules, which speeds up its development.

0.0.14-alpha

- Utilities: DIST, AREA, ID, LIST, MEASUREGEOM, text window (F2), PURGE, UNITS, clipboard, QSELECT, SELECTSIMILAR, SETVAR, SCRIPT and DRAWORDER.
- Annotation and drawing: MLEADER and its styles, REVCLOUD, WIPEOUT, DONUT, SOLID, DIVIDE, MEASURE, PTYPE, BOUNDARY, REGION, and complete ELLIPSE and SPLINE.
- **Several open drawings**, with tabs, CLOSE, CLOSEALL and copy and paste between drawings.
- Plotting and saving, much faster using several cores.

0.0.13-alpha

- Complete Layer Properties Manager: all columns, filters, layer states and the layer commands.
- Plotting: Window and Limits areas, PAGESETUP, CTB/STB plot styles, PREVIEW and PNG/SVG output.
- Visible name **LinceCAD** and a smaller executable.

0.0.12-alpha

- Progress bar when opening large drawings.

0.0.11-alpha

- Editable grips (stretch, move, rotate, scale, mirror), **POLAR** (F10), **OTRACK** (F11) and dynamic input (F12).
- Tangent, Nearest and Extension snaps, and typed snap overrides (END, MID...).
- CIRCLE by 3 points, 2 points and tangent-tangent-radius; ARC in four variants; PLINE with arcs and width.
- Clickable options, shortcut menu, double-click to edit, Shift to remove from the selection, Ctrl+A and ZOOM.

0.0.10-alpha

- HATCH and HATCHEDIT; MTEXT with an editor; TEXTEDIT; DIMCONTINUE, DIMBASELINE and DIMORDINATE.
- Construction lines, rays, multilines, multileaders and tables are displayed.
- Attributes (ATTDEF, ATTEDIT, EATTEDIT), block redefinition, inserting a DWG as a block, WBLOCK, read-only external references (XREF), AUDIT and RECOVER.

0.0.9-alpha

- Interface in Spanish and English, extensible with more languages without recompiling; number format; OPTIONS.

0.0.8-alpha

- Plotting to vector PDF, layouts and viewports, MSPACE and PSPACE.
- Editable dimensions, DIMSTYLE, STYLE and LINETYPE with `.lin` loading.
- Text with the real SHX and TrueType fonts.
- Unsaved-changes warning, safe saving with `.bak`, automatic save and protection against damaged files.
- Linux version.

0.0.7-alpha

- BLOCK and INSERT; linear, aligned, radius, diameter and angular dimensions.

0.0.6-alpha

- PROPERTIES palette, MATCHPROP, COLOR, LINETYPE and LWEIGHT.
- Hatches, linetypes and lineweights drawn as in AutoCAD.
- RECTANG, POLYGON, ELLIPSE, POINT, SPLINE, MIRROR, ARRAY, TRIM, EXTEND, FILLET, CHAMFER, EXPLODE, JOIN, BREAK and STRETCH.

0.0.1-alpha to 0.0.5-alpha (October 2026)

- Opening, viewing, drawing and saving DWG and DXF; layers; selection, editing, undo, object snaps and Ortho.
- Ribbon, menus and Classic workspace with all commands, own icons and autocomplete; layer control and manager; installer.

Coming soon

A collapsible ribbon with editable aliases and shortcuts, and this help (F1).

See also

File formats · The user interface · ABOUT

Automation (API and MCP)

LinceCAD can be driven by other programs on the same computer: Python scripts, other programs and **AI assistants** (Claude, for example) through **MCP** (*Model Context Protocol*). They can open and save drawings, query and create entities, change layers, run commands, evaluate AutoLISP, plot to PDF and see the drawing as an image. Everything goes through local channels: nothing leaves the computer.

Enabling it

Automation is **off** by default. Turn it on in OPTIONS, **Automation** tab:

Setting	What it does
Allow automation (API/MCP)	Opens the local API channel. Without it, no program can connect to the window
Read-only (they can read, not change)	Programs can read the drawing but not change it, save it or run commands
Confirm the changes of AI assistants (AI Actions palette)	Whatever an AI assistant wants to change waits in the AIACTIONS palette until you accept it
Allowed folders	Drawings are only opened, saved and plotted in those folders (empty: anywhere)
MCP server over HTTP on 127.0.0.1, port	MCP server over HTTP, on `127.0.0.1` only, to connect an assistant to the running LinceCAD

The settings are saved at once in `api.toml`, in the user configuration folder (`%APPDATA%\lince` on Windows, `~/.config/lince` on Linux), together with the **token**: the password programs present when they connect. Only your user can read it. **Generate a new token** changes it (programs holding the old one can no longer get in).

Control over what the AI does

- **One undo step per call**: everything a call changes is undone with a single UNDO.
- **AI Actions** (AIACTIONS): with confirmation on, each change requested by an assistant shows up with what it wants to do; **Accept** runs it and **Reject** denies it. **Accept all in this session** stops asking until LinceCAD is closed. Below is the log of every call.
- If you are in the middle of a command, changes wait: the API answers "busy".
- An internal error during a call is undone entirely and LinceCAD keeps running.

Without a window

- `lince --servidor` serves the API with no visible window (for batch processes and servers). `--canal NAME` changes the channel name.
- `lince --mcp` is an MCP server over standard input and output, for assistants that start it themselves.

Both are the same LinceCAD with its window off-screen: same commands and same view (for captures). They open no dialogs and ask nothing; they honour the read-only mode and allowed folders of `api.toml`. On Linux they need a graphical session (or `xvfb-run`).

Connecting an AI assistant (MCP)

Claude Desktop: in `%APPDATA%\Claude\claude_desktop_config.json` (Windows):

```
{
  "mcpServers": {
    "lince": { "command": "C:\\Program Files\\lince\\lince.exe", "args": ["--mcp"] }
  }
}
```

Claude Code (no window, in a project): `claude mcp add --scope project lince -- "C:\Program Files\lince\lince.exe" --mcp`

Claude Code with LinceCAD open (enable MCP server over HTTP on 127.0.0.1, port; the token is in `api.toml`): `claude mcp add --transport http lince http://127.0.0.1:7341/mcp --header "Authorization: Bearer <token>"`

The assistant sees the **tools** (open, query, create, dimension, insert blocks, plot to PDF, capture the view, undo…), the **resources** (summary, entities, layers and blocks of each open drawing, and these help pages as documentation) and three **prompts**: review the drawing, dimension a floor plan and clean the layers.

Python

There are two Python packages (Python 3.9 or newer):

- `lince` (client): drives LinceCAD through the API. `pip install lince`. No dependencies.

```
import lince
with lince.iniciar() as s:              # starts "lince --servidor" with no window
    dib = s.nuevo()
    dib.crear_capa("MUROS", color=1)
    with dib.transaccion():             # a single undo step
        dib.linea([0, 0], [100, 0], layer="MUROS")
        dib.circulo([50, 50], 20)
    dib.trazar_pdf("plan.pdf")
    dib.guardar_como("plan.dwg")
```

For a running instance (with automation enabled): `s = lince.conectar()`; `dib = s.dibujo`.

- `lince_nucleo` (PyO3 wheel): read, modify, save and query DWG/DXF **without opening LinceCAD** (batch, servers, Jupyter). `pip install lince-nucleo`.

```
import lince_nucleo as ln
dib = ln.abrir("plan.dwg")
for e in dib.consultar(tipo="CIRCLE"):
    print(e["handle"], e["radius"])
dib.guardar("out.dwg", "2018")
```

Entities use the same typed format of §5.1. Examples in `python/ejemplos` of the source.

Your own programs (JSON-RPC)

Programs connect through the pipe `\\.\pipe\lince-<user>` (Windows) or the socket `$XDG_RUNTIME_DIR/lince.sock` (Linux) and send one JSON-RPC 2.0 message per line. The first one is `session.hello` with the token:

```
    {"jsonrpc":"2.0","id":1,"method":"session.hello","params":{"token":"<token>"}}
    {"jsonrpc":"2.0","id":2,"method":"entity.create","params":{"entity":{"type":"LINE","star
t":[0,0],"end":[100,0]}}}
    {"jsonrpc":"2.0","id":3,"method":"document.saveAs","params":{"path":"C:\\drawings\\test.
dwg"}}
```

There are methods for drawings (`document.*`), commands (`command.run`), AutoLISP (`lisp.eval`), entities (`entity.query/get/create/set/delete`, as objects or as `entget` DXF lists), layers, blocks, styles, variables, selection, undo, plotting, view capture and events. The full specification is in `docs/api.md` of the source code.

See also

OPTIONS · AIACTIONS · AutoLISP · SCRIPT

Map servers by country

Official map services (cadastre, national mapping agency, national SDI, orthophoto) in Europe, Latin America and North Africa that are the equivalent of Spain's Catastro and IGN, with the URL to add in MAPSERVICE, WMS, WFS or WMTS. Every URL was checked one by one with the same requests LinceCAD makes (see the date at the bottom).

How to add a server

1. Open the command for the service type: WMS, WFS or WMTS. The new server is saved with the type of the command you add it from. From MAPSERVICE it is saved as **XYZ** if the URL contains `{z}` and as **WMS** otherwise.
2. Expand **Add another server…**, type a name and the URL from the list below and click **Add and save**.
3. Click **Connect**, choose the recommended layer and click **Load**.

Which type to choose and what the service must offer:

Type	What it brings	Requirement for LinceCAD
WMS	Image of the view	The server must offer the **drawing's coordinate system**: LinceCAD requests the image (GetMap 1.3.0) directly in that system
WMTS	Tiled image	`GoogleMapsCompatible` tile matrix set (EPSG:3857) in PNG; if the service uses another name, add it as XYZ with the template given
XYZ	Tiled image	Template with `{z}`, `{x}` and `{y}`; 256-pixel tiles in EPSG:3857, PNG or JPEG
WFS	Vectors with attributes	Must answer in **GeoJSON** (`outputFormat=application/json`) and accept WGS84 coordinates (CRS84); services that only return GML do not work

Coordinate system. The drawing must have one (GEOGRAPHICLOCATION) that is in LinceCAD's catalogue (see Georeferencing). WMTS and XYZ tiles are reprojected to any system in the catalogue. A WMS, however, only works if it offers the drawing's system: the relevant ones are given for each service. Almost all offer Web Mercator (EPSG:3857), but a drawing in 3857 does not measure true distances (it enlarges them by a factor of 1/cos(latitude)): use it only as a reference background. Draw in the country's **national system**, which the catalogue already includes (it is given for each country): WMTS and XYZ tiles are reprojected to it, and the WMS that offer it serve it directly. It is advertised in the GetCapabilities (checked on 2026-11-02) of the WMS of France (2154), Switzerland (2056), the Netherlands (28992), Belgium (31370 and 3812), Austria (31254–31256), Germany (25832, 25833 and 31467), Poland (2180), Luxembourg (2169), Portugal (3763), Italy (3003, 3004 and 25832–25834), Argentina (5343–5349), Uruguay (5382), Brazil (4674) and Ecuador (32717).

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Europe

To find more services in the European Union, use the **INSPIRE geoportal**, `https://inspire-geoportal.ec.europa.eu`: it catalogues the view (WMS/WMTS) and download (WFS) services of every member state. On the day of the check its HTTPS certificate had expired and the browser may warn about it.

Portugal: DGT (Direção-Geral do Território)

- **OrtoSat 2023 satellite orthoimage** (WMS): `https://ortos.dgterritorio.gov.pt/wms/ortosat2023` · layer `ortoSat2023-CorVerdadeira` · CRS 3857 and 3763.
- **2021 orthophotos** (WMS, northern mainland): `https://cartografia.dgterritorio.gov.pt/ortos2021/service` · layer `Ortos2021-RGB` · CRS 3857 and 3763.
- **INSPIRE land cadastre** (WMS and WFS at the same URL):

`https://snicws.dgterritorio.gov.pt/geoserver/inspire/ows` · WMS layer `cadastralparcel`, WFS layer `inspire:cadastralparcel` (GeoJSON) · CRS 3857, 3763 and others. Coverage is partial: only municipalities with a geometric cadastre.

- **CAOP administrative boundaries** (WMS): `https://geo2.dgterritorio.gov.pt/geoserver/caop_continente/wms` · layer `cont_municipios`.
- Licence: DGT open data, free of charge (DGT publishes its products under CC BY 4.0; check each one on dados.gov.pt). Attribution: "©DGT – Direção-Geral do Território".
- National system in LinceCAD: **PT-TM06 (EPSG:3763)**, which these WMS offer: work with the drawing in 3763.

France: IGN (Géoplateforme) and cadastre (PCI)

- **Raster WMS** `https://data.geopf.fr/wms-r/wms` · layers `ORTHOIMAGERY.ORTHOPHOTOS` (orthophoto), `GEOGRAPHICALGRIDSYSTEMS.PLANIGNV2` (base map) and `CADASTRALPARCELS.PARCELLAIRE_EXPRESS` (cadastre) · CRS 3857.
- **WFS** `https://data.geopf.fr/wfs/ows` · layers `CADASTRALPARCELS.PARCELLAIRE_EXPRESS:parcelle` (parcels) and `BDTOPO_V3:batiment` (buildings) · GeoJSON.
- **WMTS** `https://data.geopf.fr/wmts`: its tile matrix set is called `PM`, not `GoogleMapsCompatible`. Add it as **XYZ** with one of the two templates below.
- **Plan IGN** (XYZ): `https://data.geopf.fr/wmts?SERVICE=WMTS&REQUEST=GetTile&VERSION=1.0.0&LAYER=GEOGRAPHICALGRIDSYSTEMS`
- **Orthophoto** (XYZ): `https://data.geopf.fr/wmts?SERVICE=WMTS&REQUEST=GetTile&VERSION=1.0.0&LAYER=ORTHOIMAGERY.ORTHOPHOTOS`
- Licence: Licence Ouverte Etalab 2.0 (free use, commercial included, crediting the source); terms at cartes.gouv.fr/cgu. Attribution: "IGN – Géoplateforme"; cadastre: "IGN, DGFiP – Parcellaire Express (PCI)".
- National system in LinceCAD: **RGF93 v1 / Lambert-93 (2154)** and the CC42–CC50 conics (3942–3950). The raster WMS also offers 2154.

Italy: Geoportale Nazionale and cadastre

- **2012 orthophoto** from the Geoportale Nazionale (MASE) (WMS, http only): `http://wms.pcn.minambiente.it/ogc?map=/ms_ogc/WMS_v1.3/raster/ortofoto_colore_12.map` · layers `OI.ORTOIMMAGINI.2012.32` and `OI.ORTOIMMAGINI.2012.33` (zones 32 and 33) · CRS 3857. No conditions stated. Attribution: "Geoportale Nazionale – MASE".
- **Cadastre** of the Agenzia delle Entrate (WMS): `https://wms.cartografia.agenziaentrate.gov.it/inspire/wms/ows01.php` · layers `CP.CadastralParcel` and `CP.CadastralZoning` · CC BY 4.0, "Agenzia delle Entrate". It offers ETRS89 / UTM 32N–34N (25832–25834), RDN2008 and similar, but not 3857: work with the drawing in 25832, 25833 or 25834, all in the catalogue. It does not cover Trento or Bolzano.
- National system in LinceCAD: **RDN2008 / UTM 32N–34N (7791–7793)** or ETRS89 / UTM 32N–34N; for older drawings, Monte Mario / Italy zone 1 and 2 (3003 and 3004, with a datum shift of about 4 m). The Geoportale orthophoto also offers 3003, 3004 and 25832.

Germany: BKG and each Land's cadastre

- **TopPlusOpen** by BKG (WMS): `https://sgx.geodatenzentrum.de/wms_topplus_open` · layer `web` · CRS 3857. As XYZ: `https://sgx.geodatenzentrum.de/wmts_topplus_open/tile/1.0.0/web/default/WEBMERCATOR/{z}/{y}/{x}.png`. Licence Datenlizenz Deutschland – Namensnennung 2.0. Attribution: "©BKG (year) dl-de/by-2-0".
- **basemap.de** (WMS): `https://sgx.geodatenzentrum.de/wms_basemapde` · layer `de_basemapde_web_raster_farbe`. CC BY 4.0. Attribution: "©basemap.de / BKG (month and year)".
- **Cadastre (ALKIS)**: there is no federal service; each Land publishes its own (look for it in the INSPIRE geoportal or geoportal.de). Example, North Rhine-Westphalia (WMS): `https://www.wms.nrw.de/geobasis/wms_nw_alkis` · layer `adv_alkis_flurstuecke` (parcels) · CRS 3857 and 25832 · Datenlizenz Deutschland – Zero 2.0. Attribution: "Geobasis NRW".
- National system in LinceCAD: **ETRS89 / UTM 32N and 33N (25832, 25833)**; for older drawings, DHDN /

Gauss-Krüger zones 2–5 (31466–31469, with a datum shift of about 3 m). TopPlusOpen also offers 25832, 25833 and 31467.

Netherlands: PDOK (Kadaster)

- **BRT base map** (XYZ): `https://service.pdok.nl/brt/achtergrondkaart/wmts/v2_0/standaard/EPSG:3857/{z}/{x}/{y}.png`. CC BY 4.0. Attribution: "Kadaster".
- **Orthophoto** (XYZ): `https://service.pdok.nl/hwh/luchtfotorgb/wmts/v1_0/Actueel_orthoHR/EPSG:3857/{z}/{x}/{y}.jpeg`; or WMS `https://service.pdok.nl/hwh/luchtfotorgb/wms/v1_0` · layer `Actueel_orthoHR`. CC BY 4.0. Attribution: "Beeldmateriaal.nl".
- **Cadastre** (WMS): `https://service.pdok.nl/kadaster/kadastralekaart/wms/v5_0` · layer `Perceel` · CRS 3857 · CC0. Suggested attribution: "Kadaster".
- National system in LinceCAD: **Amersfoort / RD New (28992)**, offered by the cadastre and orthophoto WMS (datum shift to WGS84 of about 1 m).
- The cadastre WFS (`.../kadastralekaart/wfs/v5_0`) returns GeoJSON but rejects the WGS84 (CRS84) coordinates LinceCAD asks for: **not usable today**.

Belgium: NGI/IGN, Flanders and Wallonia

- **Cartoweb** by NGI/IGN (WMS): `https://cartoweb.wms.ngi.be/service` · layer `topo` · CRS 3857. As XYZ: `https://cartoweb.wmts.ngi.be/1.0.0/topo/default/3857/{z}/{y}/{x}.png`. Free use under the cartoweb.be terms (ngi.be). Attribution: "©NGI/IGN".
- **Flanders, orthophoto** (WMS): `https://geo.api.vlaanderen.be/OMWRGBMRVL/wms` · layer `Ortho`.
- **Flanders, cadastral parcels** (WMS): `https://geo.api.vlaanderen.be/Adpf/wms` · layer `Adpf`; and WFS `https://geo.api.vlaanderen.be/Adpf/wfs` · layer `Adpf:Adpf` (GeoJSON).
- Flanders: free use (Digitaal Vlaanderen terms). Attribution: "Digitaal Vlaanderen".
- **Wallonia**, orthophoto (WMS): `https://geoservices.wallonie.be/arcgis/services/IMAGERIE/ORTHO_LAST/MapServer/WMSServer` · CRS 31370 and 4326 (not 3857): work with the drawing in 31370.
- The federal cadastre (FPS Finance) has no public WMS that answers today.
- National system in LinceCAD: **BD72 / Lambert 72 (31370)** and **ETRS89 / Lambert 2008 (3812)**. Cartoweb offers both, and the Wallonia orthophoto, which only offers 31370, can now be used with the drawing in 31370.

Switzerland: swisstopo

- **WMS** `https://wms.geo.admin.ch/` · layers `ch.swisstopo.pixelkarte-farbe` (national map) and `ch.swisstopo.swissimage` (orthophoto) · CRS 3857.
- **National map** (XYZ): `https://wmts.geo.admin.ch/1.0.0/ch.swisstopo.pixelkarte-farbe/default/current/3857/{z}/{x}/{y}.jpeg`
- **Orthophoto** (XYZ): `https://wmts.geo.admin.ch/1.0.0/ch.swisstopo.swissimage/default/current/3857/{z}/{x}/{y}.jpeg`
- **Official cadastral survey** (XYZ): `https://wmts.geo.admin.ch/1.0.0/ch.kantone.cadastralwebmap-farbe/default/current/3857/{z}/{x}/{y}.pn`
- Licence: swisstopo open data, free use (commercial included) crediting the source. Attribution: "©swisstopo"; cadastre: "©swisstopo, official cadastral survey of the cantons".
- National system in LinceCAD: **CH1903+ / LV95 (2056)**, which the WMS also offers.

Austria: basemap.at and BEV

- **basemap.at, base map** (XYZ): `https://mapsneu.wien.gv.at/basemap/geolandbasemap/normal/google3857/{z}/{y}/{x}.png`
- **basemap.at, orthophoto** (XYZ): `https://mapsneu.wien.gv.at/basemap/bmaporthofoto30cm/normal/google3857/{z}/{y}/{x}.jpeg`

- basemap.at: CC BY 4.0. Attribution: "Datenquelle: basemap.at".
- **DKM cadastre** by BEV (WMS): `https://data.bev.gv.at/geoserver/BEVdataKAT/wms` · layer `KAT_DKM_GST-NFL` · CRS 3857 and 31254–31256. CC BY 4.0. Attribution: "©BEV".
- National system in LinceCAD: **MGI / Austria GK West, Central and East (31254–31256)**, with a datum shift of about 1.5 m; or ETRS89 / UTM 32N and 33N.

Poland: GUGiK

- **Parcels (KIEG)** (WMS): `https://integracja.gugik.gov.pl/cgi-bin/KrajowaIntegracjaEwidencjiGruntow` · layers `dzialki` and `numery_dzialek` · CRS 3857. It only draws at large scales (zoom in before loading).
- **Orthophoto** (WMS): `https://mapy.geoportal.gov.pl/wss/service/PZGIK/ORTO/WMS/HighResolution` · CRS 2180 and 4326 (not 3857): work with the drawing in 2180. The same goes for the topographic map.
- National system in LinceCAD: **PL-1992 (2180)** and PL-2000 zones 15–24 (2176–2179). In these systems EPSG puts north first, and LinceCAD already asks for the WMS box in that order.
- Licence: GUGiK data free of charge and reusable. Attribution: "?ród?o: GUGiK (geoportal.gov.pl)".

United Kingdom: Ordnance Survey

- There is no open map service without registration. The **OS Maps API** needs a key (free on the OS OpenData plan of the OS Data Hub, osdatahub.os.uk); without it the answer is "401". With a key, as XYZ: `https://api.os.uk/maps/raster/v1/zxy/Light_3857/{z}/{x}/{y}.png?key=YOUR_KEY`.
- Only **OS OpenData** is open (Open Government Licence v3). Styles and zoom levels with Premium data (OS MasterMap) are paid. Attribution: "Contains OS data ©Crown copyright and database right (year)".
- The cadastre (HM Land Registry, INSPIRE Index Polygons) is published only as a download, not as a map service.
- National system in LinceCAD: **OSGB36 / British National Grid (27700)**. The datum shift uses 7 parameters (2 m nominal, up to 3–4 m): the official OSTN15 grid is not included.

Ireland: Tailte Éireann

- No open service could be verified. Tailte Éireann (formerly OSi) maps and orthophotos are licensed; its INSPIRE orthophoto service did not answer. Use OpenStreetMap or the global alternative below.
- National system in LinceCAD: **IRENET95 / Irish Transverse Mercator (2157)**.

Andorra: Govern d'Andorra (IDE Andorra)

- **2025 orthophoto** (WMS): `https://www.ideandorra.ad/Serveis/wmsorto2025/wms` · layer `Andorra_orto_25cmGSD_2025` · CRS **25831** (ETRS89 / UTM 31N, in LinceCAD's catalogue), 23031 and 4326. It does not advertise 3857: work with the drawing in 25831.
- **General map** (WMS): `https://www.ideandorra.ad/Serveis/wms_cartogeneral/wms` (roads, rivers, boundaries, towns). The full list of services is at cartografia.ad/serveis-ogc.
- Terms: "©Govern d'Andorra"; see the legal notice on cartografia.ad.

Other European countries checked

- **Luxembourg** (ACT), orthophoto (WMS): `https://wms.geoportail.lu/opendata/service` · layer `ortho_latest` · CRS 3857 and 2169 (LUREF / Luxembourg TM, in the catalogue). Open data (CC0). Attribution: "©ACT Luxembourg".
- **Czechia** (?ÚZK), orthophoto (WMS): `https://ags.cuzk.cz/arcgis1/services/ORTOFOTO/MapServer/WMSServer` · layer `0`; cadastre (WMS): `https://services.cuzk.cz/wms/wms.asp` · layer `KN` · CRS 3857 and 32633. Open data (CC BY 4.0). Attribution: "©?ÚZK". The national S-JTSK / Krovak system (5514) is not in the catalogue: use WGS 84 / UTM 33N (32633), which the cadastre offers.
- **Norway** (Kartverket), topographic map (XYZ): `https://cache.kartverket.no/v1/wmts/1.0.0/topo/default/webmercator/{z}/{y}/{x}.png`. CC BY

4.0. Attribution: "©Kartverket".

Latin America

Mexico: INEGI

- **Mapa Digital de México** (WMS): `https://gaia.inegi.org.mx/NLB/tunnel/wms/wms61` · layers `Ortofotos`, `c100` (states), `c101` (municipalities), `c104` (blocks), `c102` (urban localities) · CRS 3857.
- Licence: INEGI's free-use terms (commercial included, crediting the source). Attribution: "Fuente: INEGI".
- National system in LinceCAD: **Mexico ITRF2008 / LCC (6372)** and Mexico ITRF2008 / UTM 11N–16N (6366–6371). This WMS does not advertise them: use the tiles, or the drawing in 3857 only as a background.

Argentina: IGN

- **WMS and WFS** (same URL): `https://wms.ign.gob.ar/geoserver/ows` · layers `ign:provincia`, `ign:departamento` and many more · CRS 3857 · WFS in GeoJSON.
- **Argenmap base map**: IGN blocks its use over WMS; add it as XYZ: `https://wms.ign.gob.ar/geoserver/gwc/service/wmts?SERVICE=WMTS&REQUEST=GetTile&VERSION=1.0.0&LAYER=c` .
- Licence: free reuse (Law 27.275 on access to public information). Attribution: "Instituto Geográfico Nacional de la República Argentina".
- National system in LinceCAD: **POSGAR 2007 / Argentina zones 1–7 (5343–5349)**, which the WMS offers.

Brazil: IBGE (INDE)

- **WMS** `https://geoservicos.ibge.gov.br/geoserver/wms` · layer `CCAR:bc250_2025` (1:250,000 base map) · CRS 3857.
- **WFS** `https://geoservicos.ibge.gov.br/geoserver/wfs` · layer `CCAR:BCIM_Unidade_Federacao_A` (states) · GeoJSON.
- Free use crediting the source. Attribution: "Fonte: IBGE". More services in the INDE catalogue (inde.gov.br).
- National system in LinceCAD: **SIRGAS 2000 / UTM 17S–26S and 11N–24N (31965–31985, 6210, 6211, 5396)** and the Brazil Polyconic (5880). The WMS offers SIRGAS 2000 geographic (4674), not its UTM zones: with the drawing in UTM, use the WFS or the tiles.

Uruguay: IDEuy

- **Orthophotos** (WMS): `https://mapas.ide.uy/geoserver-raster/ortofotos/ows` · layer `ORTOFOTOS_2019` · CRS 3857. Open data. Attribution: "IDEuy – Infraestructura de Datos Espaciales del Uruguay".
- National system in LinceCAD: **SIRGAS-ROU98 / UTM 21S (5382)** and 22S (5383); the orthophoto WMS offers 5382.

Ecuador: IGM

- **National map** (WMS and WFS, same URL): `https://www.geoportaligm.gob.ec/nacional/wms` · layer `igm:Nacional` · CRS 3857 · WFS in GeoJSON.
- **2021 orthophotos** (WMS): `https://www.geoportaligm.gob.ec/orto/toma2021/wms` · one layer per area (`otf_...`) · CRS 3857.
- The service states no licence: check the terms of the IGM geoportal. Attribution: "Instituto Geográfico Militar del Ecuador".
- National system in LinceCAD: **WGS 84 / UTM 17S (32717)**, which the national WMS offers, or SIRGAS 2000 / UTM 17S (31977).

Peru: IGN (through the IDEP)

- **1:100,000 maps** (WMS): `https://www.idep.gob.pe/geoportal/services/DATOS_GEOESPACIALES/PERU_100K/MapServer/WMSServer`

· numbered layers (`1`, `2`…; choose them when connecting) · CRS 3857.

- **1:500,000 raster map** (WMS): `https://www.idep.gob.pe/geoportal/services/MAPA_BASE/PER%C3%9A_RASTER_500K/MapServer/WMSServer` · layer `0`.
- No licence stated. Attribution: "Instituto Geográfico Nacional del Perú". Catalogue of government services at geoidep.gob.pe.
- National system in LinceCAD: **Peru96 / UTM 17S–19S (5839, 5387, 5389)** or WGS 84 / UTM 17S–19S.

Dominican Republic: IGN-JJHM (IDERD)

- **WMS and WFS** (same URL): `https://geoportal.iderd.gob.do/geoserver/ows` · layer `ign:Provincias` · CRS 3857 · WFS in GeoJSON. No licence stated. Attribution: "IGN José Joaquín Hungría Morell – IDERD".

Countries without a verified compatible service

- **Chile**: IDE Chile catalogues its services at geoportal.cl, but no national base map or orthophoto WMS that answered could be found. National system in LinceCAD: **SIRGAS-Chile 2021 / UTM 18S and 19S (20048, 20049)**.
- **Colombia** (IGAC/ICDE): IGAC's ArcGIS services (for example, `https://mapas.igac.gov.co/server/services/Dato_Fundamental_Catastro/MapServer/WMSServer`, cadastre CC BY-SA 4.0) answer, but they do not advertise 3857 and the image request timed out. Catalogue at colombiaenmapas.gov.co. National system in LinceCAD: **MAGNA-SIRGAS 2018 / Origen-Nacional (9377)** and the MAGNA-SIRGAS zones (3114–3118; Bogota, 3116).
- **Costa Rica** (SNIT/IGN): according to its documentation, it publishes in CRTM05 and WGS84; CR05 / CRTM05 (5367) is now in the catalogue, but its server did not answer.
- **Bolivia** (GeoBolivia): its domain did not answer.
- **Paraguay, Venezuela, Panama, Guatemala and Cuba**: no reliable public service could be found. For Venezuela the catalogue includes REGVEN / UTM 18N–20N (2201–2203); for the rest, WGS 84 / UTM or SIRGAS 2000 / UTM of their zone.

For these countries, use the global alternative below.

North Africa

Morocco (ANCFCC), **Algeria** (INCT), **Tunisia** (OTC), **Egypt**, **Libya** and **Mauritania** do not publish open map services today: their maps and cadastre are sold or consulted at their offices. Use the global alternative below and ask each country's agency for the official maps when the project requires them.

National systems in LinceCAD (to draw over that official mapping and over the global tiles):

- **Morocco**: Merchich / Nord Maroc, Sud Maroc, Sahara Nord and Sahara Sud (26191, 26192, 26194, 26195). 3-parameter datum shift (about 7 m).
- **Algeria**: Nord Sahara 1959 / UTM 29N–32N (30729–30732), Nord Algérie and Sud Algérie (30791, 30792). About 1 m.
- **Tunisia**: Carthage / UTM 32N, Nord Tunisie and Sud Tunisie (22332, 22391, 22392). About 14 m.
- **Egypt**: Egypt 1907 / Red, Blue, Purple and Extended Purple Belt (22991–22994). About 1.2 m.
- **Libya**: LGD2006 / Libya TM and UTM 32N–35N (3177, 3199, 3201–3203).
- **Mauritania** and the rest: WGS 84 / UTM of their zone.

Global alternatives

- **OpenStreetMap** (XYZ, already built in): base map of the whole world, ODbL licence. For occasional use only, under the OpenStreetMap Foundation tile policy. Attribution: "OpenStreetMap (openstreetmap.org/copyright)" (LinceCAD writes it automatically).
- **Sentinel-2 cloudless** by EOX, cloud-free 10 m satellite imagery of the whole world. Its WMTS is `GoogleMapsCompatible` and works directly: `https://tiles.maps.eox.at/wmts`.
- **Sentinel-2 cloudless 2016**, layer `s2cloudless_3857`: CC BY 4.0, commercial use allowed. Attribution:

"Sentinel-2 cloudless – https://s2maps.eu by EOX IT Services GmbH (Contains modified Copernicus Sentinel data 2016)".

- **Sentinel-2 cloudless 2018 to 2025**, layers `s2cloudless-2018_3857` to `s2cloudless-2025_3857`: CC BY-NC-SA 4.0, **non-commercial use only**; for commercial use, EOX sells a licence (cloudless.eox.at). Attribution: the same, with the year of the data.
- **Sentinel-2 cloudless 2016** as XYZ: `https://tiles.maps.eox.at/wmts/1.0.0/s2cloudless_3857/default/g/{z}/{y}/{x}.jpg`.

Digital elevation models (DEM)

IMPORTDEM reads the DEMs in ESRI ASCII Grid (`.asc`) and GeoTIFF (`.tif`) published by these agencies, and IGNDEM downloads the Spanish one directly from the IGN service. On each portal, check the coordinate system (the zone of the sheet) and the licence before using the data.

Area	Agency and product	Address
Spain	CNIG download centre: MDT05, MDT25 and MDT200 (ETRS89 / UTM in the zone of the sheet; Canary Islands, REGCAN95)	`https://centrodedescargas.cnig.es`
Spain	IGN: elevation WCS service (the one IGNDEM uses)	`https://servicios.idee.es/wcs-inspire/mdt`
Portugal	DGT: data centre (LiDAR DTM) and SNIG	`https://cdd.dgterritorio.gov.pt` · `https://snig.dgterritorio.gov.pt`
France	IGN: RGE ALTI and LiDAR HD	`https://geoservices.ign.fr/rgealti` · `https://geoservices.ign.fr/lidarhd`
Italy	INGV: TINITALY; Geoportale Nazionale (MASE)	`https://tinitaly.pi.ingv.it` · `https://gn.mase.gov.it`
Mexico	INEGI: Continuo de Elevaciones Mexicano (CEM)	`https://www.inegi.org.mx/app/geo2/elevacionesmex/`
Colombia	IGAC: Colombia en Mapas	`https://www.colombiaenmapas.gov.co`
Argentina	IGN Argentina: MDE-Ar	`https://www.ign.gob.ar/NuestrasActividades/Geodesia/Modelo`
Brazil	INPE: Topodata	`http://www.dsr.inpe.br/topodata/`
Chile and Peru	National spatial data infrastructures	`https://www.ide.cl` · `https://www.idep.gob.pe`
Morocco	ANCFCC (official mapping); for a freely downloadable DEM, the Copernicus DEM	`https://www.ancfcc.gov.ma`
Worldwide	Copernicus DEM GLO-30 (ESA) and SRTM (USGS EarthExplorer)	`https://dataspace.copernicus.eu` · `https://earthexplorer.usgs.gov`

Verification and validity

Checked on 2026-11-02: GetCapabilities of every WMS, WMTS and WFS; for WMS, a GetMap image in EPSG:3857 (or 25831 in Andorra); for WFS, a GetFeature query in GeoJSON with CRS84 coordinates; for XYZ, one tile. Agencies change their URLs, layers and terms without notice: if a service stops answering, look for it on the agency's geoportal or the INSPIRE geoportal, and always check the current licence.

See also

MAPSERVICE · WMS · WFS · WMTS · GEOGRAPHICLOCATION · Georeferencing

License and maintenance

The LinceCAD license is **perpetual**: it does not expire and keeps working without paying anything else. It includes **one year of maintenance**, that is, the right to the new versions released during that year.

What it includes

- During the maintenance year, CHECKFORUPDATES offers every new version.
- When it ends, LinceCAD **keeps working** in the version you have and in any other version **released up to the end of maintenance**. Versions released later are no longer offered: the update dialog says which one requires renewing.
- **Conformity or security fixes** (labelled «fix») are also offered with expired maintenance, during the **2-year warranty from purchase**, and can be installed without LinceCAD switching to read-only.
- 30-day **trials** get every update while they last.
- Licenses bought before maintenance existed get one year from their issue date.

Where to see it

In LICENSE, the **Updates until** row and the **Maintenance** section show **Updates included until: …**or, if it has passed, **Maintenance expired on …**

Renewing

1. In LICENSE, click **Renew maintenance…**(it also appears in the update dialog and on the start screen when needed).
2. The secure Stripe payment page opens, as for the purchase. It costs €50 a year plus your country's taxes.
3. Once the payment is confirmed, maintenance is extended **one year from its end date** or, if it had already expired, **from today**. LinceCAD picks up the new date within seconds and you receive a confirmation email.

If you manually install a later version

If you install a version **released after** the end of your maintenance, LinceCAD detects it at startup, also offline (it compares the release date built into the program with the one in the signed license):

- It switches to **read-only** and grants the **courtesy save**: no work is lost.
- The start screen and LICENSE explain it: «This version (released on F) is later than your maintenance (until D). Renew maintenance or reinstall version X».
- CHECKFORUPDATES offers to **install the latest entitled version**. Once you renew, the full license comes back without reinstalling anything.

See also

LICENSE · CHECKFORUPDATES · Options

Command Reference

Draw

3DPOLY —3D polyline

Draws a polyline with 3D points.

- **Command:** 3DPOLY · **Global name:** _3DPOLY · **Alias:** 3P

Draws a polyline whose vertices have 3D coordinates. It is a different object from the lightweight polyline of PLINE.

Usage

1. Enter 3DPOLY (or 3P).
2. Specify the start point and the next ones, as `x,y,z` (a missing Z is the current elevation).

```
Specify start point of polyline:
Specify endpoint of line or [Close/Undo]:
```

3. Press Enter to finish.

Options

- **Close** (C): joins the last point to the first (three are needed).
- **Undo** (U): removes the last point.

Notes

- It has no arcs or width. To draw on the plane use PLINE.

See also

PLINE · LINE · PEDIT

ARC —Arc

Creates an arc through three points.

- **Command:** ARC · **Global name:** _ARC · **Alias:** A
- Ribbon: Home › Draw
- Menu: Draw
- Toolbar: Draw

Draws a circular arc. By default it passes through three points: start, intermediate and end. With its options it can also be drawn by its center, or by its start, its end and the center or the radius.

Usage

1. Enter ARC (or A) or click **Arc**.
2. Specify the start point, or enter Center to begin with the center.

```
Specify start point of arc or [Center]:
```

3. Specify a second point the arc passes through, or enter Center or End.

```
Specify second point of arc or [Center/End]:
```

4. Specify the end point.

```
Specify end point of arc:
```

Options

- **Center** (C) at the first prompt: center, start point and end point.

```
Specify center point of arc:
Specify start point of arc:
```

- **Center** (C) at the second prompt: after the start point, specify the center and the end point.
- **End** (E): after the start point, specify the end and then the center or, with the Radius option (R), the radius. A negative radius draws the major arc.

```
Specify center point of arc or [Angle/Direction/Radius]:
Specify radius of arc:
```

- **Angle** (A): in start-center and in start-end, the included angle (positive: counterclockwise). **chord Length** (L): after the center, the length of the chord. **Direction** (D): in start-end, the tangent direction at the start.

```
Specify end point of arc or [Angle/chord Length]:
Specify tangent direction for the start point of arc:
```

- **Enter** at the first prompt: continues the last object (line, arc or open polyline) with a tangent arc; only the end point is asked.

Notes

- With a center, arcs are drawn counterclockwise from the start point; the end point only gives the direction in which the arc ends and does not have to lie on it.
- If the three points are collinear there is no arc and the program says so.
- The AutoCAD methods are available except Ctrl to switch direction.

See also

CIRCLE · PLINE · ELLIPSE · Object snaps and precision

BOUNDARY —Boundary

Creates a polyline or a region from the boundary around a point.

- **Command:** `BOUNDARY` · **Global name:** `_BOUNDARY` · **Alias:** `BO`, `BPOLY`
- Ribbon: Home › Draw
- Menu: Draw

Creates a closed polyline (or a region) from the boundary around an internal point, using the objects that enclose it. The original objects are not modified.

Usage

1. Enter BOUNDARY (or BO) or click **Boundary**.
2. Click inside each closed area. For each one, its outer boundary and the boundary of each island it contains are created.

```
Specify internal point or [Advanced options]:
```

3. Press Enter to finish; the program reports how many objects it has created.

Options

- **Advanced options** (A): like AutoCAD -BOUNDARY; Enter goes back to asking for points.

```
Enter an option [Boundary set/Island detection/Object type]:
```

- **Boundary set** (B): **New** selects the objects that count when looking for the boundary (the rest is ignored); **Everything** counts everything visible again.

```
Specify candidate set for boundary [New/Everything] <Everything>:
```

- **Island detection** (I): with No, only the outer boundary is created, without the islands.
- **Object type** (O): chooses what is created, polylines or regions (also directly with O when asked for the point).

```
Enter type of boundary object [Region/Polyline] <Polyline>:
```

Notes

- The boundary is found in the same way as in HATCH with an internal point: lines, arcs, circles, polylines, ellipses, splines and blocks count; text and dimensions do not.
- If there is no closed boundary around the point, the program says so and you can try another point.
- The resulting polylines are useful, for example, to measure the area of a room with AREA.
- No dialog box in this version: it is used from the command line, with the options of the AutoCAD dialog box.

See also

REGION · HATCH · AREA · PLINE

CIRCLE —Circle

Creates a circle by center and radius.

- **Command:** `CIRCLE` · **Global name:** `_CIRCLE` · **Alias:** `C`
- Ribbon: Home › Draw
- Menu: Draw
- Toolbar: Draw

Draws a circle. By default you specify the center and the radius; with its options, also by three points, by the endpoints of a diameter or tangent to two objects with a given radius.

Usage

1. Enter CIRCLE (or C) or click **Circle**.
2. Specify the center, or enter an option.

```
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
```

3. Specify the radius with a point on the circle or by typing its value.

```
Specify radius of circle or [Diameter]:
```

Options

- **3P**: the circle passes through three points.

```
Specify first point on circle:
Specify second point on circle:
```

```
Specify third point on circle:
```

- **2P**: the two points are the endpoints of a diameter.

```
Specify first end point of circle's diameter:
Specify second end point of circle's diameter:
```

- **Ttr** (T): tangent to two objects (lines, circles, arcs or polyline segments) with the radius you enter. Select each object near where the circle should touch it and enter the radius (or give a point: the radius is its distance to the second point selected).

```
Specify point on object for first tangent of circle:
Specify point on object for second tangent of circle:
Specify radius of circle or [Diameter]:
```

Notes

- Type 3P and 2P in full.
- With Ttr there are usually several solutions: the one nearest the points where the objects were selected is used. If there is none with that radius, the program says so.
- With the **Diameter** option (D), after the center, give the diameter by typing it or with a point (its distance from the center).
- Tangent to three objects (Tan, Tan, Tan), as in AutoCAD: in 3P, type TAN before each point and select the object (line, circle, arc or polyline segment) near where it should touch it; points and TAN can be mixed.

See also

ARC · ELLIPSE · DONUT · POLYGON · Object snaps and precision

DIVIDE —Divide

Places points or blocks that divide an object into equal parts.

- **Command:** `DIVIDE` · **Global name:** `_DIVIDE` · **Alias:** `DIV`
- Ribbon: Home › Draw
- Menu: Draw

Divides an object into a number of parts of equal length and places points (or a block) at the divisions. The object is not cut.

Usage

1. Enter DIVIDE (or DIV) or click **Divide**.
2. Select a line, arc, circle, polyline, ellipse or spline.

```
Select object to divide:
```

3. Enter the number of parts, from 2 to 32767.

```
Enter the number of segments or [Block]:
```

Options

- **Block** (B): instead of points, inserts a block of the drawing at each division, rotated to follow the direction of the object at that point or not rotated.

```
Enter name of block to insert:
Align block with object? [Yes/No] <Y>:
Enter the number of segments:
```

Notes

- On an open object, n parts give n ? 1 points. On a closed one you get n points, the first one at the start (on a circle, at 0°).
- The points are shown with the style set by PTYPE; with the default style they can hardly be seen. They can be used with the Node object snap.
- On ellipses and splines the length is measured on a very fine approximation of the curve.

See also

MEASURE · POINT · PTYPE · BLOCK

DONUT —Donut

Draws filled rings or filled circles.

- **Command:** `DONUT` · **Global name:** `_DONUT` · **Alias:** `DO`, `DOUGHNUT`
- Ribbon: Home › Draw
- Menu: Draw

Draws donuts (filled rings) or filled circles. You give the two diameters and then as many centers as you want.

Usage

1. Enter DONUT (or DO) or click **Donut**.
2. Enter the inside diameter; Enter accepts the one shown.

```
Specify inside diameter of donut <…>:
```

3. Enter the outside diameter, greater than the inside one.

```
Specify outside diameter of donut <…>:
```

4. Specify the center of each donut. Press Enter or Esc to finish.

```
Specify center of donut or <exit>:
```

Notes

- With an inside diameter of 0 you get a filled circle.
- The diameters (0.5 and 1 at the start) are remembered for the next donut during the session.
- Each donut is a closed polyline of two arcs with a width, as in AutoCAD; it can be changed with PEDIT.

See also

CIRCLE · SOLID · PLINE · PEDIT

ELLIPSE —Ellipse

Creates an ellipse or an elliptical arc.

- **Command:** `ELLIPSE` · **Global name:** `_ELLIPSE` · **Alias:** `EL`
- Ribbon: Home › Draw
- Menu: Draw
- Toolbar: Draw

Draws an ellipse or an elliptical arc. The ellipse is defined by one axis (its two endpoints, or the center and one endpoint) and by the distance to the other axis or by a rotation.

Usage

1. Enter ELLIPSE (or EL) or click **Ellipse**.
2. Specify one endpoint of the first axis, or enter an option.

```
Specify axis endpoint of ellipse or [Arc/Center]:
```

3. Specify the other endpoint of that axis.

```
Specify other endpoint of axis:
```

4. Specify the distance from the center to the other axis, with a point or a number.

```
Specify distance to other axis or [Rotation]:
```

Options

- **Center** (C): starts with the center and then one endpoint of the first axis.

```
Specify center of ellipse:
Specify endpoint of axis:
```

- **Rotation** (R): instead of the distance, the rotation of a circle about the first axis, from 0 to 89.4 degrees (0 gives a circle).

```
Specify rotation around major axis:
```

- **Arc** (A): draws an elliptical arc. After the ellipse is defined, it asks where it starts and where it ends.

```
Specify start angle or [Parameter]:
Specify end angle or [Parameter/Included angle]:
```

- **Parameter** (P) and **Angle** (A): switch between giving angles (the direction from the center) and parameters (the angle in the parametric equation of the ellipse).
- **Included angle** (I): the end as the angle swept from the start.
- **Isocircle** (I): only with isometric drafting on (ISODRAFT); the first prompt offers it. It draws the ellipse that represents a circle in the current isometric plane: specify the center and the radius, or choose **Diameter** (D) and give the diameter.

```
Specify axis endpoint of ellipse or [Arc/Center/Isocircle]:
Specify radius of isocircle or [Diameter]:
```

Notes

- The first axis may turn out to be the major or the minor one: if the distance to the other axis is greater than half the first one, the other becomes the major axis.
- The angles and parameters of the arc are measured from the first axis specified and counterclockwise, as in AutoCAD.
- A true ellipse (ELLIPSE) is created, not a polyline that approximates it.

See also

CIRCLE · ARC · SPLINE · Object snaps and precision

EXTRUDE —Extrude

Creates a solid by extruding a closed profile a given height.

- **Command:** `EXTRUDE` · **Global name:** `_EXTRUDE` · **Alias:** `EXT`
- Ribbon: Home › Draw

Creates a **3D solid** by extruding one or more closed profiles a given height, like AutoCAD's EXTRUDE. Modeling uses the `truck` kernel (pure Rust) and the result is stored as an ACIS 3DSOLID, just like AutoCAD.

Usage

1. Type EXTRUDE or click it in the ribbon.
2. Select one or more closed profiles (polylines, circles, ellipses…) and press Enter.

```
Select the closed profiles and press Enter:
```

3. Enter the height (positive towards +Z, negative towards ?Z).

```
Extrusion height:
```

Exact vs faceted

- **Polygonal** profiles (straight edges only): the solid is **exact**; AutoCAD audits it with no errors and MASSPROP reports the exact volume.
- Profiles with **arcs or curves**: the outline is approximated with segments and the solid comes out **faceted**.

Notes

- The extrusion starts at the profile's Z (the elevation of the polyline or circle: the ELEV one if it was drawn with it) and is vertical. Profiles on tilted planes, path and taper angle come in a later phase.
- See also REVOLVE, UNION, SUBTRACT and INTERSECT.

FLATSHOT — Flatshot

Projects the view's 3D solids to a 2D block with visible and hidden lines.

- **Command:** `FLATSHOT` · **Global name:** `_FLATSHOT` · **Alias:** `FSHOT`
- Ribbon: Home › Draw

Projects the **3D solids** of the current view to **2D geometry** (visible and hidden lines) and inserts it as a **block** in the current space, like AutoCAD's FLATSHOT. Hidden-line removal reuses the same z-buffer as the 3D plot.

Usage

1. Orbit to the 3D view you want to flatten (for example an isometric).
2. Type FLATSHOT (or APLANADA) or click it in the ribbon.
3. Specify the insertion point of the flattened drawing.

```
Specify the insertion point of the flattened drawing:
```

Result

- **Visible lines** go on the current layer.
- **Hidden lines** go on the **OCULTAS** layer (gray), created if it does not exist.
- Everything is grouped into a block and inserted at the given point.

Notes

- The projection is orthographic, on the view plane.
- See also PRESSPULL and SLICE.

Notes

- No dialog box in this version: it is used from the command line.

GRADIENT —Gradient

Fills an enclosed area with a gradient fill.

- **Command:** `GRADIENT` · **Global name:** `_GRADIENT` · **Alias:** `GD`
- Ribbon: Home › Draw
- Menu: Draw

Fills a closed area with a color gradient: from one color to another, or from a color towards black or white. It is HATCH with its dialog open on the **Gradient** tab: the boundary is given in the same way.

Usage

1. Enter GRADIENT (or GD) or click the button.
2. In the **Hatch and Gradient** dialog, choose **One color** or **Two color** and the colors (from the AutoCAD palette or a true color).
3. With one color, move the slider between **Shade** (towards black) and **Tint** (towards white).
4. Choose one of the nine types by clicking its sample, and set **Centered** and the **Angle**.
5. Click inside each area you want to fill (or select its boundaries, as in HATCH).

```
Pick internal point or [Select objects/Pattern/sCale/Angle/Gradient/Origin/inHerit/adVanced/
dRaw order]:
```

6. Press Enter or **OK**.

Types

LINEAR, CYLINDER, INVCYLINDER, SPHERICAL, INVSPHERICAL, HEMISPHERICAL, INVHEMISPHERICAL, CURVED and INVCURVED: the same as in AutoCAD. When not centered, the gradient is shifted towards one side, as in AutoCAD.

On the command line

The **Gradient** (G) option of HATCH and HATCHEDIT asks for the type (? lists them), the angle and whether it is centered; it uses the colors of the last gradient.

```
Enter gradient name or [?] <…>:
Specify gradient angle <…>:
Centered gradient? [Yes/No] <Yes>:
```

Notes

- The gradient is saved as in AutoCAD (a solid hatch with the gradient data) and AutoCAD draws it the same way.
- When plotting to PDF, PNG or SVG it comes out in thin bands of flat color, which cannot be told apart by eye.
- The settings are remembered for the next gradient. They are changed later with HATCHEDIT.

See also

HATCH · HATCHEDIT · HATCHGENERATEBOUNDARY · Hatches

HATCH —Hatch

Fills a closed boundary with a pattern.

- **Command:** `HATCH` · **Global name:** `_HATCH` · **Alias:** `H`, `BH`

- Ribbon: Home › Draw
- Menu: Draw
- Toolbar: Draw

Fills a closed area with a pattern of lines, a solid fill or a gradient. While the command runs, the **Hatch and Gradient** dialog box is shown, with its **Hatch** tab (the pattern swatches, the scale and the angle) and its **Gradient** tab; the same can be done on the command line.

Usage

1. Enter HATCH (or H) or click **Hatch**.
2. In the dialog box, choose the pattern by clicking its swatch and set the scale and the angle (or switch to the Gradient tab: see GRADIENT).
3. Click inside each area you want to hatch (or, with the Select objects option, select its boundaries). The dialog box shows how many boundaries have been found.

```
Pick internal point or [Select objects/Pattern/sCale/Angle/Gradient/Origin/inHerit/adVanced/
dRaw order]:
```

4. Press Enter or **OK** to create the hatch. **Cancel** exits without creating it.

Options

- **Select objects** (S): select closed polylines, circles, ellipses or splines as boundaries.

```
Select objects or [Internal point/Pattern/sCale/Angle/Gradient/Origin/inHerit/adVanced/dRaw
order]:
```

- **Internal point** (I): goes back to picking internal points.
- **Pattern** (P): enter the pattern name; ? lists them.

```
Enter a pattern name or [?] <…>:
```

- **sCale** (C) and **Angle** (A): pattern scale (greater than 0) and its rotation in degrees.
- **Gradient** (G): a gradient instead of the pattern: type, angle and whether it is centered.

```
Enter gradient name or [?] <…>:
```

- **Origin** (O): the point the pattern passes through (for example, the corner of a row of bricks).

```
Specify new hatch origin:
```

- **inHerit** (H): select a hatch and the new one takes its pattern, scale, angle or gradient (not its origin). In the dialog box, **Inherit properties** button.

```
Select hatch object:
```

- **adVanced** (V): like the advanced options of AutoCAD -HATCH; Enter goes back to the main prompt.

```
Enter an option [Island detection/Style/Gap tolerance/separate Hatches]:
```

- **Island detection** (I): with No, islands are not taken into account.
- **Style** (S): which islands are filled: **Normal** (nested ones alternate), **Outer** (only the area between the boundary and the outermost islands; AutoCAD's default) or **Ignore** (the whole boundary). All islands and the style are stored, as in AutoCAD.

```
Enter hatching style [Ignore/Outer/Normal] <…>:
```

- **Gap tolerance** (G): the gap tolerance (HPGAPTOL).
- **separate Hatches** (H): with Yes, each area (each outer boundary with its islands) is a separate hatch.
- **dRaw order** (R): draw order of the new hatch; by default, behind its boundary, as in AutoCAD.

```
Enter draw order [do Not assign/send to Back/bring to Front/send beHind boundary/bring in fr
```

```
ont of bounDary] <…>:
```

Patterns

SOLID (solid fill), ANSI31, ANSI32, ANSI37, LINE, NET, DOTS, CROSS, BRICK, HONEY and HORMIGON (concrete). The definitions are LinceCAD's own; the names match AutoCAD's when the drawing is the same. HORMIGON is saved as a custom pattern.

Notes

- With an internal point, the program looks for the nearest boundary around it, with its islands (Outer style, unless another one is chosen). Lines, arcs, circles, polylines, ellipses, splines and blocks count; text and dimensions do not. If the boundary has small gaps, HPGAPTOL closes them.
- The hatch is **associative**: if an object of its boundary is edited (with grips, MOVE, STRETCH...), the hatch is rebuilt to follow it. It stops being associative if the hatch is moved on its own or a boundary object is erased.
- Besides the library, patterns from AutoCAD `.pat` files are read: `acadiso.pat` and `acad.pat` (with several patterns) or `NAME.pat` (a single one) in the current folder, in those of the `LINCE_PATRONES` environment variable or in the `patrones` folder of the user configuration. Their measures are used as they are, with the hatch scale and angle, and the result is a custom pattern.
- TRIM trims hatches: it removes the selected piece between the cutting edges.

See also

HATCHEDIT · GRADIENT · HATCHGENERATEBOUNDARY · BOUNDARY · Hatches

HATCHEDIT —Edit Hatch

Modifies the pattern, scale, angle, gradient or boundary of a hatch.

- **Command:** `HATCHEDIT` · **Global name:** `_HATCHEDIT` · **Alias:** `HE`
- Ribbon: Home › Modify

Changes the pattern, scale, angle, gradient, origin or boundary of an existing hatch. It also opens when you double-click the hatch.

Usage

1. Enter HATCHEDIT (or HE), or double-click a hatch. If a hatch was already selected, that one is edited.
2. Select the hatch.

```
Select hatch object:
```

3. Change the values in the **Hatch and Gradient** dialog box (Hatch or Gradient tab) or with the command-line options.

```
Enter an option [Pattern/sCale/Angle/Gradient/Origin/inHerit/Disassociate/ASsociate/Separate
/Boundary recreate] <exit>:
```

4. Press Enter or **OK** to apply the changes.

Options

- **Pattern** (P): name of the new pattern (? lists them).
- **sCale** (C): new scale, greater than 0.
- **Angle** (A): new angle in degrees.
- **Gradient** (G): switches to a gradient, or changes its type, angle and whether it is centered.
- **Origin** (O): new origin point of the pattern; it is applied at once.

```
Specify new hatch origin:
```

- **Disassociate** (D): the hatch stops following its boundary.
- **ASsociate** (AS): select closed objects; they become the boundary of the hatch, which is associated with them.

```
Select boundary objects:
```

- **Separate** (S): a hatch with several separate areas is split into one hatch per area (each with its islands).
- **Boundary recreate** (B): creates a polyline for each boundary of the hatch and associates the hatch with them.

Notes

- A hatch read from a drawing with a pattern that is not in the LinceCAD library keeps its definition: its scale and angle can be changed.
- When the scale is changed, the pattern goes back to its definition origin (the origin chosen with Origin is lost).
- To create the boundary polylines without associating them, use HATCHGENERATEBOUNDARY.

See also

HATCH · GRADIENT · HATCHGENERATEBOUNDARY · PROPERTIES · Hatches

HATCHGENERATEBOUNDARY —Generate Boundary

Creates non-associated polylines around selected hatches.

- **Command:** `HATCHGENERATEBOUNDARY` · **Global name:** `_HATCHGENERATEBOUNDARY`
- Ribbon: Home › Draw
- Menu: Draw

Creates closed polylines around the selected hatches: one for each boundary, the outer one and that of each island. It is used to recover the boundary of a hatch that has lost it (for example, one that comes from another program or has been disassociated).

Usage

1. Enter HATCHGENERATEBOUNDARY or click the button.
2. Select the hatches and press Enter. If hatches were already selected, those are used.

```
Select hatch objects:
```

3. The program reports how many boundaries it has created.

```
Boundaries created: ….
```

Notes

- The polylines follow the straight segments and arcs of the boundary; ellipses and splines in the boundary become short straight segments.
- They are created on the current layer. The hatch is not associated with them; for that, use the **Boundary recreate** option of HATCHEDIT.
- Other objects can be included in the selection: only hatches count.

See also

HATCHEDIT · HATCH · BOUNDARY · Hatches

HATCHSETBOUNDARY —Set Hatch Boundary

Replaces the boundary of a hatch with other closed objects.

- **Command:** `HATCHSETBOUNDARY` · **Global name:** `_HATCHSETBOUNDARY`
- Ribbon: Home › Draw

Replaces the boundary of a hatch or fill with other closed objects, as in AutoCAD 2027. The pattern, scale, angle and the other properties of the hatch do not change.

Usage

1. Enter HATCHSETBOUNDARY or click the button.
2. Select the hatch.

```
Select hatch object:
```

3. Select the closed objects of the new boundary and press Enter. Only closed polylines, circles, ellipses and closed splines are valid; other objects are not taken.

```
Select objects to use for the new boundary:
```

4. Say whether the selected objects are deleted (Enter: No).

```
Delete selected lines? [Yes/No] <N>:
```

Notes

- With **No**, the hatch stays associative to the new objects: when they are edited, the hatch follows them.
- With **Yes**, the objects are deleted and the hatch is no longer associative.

```
Hatch boundary associativity removed.
```

- Each closed object is a boundary; one inside another is an island.

See also

HATCH · HATCHEDIT · HATCHGENERATEBOUNDARY · Hatches

HATCHSETORIGIN —Set Hatch Origin

Changes the pattern origin of the selected hatches.

- **Command:** `HATCHSETORIGIN` · **Global name:** `_HATCHSETORIGIN`
- Ribbon: Home › Draw

Changes the pattern origin of one or more hatches, as in AutoCAD 2027: the pattern shifts to pass through the given point; the boundary does not change.

Usage

1. Enter HATCHSETORIGIN or click the button.
2. Select the hatches and press Enter.

```
Select hatch objects:
```

3. Specify the new origin.

```
Select new hatch origin:
```

Notes

- It is the same as the Origin option of HATCHEDIT, for several hatches at once.

See also

HATCH · HATCHEDIT · Hatches

HATCHTOBACK —Hatches to Back

Puts all hatches behind the other objects.

- **Command:** `HATCHTOBACK` · **Global name:** `_HATCHTOBACK` · **Alias:** `HB`
- Ribbon: Home › Draw

Puts all hatches and gradients in model space behind the other objects, as in AutoCAD 2027. It asks nothing.

Usage

1. Enter HATCHTOBACK (alias HB) or click the button.

```
… hatches sent to back.
```

Notes

- It changes the draw order, like DRAWORDER › Back with all hatches selected.

See also

DRAWORDER · HATCH · Hatches

HELIX —Helix

Draws a helix or a flat spiral.

- **Command:** `HELIX` · **Global name:** `_HELIX`
- Ribbon: Home › Draw

Draws a helix, as in AutoCAD 2027. With height 0 it is a flat spiral; with a height, its projection is seen in plan (the spiral from the base to the top).

Usage

1. Enter HELIX or click the button. The program remembers the number of turns and the twist.

```
Number of turns = …    Twist=…
```

2. Specify the center point of the base.

```
Specify center point of base:
```

3. Specify the base radius (or **Diameter**). If it is given with a point, the helix starts there.

```
Specify base radius or [Diameter] <…>:
```

4. Specify the top radius (Enter: the base one).

```
Specify top radius or [Diameter] <…>:
```

5. Specify the height (0 for a flat spiral) or an option.

```
Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <…>:
```

Options

- **Axis endpoint**: the height and the direction of the axis come from a point (it may have Z).
- **Turns**: number of turns (up to 500).

```
Enter number of turns <…>:
```

- **turn Height**: distance between turns; it ends the helix with height = turns × distance.

```
Specify distance between turns <…>:
```

- **tWist**: twist direction, clockwise or counterclockwise.

```
Enter twist direction of helix [CW/CCW] <…>:
```

Notes

- It is saved as the AutoCAD HELIX entity (with its spline), which AutoCAD opens as a helix.

See also

SPLINE · PLINE

LINE — Line

Creates straight line segments.

- **Command:** `LINE` · **Global name:** `_LINE` · **Alias:** `L`
- Ribbon: Home › Draw
- Menu: Draw
- Toolbar: Draw

Draws straight line segments. Each segment is a separate object that can be moved, erased or put on another layer on its own. For an outline that is a single object, use PLINE.

Usage

1. Enter LINE (or L) or click **Line** on the ribbon.
2. Specify the first point with a click, an object snap or by typing its coordinates.

```
Specify first point:
```

3. Specify the next point. A segment is created and the prompt repeats from the last point.

```
Specify next point or [Undo]:
```

4. From the third point on you can close the figure.

```
Specify next point or [Close/Undo]:
```

5. Press Enter, Space or Esc to finish.

Options

- **Close** (C): joins the last point to the first one and ends the command.
- **Undo** (U): removes the last segment without leaving the command; it can be repeated.
- Enter at the first point: continues from the end of the last line or arc drawn. After an arc, the line is tangent and its length is asked.

```
Length of line:
```

Notes

- Coordinates can be typed as absolute (`100,50`), relative (`@20,0`) or polar (`@50<30`). See Coordinates and input .
- With Ortho (F8) or polar tracking (F10) the segments snap to the horizontal and vertical directions or to the polar angles.
- Lines are created on the current layer, with the current color, linetype and lineweight.

See also

PLINE · RECTANG · PEDIT · Object snaps and precision

MEASURE —Measure

Places points or blocks at measured intervals along an object.

- **Command:** `MEASURE` · **Global name:** `_MEASURE` · **Alias:** `ME`
- Ribbon: Home › Draw
- Menu: Draw

Places points (or a block) along an object at intervals of a given length, measured from one of its ends. The object is not cut.

Usage

1. Enter MEASURE (or ME) or click **Measure**.
2. Select the object near the end you want to measure from.

```
Select object to measure:
```

3. Enter the length of each segment.

```
Specify length of segment or [Block]:
```

Options

- **Block** (B): instead of points, inserts a block of the drawing at each mark, rotated to follow the direction of the object at that point or not rotated.

```
Enter name of block to insert:
Align block with object? [Yes/No] <Y>:
Specify length of segment:
```

Notes

- Lines, arcs, circles, polylines, ellipses and splines can be used. On an open object, measuring starts at the end nearest the point where it was selected; the last segment may be shorter. If the last mark falls exactly on the far end, it is placed too.
- If the length is so small that more than 100,000 objects would be created, the program says so and creates none.
- The points are shown with the style set by PTYPE.

See also

DIVIDE · POINT · PTYPE · BLOCK

MLINE —Multiline

Draws multilines (parallel lines following the multiline style).

- **Command:** `MLINE` · **Global name:** `_MLINE` · **Alias:** `ML`
- Ribbon: Home › Draw
- Menu: Draw

Draws multilines: several parallel lines that run together, according to a multiline style. They are used, for example, for walls or roads.

Usage

1. Enter MLINE (or ML) or click **Multiline**. The prompt shows the current justification, scale and style.

```
Current settings: Justification = …, Scale = …, Style = …. Specify start point or [Justification/Scale/STyle]:
```

2. Specify the start point and the following ones.

```
Specify next point:
Specify next point or [Close/Undo]:
```

3. Press Enter to finish.

Options

- **Justification** (J): which line passes through the points you give.

```
Enter justification type [Top/Zero/Bottom] <…>:
```

- **Top** (T), **Zero** (Z) and **Bottom** (B): the top line, the axis of the style or the bottom line.
- **Scale** (S): multiplies the spacing of the style. A negative scale reverses the order of the lines.

```
Enter mline scale <…>:
```

- **STyle** (ST): multiline style; `?` lists those in the drawing.

```
Enter mline style name or [?]:
```

- **Close** (C): with three points or more, joins back to the first one and ends the command.
- **Undo** (U): removes the last point.

Notes

- Justification, scale and style are changed before the first point and saved in the drawing for the next time.
- As in AutoCAD 2027, Style only picks one of the styles the drawing already has (Standard, two lines at ±0.5, if there is no other); a name that does not exist is rejected and asked again. Styles are created and loaded from .mln files with MLSTYLE.
- For crossings, tees, cuts and welds of multilines, use MLEDIT.

See also

MLEDIT · PLINE · LINE · EXPLODE

PLINE — Polyline

Creates a polyline with several segments.

- **Command:** `PLINE` · **Global name:** `_PLINE` · **Alias:** `PL`
- Ribbon: Home › Draw
- Menu: Draw
- Toolbar: Draw

Draws a polyline: a sequence of straight and arc segments that form a single object. Each segment can have a width, constant or tapering from its start to its end.

Usage

1. Enter PLINE (or PL) or click **Polyline**.
2. Specify the first point.

```
Specify first point:
```

3. Specify the next points. Each point adds a straight segment.

```
Specify next point or [Arc/Halfwidth/Length/Undo/Width]:
```

4. With the Arc option, the following segments are arcs tangent to the previous segment: specify the end point of each arc.

```
Specify endpoint of arc or [Angle/CEnter/Direction/Halfwidth/Line/Radius/Second pt/Undo/Widt
h]:
```

5. Press Enter, Space or Esc to finish.

Options

With straight segments:

- **Arc** (A): switches to arcs.
- **Halfwidth** (H) and **Width** (W): ask for the starting and ending value of the next segment (the half-width is half the width); Enter accepts the value shown and a click gives it as the distance from the last point. Later segments keep the ending value, which becomes the default width (PLINEWID).

```
Specify starting width <…>:
Specify ending width <…>:
```

- **Length** (L): adds a segment of that length in the direction of the previous one.
- **Close** (C): joins the last point to the first and finishes.
- **Undo** (U): removes the last point without leaving the command.

With arcs (the default, tangent to the previous segment):

- **Angle** (A): included angle and end point. **CEnter** (CE): center and end point (counterclockwise). **Direction** (D): tangent direction at the start and end point. **Radius** (R): radius and end point. **Second pt** (S): second point of the arc and end point. **Line** (L): back to straight segments.

```
Specify endpoint of arc or [Angle/CEnter/Direction/Halfwidth/Line/Radius/Second pt/Undo/Widt
h]:
```

Notes

- A lightweight polyline (LWPOLYLINE) is created, as in AutoCAD. As there, the last vertex of an open polyline repeats the bulge of the previous segment.
- Pending: the width is not taken from PLINEWID at the start, and the Angle/Length sub-options after CEnter.

- To change the vertices or the width or to join segments later, use PEDIT.

See also

LINE · RECTANG · POLYGON · PEDIT · EXPLODE · Object snaps and precision

POINT —Point

Creates point objects.

- **Command:** `POINT` · **Global name:** `_POINT` · **Alias:** `PO`
- Ribbon: Home › Draw
- Menu: Draw
- Toolbar: Draw

Places point objects. Each click creates a point; points are used as marks and as targets for the Node object snap.

Usage

1. Enter POINT (or PO) or click **Point**.
2. Specify a point with a click or by typing its coordinates. The prompt repeats so you can place more points.

```
Specify a point:
```

3. Press Enter, Space or Esc to finish.

Notes

- The symbol used to draw all points and its size are chosen with PTYPE (variables PDMODE and PDSIZE). With the default style a point is a single pixel, hard to see.
- To place points along an object, use DIVIDE or MEASURE.
- Points are created on the current layer, with the current color and linetype.

See also

PTYPE · DIVIDE · MEASURE · Object snaps and precision

POLYGON —Polygon

Creates a regular polygon.

- **Command:** `POLYGON` · **Global name:** `_POLYGON` · **Alias:** `POL`
- Ribbon: Home › Draw
- Menu: Draw
- Toolbar: Draw

Draws a regular polygon, with 3 to 1024 sides, as a closed polyline.

Usage

1. Enter POLYGON (or POL) or click **Polygon**.
2. Enter the number of sides; Enter accepts the one shown.

```
Enter number of sides <…>:
```

3. Specify the center.

```
Specify center of polygon or [Edge]:
```

4. Choose whether the polygon is inscribed in the circle or circumscribed about it.

```
Enter an option [Inscribed in circle/Circumscribed about circle] <I>:
```

5. Specify the radius with a point or by typing its value.

```
Specify radius of circle or [Diameter]:
```

Options

- **Inscribed** (I): the radius reaches the vertices; the point you give is a vertex.
- **Circumscribed** (C): the radius reaches the midpoints of the sides; the point you give is the midpoint of a side.

Notes

- If you type the radius, the bottom side is horizontal. With a point, the polygon rotates to pass through it.
- With the **Edge** option (E) you give the two endpoints of a side and the polygon is drawn to its left.

See also

RECTANG · CIRCLE · PLINE · PEDIT

PRESSPULL — Press/Pull

Pushes or pulls a closed region a distance to create a solid.

- **Command:** PRESSPULL · **Global name:** _PRESSPULL
- Ribbon: Home › Draw

Creates or modifies a **3D solid** by pushing or pulling a **closed region** a given distance, like AutoCAD's PRESSPULL. The region is extruded with the `truck` kernel (pure Rust) and stored as an ACIS 3DSOLID.

Usage

1. Type PRESSPULL (or PULSARTIRAR) or click it in the ribbon.
2. Select one or more closed areas (polylines, circles, ellipses…) and press Enter.

```
Select the closed areas and press Enter:
```

3. Enter the distance (positive towards +Z, negative towards ?Z).

```
Press and pull distance:
```

Exact vs faceted

- **Polygonal** regions (straight edges only): the solid is **exact**; AutoCAD audits it with no errors.
- Regions with **arcs or curves**: the outline is approximated with segments and the solid comes out **faceted**.

Notes

- Pushing a specific **face** of an existing solid will come with subentity picking; for now PRESSPULL acts on closed areas of the drawing.
- See also EXTRUDE, SLICE and UNION.

RAY — Ray

Creates lines that start at a point and continue to infinity.

- **Command:** `RAY` · **Global name:** `_RAY`
- Ribbon: Home › Draw
- Menu: Draw

Draws rays: half-lines that start at a point and go on endlessly in one direction. Like construction lines, they are construction guides, for when only one side matters.

Usage

1. Enter RAY or click the button.
2. Specify the start point.

```
Specify start point:
```

3. Specify a point the ray passes through. Each point creates another ray from the same start.

```
Specify through point:
```

4. Press Enter, Space or Esc to finish.

Notes

- RAY objects are created, as in AutoCAD. They do not count for zoom extents.
- With ortho (F8) or polar tracking (F10) it is easy to draw horizontal, vertical or fixed-angle rays.

See also

XLINE · LINE · Object snaps and precision

RECTANG — Rectangle

Creates a rectangle from two opposite corners.

- **Command:** `RECTANG` · **Global name:** `_RECTANG` · **Alias:** `REC`, `RECTANGLE`
- Ribbon: Home › Draw
- Menu: Draw
- Toolbar: Draw

Draws a rectangle with its sides parallel to the X and Y axes of the current UCS from two opposite corners. The rectangle is a closed polyline with four vertices.

Usage

1. Enter RECTANG (or REC) or click **Rectangle**.
2. Specify the first corner.

```
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
```

3. Specify the opposite corner, with a click or with relative coordinates: `@40,25` draws a 40 × 25 rectangle.

```
Specify other corner point or [Area/Dimensions/Rotation]:
```

Options

Before the first corner:

- **Chamfer**: two distances (the first counts on the side that arrives at each corner). **Fillet**: radius. One cancels the other.
- **Elevation**, **Thickness** (extrusion height) and **Width** (line width).

```
Specify first chamfer distance for rectangles <…>:
```

After the first corner:

- **Area**: the area and the length or width; the rectangle is created at once, up and to the right.
- **Dimensions**: length and width; the opposite corner gives the quadrant.
- **Rotation**: rotation of the rectangle (an angle, or a point giving the direction).

Notes

- The two corners must have different X and different Y (in the UCS). With a rotated UCS (see UCS), the rectangle is rotated with it.
- The Chamfer, Elevation, Fillet, Thickness and Width modes are remembered for the next rectangles (as in AutoCAD). With chamfer or fillet the rectangle has eight vertices; if they do not fit the sides, it is drawn plain.
- As it is a polyline, it is edited with PEDIT and exploded into four lines with EXPLODE.

See also

PLINE · POLYGON · PEDIT · Coordinates and input

REGION — Region

Converts closed curves into regions.

- **Command:** `REGION` · **Global name:** `_REGION` · **Alias:** `REG`
- Ribbon: Home › Draw
- Menu: Draw

Converts closed curves into regions: flat surfaces saved in the same way as in AutoCAD. Each closed curve selected becomes a region and the original curve is erased.

Usage

1. Enter REGION (or REG) or click **Region**. If objects were already selected, those are used.
2. Select closed polylines, circles, ellipses or closed splines.

```
Select objects:
```

3. Press Enter. The program reports how many loops it found and how many regions it created.

Notes

- Each region takes the layer, color and linetype of the curve it comes from.
- Only closed curves made of a single object can be used: separate lines or arcs that enclose an area are not joined into a region. For that, use BOUNDARY with the Region type or join the segments with PEDIT.
- Ellipses and splines are turned into a very fine polygon that approximates them.
- There are no operations between regions (union, subtraction, intersection).
- To get curves back, explode the region with EXPLODE.

See also

BOUNDARY · EXPLODE · AREA · Selecting objects

REVCLOUD — Revision Cloud

Draws a revision cloud: rectangular, polygonal, freehand or from an object.

- **Command:** `REVCLOUD` · **Global name:** `_REVCLOUD`
- Ribbon: Home › Draw
- Menu: Draw

Draws a revision cloud: a polyline made of arcs used to mark the changes on a drawing. The cloud can be rectangular, polygonal or freehand, or made from an existing object.

Usage

1. Enter REVCLOUD or click **Revision Cloud**.
2. Choose the type of cloud with its option, if it is not the current one, and specify the first point. The prompt shows the current type; Enter goes to the Object option.

```
Specify first corner point or [Arc length/Object/Rectangular/Polygonal/Freehand/Style/Modify
] <Object>:
```

3. Rectangular: specify the opposite corner and the cloud is created.

```
Specify opposite corner:
```

4. Polygonal and freehand: specify the next points and press Enter to finish.

```
Specify next point or [Undo]:
```

Options

- **Arc length** (A): minimum and maximum arc length. The maximum cannot exceed three times the minimum.

```
Specify minimum length of arc <…>:
Specify maximum length of arc <…>:
```

- **Object** (O): turns a polyline, circle, ellipse or spline into a cloud. The cloud replaces the object and takes its layer, color and linetype; the direction of the arcs can be reversed.

```
Reverse direction [Yes/No] <No>:
```

- **Rectangular** (R), **Polygonal** (P) and **Freehand** (F): type of cloud.
- **Style** (S): Normal or Calligraphy (arcs of varying width, as if drawn with a pen nib).
- **Modify** (M): changes a piece of a cloud. Select the cloud at the point where the new part starts (or specify it with First point), give the points of the new part until a segment crosses the cloud again and pick the side of the cloud to erase; the new part, with its arcs, takes its place in the same polyline.

```
Select polyline to modify:
Specify next point or [First point]:
Pick a side to erase:
Reverse direction [Yes/No] <No>:
Select arc style [Normal/Calligraphy] <Normal>:
```

- **Undo** (U): removes the last point of a polygonal or freehand cloud.

Notes

- A polygonal cloud closes itself when you press Enter. A freehand cloud closes when you come back near the first point; if you press Enter before, it stays open. Freehand clouds are also drawn with clicks, point by point.
- The arc lengths, the style and the type are remembered for the next cloud.
- The cloud is saved like AutoCAD's (a polyline with 110° arcs) and is edited with PEDIT.

See also

PLINE · RECTANG · PEDIT · WIPEOUT

REVOLVE —Revolve

Creates a solid by revolving a closed profile about an axis.

- **Command:** REVOLVE · **Global name:** _REVOLVE · **Alias:** REV
- Ribbon: Home › Draw

Creates a **3D solid** by revolving a closed profile about an axis, like AutoCAD's REVOLVE. Modeling uses the `truck` kernel (pure Rust).

Usage

1. Type REVOLVE or click it in the ribbon.
2. Select the closed profile and press Enter.

```
Select the closed profiles and press Enter:
```

3. Enter the axis: **X**, **Y** or two points "x1,y1,z1 x2,y2,z2".

```
Revolution axis [X/Y] or two points "x1,y1,z1 x2,y2,z2":
```

4. Enter the angle (360° by default).

```
Revolution angle <360>:
```

Notes

- The revolved solid comes out **faceted**: exact curved surfaces (cylinders, cones, tori as a B-spline surface) come in a later phase. AutoCAD audits it with no errors.
- The axis must lie in the profile's plane and must not cross it.
- See also EXTRUDE, UNION, SUBTRACT and INTERSECT.

SKETCH —Sketch

Draws freehand.

- **Command:** SKETCH · **Global name:** _SKETCH
- Ribbon: Home › Draw

Draws freehand, as in AutoCAD 2027.

Usage

1. Enter SKETCH or click the button. The program shows the current type, increment and tolerance.

```
Type = … Increment = … Tolerance = …
```

2. Click to lower the pen and move the mouse: it leaves a point each time it moves the increment away from the previous one. Another click raises the pen; you can make several strokes.

```
Specify sketch or [Type/Increment/toLerance]:
```

3. Press Enter to record the strokes.

```
… objects recorded.
```

Options

- **Type**: lines, polylines (one per stroke) or splines (one per stroke, fitted with the tolerance).

```
Enter sketch type [Line/Polyline/Spline] <…>:
```

- **Increment**: distance between the points of the stroke.

```
Specify sketch increment <…>:
```

- **toLerance**: how far the spline may deviate from the stroke.

```
Specify spline fit tolerance <…>:
```

Notes

- The options are the variables SKPOLY, SKETCHINC and SKTOLERANCE.
- In a script, each typed point lowers or raises the pen; the stroke joins the typed points.

See also

PLINE · SPLINE · LINE

SOLID —2D Solid

Draws filled triangles and quadrilaterals.

- **Command:** `SOLID` · **Global name:** `_SOLID` · **Alias:** `SO`
- Ribbon: Home › Draw
- Menu: Draw

Draws filled triangles and quadrilaterals (2D solids). They can be chained to form a strip.

Usage

1. Enter SOLID (or SO) or click **2D Solid**.
2. Specify the first and second points: they form one side.

```
Specify first point:
Specify second point:
```

3. Specify the third point.

```
Specify third point:
```

4. Specify the fourth point, or press Enter to create a triangle and finish.

```
Specify fourth point or <exit>:
```

5. After the fourth point, the third and fourth become the first side of the next solid: go on with more third and fourth points, or press Enter to finish.

Notes

- The order of the points is AutoCAD's: the first and second form one side and the third and fourth the opposite side, in the same direction. If you give them in circular order (1, 2, 3, 4 around the figure), you get a bow tie.
- To fill areas of any shape, use HATCH with the SOLID pattern.

See also

HATCH · DONUT · WIPEOUT

SPLINE —Spline

Creates a smooth curve through the specified points.

- **Command:** `SPLINE` · **Global name:** `_SPLINE` · **Alias:** `SPL`
- Ribbon: Home › Draw
- Menu: Draw
- Toolbar: Draw

Draws a smooth curve (spline). By default the curve passes through the points you specify (Fit method); with the CV method, the points are control vertices that pull the curve.

Usage

1. Enter SPLINE (or SPL) or click **Spline**.
2. Specify the first point, or first change the method, the knots or the degree.

```
Specify first point or [Method/Knots/Object]:
```

3. Specify the next points.

```
Enter next point or [start Tangency/toLerance]:
Enter next point or [end Tangency/toLerance/Undo/Close]:
```

4. Press Enter to finish.

Options

- **Method** (M): chooses the method.

```
Enter spline creation method [Fit/CV] <Fit>:
```

- **Knots** (K): in the Fit method, how the knots are spaced between the points: by the distance between them (Chord), by its square root (S) or evenly (Uniform).

```
Enter knot parameterization [Chord/Square root/Uniform] <Chord>:
```

- **Degree** (D): in the CV method, the degree of the curve, from 1 to 10 (3 by default).

```
Enter degree of spline <3>:
```

- **start Tangency** (T): after the first point, the direction in which the curve leaves.
- **end Tangency** (T): the direction in which it reaches the last point; ends the spline. Enter at a tangent prompt leaves it free.
- **Close** (C): with three points or more, closes the curve and ends the command. By CVs, with any degree, the closed spline is periodic (smooth at the joint), as in AutoCAD.
- **toLerance** (L): with a value greater than 0, the curve no longer passes through the points but within that distance of them, smoother; the points entered and the tolerance are stored.

```
Specify fit tolerance <…>:
```

- **Object** (O): select spline-fit polylines (AutoCAD PEDIT Spline) and press Enter: each one becomes the equivalent quadratic or cubic spline, with its frame as control vertices, and the polyline is erased.

```
Select spline-fit polyline:
```

- **Undo** (U): removes the last point.

Notes

- A fit spline stores its fit points and tangents as AutoCAD does, which recognizes and edits it in the same way.

- With toLerance, the curve is smoothed differently from AutoCAD; when it opens the DWG, AutoCAD recalculates it from the stored points and tolerance.
- To edit the curve later, use SPLINEDIT.

See also

SPLINEDIT · PLINE · ELLIPSE · REVCLOUD

SUPERHATCH —Super hatch

Fills boundaries by repeating a block or an image as the pattern, or with a wipeout.

- **Command:** SUPERHATCH · **Global name:** _SUPERHATCH
- Ribbon: Home › Draw

Fills closed boundaries by repeating a block or an image as the pattern, or covers them with a wipeout, like the Express Tools SUPERHATCH; in addition to the HATCH patterns.

Usage

1. Type SUPERHATCH or click **Super hatch**.
2. Select the closed boundaries and press Enter; those inside another one are islands and are not filled.

```
Select the closed boundaries to fill or [Internal point]:
```

3. Type the name of a block in the drawing (or choose an option).

```
Name of the block to fill with or [Image/Wipeout]:
```

Options

- **Internal point** (I): instead of selecting, click inside the areas, as in HATCH: their boundary and islands are taken. Enter goes on.

```
Specify internal point:
```

- **Image** (I): type the file of a raster image; it is tiled at the size IMAGEATTACH inserts it at scale 1.

```
Image file:
```

- **Wipeout** (W): a wipeout (WIPEOUT) for each outer boundary, which hides what is below.

Notes

- The boundary can be a closed polyline, a circle, an ellipse, a closed spline or a region.
- The pattern is tiled over the bounding box of the boundaries at scale 1 and the copies whose center falls in the filled area (outside the islands) are kept. Copies crossing the boundary are not clipped.
- An external reference works as a block (it is inserted the same way).
- The copies are plain INSERT objects (or images); a single UNDO removes them all.
- No dialog box: the options of the Express Tools dialog box are on the command line. The wipeout leaves no holes for the islands.

See also

HATCH · INSERT · BOUNDARY · WIPEOUT

WIPEOUT —Wipeout

Covers what lies under an outline with the background color.

- **Command:** `WIPEOUT` · **Global name:** `_WIPEOUT`
- Ribbon: Home › Draw
- Menu: Draw

Creates a wipeout: a polygonal area that covers everything drawn beneath it with the background color. It is used, for example, to clear the space for a text or a table on a drawing.

Usage

1. Enter WIPEOUT or click **Wipeout**.
2. Specify the first point of the boundary, or press Enter to convert a polyline.

```
Specify first point or [Frames/Polyline] <Polyline>:
```

3. Specify the next points.

```
Specify next point:
Specify next point or [Close/Undo]:
```

4. Press Enter (or enter C) to close the boundary and create the wipeout. At least three points are required.

Options

- **Polyline** (P): uses a closed polyline as the boundary; then asks whether to erase the polyline.

```
Select a closed polyline:
Erase polyline? [Yes/No] <No>:
```

- **Frames** (F): shows or hides the frame of all the wipeouts in the drawing (WIPEOUTFRAME).

```
Enter mode [ON/OFF/Display but not plot] <ON>:
```

- **Undo** (U): removes the last point.

Notes

- A wipeout covers what was drawn before it; to change what stays on top, use DRAWORDER.
- The arcs of the polyline are turned into straight segments, as AutoCAD does.
- The wipeout and its frame mode are saved in the drawing like AutoCAD's.

See also

DRAWORDER · REVCLOUD · SOLID · HATCH

XLINE —Construction Line

Creates lines of infinite length in both directions.

- **Command:** `XLINE` · **Global name:** `_XLINE` · **Alias:** `XL`
- Ribbon: Home › Draw
- Menu: Draw

Draws construction lines: infinite straight lines in both directions, useful as construction guides (axes, alignments, projections between views). They do not count for zoom extents.

Usage

1. Enter XLINE (or XL) or click the button.
2. Specify a point on the line, or choose an option.

```
Specify a point or [Hor/Ver/Ang/Bisect/Offset]:
```

3. Specify a point it passes through. Each point creates another construction line through the first one; Enter ends.

```
Specify through point:
```

Options

- **Hor** (H) and **Ver** (V): horizontal or vertical lines through each point you specify.
- **Ang** (A): lines at a fixed angle, in degrees, through each point.

```
Enter angle of xline (0) or [Reference]:
```

- **Reference** (R), inside Ang: the angle is measured from a line, a polyline (its straight segment nearest to the selected point), a ray or another construction line.

```
Select a line object:
Enter angle of xline <0>:
```

- **Bisect** (B): the bisector of an angle, given by its vertex and a point on each side. Each further end point creates another bisector with the same vertex and start.

```
Specify angle vertex point:
Specify angle start point:
Specify angle end point:
```

- **Offset** (O): parallels to a line, at a distance or through a point (Enter). After each one you can select another line.

```
Specify offset distance or [Through] <Through>:
Specify side to offset:
```

Notes

- XLINE objects are created, as in AutoCAD. They cannot be trimmed or extended yet.
- For a half-line starting at a point, use RAY.

See also

RAY · LINE · OFFSET · Object snaps and precision

Modify

-MLEDIT —Edit Multiline (command line)

Edits multilines from the command line.

- **Command:** `-MLEDIT` · **Global name:** `_-MLEDIT`

Command-line version of MLEDIT: edits crossings, tees, corners, vertices, cuts and welds of multilines by typing the code of the tool. It is the way to use it in scripts and in AutoLISP.

Usage

1. Enter -MLEDIT.
2. Type the code: CC, OC or MC (crosses), CT, OT or MT (tees), CJ (corner), AV or DV (add or delete vertex), CS or CA (cut one or all) or WA (weld all).

```
Enter mline editing option [CC/OC/MC/CT/OT/MT/CJ/AV/DV/CS/CA/WA]:
```

3. Select the multilines and the points as in MLEDIT. The prompt repeats; Enter finishes.

```
Select first mline or [Undo]:
```

Options

- **Undo** (U): undoes the last edit.

Notes

- In LinceCAD MLEDIT opens no dialog box either, so both commands do the same.

See also

MLEDIT · MLINE · SCRIPT

-OVERKILL —Delete Duplicate Objects (command line)

Deletes duplicates from the command line.

- **Command:** `-OVERKILL` · **Global name:** `_-OVERKILL`

Command-line version of OVERKILL: erases duplicates and combines overlapping lines and arcs with typed options, without dialog boxes. It is the way to use it in scripts and in AutoLISP.

Usage

1. Enter -OVERKILL.
2. Select the objects and press Enter (or start the command with them already selected).

```
Select objects:
```

3. Change the settings you want and press Enter to clean up.

```
Current settings: Tolerance=…, Ignore=…, Optimize polylines=…, Combine partial overlap=…, Co
mbine end-to-end=…. Enter an option to change [Done/Ignore/tOlerance/optimize Polylines/comb
ine parTial overlap/combine End-to-end] <done>:
```

Options

The same as OVERKILL: **tOlerance** (O), **Ignore** (I), **Polylines** (P), **parTial** (T), **End-to-end** (E) and **Done** (D).

Notes

- In LinceCAD OVERKILL has no dialog box either, so both commands do exactly the same.
- In a script, for example, `-OVERKILL` · `ALL` · Enter · `I` · `layer,color` · Enter cleans the whole drawing ignoring layer and color.

See also

OVERKILL · SCRIPT · AutoLISP

ALIGN —Align

Moves, rotates and scales objects to align them with others.

- **Command:** `ALIGN` · **Global name:** `_ALIGN` · **Alias:** `AL`
- Ribbon: Home › Modify
- Menu: Modify

Moves, rotates and, if you ask for it, scales the selected objects so that some source points match some destination points. Use it to fit a part or a block against another object without working out angles.

Usage

1. Enter ALIGN (or AL).
2. Select the objects and press Enter. If they were already selected before starting the command, they are used without asking.

```
Select objects:
```

3. Specify the first pair of points: one on the objects (source) and the place where it must go (destination).

```
Specify first source point:
Specify first destination point:
```

4. Specify the second pair, or press Enter to just move.

```
Specify second source point or <continue>:
Specify second destination point:
```

5. Specify the third pair, or press Enter and choose whether the objects are scaled so that the second source lands on the second destination (Enter: no).

```
Specify third source point or <continue>:
Scale objects based on alignment points? [Yes/No] <N>:
```

Notes

- With one pair of points, the objects are moved; with two, they are also rotated around the first destination so that the direction from the first source to the second matches that of the destinations.
- With three pairs, the third one also decides whether the objects must be flipped (mirrored about the line of the first pair) so that they end up on the same side as the third destination.
- Use object snaps (F3) so that the points land exactly on the vertices.

See also

MOVE · ROTATE · SCALE · MIRROR · Object snaps and precision

ARRAY —Array

Repeats objects in rows and columns or in a circle.

- **Command:** ARRAY · **Global name:** _ARRAY · **Alias:** AR
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Repeats the selected objects in rows and columns (rectangular array) or around a center (polar array). The copies are independent objects.

Usage

1. Enter ARRAY (or AR) or click **Array** on the ribbon.
2. Select the objects and press Enter. If they were already selected before starting the command, they are used without asking.

```
Select objects:
```

3. Choose the type of array (Enter: rectangular).

```
Enter the type of array [Rectangular/PAth/POlar] <R>:
```

4. Rectangular: enter the number of rows and columns (Enter: 1) and the distances between them. The original stays in a corner; with negative distances the copies go down or to the left.

```
Enter the number of rows (---) <1>:
Enter the number of columns (|||) <1>:
Specify the distance between rows:
Specify the distance between columns:
```

5. Polar: specify the center, the number of items (including the original) and the angle to fill (Enter: 360; negative, clockwise). The copies are rotated around the center.

```
Specify center point of array:
Enter the number of items in the array:
Specify the angle to fill (+=ccw, -=cw) <360>:
```

Options

- **Rectangular** (R): rows and columns at a fixed spacing.
- **Polar** (PO): copies spread around a circle or an arc.
- **PAth** (PA): continues as ARRAYPATH with the objects already selected.

Notes

- With a single row or column its distance is not asked. The total number of items goes from 2 to 100,000.
- The array is not associative (it cannot be edited later as a whole) and there is no path array.

See also

COPY · OFFSET · ROTATE · Selecting objects

ARRAYCLASSIC —Classic array

Rectangular or polar array in the classic style.

- **Command:** `ARRAYCLASSIC` · **Global name:** `_ARRAYCLASSIC`

Array in the classic AutoCAD style (the dialog box from before 2012). In LinceCAD it does the same as ARRAY: it asks the type, rectangular or polar, on the command line.

Usage

1. Enter ARRAYCLASSIC (MATRIZCLASICA).
2. Select the objects and press Enter.

```
Select objects:
```

3. Choose the array type.

```
Enter the type of array [Rectangular/PAth/POlar] <R>:
```

See also

ARRAY · ARRAYRECT · ARRAYPOLAR

ARRAYCLOSE —Close array

Ends editing the source of an array, saving or discarding the changes.

- **Command:** `ARRAYCLOSE` · **Global name:** `_ARRAYCLOSE`

Ends editing the source of an associative array, like AutoCAD ARRAYCLOSE: saves or discards the changes and goes back to the drawing.

Usage

1. With ARRAYEDIT, select the array and choose **Source**.
2. Change the source objects and enter ARRAYCLOSE.

See also

ARRAYEDIT · ARRAY

ARRAYEDIT —Edit array

Changes an array's parameters and regenerates it.

- **Command:** `ARRAYEDIT` · **Global name:** `_ARRAYEDIT` · **Alias:** `EDITARRAY`
- Ribbon: Home › Modify

Changes the parameters of an array created with ARRAY or ARRAYPATH and lays it out again, without reselecting the objects.

Usage

1. Type ARRAYEDIT (or EDITARRAY).
2. Select the array to edit. If it was selected before the command, it is used without asking.

```
Select the array to edit:
```

3. Depending on the array type, re-enter its values. Press Enter in a field to keep its current value. - Rectangular: rows, columns and spacing. > Enter the number of rows (---) <1>: > Enter the number of columns (|||) <1>: - Polar: number of items and angle (the center is kept). > Enter the number of items in the array: > Specify the angle to fill (+=ccw, -=cw) <360>: - Path: number of items or, with Distance, the spacing; the path is kept. > Enter the number of items or [Distance/Align]:

Options

- **Distance** / **Number** / **Align**: on a path array, switch between laying out by number or by distance and turn each copy's rotation on or off.

Source, Replace and Reset

At the first prompt these also work:

- **Source**: opens the source objects of the array for editing (like the block editor); when done, ARRAYCLOSE saves or discards the changes and every item changes.
- **REPlace**: select the replacement objects and their base point and then, as in AutoCAD 2027, the items of the array to replace (one click on each; Enter ends), or **Source** to replace the source of all of them. The base point goes to the reference point of each item and the replacement objects are erased.

```
Select replacement objects:
Select base point of replacement objects:
Select an item in array to replace or [Source objects] <exit>:
```

- **RESet**: lays the items out again with the array parameters (undoes single item replacements).

Notes

- A replaced item becomes like the others again if the array numbers are changed afterwards (AutoCAD keeps it).
- It only edits Lince's own arrays (blocks with their parameters stored). Selecting any other object warns and does nothing.
- A path array's path must still be in the drawing; if it was deleted, the array cannot be regenerated.

See also

ARRAY · ARRAYPATH

ARRAYPATH —Path array

Distributes copies of objects along a path.

- **Command:** ARRAYPATH · **Global name:** _ARRAYPATH · **Alias:** PATHARRAY
- Ribbon: Home › Modify

Distributes copies of the selected objects along a path curve (a line, arc, circle, polyline, ellipse or spline), as in AutoCAD 2027.

Usage

1. Type ARRAYPATH (or PATHARRAY) or click **Path array**.
2. Select the objects to distribute and press Enter. If they were already selected, they are used without asking.

```
Select objects:
```

3. Select the path curve. The array starts with the default values: items spaced 1.5 times the size of the selection along the path, until it is filled.

```
Select path curve:
```

4. Change what you need with the options and press Enter (or eXit) to create the array.

```
Select grip to edit array or [ASsociative/Method/Base point/Tangent direction/Items/Rows/Lev
els/Align items/Z direction/eXit] <eXit>:
```

Options

- **ASsociative** (AS): associative array (a single object, editable with ARRAYEDIT) or separate copies (ARRAYASSOCIATIVITY variable).

```
Create associative array [Yes/No] <…>:
```

- **Method** (M): Divide spreads the number of items over the whole path; Measure places them at the given distance from the start.

```
Enter path method [Divide/Measure] <…>:
```

- **Base** (B): point of the selection that is taken to the path. Without it, items are placed relative to the start of the path and the original does not move.

```
Specify base point <end of path curve>:
```

- **Tangent** (T): two points giving the direction of the selection that is aligned with the path tangent (by default, the tangent at the start).

```
Specify first point of tangent vector <path tangent>:
```

- **Items** (I): with Divide, how many items; with Measure, the distance between them and then how many (at most, those that fit; Fill places all of them).

```
Specify the distance between items along path <…>:
Specify number of items or [Fill entire path] <…>:
```

- **Rows** (R): rows parallel to the path, at the given distance (to the left of the path direction), and elevation between rows.
- **Levels** (L): levels and distance between them (in Z).
- **Align** (A): each item rotates with the path tangent (Yes, by default) or is only moved.

```
Align arrayed items to the path? [Yes/No] <…>:
```

- **Z** (Z): keep the Z direction of the items (no effect in 2D).

Notes

- At a path vertex the item takes the tangent of the incoming segment, as in AutoCAD.
- Associative, with a single row and level, no base point and no own tangent, it is a LinceCAD array stored as a block: AutoCAD opens it the same, but only LinceCAD edits it with ARRAYEDIT. Otherwise separate copies are created.
- Expression and Key point are not available. On ellipses and splines the length is measured on a very fine approximation of the curve.

See also

ARRAYEDIT · ARRAY · DIVIDE · MEASURE · COPY

ARRAYPOLAR —Polar array

Distributes copies of objects around a circle or arc.

- **Command:** ARRAYPOLAR · **Global name:** _ARRAYPOLAR

Creates a polar array: copies of the objects spread around a circle or an arc. It is ARRAY with the polar type

already chosen.

Usage

1. Enter ARRAYPOLAR (MATRIZPOLAR) or click **Polar Array** on the ribbon.
2. Select the objects and press Enter (if they were already selected, they are used without asking).

```
Select objects:
```

3. Specify the center, the number of items (counting the original) and the angle to fill.

```
Specify center point of array:
Enter the number of items in the array:
Specify the angle to fill (+=ccw, -=cw) <360>:
```

Notes

- It works like ARRAY with the Polar option; the notes of that page apply here.
- It does not have AutoCAD's interactive options yet (Base point, Rotate items, Direction, Rows, Levels).

See also

ARRAY · ARRAYRECT · ARRAYCLASSIC · ARRAYPATH

ARRAYRECT — Rectangular array

Creates a rectangular array of rows and columns.

- **Command:** `ARRAYRECT` · **Global name:** `_ARRAYRECT`

Creates a rectangular array: copies of the objects in rows and columns. It is ARRAY with the rectangular type already chosen.

Usage

1. Enter ARRAYRECT (MATRIZRECTANG) or click **Rectangular Array** on the ribbon.
2. Select the objects and press Enter (if they were already selected, they are used without asking).

```
Select objects:
```

3. Give the rows, columns and the distances between them.

```
Enter the number of rows (---) <1>:
Enter the number of columns (|||) <1>:
```

Notes

- It works like ARRAY with the Rectangular option; the notes of that page apply here.
- It does not have AutoCAD's interactive options yet (Base, Angle, Count, Spacing, Associative, Levels).

See also

ARRAY · ARRAYPOLAR · ARRAYCLASSIC · ARRAYPATH

BLEND — Blend curves

Creates a spline joining two open curves.

- **Command:** `BLEND` · **Global name:** `_BLEND`

Creates a spline joining two open curves at the ends nearest to where you picked them, tangent to both.

Usage

1. Enter BLEND (FUSIONAR).
2. Select the first curve near the end you want to join.

```
Select first object or [CONtinuity]:
```

3. Select the second curve, also near its end.

```
Select second object:
```

Options

- **CONtinuity**: Tangent (the default) or Smooth.

```
Enter a continuity [Tangent/Smooth] <…>:
```

Notes

- Works with lines, arcs and open polylines. The spline is a cubic with the end tangents; the Smooth type gives the same result as Tangent today (curvature continuity is missing).
- It does not trim or extend the curves.

See also

SPLINE · FILLET · JOIN

BREAK —Break

Breaks an object between two points.

- **Command:** BREAK · **Global name:** _BREAK · **Alias:** BR
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Breaks an object by removing the piece between two points, or splits it in two at a single point without removing anything.

Usage

1. Enter BREAK (or BR) or click **Break** on the ribbon.
2. Click the object. The point you click is the first break point.

```
Select object:
```

3. Specify the second break point: whatever lies between both points is removed.

```
Specify second break point or [First point] (@: at the same point):
```

Options

- **First point** (F): discards the clicked point and asks for the first break point, for example to give it with an object snap.

```
Specify first break point:
```

- @: uses the same point as the first one; the object is split into two objects that touch.

Notes

- Breaks lines, arcs, circles and polylines. The points are projected onto the object, so they do not have to lie exactly on it.
- On a circle the piece from the first point to the second one counterclockwise is removed and an arc remains; a closed polyline becomes open. Closed objects cannot be broken at a single point (@).
- It does not use preselection: the object is picked during the command.

See also

TRIM · JOIN · LENGTHEN · Object snaps and precision

BREAKATPOINT — Break at point

Breaks an object in two at a point.

- **Command:** `BREAKATPOINT` · **Global name:** `_BREAKATPOINT`

Breaks an object in two at a point, removing nothing. It is BREAK with the second point equal to the first.

Usage

1. Enter BREAKATPOINT (PARTIRENPUNTO) or click **Break at Point** on the ribbon.
2. Select the object (a line, an arc, a circle or an open polyline).

```
Select object:
```

3. Specify the break point (it is projected onto the object).

```
Specify break point:
```

Notes

- A closed polyline or a circle cannot be broken with a single point: use BREAK with two.
- A point at an end of the object changes nothing.

See also

BREAK · JOIN · TRIM

BURST — Burst

Explodes blocks keeping attributes as texts with their values.

- **Command:** `BURST` · **Global name:** `_BURST`
- Ribbon: Home › Modify

Explodes blocks like EXPLODE, but visible attributes become texts with their values (EXPLODE turns them into attribute definitions that show the tag). It is the BURST tool of the Express Tools.

Usage

1. Enter BURST.
2. Select the blocks and press Enter. If they were already selected, they are exploded without asking.

```
Select objects:
```

Notes

- What is on layer 0 or has ByBlock color, linetype or lineweight takes those of the insert, as when exploding in AutoCAD.
- Invisible or empty attributes disappear. Multiline attributes come out as a single-line text.
- One level is exploded: nested blocks stay as blocks. Other selected objects do not change.

See also

EXPLODE · TXTEXP · ATTEDIT · BATTMAN

CHAMFER —Chamfer

Bevels the corner between two lines.

- **Command:** `CHAMFER` · **Global name:** `_CHAMFER` · **Alias:** `CHA`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Joins two lines with a chamfer (a straight line) at the given distances, trimming or extending them to it. With distances 0 it forms a sharp corner.

Usage

1. Enter CHAMFER (or CHA) or click **Chamfer** on the ribbon.

```
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]:
```

2. Select the first line and then the second, each on the part you want to keep. The first distance (or the length) counts on the first line.

```
Select second line or shift-select to apply corner or [Distance/Angle/Method]:
```

3. The chamfer is created and the command ends (with Multiple it keeps asking for pairs until Enter).

Options

- **Distance**: the two distances (CHAMFERA and CHAMFERB; Enter on the second repeats the first).

```
Specify first chamfer distance <…>:
```

- **Angle**: length on the first line (CHAMFERC) and angle with it (CHAMFERD, in radians).

```
Specify chamfer length on the first line <…>:
```

- **Method**: Distance or Angle (CHAMMODE).
- **Polyline**: chamfers all straight vertices of a 2D polyline.
- **Trim**: Trim or No trim (TRIMMODE). Without trimming only the chamfer is added.
- **Multiple** and **Undo**.

Notes

- Works only with lines and polylines. If the distances do not fit, a message is shown.
- The chamfer goes on the layer of the first line if both share a layer; otherwise on the current layer.
- With Shift held when selecting the second line, a sharp corner. Two straight segments of the same polyline, adjacent or separated by an arc (which is replaced by the chamfer), or the first or last segment of an open polyline with a line, are chamfered as in FILLET, as in AutoCAD 2027. The Polyline option leaves the vertices next to an arc unchanged.

See also

FILLET · TRIM · EXTEND · Editing

CHANGE —Change

Changes points and properties of objects.

- **Command:** `CHANGE` · **Global name:** `_CHANGE` · **Alias:** `-CH`
- Ribbon: Home › Modify
- Menu: Modify

Changes the geometry or the properties of the selected objects: with a change point it takes line endpoints and circle radii to that point and lets you change texts and blocks one by one; with Properties it changes the color, layer, linetype and other properties of everything selected.

Usage

1. Enter CHANGE (or -CH).
2. Select the objects and press Enter.

```
Select objects:
```

3. Specify the change point, press Enter to go on to texts, blocks and circles, or choose Properties.

```
Specify change point or [Properties]:
```

Change point

- **Lines**: the endpoint of each line closest to the point is moved to it.
- **Circles**: the radius changes so that the circle passes through the point. Without a point (Enter), the radius of each circle is asked.

```
Specify new circle radius <no change>:
```

- **Texts**: the insertion point, style, height, rotation angle and text are asked one by one. Enter keeps each value.

```
Specify new insertion point <no change>:
Enter new text style <no change>:
Specify new height <no change>:
Specify new rotation angle <no change>:
Enter new text <no change>:
```

- **Blocks**: the insertion point and the rotation angle.

Properties

```
Enter property to change [Color/Elev/LAyer/LType/ltScale/LWeight/Thickness/TRansparency/Mate
rial/Annotative]:
```

- **Color** (C), **LAyer** (LA), **LType** (LT), **ltScale** (S), **LWeight** (LW) and **TRansparency** (TR): as in the PROPERTIES palette. The layer and the linetype must exist in the drawing.
- **Elev** (E): the elevation (Z) of lines (if both endpoints are at the same one), circles, arcs, polylines, texts, blocks, points, ellipses, hatches, 2D solids and shapes.
- **Material** (M): the name of a material of the drawing, ByLayer or ByBlock. **Annotative** (A... see the prompt): Yes or No.
- **Thickness** (T): the extrusion thickness of lines, circles, arcs, polylines, texts and points.

After each property you return to the list; Enter ends.

Notes

- Everything a CHANGE does is undone at once with U.
- It does not change the material, the plot style or the annotative property yet; of attributes, only what they have of text; of multiline texts, only the insertion point. ORTHO mode does not make the lines parallel.

See also

PROPERTIES · MATCHPROP · STRETCH · TEXTEDIT

CHPROP —Change properties

Changes color, layer, linetype and other properties of objects.

- **Command:** `CHPROP` · **Global name:** `_CHPROP`
- Ribbon: Home › Modify

Changes the properties of the selected objects from the command line, like AutoCAD CHPROP: color, layer, linetype and its scale, lineweight, thickness, transparency, material and annotative.

Usage

1. Enter CHPROP.
2. Select the objects and press Enter.

```
Select objects:
```

3. Choose a property, enter its value and repeat with another; Enter ends.

```
Enter property to change [Color/LAyer/LType/ltScale/LWeight/Thickness/TRansparency/Material/
Annotative]:
```

Notes

- Options and values are those of the Properties option of CHANGE, without elevation.

See also

CHANGE · PROPERTIES · MATCHPROP

COPY —Copy

Copies objects from a base point to another point.

- **Command:** `COPY` · **Global name:** `_COPY` · **Alias:** `CO`, `CP`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Copies the selected objects from a base point to one or more destination points. Each point you specify creates a new copy; the originals do not change.

Usage

1. Enter COPY (or CO, CP) or click **Copy** on the ribbon.
2. Select the objects and press Enter. If they were already selected before starting the command, they are used without asking.

```
Select objects:
```

3. Specify the base point.

```
Specify base point or [Displacement/mOde] <Displacement>:
```

4. Specify the destination point. A copy is created and the prompt repeats to make another one from the same base point.

```
Specify second point or [Array] <use first point as displacement>:
```

5. Press Enter, Space or Esc to finish.

Notes

- As in AutoCAD, copying is multiple: the command keeps asking for destinations until you finish it.
- Destinations can be typed as relative or polar coordinates from the base point, or as a distance in the direction of the cursor.
- In 3D points accept Z (`x,y,z`, `@dx,dy,dz`), as in MOVE: the copy is raised or lowered.
- Grips copy too: in any mode (stretch, move, rotate, scale, mirror) choose **Copy** (C) and each point leaves a copy.
- There is no Array option as in AutoCAD; for copies in rows, columns or around a circle use ARRAY.
- To copy between drawings use COPYCLIP and PASTECLIP.
- AutoCAD's Displacement, mOde and Undo options are not available.
- Options: **Displacement** (a vector, a single copy), **mOde** (Single or Multiple, variable COPYMODE), **Array** (number of items and second point; with **Fit** the point is the last item), **Exit** and **Undo** (removes the last copy or array). Enter at the first second point uses the base point as displacement.

See also

MOVE · ARRAY · OFFSET · MIRROR · Selecting objects

COPYM —Multiple copy

Multiple copies with repeat, divide, measure and array (Express Tools).

- **Command:** `COPYM` · **Global name:** `_COPYM`
- Ribbon: Home › Modify

Makes multiple copies of the selection, like the Express Tools COPYM: one copy per point, Repeat, Divide, Measure and Array.

Usage

1. Enter COPYM, select the objects and press Enter, then specify the base point.

```
Second point or [Repeat/Divide/Measure/Array/Undo] <eXit>:
```

See also

COPY · ARRAY

DRAWORDER —Draw Order

Changes the draw order of objects.

- **Command:** `DRAWORDER` · **Global name:** `_DRAWORDER` · **Alias:** `DR`
- Ribbon: Home › Modify

- Menu: Tools

Changes the order in which overlapping objects are drawn: it brings the selection to the front or sends it to the back of everything, or above or under other specific objects. It is useful with images, solid hatches and wipeouts.

Usage

1. Enter DRAWORDER (or DR).
2. Select the objects and press Enter. If they were already selected before starting the command, they are used without asking.

```
Select objects:
```

3. Choose where they go (Enter: to the back).

```
Enter object ordering option [Above objects/Under objects/Front/Back] <Back>:
```

4. With Above or Under, select the reference objects and press Enter.

```
Select reference objects:
```

Options

- **Above** (A): just above the highest of the reference objects.
- **Under** (U): just under the lowest of the reference objects.
- **Front** (F): in front of every object in its space.
- **Back** (B): behind every object in its space.

Notes

- The order is saved in the drawing (SORTENTS table of model space or of the layout), as in AutoCAD, which reads it the same way. It can be undone.
- The selected objects keep the order they had among themselves.
- When it finishes, the view is regenerated so that the new order shows. After undoing it, use REGEN if the order on screen is not up to date.

See also

WIPEOUT · HATCH · IMAGEATTACH · Editing

ERASE —Erase

Removes objects from the drawing.

- **Command:** ERASE · **Global name:** _ERASE · **Alias:** E, DELETE · **Shortcut:** Del
- Ribbon: Home › Modify
- Menu: Edit
- Menu: Modify
- Toolbar: Modify

Removes the selected objects from the drawing.

Usage

1. Enter ERASE (or E) or click **Erase** on the ribbon.
2. Select the objects with clicks, windows or crossing windows and press Enter to erase them.

```
Select objects:
```

The quickest way is preselection: select the objects with no command running and press **Delete**, or start ERASE,

which erases them without asking anything else.

Notes

- Erased objects come back with UNDO (Ctrl+Z).
- Objects on locked layers cannot be selected and so are not erased.
- Erasing a member of a group removes it from the group; an unnamed group that becomes empty disappears.
- To remove unused definitions (blocks, layers, styles) use PURGE; to erase repeated objects, OVERKILL.

See also

UNDO · OVERKILL · PURGE · CUTCLIP · Selecting objects

EXOFFSET —Extended offset

OFFSET with the gap type option (Express Tools).

- **Command:** `EXOFFSET` · **Global name:** `_EXOFFSET`
- Ribbon: Home › Modify

OFFSET with the Express Tools gap type option (OFFSETGAPTYPE).

Notes

- Extend prolongs the segments until they meet; Fillet adds an arc whose radius is the distance, centred on the vertex; Chamfer, a segment perpendicular to the bisector at the distance from the vertex. OFFSET does the same with OFFSETGAPTYPE, which is how it was compared with AutoCAD 2027.
- Express Tools do not run in AutoCAD's console without interface: this command has unit tests.

See also

OFFSET

EXPLODE —Explode

Breaks a compound object into its parts.

- **Command:** `EXPLODE` · **Global name:** `_EXPLODE` · **Alias:** `X`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Breaks each selected compound object into the pieces it is made of: a polyline into lines and arcs, a block into its objects, a multiline into its lines, a dimension into its lines, arrowheads and text, a hatch into its lines and a table into its lines and texts.

Usage

1. Enter EXPLODE (or X) or click **Explode** on the ribbon.
2. Select the objects and press Enter. If they were already selected before starting the command, they are exploded without asking.

```
Select objects:
```

Notes

- Each use explodes a single level: a block that contains other blocks leaves those blocks as references, which

can be exploded again.
- Pieces of a dimension that were on layer 0 go to the layer of the dimension.
- Simple objects (lines, arcs, circles, texts) do not change; if nothing selected can be exploded, a message says so.
- Exploding loses what the compound object provides: a dimension stops measuring, a block no longer updates when it is redefined and a polyline loses its width.

See also

JOIN · PEDIT · BLOCK · UNGROUP · Blocks

EXTEND —Extend

Extends objects to the nearest boundary.

- **Command:** `EXTEND` · **Global name:** `_EXTEND` · **Alias:** `EX`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Extends an object from the end you pick until it touches the first object it meets (the boundary edges).

Usage

1. Enter EXTEND (or EX) or click **Extend** on the ribbon.
2. In **Quick mode** (default, TRIMEXTENDMODE=1) every object in the drawing is a boundary: click each object near the end you want to extend. The prompt repeats to extend more.

```
Select object to extend or shift-select to trim or [Boundary edges/Crossing/mOde/Project/eRase]:
```

3. In **Standard mode** you first select the boundary edges (Enter = all objects).

```
Select cutting edges... Select objects or [mOde] <select all>:
```

4. Press Enter or Esc to finish.

Options

The same as TRIM: boundary edges, Fence/Crossing, mOde, Project, Edge (Extend), eRase and Undo.

Notes

- Extends lines, arcs and open polylines; boundaries can be lines, arcs, circles and polylines. An arc is extended along its circle.
- If the object, once extended, reaches no boundary, a message is shown and nothing changes.
- Objects are picked one at a time with a click near the end; the previous selection is not used.
- An elliptical arc is extended along its ellipse. An open cubic spline is extended, as in AutoCAD, with one more span that follows its curvature (the existing part does not change). Ellipses and splines also work as boundaries.
- With Shift held while clicking, it trims instead of extending. To extend by a distance or to a total length use LENGTHEN.

See also

TRIM · LENGTHEN · FILLET · Editing

EXTRIM — Extended trim

Trims on one side everything that crosses a cutting edge (Express Tools).

- **Command:** `EXTRIM` · **Global name:** `_EXTRIM`
- Ribbon: Home › Modify

Trims at once everything that crosses a cutting edge, on the side you specify, like the Express Tools EXTRIM.

Usage

1. Enter EXTRIM and select the cutting edge (line, polyline, arc or circle).
2. Specify a point on the side to trim.

See also

TRIM

FILLET — Fillet

Joins two objects with a tangent arc.

- **Command:** `FILLET` · **Global name:** `_FILLET` · **Alias:** `F`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Joins two objects with an arc tangent to both, of the current radius, trimming or extending them to the arc. With radius 0 it brings them to their intersection and forms a sharp corner.

Usage

1. Enter FILLET (or F) or click **Fillet** on the ribbon.

```
Select first object or [Undo/Polyline/Radius/Trim/Multiple]:
```

2. Select the first object and then the second, each on the part you want to keep.

```
Select second object or shift-select to apply corner or [Radius]:
```

3. The arc is created and the command ends (with Multiple it keeps asking for pairs until Enter).

Options

- **Radius**: changes the fillet radius (0 or positive; Enter keeps the current one). Stored in FILLETRAD.

```
Specify fillet radius <…>:
```

- **Polyline**: select a 2D polyline and fillet all its vertices between straight segments (those that do not fit are left).
- **Trim**: Trim or No trim (variable TRIMMODE). Without trimming only the arc is added.

```
Enter Trim mode option [Trim/No trim] <…>:
```

- **Multiple**: keeps asking for pairs of objects.
- **Undo**: undoes the last fillet.

Notes

- Works between lines, arcs and circles (also arc with arc and circle with circle). Circles are not trimmed. Between two parallel lines a semicircle with the separation as diameter is formed, whatever the radius.

- If there are several solutions, the one on the side where the objects were picked is used.
- If the radius does not fit, a message is shown. The arc goes on the layer of the first object if both share a layer; otherwise on the current layer.
- It does not use the previous selection. With Shift held when selecting the second object, a sharp corner (radius 0). Two straight segments of the same polyline, adjacent or separated by an arc (which is replaced by the new fillet), are filleted inside it, as in AutoCAD 2027; so is the first or last segment of an open polyline with a line, which becomes part of the polyline. With the Polyline option, arcs tangent to both straight segments they join (earlier fillets) are redone with the current radius.

See also

CHAMFER · TRIM · EXTEND · Editing

FLATTEN —Flatten

Flattens the selected objects onto the XY plane (Express Tools).

- **Command:** `FLATTEN` · **Global name:** `_FLATTEN`
- Ribbon: Home › Modify

Flattens the selected objects onto the XY plane, like the AutoCAD Express Tools FLATTEN: the Z of their points, their elevation and their thickness become 0.

Usage

1. Enter FLATTEN.
2. Select the objects and press Enter.
3. Answer the hidden lines question (Enter: No).

```
Remove hidden lines? [Yes/No] <No>:
```

Notes

- Hidden line removal is not available yet; use FLATSHOT (FLATSHOT) for a flat view of a 3D model.

See also

FLATSHOT · CHANGE

GROUP —Group

Creates a group with the selected objects.

- **Command:** `GROUP` · **Global name:** `_GROUP` · **Alias:** `G`
- Ribbon: Home › Groups
- Menu: Modify

Creates a group with the selected objects: a named (or unnamed) set that is selected as a whole with one click, while its objects stay independent and can be edited separately.

Usage

1. Enter GROUP (or G).
2. If you want, choose Name or Description before selecting.

```
Select objects or [Name/Description]:
Enter a group name or [?]:
```

```
Enter a group description:
```

3. Select the objects and press Enter: the group is created. If the objects were already selected before starting the command, the group is created at once, unnamed.

Options

- **Name** (N): name of the group; it cannot repeat another one. `?` lists the groups in the drawing. Without a name, the group is anonymous (like AutoCAD's `*A1`, `*A2`...).
- **Description** (D): free text saved with the group.

Notes

- While group selection is on, selecting a member selects the whole group. Ctrl+Shift+A turns it on or off; it is stored in the PICKSTYLE variable.
- Groups are saved in DWG and DXF like AutoCAD's (ACAD_GROUP dictionary).
- An erased object stops being a member; an anonymous group that becomes empty disappears.
- To change the members or the name use GROUPEDIT; to undo the group, UNGROUP. Unlike a block, a group creates no definition.

See also

GROUPEDIT · UNGROUP · BLOCK · Selecting objects

GROUPEDIT — Edit Group

Adds or removes objects from a group, or renames it.

- **Command:** `GROUPEDIT` · **Global name:** `_GROUPEDIT`
- Ribbon: Home › Groups
- Menu: Modify

Changes an existing group: adds objects to it or removes them, or renames it.

Usage

1. Enter GROUPEDIT.
2. Select a member of the group, or choose Name and type the name of the group (`?` lists them). If objects of a single group were already selected before starting the command, that group is edited without asking.

```
Select group or [Name]:
```

3. Choose what to do.

```
Enter an option [Add objects/Remove objects/REName]:
```

4. Select the objects to add or remove and press Enter, or type the new name.

```
Select objects to add to group:
Select objects to remove from group:
Enter new name for group or [?]:
```

Options

- **Add** (A): adds the objects you select to the group.
- **Remove** (R): takes the objects you select out of the group; they stay in the drawing.
- **REName** (REN): gives the group another name (also to an anonymous one, which becomes named).

Notes

- To select just some members of a group, first turn group selection off with Ctrl+Shift+A (PICKSTYLE).
- Changes can be undone and the drawing is marked as modified.

See also

GROUP · UNGROUP · Selecting objects

INTERSECT —Intersect

Keeps the volume common to several solids.

- **Command:** `INTERSECT` · **Global name:** `_INTERSECT` · **Alias:** `IN`
- Ribbon: Home › Modify

Keeps the volume common to several solids (boolean intersection), like AutoCAD's INTERSECT. Between polyhedra it is computed by an in-house BSP modeler (pure Rust), exact even with coplanar faces.

Usage

1. Type INTERSECT or click it in the ribbon.
2. Select two or more solids and press Enter.

```
Select the solids and press Enter:
```

The selected solids are replaced by their intersection. The result keeps the properties (layer, color, material…) of the first solid selected.

Notes

- Between **polyhedra** (planar-faced solids) the result is **exact**: AutoCAD audits it with no errors and MASSPROP reports the exact volume, also with coplanar faces (walls flush at the top and bottom, solids touching along a face).
- Solids with curved faces are not yet supported in booleans; they come in a later phase.
- See also UNION, SUBTRACT, EXTRUDE and REVOLVE.

JOIN —Join

Joins touching objects into one.

- **Command:** `JOIN` · **Global name:** `_JOIN` · **Alias:** `J`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Joins into a single object several objects that touch at their ends or lie on the same line, circle or ellipse: lines, arcs, elliptical arcs, polylines and splines.

Usage

1. Enter JOIN (or J) or click **Join** on the ribbon.
2. Select the source object or all the objects to join and press Enter. If they were already selected before starting the command, they are joined without asking.

```
Select source object or multiple objects to join at once:
```

3. If you selected a single arc or a single elliptical arc, you can select more arcs or enter C (Close) to turn it into a

circle or a full ellipse.

```
Select arcs to join to source or [Close]:
```

Notes

- Lines on the same straight line are joined into one line from end to end, even with gaps between them.
- Arcs of the same circle are joined into one arc, counterclockwise from the first one selected, even with gaps; if they cover the whole turn, the result is a circle. The same for arcs of the same ellipse.
- Objects that touch at their ends form a chain: a polyline if they are lines, arcs and polylines, or a spline if there is any spline or elliptical arc. If the chain ends where it starts, the result is closed. More than one object can result.
- The result keeps the properties (layer, color...) of the first object of each group, the source.
- Objects that are already closed (circles, ellipses, closed polylines and splines) are not joined. As in AutoCAD there is no fuzz distance: to add objects with gaps to a polyline, use the Join option of PEDIT.
- The command line says how many objects were joined and into how many; if there is nothing to join, it says so.

See also

PEDIT · SPLINEDIT · EXPLODE · BREAK · PLINE · Selecting objects

LENGTHEN —Lengthen

Lengthens or shortens lines, arcs and open polylines.

- **Command:** `LENGTHEN` · **Global name:** `_LENGTHEN` · **Alias:** `LEN`
- Ribbon: Home › Modify
- Menu: Modify

Measures or changes the length of lines, arcs, open polylines, elliptical arcs and open splines: it lengthens or shortens the end nearest to the point where you select, along the curve itself.

Usage

1. Enter LENGTHEN (or LEN).
2. Select an object to see its length (and the included angle, for an arc), or choose how to change it. Enter repeats the previous mode and value.

```
Select an object to measure or [DElta/Percent/Total/DYnamic] <…>:
```

3. Enter the value of the chosen mode.
4. Select each object near the end you want to change. The prompt repeats; Enter finishes.

```
Select an object to change or [Undo]:
```

Options

- **DElta** (DE): adds that length (negative shortens). With **Angle** (A), an angle in degrees is added to an arc.

```
Enter delta length or [Angle] <…>:
```

- **Percent** (P): the new length is that percentage of the current one (50 halves it).

```
Enter percentage length <…>:
```

- **Total** (T): leaves the object with that total length (or, with Angle, an arc with that total angle).

```
Specify total length or [Angle] <…>:
```

- **DYnamic** (DY): after you select the object, its end follows the cursor up to the point you specify.

```
Specify new end point:
```

- **Undo** (U): undoes the last change without leaving the command.

Notes

- A line is lengthened in its direction and an arc, along its circle; in a polyline the end segment is lengthened or the polyline is trimmed along itself.
- An elliptical arc is lengthened or shortened along its ellipse. An open spline can only be shortened: as in AutoCAD, lengthening it leaves it unchanged. Whole ellipses and closed splines are not changed. Preselection is not used.

See also

EXTEND · TRIM · BREAK · LIST · Editing

MIRROR —Mirror

Creates a mirrored copy about an axis.

- **Command:** `MIRROR` · **Global name:** `_MIRROR` · **Alias:** `MI`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Creates a mirrored copy of the selected objects about an axis defined by two points and, if you ask for it, erases the originals.

Usage

1. Enter MIRROR (or MI) or click **Mirror** on the ribbon.
2. Select the objects and press Enter. If they were already selected before starting the command, they are used without asking.

```
Select objects:
```

3. Specify the two points of the mirror line. While you specify the second one the result is previewed.

```
Specify first point of mirror line:
Specify second point of mirror line:
```

4. Choose whether the originals are kept (Enter: they are kept).

```
Erase source objects? [Yes/No] <No>:
```

Options

- **Yes** (Y): erases the source objects; only the mirrored copy remains.
- **No** (N): keeps the originals next to the copy.

Notes

- With Ortho (F8) the mirror line is horizontal or vertical, the most common case.
- Texts are mirrored or not depending on the MIRRTEXT variable: with 0 (the usual value) they still read correctly; with 1 they look as in a mirror. Dimensions are regenerated readable.
- Grips also have a mirror mode (press Space until you reach it); the grip is the first point of the mirror line.

```
Specify second point or [Base point/Copy/eXit]:
```

See also

COPY · ROTATE · ARRAY · Selecting objects

MLEDIT —Edit Multiline

Crosses, tees, corners, vertices, cuts and welds of multilines.

- **Command:** `MLEDIT` · **Global name:** `_MLEDIT`
- Ribbon: Home › Modify
- Menu: Modify

Edits multilines: resolves their crossings, tee junctions and corners, adds or deletes vertices and cuts or welds their elements. The tool is chosen in the **Multiline Edit Tools** dialog box, as in AutoCAD 2027, or, on the command line (-MLEDIT and scripts), by its two-letter code, the same ones as in AutoCAD.

Usage

1. Enter MLEDIT.
2. Click the tool in the dialog box (or, on the command line, type its code: table below).

```
Enter mline editing option [CC/OC/MC/CT/OT/MT/CJ/AV/DV/CS/CA/WA]:
```

3. For crosses, tees and corners, select the first multiline and the second one. For the other tools, select the multiline at the point where the tool acts and, if asked, specify the second point.

```
Select first mline or [Undo]:
Select second mline:
Select mline or [Undo]:
Select second point:
```

4. The prompt repeats with the same tool to keep editing; Enter finishes.

Code	Tool
CC / OC / MC	Closed, open or merged cross between two multilines
CT / OT / MT	Closed, open or merged tee (the first one ends on the second)
CJ	Corner joint: both multilines are trimmed or extended until they meet
AV / DV	Add or delete a vertex at the selected point
CS / CA	Cut one element or all of them between the selected point and the second point
WA	Weld all the elements between both points (removes the cuts)

Options

- **Undo** (U): undoes the last edit without leaving the command.

Notes

- It does not use preselection. If the selected object is not a multiline, a message says so.
- In a tee, the piece left after the junction loses its cuts; deleting a vertex recalculates the miters of the whole multiline.
- -MLEDIT does the same without the dialog box. The dialog box shows the tool names (AutoCAD shows pictures).

See also

MLINE · -MLEDIT · EXPLODE · Editing

MOCORO —Move, copy, rotate and scale

Moves, copies, rotates and scales in a single command (Express Tools).

- **Command:** `MOCORO` · **Global name:** `_MOCORO`
- Ribbon: Home › Modify

Moves, copies, rotates and scales the selection in a single command, like the Express Tools MOCORO.

Usage

1. Enter MOCORO, select the objects and press Enter, then specify the base point.
2. Each point moves the objects; Rotate and Scale change the operation; Copy toggles copy mode; Base asks for a new base point; Undo undoes the last operation. Enter ends.

```
Second point or [Copy/Rotate/Scale/Base/Undo] <eXit>:
```

See also

MOVE · COPY

MOVE —Move

Moves objects from a base point to another point.

- **Command:** `MOVE` · **Global name:** `_MOVE` · **Alias:** `M`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Moves the selected objects a distance and direction given by two points: the base point and the destination point. The objects keep their shape, size and orientation.

Usage

1. Enter MOVE (or M) or click **Move** on the ribbon.
2. Select the objects and press Enter. If they were already selected before starting the command, they are used without asking.

```
Select objects:
```

3. Specify the base point: the point of the objects you want to take somewhere else.

```
Specify base point or [Displacement] <Displacement>:
```

4. Specify the destination point. While you move the cursor you see where the objects will go.

```
Specify second point or <use first point as displacement>:
```

Notes

- The second point can be typed as relative (`@100,0`) or polar (`@50<45`) coordinates from the base point, or as a single distance, applied in the direction of the cursor (handy with Ortho, F8).
- With no command running you can also move with grips: select the objects, click a grip and press Space until the move mode.

```
Specify move point or [Base point/Copy/eXit]:
```

- **In 3D** points can be typed with Z: `x,y,z` or `@dx,dy,dz` (e.g. base `0,0,0` and destination `@0,0,3` lifts the

objects 3 units). A point without Z lies at the current elevation (ELEV), and a relative `@dx,dy` after a 3D point keeps its Z, as in AutoCAD.

- Moving a dimension also moves its block; texts and blocks keep their rotation.
- There is no Displacement option as in AutoCAD: use a base point at `0,0` and type the displacement as destination.
- **Displacement** option: type the vector (`3,4`) instead of two points. Enter at the second point uses the base point as displacement.

See also

COPY · ROTATE · ALIGN · STRETCH · Selecting objects · Editing

MSTRETCH —Multiple stretch

Stretches with several crossing windows and polygons (Express Tools).

- **Command:** `MSTRETCH` · **Global name:** `_MSTRETCH`
- Ribbon: Home › Modify

Stretches with several crossing windows and polygons at once, like the Express Tools MSTRETCH.

Usage

1. Enter MSTRETCH and define the crossing windows (or CPolygon); Enter finishes.
2. Remove objects if needed, then give the base point and the second point.

See also

STRETCH

NCOPY —Copy nested

Copies objects from inside blocks and external references.

- **Command:** `NCOPY` · **Global name:** `_NCOPY`
- Ribbon: Home › Modify

Copies objects from inside blocks, external references or underlays, like AutoCAD NCOPY: select them one by one, they are copied in place into the drawing and the command goes on as COPY with the copies.

Usage

1. Enter NCOPY and click each nested object; press Enter.

```
Select nested objects to copy or [Settings]:
```

2. Go on as in COPY.

See also

COPY · EXPLODE

OFFSET —Offset

Creates a parallel copy at a distance.

- **Command:** `OFFSET` · **Global name:** `_OFFSET` · **Alias:** `O`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Creates a parallel copy of an object, at a distance or through a point: parallel lines, concentric circles and arcs or equidistant polylines.

Usage

1. Type OFFSET (or O) or click **Offset** on the ribbon.
2. Give the distance (a number or two points), or press Enter to use the remembered one (variable OFFSETDIST); if you last chose **Through**, that is the default.

```
Specify offset distance or [Through/Erase/Layer] <…>:
```

3. Click the object to offset.

```
Select object to offset or [Exit/Undo] <Exit>:
```

4. Specify a point on the side where the copy goes (with a distance) or the point it must pass through.

```
Specify point on side to offset or [Exit/Multiple/Undo] <Exit>:
```

5. The prompt of step 3 repeats so you can offset other objects. Press Enter to finish.

Options

- **Through**: the offset passes through the point you give; the distance is measured to the object.
- **Erase**: erases the source object after offsetting it (Yes/No).
- **Layer**: the copies go on the Current layer or on the layer of the Source object.
- **Multiple**: keeps asking for points; each copy is offset from the previous one.
- **Undo**: removes the last offset (and restores the source if it had been erased).

Notes

- Offsets lines, circles, arcs, polylines, construction lines and rays. In polylines, outside corners are extended until they meet, inside corners are trimmed, arcs that would get a negative radius disappear, and whatever lies closer than the distance to the original (self-crossing polylines) is discarded; the result can be more than one piece.
- A circle or arc offset inward by more than its radius cannot be created and a message is shown.
- It does not use the previous selection: objects are picked one at a time during the command.
- Variables: OFFSETDIST (remembered distance, negative = through) and OFFSETGAPTYPE: how the outside corners of polylines are closed (0 extending the segments, 1 with an arc whose radius is the distance, 2 with a chamfer at the distance from the vertex), as in AutoCAD 2027.
- An ellipse or a spline is offset into a spline, as in AutoCAD: it follows the offset curve (within a millionth of the drawing size). Inward beyond its smallest radius of curvature it is not offset.

See also

COPY · ARRAY · PLINE · Editing

OVERKILL —Delete Duplicate Objects

Deletes duplicate objects and combines overlapping lines and arcs.

- **Command:** `OVERKILL` · **Global name:** `_OVERKILL`
- Ribbon: Home › Modify
- Menu: Modify

Cleans a drawing of repeated objects: it erases exact duplicates and combines overlapping lines and arcs into a single one. It is useful with imported drawings or with objects drawn several times on top of each other.

Usage

1. Enter OVERKILL.
2. Select the objects to check and press Enter. If they were already selected before starting the command, they are used without asking.

```
Select objects:
```

3. The **Delete Duplicate Objects** dialog box opens, as in AutoCAD 2027: the tolerance, the properties ignored when comparing and the combine options. **OK** cleans up. In scripts (and with -OVERKILL) the prompt shows the current settings; change the ones you want and press Enter (or choose Done) to clean up.

```
Current settings: Tolerance=…, Ignore=…, Optimize polylines=…, Combine partial overlap=…, Co
mbine end-to-end=…. Enter an option to change [Done/Ignore/tOlerance/optimize Polylines/comb
ine parTial overlap/combine End-to-end] <done>:
```

4. When it finishes, the command line reports how many duplicates and how many overlapping objects or segments were deleted.

Options

- **tOlerance** (O): distance below which two points are considered equal.

```
Specify Tolerance <…>:
```

- **Ignore** (I): properties that do not count when comparing, separated by commas, or All or None. With none ignored, two objects on different layers are not duplicates.

```
Specify properties to ignore (…) or [All/None] <…>:
```

- **Polylines** (P): also looks for repeated segments within polylines.
- **parTial** (T): combines into one the aligned objects that partially overlap.
- **End-to-end** (E): combines aligned objects that only touch at their ends.
- **Done** (D): cleans up with the settings shown.

Notes

- The options start each time with their default values. Deleted objects come back with UNDO.
- The dialog box and the command line have the same options. AutoCAD's "Maintain associative objects" and "Do not break polylines" are not available.
- Only lines, arcs, circles, points and polylines are compared with tolerance; other types must be identical. A block with attributes is not recognized as a duplicate, and lines are not combined with polyline segments.

See also

-OVERKILL · ERASE · JOIN · PURGE · Selecting objects

PEDIT —Edit Polyline

Edits polylines: close, join, width, vertices, fit and spline.

- **Command:** `PEDIT` · **Global name:** `_PEDIT` · **Alias:** `PE`
- Ribbon: Home › Modify
- Menu: Modify

Edits polylines: closes or opens them, joins other objects to them, changes their width, edits their vertices and turns them into curves. It also converts lines and arcs into polylines.

Usage

1. Enter PEDIT (or PE).
2. Select a polyline, or choose Multiple to select several. If you select a line or an arc, you are offered to turn it into a polyline. With preselection: one object is edited directly and several go into Multiple.

```
Select polyline or [Multiple]:
Object selected is not a polyline. Do you want to turn it into one? [Yes/No] <Y>:
```

3. Choose options; the prompt repeats until you press Enter.

```
Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Reverse/Undo]:
```

Options

- **Close** (C) / **Open** (O): closes the polyline with a last segment or opens it.
- **Join** (J): adds lines, arcs and open polylines that touch its ends. In Multiple it also asks for a fuzz distance and the join type (Extend, Add or Both) to fill the gaps.
- **Width** (W): the same width for every segment.
- **Edit vertex** (E): walks the vertices with a cross (Next, Previous) and lets you Break, Insert, Move, Straighten, set the Tangent or the Width of the segment; eXit goes back. Only with one polyline.

```
Enter a vertex editing option [Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Widt
h/eXit] <N>:
```

- **Fit** (F): curve of tangent arcs through the vertices.
- **Spline** (S): curve that uses the vertices as its frame (SPLINETYPE, SPLINESEGS).
- **Decurve** (D): removes the fit or the curve and leaves straight segments.
- **Ltype gen** (L): continuous linetype generation (ON/OFF).
- **Reverse** (R): reverses the direction. **Undo** (U): undoes the last option.

Notes

- Decurve recovers the original frame only within the same PEDIT; afterwards it straightens the segments.
- Only lightweight polylines (LWPOLYLINE), the ones PLINE creates, can be edited.

See also

PLINE · JOIN · EXPLODE · SPLINEDIT · Editing

REVERSE —Reverse

Reverses the direction of lines, polylines and splines.

- **Command:** `REVERSE` · **Global name:** `_REVERSE`
- Ribbon: Home › Modify

Reverses the direction of lines, polylines (lightweight, 2D and 3D) and splines, like AutoCAD REVERSE: the start becomes the end. Useful, for example, so that the text of a linetype reads the right way up.

Usage

1. Enter REVERSE.
2. Select the objects and press Enter.

```
Select objects:
```

See also

PEDIT · SPLINEDIT

ROTATE — Rotate

Rotates objects around a base point.

- **Command:** `ROTATE` · **Global name:** `_ROTATE` · **Alias:** `RO`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Rotates the selected objects around a base point. The angle is given with a point or typed in degrees.

Usage

1. Enter ROTATE (or RO) or click **Rotate** on the ribbon.
2. Select the objects and press Enter. If they were already selected before starting the command, they are used without asking.

```
Select objects:
```

3. Specify the base point: the center of rotation.

```
Specify base point:
```

4. Specify the angle. Typed, in degrees (positive is counterclockwise); with a click, the angle is that of the direction from the base point to the point. The preview follows the cursor.

```
Specify rotation angle or [Copy/Reference] <…>:
```

With **Copy** a copy is rotated and the original stays; with **Reference** you give the reference angle (a number or two points) and the new angle (Specify the reference angle <0>:), and the difference is applied.

Notes

- With Ortho (F8) or polar tracking (F10) the rotation with the cursor snaps to 90° or to the polar angles.
- With no command running you can rotate with grips: select the objects, click a grip (it becomes the base point) and press Space until the rotate mode.

```
Specify rotation angle or [Base point/Copy/eXit]:
```

- Texts, blocks and arcs update their angle; dimensions are regenerated and their text stays readable.
- The Copy and Reference options are the AutoCAD ones; in Reference, **Points** gives the angle with two points. Enter uses the last rotated angle (the value between < >).

See also

MOVE · SCALE · ALIGN · MIRROR · Selecting objects

SCALE —Scale

Enlarges or reduces objects from a base point.

- **Command:** `SCALE` · **Global name:** `_SCALE` · **Alias:** `SC`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Enlarges or reduces the selected objects from a base point, the only point that does not move.

Usage

1. Enter SCALE (or SC) or click **Scale** on the ribbon.
2. Select the objects and press Enter. If they were already selected before starting the command, they are used without asking.

```
Select objects:
```

3. Specify the base point.

```
Specify base point:
```

4. Specify the scale factor: typed (greater than 0; `2` doubles, `0.5` halves) or with a click, and then the factor is the distance from the base point to the point.

```
Specify scale factor or [Copy/Reference]:
```

With **Copy** a copy is scaled and the original stays; with **Reference** you give the reference length and the new one (Specify reference length <1.0000>:), and the factor is new / reference.

Notes

- The factor is the same in X and Y: circles stay circles and blocks change their scale.
- With no command running you can scale with grips: select the objects, click a grip (it becomes the base point) and press Space until the scale mode.

```
Specify scale factor or [Base point/Copy/eXit]:
```

- Scaling a table enlarges its rows and columns, but the height of its texts comes from its style and does not change.
- The Copy and Reference options are the AutoCAD ones; in Reference, **Points** gives the new length with two points.

See also

ROTATE · ALIGN · STRETCH · LENGTHEN · Selecting objects

SLICE —Slice

Slices 3D solids by a plane, keeping one side or both.

- **Command:** `SLICE` · **Global name:** `_SLICE` · **Alias:** `SL`
- Ribbon: Home › Modify

Slices one or more **3D solids** with a **plane**, keeping one side or both, like AutoCAD's SLICE. The cut is computed as a boolean against a half-space with the `truck` kernel; between polyhedra (planar faces) the result is **exact**.

Usage

1. Type SLICE (or CORTE) or click it in the ribbon.
2. Select the solids to slice and press Enter.

```
Select the solids to slice and press Enter:
```

3. Define the cutting plane: a keyword (XY, YZ or ZX, through the center of the solids) or three points.

```
Cutting plane [XY/YZ/ZX] or three points "x1,y1,z1 x2,y2,z2 x3,y3,z3":
```

4. Choose the side: click a point on the side you want to keep, or press Enter to keep both.

```
Side to keep: a point on that side, or Enter for [Both]:
```

Notes

- With three points, the plane is the one that contains them (their order sets the normal direction).
- Only **polyhedra** with planar faces are sliced; exact curved-face solids are not supported yet (a later phase).
- See also PRESSPULL, SUBTRACT and INTERSECT.

SPLINEDIT —Edit Spline

Edits splines: fit points, control vertices, close, reverse and convert to polyline.

- **Command:** `SPLINEDIT` · **Global name:** `_SPLINEDIT` · **Alias:** `SPE`
- Ribbon: Home › Modify
- Menu: Modify

Edits a spline, as in AutoCAD 2027: closes or opens it, joins other curves to it, changes its fit points or its control vertices, elevates its order, reverses its direction or converts it into a polyline.

Usage

1. Enter SPLINEDIT (or SPE).
2. Select the spline. If it was selected (on its own) before starting the command, it is used without asking.

```
Select spline:
```

3. Choose options; the prompt repeats until you choose eXit or press Enter.

```
Enter an option […/Join/…Edit vertex/convert to Polyline/Reverse/Undo/eXit] <eXit>:
```

Options

- **Close** (C) / **Open** (O): closes or opens the spline (the prompt offers the one that applies).
- **Join** (J): joins to the spline the selected open splines and lines that start or end at one of its ends (reversing them if needed), as in AutoCAD: the knots of the joined curve follow those of the spline and the joint keeps a multiplicity equal to the degree; joining a line makes the spline rational.

```
Select any open curves to join to the source:
```

- **Fit data** (F): only if the spline has fit points. Lets you Add, Delete and Move fit points, add a Kink (a corner at a point of the curve), change the end Tangents, the toLerance or Purge the fit data (only the control vertices remain). The curve is recalculated.

```
Enter a fit data option [Add/…/Delete/Kink/Move/Purge/Tangents/toLerance/eXit] <eXit>:
Specify point on the spline <exit>:
```

- **Edit vertex** (E): Add a control vertex without changing the shape, Delete or Move vertices, Elevate order (higher degree, same curve) and change their Weight. Editing the vertices removes the fit data.

```
Enter a vertex editing option [Add/Delete/Elevate order/Move/Weight/eXit] <eXit>:
Enter new order <…>:
```

- **Polyline** (P): replaces the spline with a polyline that approximates it.

```
Specify a precision <10>:
```

- **Reverse** (R): reverses the direction of the curve.
- **Undo** (U): undoes the last option. **eXit** (X): ends.

Notes

- When moving points you walk them with Next and Previous, or pick one with Select point.
- A kink leaves the spline without fit data (two fit splines joined at the corner); AutoCAD keeps it with them. Join does not join arcs, polylines or ellipses yet.
- The tolerance is saved, but the curve still goes through the fit points.
- With no command running, the spline grips can also be dragged.

See also

SPLINE · PEDIT · PLINE · Editing

STRETCH —Stretch

Stretches objects by moving the vertices inside a crossing window.

- **Command:** `STRETCH` · **Global name:** `_STRETCH` · **Alias:** `S`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Stretches objects by moving only the vertices and endpoints that lie inside a crossing window; the rest of each object stays where it was. Use it, for example, to lengthen a part or move a door without redrawing the walls.

Usage

1. Enter STRETCH (or S) or click **Stretch** on the ribbon.
2. Select the objects with a crossing window (dragged from right to left) that encloses the points to move. You can make several crossing windows. Press Enter.

```
Select objects to stretch by crossing-window:
```

3. Specify the base point and the second point, as in MOVE. The preview moves only the captured points.

```
Specify base point or [Displacement] <Displacement>:
Specify second point or <use first point as displacement>:
```

Notes

- The endpoints of lines, arcs (which keep their angle) and the vertices of polylines that fall inside a crossing window are moved. An object that lies completely inside a crossing window is moved as a whole.
- Elliptical arcs: the endpoints inside the window move and the arc keeps its shape (it rotates and scales about the other endpoint). A full ellipse, a circle, a text, a block or a point moves if its center or insertion point is inside, as in AutoCAD.
- Splines: the fit points inside the window move and the curve is refitted; without fit data, the control vertices.
- Dimensions: their definition points (extension line origins, dimension line and text) move and the dimension is measured again. An associative dimension stretched with its geometry follows it; stretched alone, it is disassociated.

- Multilines (MLINE): their vertices move. Hatches: the vertices of their boundary move; if the boundary object is stretched too, an associative hatch follows it.
- At least one crossing window is needed: if you only select with clicks, a message says so. Preselection is not used.
- With no command running, grips stretch point by point: select the object, click a grip (it turns red) and take the point to its new place. It is the first mode; Space switches to move, rotate, scale and mirror.

```
Specify stretch point or [Base point/Copy/eXit]:
```

- With the **CPolygon** option you draw a crossing polygon (Enter closes it; **Undo** removes the last point); it also selects what it touches. **Displacement** option as in MOVE.

See also

MOVE · LENGTHEN · SCALE · Selecting objects · Editing

SUBTRACT — Subtract

Subtracts some solids from another.

- **Command:** SUBTRACT · **Global name:** _SUBTRACT · **Alias:** SU
- Ribbon: Home › Modify

Subtracts some solids from another (boolean subtraction), like AutoCAD's SUBTRACT. Between polyhedra it is computed by an in-house BSP modeler (pure Rust), exact even with coplanar faces.

Usage

1. Type SUBTRACT or click it in the ribbon.
2. Select the **base** solid (to subtract from) and press Enter.

```
Select the solid to subtract from (or the base and then the ones to subtract) and press Ente
r:
```

3. Select the solids to **subtract** from it (one or more) and press Enter.

```
Select the solids to subtract and press Enter:
```

Shortcut: if in step 2 you select two or more solids and press Enter, the **first one selected** is the base and the rest are subtracted from it (with a window, the order is the drawing order: the oldest first). The same with a pre-selection.

The result keeps the base's properties (layer, color, linetype, material…), not the current ones.

Notes

- Between **polyhedra** (planar-faced solids) the result is **exact**, also with coplanar faces (a tool flush with the top, the bottom or a facade, a through hole) and when subtracting several tools at once (they are subtracted one by one): AutoCAD audits it with no errors and MASSPROP reports the exact volume. The result can be subtracted from again.
- A face with a hole (e.g. the top pierced by an opening) is stored split into several planar faces, so seam lines show; the geometry and the volume are exact.
- Solids with curved faces are not yet supported in booleans; they come in a later phase.
- See also UNION, INTERSECT, EXTRUDE and REVOLVE.

TEXTTOFRONT —Text to front

Brings texts, dimensions or leaders in front of other objects.

- **Command:** `TEXTTOFRONT` · **Global name:** `_TEXTTOFRONT`
- Ribbon: Home › Modify

Brings all texts, dimensions or leaders of the drawing to the front of the draw order, like AutoCAD TEXTTOFRONT, so that hatches, images and other objects do not hide them.

Usage

1. Enter TEXTTOFRONT.

```
Bring to front [Text/Dimensions/Leaders/All] <All>:
```

2. Choose Text, Dimensions, Leaders or All (Enter).

See also

DRAWORDER

TRIM —Trim

Trims objects at their intersections.

- **Command:** `TRIM` · **Global name:** `_TRIM` · **Alias:** `TR`
- Ribbon: Home › Modify
- Menu: Modify
- Toolbar: Modify

Trims the part of an object that lies between its intersections with other objects (the cutting edges) or between an intersection and its end.

Usage

1. Enter TRIM (or TR) or click **Trim** on the ribbon.
2. In **Quick mode** (the AutoCAD 2027 default: variable TRIMEXTENDMODE=1) you do not select the edges: every object in the drawing is a cutting edge. Click the part to remove from each object: the clicked piece is removed up to the nearest intersections. The prompt repeats to trim more.

```
Select object to trim or shift-select to extend or [cuTting edges/Crossing/mOde/Project/eRas
e]:
```

3. In **Standard mode** you first select the cutting edges (Enter = all objects).

```
Select cutting edges... Select objects or [mOde] <select all>:
```

4. Press Enter or Esc to finish.

Options

- **cuTting edges**: select the cutting edges instead of using every object.
- **Fence** (Standard mode) and **Crossing**: draw a fence; each object it crosses is trimmed where it crosses.
- **mOde**: switch between Quick and Standard.
- **Project**: None, Ucs or View (PROJMODE; stored: it does not change the result in 2D).
- **Edge** (Standard mode): Extend prolongs the edges (EDGEMODE) to trim against their extension.
- **eRase**: erases the objects you select.

- **Undo**: undoes the last trim.

Notes

- Lines, arcs, circles and polylines can be trimmed and serve as edges. Trimming a circle leaves an arc; trimming the inside of a line or polyline splits it into two objects.
- Hatches are trimmed too: the clicked piece is removed between the edges and the rest stays a single, non-associative hatch.
- Objects to trim are picked one at a time with a click on the part to remove. The previous selection is not used.
- With Shift held while clicking, it extends instead of trimming. To extend with the same edges use EXTEND.
- Ellipses (whole or arcs) and open splines are trimmed like other objects and also act as cutting edges. A closed spline is not trimmed.

See also

EXTEND · BREAK · LENGTHEN · FILLET · Editing

TXTEXP —Explode text

Turns texts into polylines with the shape of their letters.

- **Command:** `TXTEXP` · **Global name:** `_TXTEXP`
- Ribbon: Home › Modify

Turns texts into polylines with the shape of their letters, like the TXTEXP tool of the Express Tools. Useful to cut or engrave lettering, or so that a text looks the same on a computer that lacks its font.

Usage

1. Enter TXTEXP.
2. Select the texts and press Enter. If they were already selected, they are converted without asking.

```
Select objects:
```

Notes

- Texts (TEXT), multiline texts (MTEXT) and attribute definitions (with their tag) are converted. Other selected objects do not change.
- With TrueType fonts the closed outlines of each letter come out; with SHX fonts, their strokes.
- The polylines keep the layer and properties of the text, and the text is erased.

See also

BURST · EXPLODE · STYLE

UNGROUP —Ungroup

Removes a group; its objects stay in the drawing.

- **Command:** `UNGROUP` · **Global name:** `_UNGROUP`
- Ribbon: Home › Groups
- Menu: Modify

Removes a group. Its objects stay in the drawing, but they are no longer selected together.

Usage

1. Enter UNGROUP.
2. Select a member of the group, or choose Name and type the name of the group (? lists them). The group is removed at once.

```
Select group or [Name]:
Enter a group name or [?]:
```

Preselection is quicker: click a member (the whole group is selected) and start UNGROUP. If the selection belongs to several groups, all of them are removed.

Options

- **Name** (N): picks the group by its name instead of selecting it.

Notes

- If the selected object belongs to no group, a message says so.
- It can be undone with UNDO.

See also

GROUP · GROUPEDIT · EXPLODE

UNION —Union

Merges several solids into one.

- **Command:** `UNION` · **Global name:** `_UNION` · **Alias:** `UNI`
- Ribbon: Home › Modify

Merges several 3D solids into one (boolean union), like AutoCAD's UNION. Between polyhedra it is computed by an in-house BSP modeler (pure Rust), exact even with coplanar faces; the result is stored as an ACIS 3DSOLID.

Usage

1. Type UNION or click it in the ribbon.
2. Select two or more solids and press Enter.

```
Select the solids and press Enter:
```

The selected solids are replaced by the union solid. The result keeps the properties (layer, color, material…) of the first solid selected.

Notes

- Between **polyhedra** (planar-faced solids) the result is **exact**: AutoCAD audits it with no errors and MASSPROP reports the exact volume, also with coplanar faces (walls flush at the top and bottom, solids touching along a face).
- Solids with curved faces (cylinders, spheres…) are not yet supported in booleans; they come in a later phase.
- See also SUBTRACT, INTERSECT, EXTRUDE and REVOLVE.

Annotation

-DIMINSPECT —Inspection dimension (command line)

Adds or removes the inspection data of dimensions at the command line.

- **Command:** `-DIMINSPECT` · **Global name:** `_-DIMINSPECT`

Adds or removes the inspection data of dimensions asking at the command line, like AutoCAD 2027 -DIMINSPECT.

Use

1. Type -DIMINSPECT.
2. Choose Add or Remove.

```
Add Inspection data or [Remove] <Add>:
```

3. Select the dimensions and press Enter.

```
Select dimensions:
```

4. When adding: shape, label and rate.

```
Enter shape option [Round/Angular/None] <Round>:
Enter label value or <None>:
Enter inspection rate <100%>:
```

See also

DIMINSPECT

-SCALELISTEDIT —Scale List (command line)

Edits the scale list from the command line.

- **Command:** `-SCALELISTEDIT` · **Global name:** `_-SCALELISTEDIT`

Command-line version of SCALELISTEDIT: edits the list of annotation scales of the drawing. It is the one used in scripts.

Usage

1. Enter -SCALELISTEDIT.
2. Choose an option (Enter: Add).

```
Enter option [?/Add/Delete/Reset/Exit] <Add>:
```

Options

- **?**: lists the scales.
- **Add** (A): asks for the name and the paper:drawing ratio (`1:50`).

```
Enter scale ratio (paper:drawing):
```

- **Delete** (D): deletes a scale by its name.
- **Reset** (R): goes back to the default list.

```
Reset scale list to default entries? [Yes/No] <No>:
```

- **Exit** (E): ends the command.

Notes

- It does the same as SCALELISTEDIT, which in LinceCAD has no dialog box either.
- The scales in the list are the ones accepted by CANNOSCALE and by annotative objects.

See also

SCALELISTEDIT · OBJECTSCALE

ACTUALIZARDATOS —Update data links

Re-reads the file of the linked tables.

- **Command:** `ACTUALIZARDATOS` · **Global name:** `_DATALINKUPDATE` · **Alias:** `DLU`
- Ribbon: Annotate › Tables

Re-reads the source file of each linked table (DATALINK) in the drawing and refreshes its contents, keeping the table position, style and layer; or, the other way round, writes what the tables hold to the file (AutoCAD's Write data link).

Use

1. Type DATALINKUPDATE or click its button.
2. Choose **Update data link** (Enter) to read the files or **Write data link** to write to them.

```
Select an option [Update data link/Write data link] <Update data link>:
```

3. It reports how many tables were updated (or written) and how many failed (for example, if the file is missing).

Notes

- It rebuilds each table with the current data from the file: if the number of rows or columns changed, the table adjusts.
- It only affects tables created with DATALINK; the others are left unchanged.
- **Write data link** only writes to CSV files: the whole file or, if the link has a range, only the cells of the range, keeping the rest of the file. An Excel workbook is not written (LinceCAD reads it with a read-only library, and rewriting it would lose its formats and formulas): an error says so.

See also

DATALINK · TABLE

ANNORESET —Reset annotation scales

Moves all scale representations of annotative objects to the position of the current one.

- **Command:** `ANNORESET` · **Global name:** `_ANNORESET`
- Ribbon: Annotate › Annotation Scaling

Moves all scale representations of annotative objects to the position of the current one, like AutoCAD 2027 ANNORESET.

Use

1. Type ANNORESET.
2. Select the annotative objects and press Enter.

```
Select objects:
```

Notes

- An annotative object can be at a different place in each scale (each representation is moved separately); ANNORESET puts them all back where the one of the current annotation scale is.

See also

ANNOUPDATE · OBJECTSCALE · SCALELISTEDIT

ANNOUPDATE —Update annotative objects

Updates annotative objects to the properties of their style.

- **Command:** ANNOUPDATE · **Global name:** _ANNOUPDATE
- Ribbon: Annotate › Annotation Scaling

Updates annotative objects to the properties of their style, like AutoCAD 2027 ANNOUPDATE.

Use

1. Type ANNOUPDATE.
2. Select the objects and press Enter.

```
Select objects:
```

Notes

- A text becomes annotative (or stops being so) as its text style and, if the style has a fixed height, it takes that height (the paper height, for annotative ones). A dimension takes the annotative property of its dimension style and is regenerated.
- Multileaders, blocks and hatches are not changed.

See also

ANNORESET · OBJECTSCALE · STYLE

ARCTEXT —Arc-aligned text

Writes text following an arc.

- **Command:** ARCTEXT · **Global name:** _ARCTEXT
- Ribbon: Annotate › Text

Writes text that follows an arc: each character is placed rotated tangent to the arc.

Use

1. Type ARCTEXT or click its button.
2. Select the arc to use as a guide.

```
Select the arc:
```

3. The **Arc-aligned text** dialog box opens: type the text and the height, choose the options and click **OK**. You can also go on at the command line: the height (Enter accepts the last one used) and the text.

```
Text height <…>:
Text:
```

The dialog box

- **Offset from arc:** distance between the arc and the text.
- **Character spacing:** extra space between each two characters.
- **Alignment:** Fit spreads the characters over the whole arc; Left, Right and Center place it at the start, at the end or in the middle, as it reads.
- **Position:** Convex side (outside the arc) or Concave side (inside).
- **Direction:** Outward from the center: the tops of the letters face outward and it reads clockwise; Inward to the center: the tops face the center and it reads counterclockwise.

Notes

- From the command line the text is centered on the convex side of the arc and reads left to right. Each character is a text entity (TEXT) rotated and centered, with the current text style (STYLE).
- Plain text entities are written (not a proxy object), so any CAD reads them and they audit clean in AutoCAD.
- For straight text, use TEXT or MTEXT.
- In scripts and with FILEDIA set to 0 the dialog box does not open.

See also

TEXT · MTEXT · TEXTFIT · ARC

CENTERLINE —Centerline

Draws an associative centerline between two lines.

- **Command:** `CENTERLINE` · **Global name:** `_CENTERLINE` · **Alias:** `CL`
- Ribbon: Home › Draw

Draws a centerline between two lines, as in AutoCAD 2027: it runs along the bisector of the angle they form (or halfway, if they are parallel), between the ends of both, and extends CENTEREXE on each side. It is associative: if one of the lines changes, the centerline is rebuilt.

Usage

1. Enter CENTERLINE (alias CL) or click the button.
2. Select the first line.

```
Select first line or [Layer]:
```

3. Select the second line.

```
Select second line:
```

Options

- **Layer**: layer of the new centerlines (`.`, the current one); it is the CENTERLAYER variable.

```
Enter layer name or select an object to specify the layer for center marks or . to use curre
nt [?/Exit] <"…">:
```

Notes

- It is saved as AutoCAD does: a reference to an anonymous block with the line, with the CENTERLTYPE linetype (CENTER2 if empty).

See also

CENTERMARK · LINE

CENTERMARK —Center Mark

Adds an associative center mark to circles and arcs.

- **Command:** `CENTERMARK` · **Global name:** `_CENTERMARK` · **Alias:** `CM`
- Ribbon: Home › Draw

Adds center marks to circles and arcs, as in AutoCAD 2027: a cross at the center and extension lines that go beyond the circle. The mark is associative: if the circle or arc changes, the mark is rebuilt.

Usage

1. Enter CENTERMARK (alias CM) or click the button.
2. Select the circles and arcs one by one and press Enter to finish.

```
Select circle or arc to add centermark or [Layer]:
```

Options

- **Layer**: layer of the new marks (`.`, the current one); it is the CENTERLAYER variable.

```
Enter layer name or select an object to specify the layer for center marks or . to use curre
nt [?/Exit] <"…">:
```

Notes

- It is saved as AutoCAD does: a reference to an anonymous block with the cross and the extensions, with the CENTERLTYPE linetype (CENTER2 if empty) and its scale CENTERLTSCALE.
- Shape: CENTERCROSSSIZE (cross), CENTERCROSSGAP (gap), CENTEREXE (overshoot) and CENTERMARKEXE (with or without extensions).
- If another object is selected:

```
The selected object is not an arc or a circle.
```

See also

CENTERLINE · CIRCLE · ARC

DATALINK —Data link

Creates a table linked to an Excel or CSV file.

- **Command:** `DATALINK` · **Global name:** `_DATALINK` · **Alias:** `DL`
- Ribbon: Annotate › Tables

Creates a table linked to an external Excel (.xlsx, .xls, .ods) or CSV file. The table stores the link (path, sheet and range) so it can be refreshed with ACTUALIZARDATOS.

Use

1. Type DATALINK or click its button. The **Data Link Manager** opens with the links of the drawing (their **Name**, **File**, **Sheet** and **Range**): - **Edit...**: changes the name, file, sheet or range of the chosen link; on OK the table is read again from the new source. - **Update**: reads the table of the chosen link again (like ACTUALIZARDATOS). - **Unlink**: removes the link; the table keeps its data. - **Create a new data link...**: goes on with the steps

below to create a linked table. In scripts and with FILEDIA set to 0 the manager does not open: it starts with the file.

2. Type the path of the data file.

```
Data file (.xlsx, .xls, .ods or .csv):
```

3. If it is an Excel workbook with several sheets, choose the sheet (Enter = the first one).

```
Sheet […]:
```

4. Enter the cell range (for example `A1:C10`) or press Enter for the whole sheet.

```
Range (e.g. A1:C10), or Enter for the whole sheet:
```

5. Specify the table insertion point.

```
Specify the insertion point:
```

Notes

- The first row read becomes the table header.
- The table is a normal table (TABLE) with the link stored as extended data: it audits clean in AutoCAD and any CAD reads it. AutoCAD shows the written values but does not refresh it by itself; use ACTUALIZARDATOS in Lince.
- Excel is read with the calamine library (read-only); date values may appear as a serial number.
- A new link takes the file name as its name; change it with **Edit...**.

See also

ACTUALIZARDATOS · TABLE · TABLEEXPORT

DIM —Dimension

Dimensions what you select or the points you specify (the "smart" dimension).

- **Command:** `DIM` · **Global name:** `_DIM`
- Ribbon: Home › Annotation
- Ribbon: Annotate › Dimensions

Dimensions what you select or the points you specify, like AutoCAD 2027 DIM (the "smart" dimension). It keeps dimensioning until you press Enter.

Use

1. Type DIM.
2. Select an object or specify the first extension line origin.

```
Select objects or specify first extension line origin or [Angular/Baseline/Continue/Ordinate
/aliGn/Distribute/Layer/Undo]:
```

3. Follow the prompts of the corresponding dimension.

Which dimension is made

- **A point**: a linear dimension, like DIMLINEAR: second origin and location (horizontal or vertical depending on where you place it).
- **A line**: its linear dimension.
- **A circle**: its diameter, like DIMDIAMETER. **An arc**: its radius, like DIMRADIUS.

Options

- **Angular**, **Baseline**, **Continue** and **Ordinate**: like DIMANGULAR, DIMBASELINE, DIMCONTINUE and DIMORDINATE.
- **aliGn**: select a base dimension and those that must be aligned with it.
- **Distribute**: **Equal** spreads them at the same distance between the base and the farthest one; **Offset**, at the distance you enter.
- **Layer**: the layer of new dimensions (. goes back to the current one).
- **Undo**: erases the last dimension made.

Notes

- It does not choose the dimension type from where you move the cursor, as AutoCAD's preview does.

See also

DIMLINEAR · DIMALIGNED · DIMSPACE · DIMSTYLE

DIMALIGNED —Aligned

Creates a dimension parallel to the points.

- **Command:** `DIMALIGNED` · **Global name:** `_DIMALIGNED` · **Alias:** `DAL`, `DIMALI`
- Ribbon: Annotate › Dimensions
- Menu: Dimension

Dimensions the true distance between two points, with the dimension line parallel to the segment that joins them, also when it is diagonal.

Usage

1. Enter DIMALIGNED (or DAL) or click **Aligned** on the ribbon.
2. Specify the first extension line origin, or press Enter to select an object.

```
Specify first extension line origin or <select object>:
```

3. Specify the second extension line origin.

```
Specify second extension line origin:
```

4. Specify how far from the points the dimension line goes.

```
Specify dimension line location or [Mtext/Text/Angle]:
```

Selecting an object

With Enter at the first step you select an object: the endpoints of a line, an arc or a polyline segment are the origins; for a circle, the diameter through the selected point.

```
Select object to dimension:
```

Notes

- The dimension uses the current dimension style (DIMSTYLE); change it with DIMSTYLE.
- To change the dimension text use DIMEDIT.
- DIMCONTINUE and DIMBASELINE can start from an aligned dimension.
- If its points lie on the geometry (endpoint, midpoint, center...), the dimension is associated with it and follows it when it is moved, stretched or scaled (DIMASSOC). With an annotative dimension style, the dimension is annotative.

- Location options, as in AutoCAD: **Mtext** (multiline editor, starting with <>), **Text** (one line; <> is the measurement) and **Angle** (text rotation). You can also change the text afterwards with DIMEDIT.

See also

DIMLINEAR · DIMEDIT · DIMSTYLE · Dimensions

DIMANGULAR —Angular

Dimensions the angle between two lines.

- **Command:** `DIMANGULAR` · **Global name:** `_DIMANGULAR` · **Alias:** `DAN`, `DIMANG`
- Ribbon: Annotate › Dimensions
- Menu: Dimension

Dimensions the angle between two non-parallel lines, or the angle defined by a vertex and two points, with a dimension arc. The text is in degrees, with the angular precision of the dimension style.

Usage

1. Enter DIMANGULAR (or DAN) or click **Angular** on the ribbon.
2. Select the first line, or press Enter to specify the angle with three points.

```
Select first line or <specify vertex>:
```

3. Select the second line. If it is parallel to the first one, a message is shown and another line is requested.

```
Select second line:
```

4. Move the cursor to choose which angle is dimensioned (depending on the quadrant it is in) and how far from the vertex the arc goes, and specify the point.

```
Specify dimension arc line location or [Mtext/Text/Angle]:
```

Three points

With Enter at the first step, the vertex and a point on each side of the angle are requested, then the location of the arc:

```
Specify angle vertex:
Specify first angle endpoint:
Specify second angle endpoint:
```

Notes

- **Quadrant** (Q), when placing the arc: specify a point of the sector to dimension; the arc can then go anywhere and the measurement is still that of the sector, as in AutoCAD.
- A selected arc is dimensioned directly (the angle between its endpoints from the center); for a circle, the angle goes from the selected point to the second point you specify.
- With two lines, the location accepts **Mtext**, **Text** (<> is the measurement) and **Angle** (text rotation), as in AutoCAD; the Quadrant option is not available.
- The AutoCAD variants that dimension an arc or a circle directly are not available; for the length of an arc use DIMARC.
- The dimension uses the current dimension style (DIMSTYLE); angular precision and zero suppression are on the **Primary Units** tab of DIMSTYLE.
- The dimension is associated with the lines or points it measures and follows them when they change (DIMASSOC).

See also

DIMLINEAR · DIMRADIUS · DIMSTYLE · Dimensions

DIMARC — Arc Length

Dimensions the length of an arc.

- **Command:** `DIMARC` · **Global name:** `_DIMARC` · **Alias:** `DAR`
- Ribbon: Annotate › Dimensions

Dimensions the length of an arc, measured along the curve. The dimension arc is concentric with the dimensioned arc and the text carries the arc length symbol (?).

Usage

1. Enter DIMARC (or DAR).
2. Select the arc with a click. Circles are not accepted.

```
Select arc:
```

3. Move the cursor to preview the dimension and specify where the dimension arc goes.

```
Specify arc length dimension location:
```

Notes

- The position of the symbol is set by the DIMARCSYM dimension variable, on the **Primary Units** tab of DIMSTYLE: before the text, above it or none.
- The dimension uses the current dimension style (DIMSTYLE) and is associated with the arc: if the arc changes, the measurement is updated.
- **Partial**: dimensions only the part between two points of the arc. **Leader**: adds a leader from the text to the arc (**No leader** removes it). **Mtext**, **Text** and **Angle** change the text and its rotation, as in the other dimensions (Mtext asks for it at the command line).

See also

DIMANGULAR · DIMSTYLE · Dimensions

DIMBASELINE — Baseline

Creates linear dimensions from the same baseline.

- **Command:** `DIMBASELINE` · **Global name:** `_DIMBASELINE` · **Alias:** `DBA`, `DIMBASE`
- Ribbon: Annotate › Dimensions

Creates several dimensions measured from the same origin. Each dimension line is offset from the previous one by the DIMDLI distance of the dimension style, so that they do not overlap.

Usage

1. Enter DIMBASELINE (or DBA).
2. The command starts from the last linear or aligned dimension in the drawing, from its first extension line. If there is none, select the base dimension.

```
Select base dimension:
```

3. Specify the second extension line origin. A dimension is created from the common origin to that point and the

prompt repeats.

```
Specify a second extension line origin or [Undo/Select] <Select>:
```

4. Press Enter to finish.

Options

- **Undo** (U): removes the last dimension created.
- **Select** (S): selects another base dimension. The common origin is the extension line closest to the point you click.

Notes

- Linear, aligned, angular (each arc DIMDLI farther from the vertex) and ordinate dimensions are accepted as the base.
- New dimensions use the current dimension style (DIMSTYLE).
- If its points lie on the geometry (endpoint, midpoint, center...), the dimension is associated with it and follows it when it is moved, stretched or scaled (DIMASSOC). With an annotative dimension style, the dimension is annotative.

See also

DIMCONTINUE · DIMLINEAR · DIMSTYLE · Dimensions

DIMBREAK —Dimension Break

Breaks dimension and extension lines where other objects cross them.

- **Command:** `DIMBREAK` · **Global name:** `_DIMBREAK` · **Alias:** `DBR`
- Ribbon: Annotate › Dimensions

Breaks the dimension and extension lines where other objects cross them, so the drawing reads better, or removes the breaks.

Usage

1. Enter DIMBREAK (or DBR).
2. Select the dimension, or choose Multiple to select several (Enter ends the selection).

```
Select dimension to add/remove break or [Multiple]:
```

3. Select the object that breaks the dimension (only that one will break it), press Enter (or choose Auto) to break it with everything that crosses it, or choose Remove to remove the breaks.

```
Select object to break dimension or [Auto/Remove] <Auto>:
```

4. The number of changed dimensions is reported.

Options

- **Multiple** (M): selects several dimensions at once.

```
Select dimensions:
```

- **Auto** (A): breaks the dimension at every crossing with other objects.
- **Remove** (R): removes the breaks from the selected dimensions.

Notes

- The breaks are redone every time the dimension is regenerated (when it is moved, stretched or its style

changes), with the crossings found at that moment.
- Lines, polylines, arcs, circles and splines break the dimension.
- With an object selected, the dimension only breaks where that object crosses it (remembered even if it moves).
- **Manual** (M), with a single dimension: specify two points on one of its lines; the part between them is removed.
- The break mark is specific to LinceCAD: AutoCAD shows the dimension broken, but does not recognize it as a dimension break and the break disappears if it edits the dimension.

See also

DIMSPACE · DIMEDIT · Dimensions

DIMCENTER —Center mark (dimension)

Draws the center mark of a circle or an arc according to DIMCEN.

- **Command:** `DIMCENTER` · **Global name:** `_DIMCENTER` · **Alias:** `DCE`
- Ribbon: Annotate › Dimensions

Draws the center mark of a circle or an arc with the size of DIMCEN, like AutoCAD 2027 DIMCENTER.

Use

1. Type DIMCENTER.
2. Select the circle or the arc.

```
Select arc or circle:
```

Notes

- With a positive DIMCEN a cross of that half size is drawn; with a negative DIMCEN, also the center lines, which start at twice |DIMCEN| from the center and end |DIMCEN| beyond the circle. With 0, nothing.
- The size is multiplied by DIMSCALE. They are loose lines on the current layer (not associated with the circle).
- For an associative mark, use MARCACENTRO.

See also

DIMRADIUS · DIMDIAMETER

DIMCONTINUE —Continue

Creates a chain of linear dimensions from the previous one.

- **Command:** `DIMCONTINUE` · **Global name:** `_DIMCONTINUE` · **Alias:** `DCO`, `DIMCONT`
- Ribbon: Annotate › Dimensions

Chains linear dimensions: each new dimension starts where the previous one ends, on the same dimension line. It is used for consecutive spans (openings in a façade, spacing of columns…).

Usage

1. Enter DIMCONTINUE (or DCO).
2. The command starts from the last linear or aligned dimension in the drawing. If there is none, select the dimension to continue.

```
Select continued dimension:
```

3. Specify the second extension line origin. A dimension is created from the previous one to that point and the

prompt repeats from the new dimension.

```
Specify a second extension line origin or [Undo/Select] <Select>:
```

4. Press Enter to finish. If no dimension has been created yet, Enter asks you to select another starting dimension.

Options

- **Undo** (U): removes the last dimension created and goes back to the previous one.
- **Select** (S): selects another starting dimension. It continues from the extension line closest to the point you click.

Notes

- Linear, aligned, angular (from the second side of the previous one, with its vertex and radius) and ordinate dimensions (with the leader end aligned with the previous one) can be continued.
- New dimensions use the current dimension style (DIMSTYLE).
- The continuation of an aligned dimension is created as a rotated linear dimension, with the same geometry.
- If its points lie on the geometry (endpoint, midpoint, center...), the dimension is associated with it and follows it when it is moved, stretched or scaled (DIMASSOC). With an annotative dimension style, the dimension is annotative.

See also

DIMBASELINE · DIMLINEAR · DIMALIGNED · Dimensions

DIMDIAMETER —Diameter

Dimensions the diameter of a circle or arc.

- **Command:** `DIMDIAMETER` · **Global name:** `_DIMDIAMETER` · **Alias:** `DDI`, `DIMDIA`
- Ribbon: Annotate › Dimensions
- Menu: Dimension

Dimensions the diameter of a circle or an arc. The text is prefixed with the diameter symbol (Ø50).

Usage

1. Enter DIMDIAMETER (or DDI) or click **Diameter** on the ribbon.
2. Select the circle or the arc with a click.

```
Select arc or circle:
```

3. Move the cursor and specify where the dimension goes. The dimension line crosses the center in the direction of the cursor.

```
Specify dimension line location or [Mtext/Text/Angle]:
```

Notes

- Circles, arcs and arc segments of polylines are dimensioned; for the radius use DIMRADIUS.
- The dimension uses the current dimension style (DIMSTYLE); change it with DIMSTYLE.
- Location options, as in AutoCAD: **Mtext**, **Text** (<> is the measurement) and **Angle** (text rotation). You can also change the text afterwards with DIMEDIT.
- If its points lie on the geometry (endpoint, midpoint, center...), the dimension is associated with it and follows it when it is moved, stretched or scaled (DIMASSOC). With an annotative dimension style, the dimension is annotative.

See also

DIMRADIUS · DIMANGULAR · DIMSTYLE · Dimensions

DIMDISASSOCIATE —Disassociate Dimensions

Removes associativity from the selected dimensions.

- **Command:** `DIMDISASSOCIATE` · **Global name:** `_DIMDISASSOCIATE` · **Alias:** `DDA`
- Ribbon: Annotate › Dimensions

Removes associativity from the selected dimensions: they stop following the geometry they measure and stay where they are.

Usage

1. Enter DIMDISASSOCIATE (or DDA).
2. Select the dimensions and press Enter. Anything that is not a dimension is discarded.

```
Select dimensions to disassociate:
```

3. The number of disassociated dimensions is reported.

Notes

- An associative dimension is also disassociated when it is moved without its geometry or when the object it is associated with is erased.
- To associate it again, use DIMREASSOCIATE.
- When DIMASSOC is not 2, new dimensions are not associated.

See also

DIMREASSOCIATE · DIMASSOC · Dimensions

DIMEDIT —Edit Dimension

Changes the text of dimensions or moves it back to its default position.

- **Command:** `DIMEDIT` · **Global name:** `_DIMEDIT` · **Alias:** `DED`, `DIMED`
- Ribbon: Annotate › Dimensions
- Menu: Dimension

Changes the text of one or more dimensions, or moves the text back to the position given by the style.

Usage

1. Enter DIMEDIT (or DED).
2. Choose the type of editing (Enter: Home).

```
Enter type of dimension editing [Home/New/Rotate/Oblique] <Home>:
```

3. With New, type the new text.

```
Enter dimension text <<>>:
```

4. Select the dimensions and press Enter. They are redrawn with the change.

```
Select objects:
```

Options

- **Home** (H): the text goes back to its default position if it had been moved (for example, with grips).
- **New** (N): replaces the text of the dimensions. <> stands for the measurement: <> `mm` writes the measurement followed by "mm", and <> alone (or Enter) shows the plain measurement again.
- **Rotate** (R): rotates the text of the dimensions by the angle you enter (0 follows the dimension line again).

```
Specify angle for dimension text:
```

- **Oblique** (O): slants the extension lines of linear and aligned dimensions; select the dimensions and then the angle (Enter: perpendicular).

```
Enter obliquing angle (press ENTER for none):
```

Notes

- Selected objects that are not dimensions are ignored.
- To move the text of a dimension, use DIMTEDIT.
- The text of a dimension can also be changed in the PROPERTIES palette.

See also

DIMLINEAR · DIMSTYLE · PROPERTIES · Dimensions

DIMINSPECT — Inspection dimension

Adds or removes the inspection data of dimensions.

- **Command:** `DIMINSPECT` · **Global name:** `_DIMINSPECT`
- Ribbon: Annotate › Dimensions

Adds the inspection data to dimensions (a frame with the label and the inspection rate) or removes it, like AutoCAD 2027 DIMINSPECT.

Use

1. Type DIMINSPECT.
2. Choose Add or Remove.

```
Add Inspection data or [Remove] <Add>:
```

3. Select the dimensions and press Enter.
4. When adding, choose the frame shape, the label (Enter: none) and the rate (Enter: 100%).

```
Enter shape option [Round/Angular/None] <Round>:
```

Notes

- In LinceCAD it asks at the command line, like -DIMINSPECT; there is no dialog as in AutoCAD.
- The data is stored as in AutoCAD (`ACAD_DSTYLE_DIMINSPECT` extended data); when removed, the dimension stops being an inspection one and keeps the rest.
- The frame is drawn in the dimension block: label on the left, measurement in the center and rate on the right.

See also

-DIMINSPECT · DIMSTYLE · TOLERANCE

DIMJOGGED —Jogged

Creates jogged radius dimensions for large arcs and circles.

- **Command:** `DIMJOGGED` · **Global name:** `_DIMJOGGED` · **Alias:** `DJO`, `JOG`
- Ribbon: Annotate › Dimensions

Dimensions the radius of a large arc or circle whose center lies outside the drawing or far from the visible area. The dimension line does not start at the real center but at an override center you choose, and it has a jog (a zigzag break) showing that it is not to scale.

Usage

1. Enter DIMJOGGED (or DJO, JOG).
2. Select the arc or the circle with a click.

```
Select arc or circle:
```

3. Specify the override center: the point the dimension line starts from.

```
Specify center location override:
```

4. Specify where the dimension line goes (and the arrow on the curve).

```
Specify dimension line location:
```

5. Specify where the jog goes, between the override center and the arrow.

```
Specify jog location:
```

Notes

- The jog angle is the DIMJOGANG dimension variable of the style (45° if it has no value).
- The text is prefixed with the letter R, like that of a normal radius dimension, and measures the real radius.
- The dimension uses the current dimension style (DIMSTYLE) and is associated with the center of the arc or circle.
- When specifying the dimension line location, **Mtext**, **Text** and **Angle** are available (Mtext asks for the text at the command line).

See also

DIMRADIUS · DIMJOGLINE · DIMSTYLE · Dimensions

DIMJOGLINE —Dimension, Jogged Linear

Adds or removes a jog line on a linear or aligned dimension.

- **Command:** `DIMJOGLINE` · **Global name:** `_DIMJOGLINE` · **Alias:** `DJL`
- Ribbon: Annotate › Dimensions

Adds a jog (a zigzag break) to the line of a linear or aligned dimension, to show that the measurement does not match the drawn length, or removes it.

Usage

1. Enter DIMJOGLINE (or DJL).
2. Select the linear or aligned dimension, or choose Remove.

```
Select dimension to add jog or [Remove]:
```

3. Specify where the jog goes on the dimension line, or press Enter to place it between the text and the first arrow.

```
Specify jog location (or press ENTER):
```

Options

- **Remove** (R): select the dimensions whose jog you want to remove and press Enter.

```
Select jogs to remove:
```

Notes

- A dimension has only one jog: if it already had one, the new one replaces it.
- The jog is saved in the dimension the way AutoCAD saves it (ACAD_DSTYLE_DIMJAG_POSITION extended data), so AutoCAD sees it and keeps it.
- Only linear and aligned dimensions are accepted.

See also

DIMJOGGED · DIMLINEAR · DIMALIGNED · Dimensions

DIMLINEAR —Linear

Creates a horizontal or vertical dimension.

- **Command:** `DIMLINEAR` · **Global name:** `_DIMLINEAR` · **Alias:** `DLI`, `DIMLIN`, `DIMHORIZONTAL`, `DIMROTATED`, `DIMVERTICAL`
- Ribbon: Home › Annotation
- Ribbon: Annotate › Dimensions
- Menu: Dimension

Dimensions the horizontal or vertical distance between two points. The dimension is horizontal or vertical depending on where you drag the dimension line: above or below the points it measures the X distance; to one side, the Y distance.

Usage

1. Enter DIMLINEAR (or DLI) or click **Linear** on the ribbon.
2. Specify the first extension line origin, or press Enter to select an object.

```
Specify first extension line origin or <select object>:
```

3. Specify the second extension line origin.

```
Specify second extension line origin:
```

4. Move the cursor to preview the dimension and specify where the dimension line goes.

```
Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:
```

Selecting an object

With Enter at the first step the object is requested: the endpoints of the selected line, arc or polyline segment are the two origins; for a circle the horizontal or vertical diameter is dimensioned, depending on where the dimension line goes.

```
Select object to dimension:
```

Notes

- Use object snaps (endpoint, intersection…) so that the origins fall exactly on the geometry.
- The dimension uses the current dimension style (DIMSTYLE): arrows, text height and placement, precision and units. Change it with DIMSTYLE.
- The measurement is written automatically; to show another text use DIMEDIT or the PROPERTIES palette.
- Location options, as in AutoCAD: **Mtext** opens the multiline text editor (it starts with <>, the measurement), **Text** asks for one line of text (<> is the measurement), **Angle** rotates the text, and **Horizontal**, **Vertical** and **Rotated** force the dimension line orientation (Rotated asks for its angle). Without them the orientation comes from the position of the dimension line. For a dimension parallel to two diagonal points, use DIMALIGNED.
- If its points lie on the geometry (endpoint, midpoint, center...), the dimension is associated with it and follows it when it is moved, stretched or scaled (DIMASSOC). With an annotative dimension style, the dimension is annotative.

See also

DIMALIGNED · DIMCONTINUE · DIMBASELINE · DIMSTYLE · Dimensions

DIMORDINATE —Ordinate

Creates an ordinate dimension.

- **Command:** `DIMORDINATE` · **Global name:** `_DIMORDINATE` · **Alias:** `DOR`, `DIMORD`
- Ribbon: Annotate › Dimensions

Dimensions the X or Y coordinate of a point relative to the drawing origin, with a leader from the point to the text. It is the usual dimension in machining or setting-out drawings.

Usage

1. Enter DIMORDINATE (or DOR).
2. Specify the point to dimension (the feature).

```
Specify feature location:
```

3. Specify the leader endpoint. If the leader runs mostly vertically, the X is dimensioned; if it runs horizontally, the Y.

```
Specify leader endpoint or [Xdatum/Ydatum/Mtext/Text/Angle]:
```

Options

- **Xdatum** (X): dimensions the X of the point even if the leader runs horizontally.
- **Ydatum** (Y): dimensions the Y of the point.

Notes

- The measurement is the coordinate in the world coordinate system.
- If the endpoint is not aligned with the point, the leader has a diagonal jog, as in AutoCAD.
- The dimension uses the current dimension style (DIMSTYLE).
- If its points lie on the geometry (endpoint, midpoint, center...), the dimension is associated with it and follows it when it is moved, stretched or scaled (DIMASSOC). With an annotative dimension style, the dimension is annotative.

See also

DIMLINEAR · DIMSTYLE · Coordinates · Dimensions

DIMOVERRIDE —Override dimension variables

Changes style variables in the selected dimensions without changing their style.

- **Command:** `DIMOVERRIDE` · **Global name:** `_DIMOVERRIDE` · **Alias:** `DOV`, `DIMOVER`
- Ribbon: Annotate › Dimensions

Changes dimension style variables in the selected dimensions without changing their style, like AutoCAD 2027 DIMOVERRIDE.

Use

1. Type DIMOVERRIDE.
2. Enter a dimension variable (DIMTXT, DIMASZ…) and its new value; repeat with others and press Enter.

```
Enter dimension variable name to override or [Clear overrides]:
```

3. Select the dimensions and press Enter.

Options

- **Clear overrides**: removes all overrides from the selected dimensions (they go back to their style).

Notes

- Overrides are stored in the dimension as in AutoCAD (`ACAD` extended data, `DSTYLE`) and show when it is opened in AutoCAD.
- It works for the numeric dimension variables (sizes, precision, fit, tolerances, alternate units); not for DIMPOST, the arrow blocks or the colors.

See also

DIMSTYLE · MATCHPROP

DIMRADIUS —Radius

Dimensions the radius of a circle or arc.

- **Command:** `DIMRADIUS` · **Global name:** `_DIMRADIUS` · **Alias:** `DRA`, `DIMRAD`
- Ribbon: Annotate › Dimensions
- Menu: Dimension

Dimensions the radius of a circle or an arc. The text is prefixed with the letter R (R25).

Usage

1. Enter DIMRADIUS (or DRA) or click **Radius** on the ribbon.
2. Select the circle or the arc with a click.

```
Select arc or circle:
```

3. Move the cursor around the circle and specify where the dimension goes. The dimension line starts at the center, in the direction of the cursor.

```
Specify dimension line location or [Mtext/Text/Angle]:
```

Notes

- Circles, arcs and arc segments of polylines are dimensioned; for the diameter use DIMDIAMETER.

- The dimension uses the current dimension style (DIMSTYLE); change it with DIMSTYLE.
- Location options, as in AutoCAD: **Mtext**, **Text** (<> is the measurement) and **Angle** (text rotation). You can also change the text afterwards with DIMEDIT.
- If its points lie on the geometry (endpoint, midpoint, center...), the dimension is associated with it and follows it when it is moved, stretched or scaled (DIMASSOC). With an annotative dimension style, the dimension is annotative.

See also

DIMDIAMETER · DIMANGULAR · DIMSTYLE · Dimensions

DIMREASSOCIATE — Reassociate Dimensions

Associates the selected dimensions to geometry.

- **Command:** `DIMREASSOCIATE` · **Global name:** `_DIMREASSOCIATE` · **Alias:** `DRE`
- Ribbon: Annotate › Dimensions

Associates dimensions with the geometry again: for each selected dimension you specify its definition points once more and the dimension becomes linked to the objects found there. From then on the dimension follows those objects when they are moved, stretched, scaled or edited with grips.

Usage

1. Enter DIMREASSOCIATE (or DRE).
2. Select the dimensions and press Enter. Anything that is not a dimension is discarded.

```
Select dimensions to reassociate:
```

3. The points of each dimension are requested one by one. Specify the point with an object snap (endpoint, midpoint, center, quadrant or node) or press Enter to keep the current one.

```
Specify definition point … or press Enter to skip:
```

4. At the end, the number of dimensions that became associative is reported.

Points requested

Dimension	Points
Linear and aligned	The two extension line origins
Arc length	The two endpoints of the arc
Three-point angular	The vertex and the two points
Radius and diameter	The center
Ordinate	The feature point

Notes

- The dimension is associated only if DIMASSOC is 2.
- The association is recognized because the point coincides with a key point of the object (endpoint, midpoint, center, quadrant, node or polyline vertex). A point at an intersection or a nearest point does not associate it.
- New dimensions are already associated when their points lie on the geometry; this command is for dimensions that were disassociated (moved on their own, their object erased) or for those of older drawings.
- Associations are saved as in AutoCAD (DIMASSOC object), and AutoCAD keeps them.

See also

DIMDISASSOCIATE · DIMASSOC · Dimensions

DIMSPACE —Adjust Space

Adjusts the spacing between parallel linear dimensions.

- **Command:** `DIMSPACE` · **Global name:** `_DIMSPACE` · **Alias:** `DSP`
- Ribbon: Annotate › Dimensions

Makes the spacing between parallel linear dimensions equal: it places them at the same distance from one another starting from a base dimension, or aligns them all on the same line.

Usage

1. Enter DIMSPACE (or DSP).
2. Select the base dimension, the one that does not move.

```
Select base dimension:
```

3. Select the dimensions to space and press Enter.

```
Select dimensions to space:
```

4. Enter the spacing, or press Enter for the automatic one.

```
Enter value or [Auto] <Auto>:
```

Options

- **Auto** (A): the spacing is twice the text height of the base dimension's style.
- A number: the distance between consecutive dimension lines. With **0** the dimensions are aligned with the base.

Notes

- Only linear and aligned dimensions parallel to the base are moved; the others are discarded.
- They are sorted by their distance to the base and placed, in that order, towards the side of the base's dimension line.
- The text of the moved dimensions returns to its default position.

See also

DIMBASELINE · DIMBREAK · Dimensions

DIMSTYLE —Dimension Style

Creates and modifies dimension styles.

- **Command:** `DIMSTYLE` · **Global name:** `_DIMSTYLE` · **Alias:** `D`, `DST`, `DIMSTY`, `DDIM`
- Ribbon: Annotate › Dimensions
- Menu: Format
- Menu: Dimension

Manages dimension styles: creates, modifies, renames and deletes styles, and sets the current one. The style sets the look of dimensions: extension lines, arrows, text, scale and number format.

Usage

1. Enter DIMSTYLE (or D, DST) or click **Dimension Style** on the ribbon. The **Dimension Style Manager** opens.
2. Choose a style in the **Styles:** list (styles used by dimensions show "(in use)"). Double-click makes it current.
3. Click one of the buttons.

Buttons

- **Set Current**: new dimensions will use the selected style.
- **New...**: opens **Create New Dimension Style**; type the name in **New Style Name:**, choose in **Start With:** the style to copy and click **Continue**.
- **Modify...**: opens **Modify Dimension Style**, with the tabs below.
- **Rename**: changes the name, also in the dimensions that use it. Standard cannot be renamed.
- **Delete**: deletes the style unless it is Standard, current or used by a dimension.

Modify Dimension Style

- **Lines**: Extend beyond dim lines, Offset from origin and Dimension line extension with ticks (DIMDLE).
- **Symbols and Arrows**: Arrow size, Tick size instead of arrowheads (DIMTSZ; 0: arrowheads), A different arrowhead at each end (DIMSAH) and the arrowheads: **Arrowheads** (or **First** and **Second** with DIMSAH) and **Leader**. They are AutoCAD's 20 (Closed filled, Closed blank, Closed, Dot, Architectural tick, Oblique, Open, Origin indicator, Origin indicator 2, Right angle, Open 30, Dot small, Dot blank, Small, Box blank, Box filled, Datum triangle, Datum triangle filled, Integral and None), drawn the way AutoCAD draws them: `_ArchTick`, `_Dot`…blocks added to the drawing (DIMBLK, DIMBLK1, DIMBLK2 and DIMLDRBLK). Dimensions from AutoCAD DWG files keep their arrowheads when edited.
- **Text**: Text height, Offset from dim line, text placement (Centered or Above) and **Text style**.
- **Fit**: what moves out when the text and the arrowheads do not fit between the extension lines (If text and arrows don't fit, move out (DIMATFIT): Both text and arrows, Arrows first, Text first or Either text or arrows (best fit)), Text always between extension lines (DIMTIX), Suppress arrowheads that don't fit (DIMSOXD), Dimension line between extension lines (DIMTOFL) and When the text is moved (DIMTMOVE) (Beside the dimension line, Over dimension line, with leader or Over dimension line, without leader). As in AutoCAD, the text width is measured with its font; what moves out goes beyond the second extension line, with the dimension line extended. Also Overall scale, which multiplies all sizes, and **Annotative**: new dimensions with the style are annotative and, instead of the overall scale, take the current annotation scale (CANNOSCALE): at 1:50 the sizes are multiplied by 50.
- **Primary Units**: precision, decimal separator, linear scale factor, round off, zero suppression (Leading and Trailing), angular precision and **Prefix and suffix (<> = measurement)**. Also the measurement tolerances: Tolerances (DIMTOL, 0/1), Limits (DIMLIM, 0/1), Upper value (DIMTP) and Lower value (DIMTM). And, for arc length dimensions, Arc length symbol (0: preceding, 1: above, 2: none).
- **Alternate Units**: Display alternate units (DIMALT) adds the measurement in other units in brackets (`40 [1.575]`), with Alternate units multiplier (DIMALTF) (25.4 from inches to millimetres; 0.03937 the other way), Alternate units decimal places (DIMALTD), Alternate units round-off (DIMALTRND), Alternate units zero suppression (DIMALTZ), Alternate tolerance decimal places (DIMALTTD) and **Alternate prefix and suffix ([] is the value)** (DIMAPOST). Unit format (DIMALTU) chooses the format as AutoCAD does: Scientific, Decimal, Engineering (`1'-0.5000"`), Architectural stacked and Architectural (`1'-0 1/2"`), Fractional stacked and Fractional (`12 1/2`) or Windows Desktop. In the feet and fraction formats the unit is the inch, Alternate units decimal places (DIMALTD) is the fraction precision (1/2, 1/4…1/256) and Alternate units zero suppression (DIMALTZ) says whether zero feet and zero inches are written.

On the right, the preview draws a linear and a radius dimension with the style as it is being edited. Click **OK**: the dimensions that use the style are redrawn.

Notes

- The current style is stored in DIMSTYLE, and the overall scale in DIMSCALE.

- Hovering over each label shows the name of its variable (DIMTXT, DIMASZ…).
- In the preview all arrowheads are drawn as closed filled ones and the text with the program font.
- The text and arrowhead fit is that of linear and aligned dimensions (angular and radius dimensions keep their own placement). Alternate unit fractions are always stacked horizontally (no DIMFRAC).
- There is no Compare nor style overrides in this dialog box: overrides are set with DIMOVERRIDE.
- Measurement tolerances (± or limits) are not geometric tolerances: the feature control frame is created with TOLERANCE.

See also

DIMLINEAR · STYLE · DIMEDIT · Dimensions

DIMTEDIT —Edit dimension text

Moves or rotates the text of a dimension.

- **Command:** `DIMTEDIT` · **Global name:** `_DIMTEDIT` · **Alias:** `DIMTED`
- Ribbon: Annotate › Dimensions

Moves or rotates the text of a dimension, like AutoCAD 2027 DIMTEDIT.

Use

1. Type DIMTEDIT.
2. Select the dimension.

```
Select dimension:
```

3. Specify the new text position or choose an option.

```
Specify new location for dimension text or [Left/Right/Center/Home/Angle]:
```

Options

- **Left** (L) and **Right** (R): the text goes next to the first or the second extension line (a DIMJUST override in the dimension, as in AutoCAD).
- **Center** (C): the text, centered on the dimension line.
- **Home** (H): the text goes back to the position given by the style.
- **Angle** (A): rotates the text by the angle you enter.

Notes

- In linear dimensions, when the text is dropped elsewhere the dimension line goes through it (with DIMTMOVE 0, the ISO-25 value).

See also

DIMEDIT · DIMSTYLE · PROPERTIES

FIELD —Field

Inserts an updatable field: date, file name, drawing properties or an object property.

- **Command:** `FIELD` · **Global name:** `_FIELD` · **Alias:** `FLD`
- Ribbon: Annotate › Text

Inserts a field: a text whose value is computed and updated automatically, such as the date, the file name, the drawing author or a property of an object (the length of a line, the area of a boundary...). The field can go in a new

text or replace the contents of an existing text.

Usage

1. Enter FIELD (or FLD). The **Field** dialog box opens.
2. Choose the field in **Field names:** and, for the date, its **Format:**. With Object, click **Select object**, select the object and, back in the dialog box, choose the **Property:**. Below you see the **Field expression:** (which can be edited) and the value it gives. Click **OK**. At the command line (scripts or FILEDIA set to 0), the same choices are typed:

```
Enter field [Date/Filename/Author/Title/Subject/Login/Object] <Date>:
Enter property [Length/Area/Radius/Diameter/anGle/laYer] <Length>:
```

3. Specify where the new text starts, or choose Select text and select a text, a multiline text, an attribute definition or a block with attributes (the field goes to the attribute nearest to the picked point).

```
Specify text start point or [Select text]:
Select the text, or the block with attributes, to hold the field:
```

Fields

Option	Value
Date	Today's date (in the short date format of the interface language)
Filename	The drawing file name
Author, **Title**, **Subject**	The drawing properties
Login	The user logged on to the computer
Object	Length, Area, Radius, Diameter, anGle or laYer of the selected object

Notes

- The new text is a multiline text with the current text style and height.
- Fields are shown on a gray background, which is not plotted; FIELDDISPLAY turns it off.
- They are updated when the drawing is opened and saved and with UPDATEFIELD; the date, only with the latter.
- They are saved as in AutoCAD, which recognizes and updates them.
- Inside the MTEXT editor, fields are inserted with its **Field** button. A field that cannot be evaluated is shown as ####.

See also

UPDATEFIELD · MTEXT · Text and styles

FIND —Find and Replace

Finds text in texts, attributes, dimensions, leaders and tables and replaces it.

- **Command:** `FIND` · **Global name:** `_FIND` · **Alias:** `FND`
- Ribbon: Annotate › Text

Finds a text in the drawing and replaces it with another. It searches texts, multiline texts, attributes, dimension override texts, multileaders and table cells.

Usage

1. Enter FIND (or FND). The **Find and Replace** dialog box opens.
2. Type what you are looking for in **Find what:**. Below it you see how many times it appears, updated as you

type.

3. Type the new text in **Replace with:** and click **Replace All**. **Close** finishes without changing anything.

The dialog box

- **Match case**: when cleared, "wall" also finds "Wall" and "WALL".
- **Find whole words only**: "wall" does not find "walls" or "drywall".
- If objects were selected when the command started, only they are searched and the dialog box says so.
- **Find Next**: goes to the next match (after the last one, back to the first), shows the text around it and zooms to the object that has it, unless **No zoom to the match** is checked.
- **Replace**: changes only the match shown and moves on to the next one.

Command line

While the dialog box is open you can also type:

```
Find what:
Replace with (Enter: find only):
```

If you press Enter at the second prompt, the matches are only counted. The typed search uses the boxes of the dialog.

Notes

- Texts with fields are skipped, because their contents come from the field; in tables, formulas are not searched.
- In multiline texts, a word split by a formatting change (one letter in bold, for example) is not found.
- Changes are undone with UNDO.

See also

SPELL · TEXTEDIT · Text and styles

JUSTIFYTEXT —Justify text

Changes the justification of texts without moving them.

- **Command:** `JUSTIFYTEXT` · **Global name:** `_JUSTIFYTEXT`
- Ribbon: Annotate › Text

Changes the justification of the selected texts without moving them, like AutoCAD 2027 JUSTIFYTEXT.

Use

1. Type JUSTIFYTEXT.
2. Select the texts and press Enter.

```
Select objects:
```

3. Choose the new justification.

```
Enter a justification option [Left/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]
 <…>:
```

Notes

- The text stays where it was: its alignment point moves to that of the chosen justification. With TrueType the width is the text advance and with SHX it reaches its ink, as in AutoCAD.
- **Align** and **Fit** span from the start to the end of the current baseline.
- For multiline texts the attachment point changes (TL…BR).

See also

SCALETEXT · TEXTALIGN · TEXT

LEADER —Leader

Draws a leader (not a multileader) with its text.

- **Command:** `LEADER` · **Global name:** `_LEADER` · **Alias:** `LEAD`
- Ribbon: Annotate › Leaders

Draws an AutoCAD leader (LEADER: a line with an arrowhead and its associated text), like AutoCAD 2027 LEADER. For leaders with their own style, use MLEADER.

Use

1. Type LEADER.
2. Specify the arrowhead point and the next points; Enter ends the line.

```
Specify next point or [Annotation/Format/Undo] <Annotation>:
```

3. Type the text lines; Enter on an empty line ends.

```
Enter first line of annotation text or <options>:
```

Options

- **Format** (F): **Spline** or **STraight**, and **Arrow** or **None** (no arrowhead).
- **Undo** (U): removes the last point.
- Enter on the first text line: **None** makes the leader without text.

Notes

- If the last segment is not horizontal, a horizontal hook of DIMASZ is added, as in AutoCAD; the text goes next to the end, at DIMGAP, above the line with DIMTAD (the ISO-25 one).
- It uses the current dimension style (arrowhead, sizes and text style).
- The Tolerance, Copy and Block annotations of AutoCAD are missing.

See also

QLEADER · MLEADER · DIMSTYLE

MLEADER —Multileader

Creates a multileader: arrowhead, landing and text or block content.

- **Command:** `MLEADER` · **Global name:** `_MLEADER` · **Alias:** `MLD`
- Ribbon: Home › Annotation
- Ribbon: Annotate › Leaders
- Menu: Dimension

Creates a multileader: an arrow pointing at a point, a landing and some content (multiline text, a block or nothing). Its look is set by the current multileader style.

Usage

1. Enter MLEADER (or MLD) or click **Multileader** on the ribbon.

2. Specify the arrowhead location, or choose what to specify first.

```
Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>:
```

3. Specify the next point. When the maximum number of points is reached (2 by default: the arrowhead and the landing), that point is the landing; with more points, Enter ends the leader at the last one.

```
Specify leader landing location:
```

4. If the content is text, the MTEXT editor opens: type the text and click **OK**.

Options

- **leader Landing first** (L): specify the landing first and then the arrowhead.
- **Content first** (C): draw the text box first (two corners), type the text and specify the arrowhead last.
- **leader arrowHead first** (H): starts again with the arrowhead.
- **Options** (O, or Enter):

```
Enter an option [Leader type/leader lAnding/Content type/Maxpoints/First angle/Second angle/
LAyer/eXit options] <eXit options>:
```

- **Content type** (C): Block, Mtext or None. With a block, type its name.
- **Leader type** (L): Straight, sPline or None (no line).
- **leader lAnding** (A): with or without landing.
- **First angle** (F) and **Second angle** (S): the first leader segment (from the arrowhead) and the second one go to multiples of that angle (0, no constraint).
- **LAyer** (LA): the layer of new multileaders (. the current one).
- **Maxpoints** (M): maximum number of leader points, from 2 to 100.
- **eXit options** (X): goes back to the first prompt.

Notes

- The chosen order (arrowhead, landing or content first) is remembered for the next MLEADER, as in AutoCAD.
- The arrowhead, landing, text style and text height are changed in MLEADERSTYLE.
- Grips move the arrowhead, the vertices, or the content together with the landing.
- If the style is annotative (those that come in AutoCAD drawings), the multileader is created annotative at the current annotation scale (CANNOSCALE). When the scale changes its content is scaled, but the leader is not rebuilt.

See also

MLEADEREDIT · MLEADERALIGN · MLEADERSTYLE · Text and styles

MLEADERALIGN —Align Multileaders

Aligns the content of several multileaders.

- **Command:** `MLEADERALIGN` · **Global name:** `_MLEADERALIGN` · **Alias:** `MLA`
- Ribbon: Annotate › Leaders

Aligns the content of several multileaders: their landings move onto the same line, the one that starts at the reference multileader in the direction you specify. The arrowheads do not move.

Usage

1. Enter MLEADERALIGN (or MLA) or click **Align Multileaders** on the ribbon.
2. Select the multileaders to align and press Enter.

```
Select multileaders:
```

3. Select the multileader used as the reference (it does not move).

```
Select multileader to align to or [Options]:
```

4. Specify the direction of the line (for example, vertical with Ortho to stack the texts in a column).

```
Specify direction:
```

Notes

- Each multileader is moved onto the line by the shortest path, together with its content.
- **Options** (O), when asked for the reference, changes the mode (remembered, as in AutoCAD): **Use current spacing** (U) moves them to the line keeping their position along it; **specify Spacing** (S) puts them at the spacing you enter from the reference; **Distribute** (D) spreads them evenly between two points (no reference); **make leader segments Parallel** (P) rotates each leader, keeping its arrowhead and length, so that its segment is parallel to the reference one.

```
Enter an option [Distribute/make leader segments Parallel/specify Spacing/Use current spacin
g] <…>:
```

See also

MLEADER · MLEADEREDIT · MLEADERSTYLE

MLEADERCOLLECT —Collect multileaders

Gathers block multileaders into a single leader.

- **Command:** `MLEADERCOLLECT` · **Global name:** `_MLEADERCOLLECT` · **Alias:** `MLC`
- Ribbon: Annotate › Leaders

Gathers block multileaders (balloons, for example) into a single leader, like AutoCAD 2027 MLEADERCOLLECT.

Use

1. Type MLEADERCOLLECT.
2. Select the block multileaders and press Enter (at least two).

```
Select multileaders:
```

3. Specify where they are gathered.

```
Specify collected multileader location or [Vertical/Horizontal/Wrap] <…>:
```

Options

- **Horizontal**: the blocks in a row. **Vertical**: in a column. **Wrap**: in rows no wider than the width you enter.

Notes

- Only the last one selected keeps its leader; the others lose it and their block is placed next to its block.
- Text multileaders are not collected.

See also

MLEADER · MLEADERALIGN · MLEADERSTYLE

MLEADEREDIT —Add or Remove Leaders

Adds leaders to a multileader or removes leaders from it.

- **Command:** `MLEADEREDIT` · **Global name:** `_MLEADEREDIT` · **Alias:** `MLE`
- Ribbon: Annotate › Leaders

Adds leaders to a multileader or removes some. All the leaders reach the same landing and the same content.

Usage

1. Enter MLEADEREDIT (or MLE) or click **Add or Remove Leaders** on the ribbon.
2. Select the multileader.

```
Select a multileader:
```

3. Specify the arrowhead of each new leader; the prompt repeats. Press Enter to finish.

```
Specify leader arrowhead location or [Remove leaders]:
```

Options

- **Remove leaders** (R): pick near the leaders you do not want; the one closest to each point is removed.

```
Specify leaders to remove or [Add leader]:
```

- **Add leader** (A): goes back to adding leaders.

Notes

- At least one leader always remains.
- New leaders are straight, from the arrowhead to the landing.

See also

MLEADER · MLEADERALIGN · MLEADERSTYLE

MLEADERSTYLE —Multileader Style

Creates, modifies and sets the current multileader style.

- **Command:** `MLEADERSTYLE` · **Global name:** `_MLEADERSTYLE` · **Alias:** `MLS`
- Ribbon: Annotate › Leaders
- Menu: Format

Manages multileader styles: creates and modifies styles and sets the current one. When a style is modified, the multileaders that use it are updated.

Usage

1. Enter MLEADERSTYLE (or MLS) or click **Multileader Style** on the ribbon. The **Multileader Style Manager** opens.
2. Choose a style in the **Styles:** list.
3. **Set Current** makes it current; to create one, type the name in the field and click **New** (a copy of the selected style); **Modify...** shows its values.
4. Change the values, click **OK** and, when done, **Close**.

Modifying the style

- **Arrowhead symbol**: one of the 20 AutoCAD arrowheads (closed filled, closed blank, dot, architectural tick, oblique, open, origin, integral, none…); **Arrowhead size**.
- **Automatically include landing**, **Set landing distance** (length of the landing) and **Landing gap** (gap between the landing and the text).
- **Multileader type**: Mtext, Block (choose the **Source block**) or None.
- **Text style** and **Text height**.
- **Left attachment** and **Right attachment**: where the landing meets the text when the leader comes from the left or from the right (Top of top line, Middle of multi-line text, Underline all text…).

Command line

While the dialog box is open you can type:

```
Multileader Style Manager (or ?, Current name, New name, Text height value):
```

- **?**: lists the styles.
- **Current** *name* (C): makes the style current.
- **New** *name* (N): creates a copy of the current style and makes it current.
- **Text height** *value* (T): changes the text height of the current style.

Notes

- The current style is stored in CMLEADERSTYLE. The arrowheads are the AutoCAD arrow blocks (_Dot, _Open…) created in the drawing when chosen.
- The Annotative property of the style is not changed here; annotative styles from AutoCAD drawings are honored and their multileaders are created annotative.

See also

MLEADER · STYLE · Text and styles

MLSTYLE — Multiline style

Creates and modifies multiline styles.

- **Command:** `MLSTYLE` · **Global name:** `_MLSTYLE`
- Ribbon: Home › Draw
- Menu: Format

Creates, modifies and manages the multiline styles used by MLINE. They are stored in the drawing as AutoCAD does (ACAD_MLINESTYLE dictionary), so they survive opening the drawing in AutoCAD and back.

Usage

1. Enter MLSTYLE.
2. In the list, choose a style and use the buttons: - **Set Current**: it will be the one of new multilines. - **New…**: first type the name in **New style name**; it is created as a copy of the chosen one. - **Modify…**: opens its elements and caps (see below). On OK, the multilines of the style are rebuilt. - **Rename**: with the name typed in **Rename to**. - **Delete**: Standard, the current one and one used by any multiline cannot be deleted.
3. **OK** closes the dialog.

Modifying a style

- **Elements**: each line, with its offset from the axis, its color (ACI number: 256 ByLayer, 0 ByBlock) and its linetype. **Add element** and **?** add and remove lines (at least one remains).

- **Caps**: at the start and at the end, a closing line, an arc joining the outer elements or arcs between the inner ones, and the angle of the caps.
- **Fill** with its color and **Display joints** at the vertices.

Notes

- **Load...** loads the styles of an AutoCAD .mln file (those not already in the drawing) and **Save...** saves the chosen style to a .mln that AutoCAD can load.
- The color of the elements is chosen by its number.

See also

MLINE · MLEDIT · RENAME

MTEXT —Multiline Text

Creates a formatted paragraph of text.

- **Command:** `MTEXT` · **Global name:** `_MTEXT` · **Alias:** `T`, `MT`
- Ribbon: Home › Annotation
- Ribbon: Annotate › Text
- Menu: Draw
- Toolbar: Draw

Creates multiline text (MTEXT): a paragraph inside a box, with line breaks and formatting (bold, italic, underline, color, height) within the same object. For one-line notes, TEXT is quicker.

Usage

1. Enter MTEXT (or T, MT) or click **Multiline Text** on the ribbon.
2. Specify the first corner of the box.

```
Specify first corner:
```

3. Specify the opposite corner or choose an option. The box width is the paragraph width: long lines wrap at the edge.

```
Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width/Columns]:
```

4. The **Text Editor** opens. Type the text and click **OK**. You can also type the lines on the command line; Enter on an empty line finishes.

```
Enter text (Enter on an empty line ends):
```

Options

- **Height** (H): text height (it offers the last one used, TEXTSIZE).
- **Justify** (J): attachment point of the box (TL, TC, TR, ML, MC, MR, BL, BC, BR).

```
Enter justification [TL/TC/TR/ML/MC/MR/BL/BC/BR] <…>:
```

- **Style** (S): text style; **?** lists the ones in the drawing.
- **Rotate** (R): angle of the box and the text.
- **Width** (W): types the box width instead of the opposite corner (0, no limit).

Text Editor

- **Style**, **Height**, **Justification** and **Width**: the values for the whole paragraph.
- **B**, **I** and **U**: bold, italic and underline for the selection (or for all the text if nothing is selected).

- Color list and **Color**: applies the chosen color to the selection.
- **×** factor and **Height** button: changes the height of the selection relative to the paragraph.
- Field list and **Field** button: writes the field code at the cursor (date, file name, author, title, subject or user, `%<\AcVar ...>%`); on OK it becomes a field of the text, which UPDATEFIELD updates. Object property fields are inserted with FIELD.
- Each line in the box is a paragraph. Formats are stored as MTEXT codes, which can also be typed by hand.

Notes

- With a fixed-height style, the style height replaces the typed one.
- With an annotative style, the text is created annotative and its height is the paper height at the current annotation scale (CANNOSCALE); OBJECTSCALE adds more scales to it.
- Fields are inserted from the editor or with FIELD, and spelling is checked with SPELL.
- To change the text later, double-click it or use TEXTEDIT.
- **Line spacing** (L), when asked for the opposite corner: At least or Exactly and the factor (`1.5x`) or the distance between lines. **Columns** (C): dynamic (width, gutter, height) or static (total width, number, gutter, height); stored as AutoCAD does, and the text flows into them when each column's height is full (dynamic columns add columns; in static ones the last column overflows).

See also

TEXT · TEXTEDIT · STYLE · Text and styles

OBJECTSCALE —Object Scale

Adds or deletes annotation scales of annotative objects.

- **Command:** `OBJECTSCALE` · **Global name:** `_OBJECTSCALE` · **Alias:** `OBJSC`
- Ribbon: Annotate › Annotation Scaling

Adds or deletes annotation scales of the selected annotative objects. An annotative object keeps one representation for each scale it supports and is shown with the one of the current annotation scale (CANNOSCALE) or of the viewport; if it does not support that scale, it is shown with its default scale.

Usage

1. Enter OBJECTSCALE (or OBJSC).
2. Select the objects and press Enter. Objects that are not annotative are discarded.

The **Annotation Object Scale List** dialog box opens with the scales the objects support: choose a drawing scale in the list box and click **Add**, or pick one in the list and click **Delete**. The changes apply at once; **Close** finishes. In scripts and with FILEDIA set to 0 it goes on at the command line:

```
Select annotative objects:
```

3. Choose whether to add or delete a scale (Enter: Add).

```
Enter an option [Add/Delete/?] <Add>:
```

4. Enter the scale name as it appears in the drawing's scale list (`1:50`). When adding, Enter takes the current annotation scale.

```
Enter named scale to add or [?] <…>:
Enter named scale to delete or [?]:
```

Options

- **Add** (A): the objects start supporting the scale. The new representation is computed from the one shown: texts and dimensions keep their size on paper.

- **Delete** (D): the objects stop supporting the scale. An object never loses its last scale.
- **?**: lists the scales supported by the selected objects.

Notes

- The scale must be in the drawing's scale list; add it with SCALELISTEDIT. If the drawing has no list, the AutoCAD metric list is created.
- The scales supported by an object are shown in the PROPERTIES palette (**Annotative scales**), where it can also be made annotative with **Annotative**.
- In the dialog box the last scale the objects support cannot be deleted.
- With ANNOAUTOSCALE, changing the annotation scale adds it automatically to the objects that supported the previous one.

See also

SCALELISTEDIT · CANNOSCALE · Annotation scales

QDIM —Quick Dimension

Quickly creates a series of dimensions from the selected geometry.

- **Command:** `QDIM` · **Global name:** `_QDIM` · **Alias:** `QD`
- Ribbon: Annotate › Dimensions

Creates a series of dimensions of the selected geometry in one go: chained dimensions, baseline dimensions, staggered dimensions, ordinate dimensions, or radius or diameter dimensions of all arcs and circles. It is the quick way to dimension a part or a façade.

Usage

1. Enter QDIM (or QD).
2. Select the geometry you want to dimension and press Enter.

```
Select geometry to dimension:
```

3. Move the cursor to preview the dimensions and specify where the dimension line goes, or choose another type of dimension first. Enter finishes without dimensioning.

```
Specify dimension line position, or [Continuous/Staggered/Baseline/Ordinate/Radius/Diameter/
datumPoint/Edit/seTtings] <…>:
```

Options

- **Continuous** (C): chained dimensions between consecutive points. This is the initial option.
- **Staggered** (S): nested symmetric dimensions, from the outer pair of points to the inner one.
- **Baseline** (B): all from the first point, each one offset from the previous one by the baseline spacing of the style (DIMDLI).
- **Ordinate** (O): one ordinate dimension per point.
- **Radius** (R) and **Diameter** (D): one dimension per arc and circle, in the direction of the cursor.

Which points are dimensioned

- Lines: their two endpoints. Polylines: their vertices. Points: their location.
- Arcs: their endpoints and, with Radius or Diameter, the arc itself. Circles: only with Radius or Diameter.
- Points that coincide in the measured direction are dimensioned only once.

Notes

- As in DIMLINEAR, the orientation comes from the position: above or below the geometry the dimensions are horizontal; to one side, vertical.
- The dimensions use the current dimension style (DIMSTYLE) and are associated with the geometry.
- **datumPoint** (P): a new datum point: the baseline starts at the dimension point nearest to it and ordinate dimensions measure from it.
- **Edit** (E): remove dimension points (the nearest to each click) or, with Add, add others.
- **seTtings** (T): the association priority, Endpoint or Intersection (remembered, but dimensions are always associated by their endpoints).

See also

DIMCONTINUE · DIMBASELINE · DIMORDINATE · Dimensions

QLEADER —Quick leader

Draws a leader with its text in a few steps.

- **Command:** `QLEADER` · **Global name:** `_QLEADER` · **Alias:** `LE`
- Ribbon: Annotate › Leaders

Draws a leader with its text in a few steps, like AutoCAD 2027 QLEADER.

Use

1. Type QLEADER.
2. Specify up to three leader points (the arrowhead first); Enter before ends the line.

```
Specify first leader point, or [Settings] <Settings>:
```

3. Enter the text width (0: no line wrapping).

```
Specify text width <…>:
```

4. Type the text lines; Enter on an empty line ends.

```
Enter first line of annotation text <Mtext>:
```

Notes

- It uses the default values of the AutoCAD Settings dialog (not in LinceCAD): three points at most, with an arrowhead and multiline text centered on its first line (to the right of the end) or on its last one (to the left).
- If the last segment is not horizontal, a horizontal hook of DIMASZ is added.

See also

LEADER · MLEADER

SCALELISTEDIT —Scale List

Edits the drawing's list of annotation scales.

- **Command:** `SCALELISTEDIT` · **Global name:** `_SCALELISTEDIT`
- Ribbon: Annotate › Annotation Scaling
- Menu: Format

Edits the list of annotation scales of the drawing: adds scales, deletes the ones not needed or goes back to the

default list. The list is saved in the drawing, as in AutoCAD (ACAD_SCALELIST dictionary).

Usage

1. Enter SCALELISTEDIT. The **Edit Drawing Scales** dialog box opens with the list of scales (name and paper and drawing units): - **Add...** and **Edit...**: below the list you type the name and the paper and drawing units of the new scale or of the chosen one; **OK** saves it. When edited, the scale remains the same object: annotative objects that support it keep it. - **Move Up** and **Move Down**: change its place in the list (the order of the scale selectors). - **Delete** and **Reset**: like the options below. Each change applies at once and can be undone; **Close** finishes. At the command line (with -SCALELISTEDIT, in scripts or with FILEDIA set to 0):
2. Choose an option (Enter: Add).

```
Enter option [?/Add/Delete/Reset/Exit] <Add>:
```

3. To add a scale, type its name and its paper:drawing ratio.

```
Enter name for new scale:
Enter scale ratio (paper:drawing):
```

Options

- **?**: lists the scales of the drawing with their paper and drawing units.
- **Add** (A): adds a scale. The ratio is typed `1:50` or `1=50`.
- **Delete** (D): deletes the scale with the given name. The last scale cannot be deleted.
- **Reset** (R): after confirmation, replaces the list with the default metric one, that of new drawings: AutoCAD's (1:1 to 1:100 and 2:1 to 100:1) followed by the UNE-EN ISO 5455 standard scales it lacks (1:25, 1:200, 1:250, 1:500, 1:1000, 1:2000, 1:2500, 1:5000, 1:10000, 5:1, 20:1 and 50:1).
- **Exit** (E): ends the command (Esc also does).

Notes

- -SCALELISTEDIT does the same without the dialog box.
- The scales in the list are the ones that can be chosen as annotation scale (CANNOSCALE, also with the selector on the status bar) and the ones added to annotative objects with OBJECTSCALE.
- LinceCAD does not check whether a scale is in use before deleting it: neither the current annotation scale nor those supported by annotative objects.
- The changes can be undone.

See also

-SCALELISTEDIT · OBJECTSCALE · Annotation scales · Layouts

SCALETEXT —Scale text

Changes the height of texts without moving them from their base point.

- **Command:** SCALETEXT · **Global name:** _SCALETEXT
- Ribbon: Annotate › Text

Changes the height of the selected texts without moving them from their base point, like AutoCAD 2027 SCALETEXT.

Use

1. Type SCALETEXT.
2. Select the texts and press Enter.

```
Select objects:
```

3. Choose the base point: **Existing** (that of the justification of each text) or one of those of TEXT.

```
Enter a base point option for scaling [Existing/Left/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <Existing>:
```

4. Enter the new height or an option.

Options

- **Paper height**: the height on paper; in model space it is divided by the current annotation scale.
- **Match object**: the height of another text you select.
- **Scale factor**: multiplies the height of each text (or **Reference**: reference length and new length).

Notes

- The base point of each text does not move; the rest of the text grows or shrinks around it.
- For multiline texts the base point is always their insertion point.

See also

JUSTIFYTEXT · SCALE · PROPERTIES

SPELL —Check Spelling

Checks the spelling of texts with the language dictionary.

- **Command:** `SPELL` · **Global name:** `_SPELL` · **Alias:** `SP`
- Ribbon: Annotate › Text

Checks the spelling of the texts in the drawing with the dictionary of the interface language (Spanish or English) and suggests corrections for every word it does not recognize.

Usage

1. Enter SPELL (or SP).
2. Select the objects you want to check and press Enter, or press Enter without selecting anything to check the whole drawing.

```
Select objects (Enter: entire drawing):
```

3. For each word not in the dictionary, the **Check Spelling** dialog box opens with the suggestions. Pick one (or type the right one in **Change to:**) and click a button.
4. At the end you are told: Spelling check complete.

Buttons

- **Change**: corrects this occurrence of the word.
- **Change All**: also corrects all the other identical occurrences.
- **Ignore**: leaves the word as it is and goes to the next one.
- **All**: leaves this word and all identical ones during this check.
- **Add**: adds it to the user dictionary; it will not be flagged again.
- **Close**: ends the check; what was already corrected stays.

Command line

The same prompt can be answered by typing:

```
"…" is not in the dictionary. Suggestions: …. Type the correction or [Ignore/All/Add] <Ignore>:
```

Type the correction (ending with `*`, for all identical occurrences) or one of the options; Enter ignores the word.

Notes

- Texts, multiline texts, attributes, dimension texts, multileaders and table cells are checked. Texts with fields are skipped.
- Single-letter words and words containing digits are not checked.
- The user dictionary is the file `ortografia_usuario.txt` in the LinceCAD configuration folder (`%APPDATA%\lince` on Windows): one word per line.
- The dictionaries come with the program (RLA-ES for Spanish and SCOWL for English). Some compound or heavily derived forms may be flagged as unknown: use Add.

See also

FIND · TEXTEDIT · Text and styles

STYLE —Text Style

Creates and modifies text styles.

- **Command:** `STYLE` · **Global name:** `_STYLE` · **Alias:** `ST`, `DDSTYLE`
- Ribbon: Annotate › Text
- Menu: Format

Creates, modifies and deletes text styles, and sets the current one. A text style holds the font, an optional fixed height, the width factor and the oblique angle; it is used by TEXT, MTEXT, dimensions and tables.

Usage

1. Enter STYLE (or ST) or click **Text Style** on the ribbon. The **Text Style** dialog box opens.
2. Choose a style in the **Styles:** list. Its values are shown on the right.
3. Change the values and click **Apply** to save them in the style.

Controls

- **Font Name**: the font, chosen from the list (the Windows TrueType and OpenType fonts, and the AutoCAD SHX fonts if AutoCAD is installed).
- **or type the font**: the font file name, typed (`txt.shx`, `arial.ttf`).
- **Height**: fixed height of the style. With 0, TEXT and MTEXT ask for the height of each text.
- **Width Factor**: 1 is the normal width; less than 1 narrows the characters and more than 1 widens them.
- **Oblique Angle**: slant of the characters, between -85° and 85°.
- **Annotative**: new texts with this style are created annotative: the height is the paper height and they are shown at the current annotation scale (CANNOSCALE). Existing texts do not change.
- **Set Current**: new texts will use the selected style.
- **New...**: creates a copy of the selected style; type the name in **Style Name:** and click **OK**.
- **Delete**: deletes the selected style. Standard, the current style and styles used by texts, attributes or dimension styles cannot be deleted.
- **Apply**: saves the changes to font, height, width and angle (enabled only when there are changes).

Notes

- The current style is stored in the TEXTSTYLE variable.
- The font file name is stored, as AutoCAD does.

See also

TEXT · MTEXT · DIMSTYLE · Text and styles

TABLE — Insert Table

Creates an empty table or one with the data of a CSV file.

- **Command:** TABLE · **Global name:** _TABLE · **Alias:** TB
- Ribbon: Home › Annotation
- Ribbon: Annotate › Tables
- Menu: Draw

Inserts a table: empty, to type in its cells, or filled with the data of a CSV file or of pasted text. The table is an AutoCAD TABLE object, with title, header and data rows according to its style.

Usage

1. Enter TABLE (or TB) or click **Insert Table** on the ribbon. The **Insert Table** dialog box opens.
2. Choose the style, where the data comes from and the size, and click **OK**.
3. Specify the insertion point (the upper-left corner) or, with **Specify window**, the two corners.

```
Specify insertion point:
```

4. In an empty table you start typing in the first cell: Tab moves to the next cell, Shift+Tab to the previous one and the Up and Down arrows to another row. **OK** finishes.

Insert Table

- **Table style:**: the table style (TABLESTYLE).
- **Insert options**: Start from empty table, From a CSV file: (type the path or click **Browse..**.) or From pasted text (CSV). In the last two the first row is the header and the columns take the width of their contents.
- **Insertion behavior**: Specify insertion point or Specify window. With a window, the columns share the width and as many data rows as fit in the height are created.
- **Column & row settings** (empty table only): Columns:, Column width:, Data rows: and Row height (lines):.
- **Set cell styles**: the style of the First row cell style: and the Second row cell style: (Title, Header or Data).

Command line

While the dialog box is open you can type the same options; Enter accepts.

```
Table options [Columns/Rows/Width/Height/Style/WIndow/CSV] <OK>:
```

Columns (C), Rows (R), Width (W), Height (H), Style (S), WIndow (WI, toggles point and window) and CSV (CSV, path of the file).

Formulas

Contents starting with = are a formula: `=Sum(A3:A5)`, `=Average(B3:B6)`, `=Count(A3:C5)`, `=A3*B3+1`. Columns are letters and rows are numbers, as in a spreadsheet. A formula that cannot be calculated shows `####`.

See also

TABLEDIT · TABLECELL · TABLESTYLE · TABLEEXPORT · Tables

TABLECELL —Table Cell

Changes the selected cells of a table.

- **Command:** `TABLECELL` · **Global name:** `_TABLECELL` · **Alias:** `CELLS`

Changes the selected cells of a table: inserts or deletes rows and columns, merges or unmerges cells, and changes the alignment, height and color of the text, the fill and the contents.

Selecting cells

With no command running, click inside a cell to select it; Shift+click or dragging from one cell to another extends the range. The **Table Cell** palette appears; its buttons run TABLECELL:

- **Rows:** Insert Above, Insert Below, Delete Rows. **Columns:** Insert Left, Insert Right, Delete Columns.
- Merge Cells, Unmerge Cells and Edit Text (opens TABLEDIT).
- **Alignment:**, **Text height:**, **Text color:** and **Fill:**.
- **Formula:** Sum, Average or Count: with a single cell selected, the formula takes the data cells above it; with a range, the range.

Command line usage

1. Enter TABLECELL (or CELLS). If no cells are selected, click one.

```
Select a table cell:
```

2. Choose the option. Each change ends the command and is undone separately.

```
Cells … [EDit/Text/ABove/BElow/LEft/RIght/DeleteRow/DeleteColumn/Merge/Unmerge/ALign/Height/
COlor/Fill/RAnge] <Exit>:
```

Options

- **EDit** (ED): edits the text. **Text** (T): writes the same contents in all the selected cells.
- **ABove** (AB), **BElow** (BE), **LEft** (LE), **RIght** (RI): insert as many rows or columns as are selected.
- **DeleteRow** (DR), **DeleteColumn** (DC): delete the selected ones; one always remains.
- **Merge** (M), **Unmerge** (U): merge the range into one cell or unmerge it.
- **ALign** (AL), **Height** (H), **COlor** (CO), **Fill** (F): text format and fill.
- **RAnge** (RA): selects another range, typed (B3 or A1:C4).

Notes

- Formulas do not shift their references when rows or columns are inserted or deleted.

See also

TABLE · TABLEDIT · TABLESTYLE · Tables

TABLEDIT —Edit Table Cell

Edits the text of a table cell.

- **Command:** `TABLEDIT` · **Global name:** `_TABLEDIT` · **Alias:** `EDITTABLE`
- Ribbon: Annotate › Tables

Edits the text of table cells with the text editor, moving from one cell to another with the keyboard.

Usage

1. Enter TABLEDIT (or EDITTABLE), or double-click a cell. If a cell was already selected, that one is edited.
2. Select the cell with a click.

```
Select a table cell:
```

3. Type in the editor. Tab moves to the next cell and Shift+Tab to the previous one; with a single line of text, the Up and Down arrows move to the row above or below. **OK** saves and finishes.
4. You can also type on the command line: each line is the contents of a cell and moves to the next one; Enter on an empty line finishes.

```
Cell …: type the contents (Enter ends):
```

Notes

- Contents starting with = are a formula (`=Sum(A3:A5)`); see TABLE.
- **Cancel** does not save the cell being edited, but keeps what was typed before in other cells. All the changes made in one TABLEDIT are undone together.
- Merged cells are edited as one; the cells covered by a merged cell are skipped.

See also

TABLE · TABLECELL · MTEXT · Tables

TABLEEXPORT —Export Table

Saves the cells of a table to a CSV file.

- **Command:** `TABLEEXPORT` · **Global name:** `_TABLEEXPORT` · **Alias:** `EXPORTTABLE`
- Ribbon: Annotate › Tables

Saves the contents of a table to a CSV file, to open it in a spreadsheet. Formulas are exported with their value.

Usage

1. Enter TABLEEXPORT (or EXPORTTABLE) or click **Export Table** on the ribbon. If cells or a table are selected, that table is exported.
2. Select the table.

```
Select a table:
```

3. Type the path of the file. If it has no extension, `.csv` is added.

```
CSV file:
```

Notes

- The file is UTF-8, with the columns separated by commas; values containing commas, semicolons or quotes are quoted.
- When finished, the number of rows written is reported.
- To create a table from a CSV file, use TABLE.

See also

TABLE · ATTEXT · Tables

TABLESTYLE —Table Style

Creates and modifies table styles.

- **Command:** `TABLESTYLE` · **Global name:** `_TABLESTYLE` · **Alias:** `TS`
- Ribbon: Annotate › Tables
- Menu: Format

Manages table styles: creates, modifies and deletes styles, and sets the current one. The style sets the cell margins and, for the data, header and title rows, the text, color, fill and alignment.

Usage

1. Enter TABLESTYLE (or TS) or click **Table Style** on the ribbon. The **Table Style** dialog box opens.
2. Choose a style in the list (styles used by tables show "(in use)"). Double-click makes it current.
3. Click one of the buttons; **Close** finishes.

Buttons

- **Set Current**: new tables will use the selected style.
- **New...**: opens **Create New Table Style**; type the name in **New Style Name:**, choose in **Start With:** the style to copy and click **Continue**.
- **Modify...**: opens the modify dialog box, described below.
- **Delete**: deletes the style unless it is Standard, current or used by a table.

Modify Table Style

- **Horizontal margin:** and **Vertical margin:**: space between the text and the cell borders.
- **Cell style:**: choose Data, Header or Title; the fields below belong to that row: **Text style:**, **Text height:**, **Text color:**, **Fill color:** and **Alignment:**.

Click **OK**: the tables that use the style are redrawn.

Command line

```
Table style (new N, current N, height N data 3, color N title 1…) <Close>:
```

`new N [base]`, `current N`, `delete N` and, for the `data`, `header` or `title` row of style N: `height N data 3.5`, `color N title 1`, `fill N header 8` (or `none`), `align N data MC`. `margin N 1.5` changes the margins. The Spanish words (nuevo, actual, altura, datos…)are also accepted.

Notes

- The current style is stored in CTABLESTYLE and can also be chosen in the list of the Tables panel on the Annotate tab.

See also

TABLE · TABLECELL · STYLE · Tables

TCOUNT —Number texts

Numbers the selected texts.

- **Command:** `TCOUNT` · **Global name:** `_TCOUNT`
- Ribbon: Annotate › Text

Numbers the selected texts, like the AutoCAD Express Tools TCOUNT.

Use

1. Type TCOUNT.
2. Select the texts and press Enter.

```
Select objects:
```

3. Choose the order: by **X**, by **Y** (top to bottom) or as selected.

```
Sort selected objects by [X/Y/Select-order] <Y>:
```

4. Enter the start number and the increment, separated by a comma.

```
Specify starting number and increment (Start,increment) <1,1>:
```

5. Choose where the number goes.

```
Placement of numbers in text [Overwrite/Prefix/Suffix] <Prefix>:
```

Options

- **Overwrite**: the number replaces the text. **Prefix**: it goes before, with a space. **Suffix**: after.

Notes

- The find and replace option of Express is not available.

See also

FIND · TEXT

TEXT —Single Line Text

Creates a single line of text.

- **Command:** `TEXT` · **Global name:** `_TEXT` · **Alias:** `DT`, `DTEXT`
- Ribbon: Home › Annotation
- Ribbon: Annotate › Text
- Menu: Draw

Creates single-line text. Each line you type is a separate text object; after Enter, the next line starts below, as in a word processor. For a formatted paragraph, use MTEXT.

Usage

1. Enter TEXT (or DT) or click **Single Line Text** on the ribbon.
2. Specify the start point (the text baseline) or choose an option.

```
Specify start point of text or [Justify/Style]:
```

3. If the text style has no fixed height, enter the height (a number or a second point).

```
Specify height <…>:
```

4. Enter the rotation angle (a number in degrees or a point).

```
Specify rotation angle of text <…>:
```

5. Type the text and press Enter. The line is created and the next one is requested, below the previous one.

```
Enter text:
```

6. Press Enter on an empty line to finish.

Options

- **Justify** (J): changes which point of the text sits on the start point. Left, Center, Right and Middle use the baseline; the two-letter options combine Top, Middle or Bottom with Left, Center or Right (TL = top left, MC = middle center, BR = bottom right…)As in AutoCAD, the justification can also be typed directly at the first prompt.

```
Enter an option [Left/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR]:
```

- **Align** (A): specify both endpoints of the baseline; the height comes from their distance and the text spans exactly that segment, with its rotation.
- **Fit** (F): like Align, but with the height you enter: the width factor changes.

```
Specify second endpoint of text baseline:
```

- **Style** (S): chooses the text style for this command. **?** lists the styles in the drawing.

```
Enter style name or [?] <…>:
```

Notes

- The text uses the current style (TEXTSTYLE) with its font, width factor and oblique angle, and the current layer, color and linetype.
- The height offered is the last one used (TEXTSIZE variable); the first time, 2.5.
- With an annotative style (STYLE check box), the text is created annotative and the height you enter is the paper height: at the 1:50 annotation scale (CANNOSCALE), a 2.5 text measures 125 drawing units.
- A text can hold fields (date, file name...): see FIELD.
- The rotation angle is measured from the X axis of the current UCS (see UCS), as in AutoCAD.
- To change the contents later, double-click the text or use TEXTEDIT.

See also

MTEXT · TEXTEDIT · STYLE · Text and styles

TEXTALIGN —Align text

Aligns texts with another text or with two points.

- **Command:** `TEXTALIGN` · **Global name:** `_TEXTALIGN` · **Alias:** `TA`
- Ribbon: Annotate › Text

Aligns several texts (single-line or multiline) with a reference text or with a line through two points, like AutoCAD 2027 TEXTALIGN.

Use

1. Type TEXTALIGN.
2. Select the texts to align and press Enter.

```
Select text objects to align [alIgnment/Options]:
```

3. Select the reference text (or **Points** and two points of the line).

```
Select text object to align to [Points]:
```

4. Pick the second point: it gives the direction of the alignment line.

```
Pick second point or [Options]:
```

Options

- **alIgnment**: the point of each text that is aligned: Left, Center or Right (at the bottom) or one of the nine (TL, TC, TR, ML, MC, MR, BL, BC, BR). It is taken from the ink box of the text.
- **Options** (spacing mode): **Distribute** spreads them evenly between the reference text and the second point; **Set spacing** leaves that space between the text boxes along the line; **Current vertical** keeps their Y and moves them onto the line; **current Horizontal** keeps their X.

Notes

- The alignment and the mode are remembered for the next time, as in AutoCAD.
- The texts are moved; their justification and rotation do not change.

See also

JUSTIFYTEXT · MLEADERALIGN · TEXT

TEXTEDIT —Edit Text

Edits the contents and format of a text object.

- **Command:** `TEXTEDIT` · **Global name:** `_TEXTEDIT` · **Alias:** `ED`, `DDEDIT`, `TEDIT`
- Ribbon: Annotate › Text

Changes the contents and formatting of an existing text, single-line or multiline, in the text editor.

Usage

1. Enter TEXTEDIT (or ED, DDEDIT), or double-click the text. If a text was already selected, that one is edited.
2. Select the text.

```
Select an annotation object:
```

3. The **Text Editor** opens with the contents. Change them and click **OK** (or **Cancel** to leave the text as it was).

Notes

- In multiline text you can change the style, height, justification, width and formatting (bold, italic, underline, color, relative height); the buttons are the ones of MTEXT.
- In single-line text the editor has a single line and only the contents, style and height change.
- It only edits texts (TEXT and MTEXT). Table cells are edited with TABLEDIT; dimensions, with DIMEDIT or the PROPERTIES palette.

See also

TEXT · MTEXT · Text and styles

TEXTFIT —Fit text

Fits the width of a text to a point.

- **Command:** `TEXTFIT` · **Global name:** `_TEXTFIT`
- Ribbon: Annotate › Text

Fits the width of a single-line text so it reaches a new point, changing its width factor and keeping the height.

Use

1. Type TEXTFIT or click its button.
2. Select the text.

```
Select the text:
```

3. Specify the end point; the text stretches or shrinks to reach from its start to there.

```
Specify the end point:
```

Notes

- Only the width factor and rotation of the text change; the height stays the same.
- The start is the text insertion point; the text rotates towards the end point you specify.
- Works with single-line text (TEXT); for a paragraph use MTEXT.

See also

TEXT · ARCTEXT · PROPERTIES

TEXTMASK —Mask text

Puts a wipeout behind each text so that it hides what is underneath.

- **Command:** `TEXTMASK` · **Global name:** `_TEXTMASK`
- Ribbon: Annotate › Text

Puts a wipeout behind each selected text so that it hides what is underneath, like the AutoCAD Express Tools TEXTMASK.

Use

1. Type TEXTMASK.
2. Select the texts and press Enter.

```
Select text objects to mask or [Offset]:
```

Options

- **Offset**: the offset factor, in text heights (0.35 by default): how far the wipeout goes beyond the text box.

Notes

- One wipeout is created per text, on its layer and just behind it in the draw order.
- Only wipeout masks are made; the text and its mask are not grouped.

See also

WIPEOUT · DRAWORDER · TEXT

TOLERANCE —Tolerance

Creates a geometric tolerance feature control frame.

- **Command:** `TOLERANCE` · **Global name:** `_TOLERANCE` · **Alias:** `TOL`
- Ribbon: Annotate › Dimensions

Creates a geometric tolerance feature control frame: a row of compartments with the geometric characteristic

symbol, the tolerance value and the datum references, as used in mechanical drawings.

Usage

1. Enter TOLERANCE (or TOL). The **Geometric Tolerance** dialog box opens.
2. Fill in the frame and click **OK**, or type it on the command line.

```
Use the dialog or type the frame (symbol tolerance datums; e.g. j %%c0.05M A B):
```

3. Specify where the frame goes.

```
Enter tolerance location:
```

The dialog box

- **Symbol**: the geometric characteristic (position, flatness, perpendicularity...).
- **Tolerance 1**: the **Ø** box puts the diameter symbol before the value; the list on the right adds the material condition (M, L or S).
- **Datum …**: up to three datum reference letters (A, B, C...). Empty ones are not drawn.

Typing the frame

Type the symbol, the tolerance and the datums separated by spaces: `j %%c0.05M A B` is a position tolerance of diameter 0.05 at maximum material condition, referred to A and B. The tolerance accepts `%%c` or `ø` before it and M, L or S after it.

Letter	Characteristic	Letter	Characteristic
j	Position	g	Cylindricity
r	Concentricity	c	Flatness
i	Symmetry	e	Circularity
f	Parallelism	u	Straightness
b	Perpendicularity	d	Profile of a surface
a	Angularity	k	Profile of a line
h	Circular runout	t	Total runout

Notes

- The text height and the compartment gap come from the current dimension style (DIMSTYLE); with an annotative style the frame is annotative too.
- The symbols are drawn with lines, without needing the AutoCAD GDT font.
- The dialog has, as AutoCAD's, two tolerances per row (with diameter and material modifier), three datums with their modifier, an optional second row and the projected tolerance zone (its height).

See also

DIMLINEAR · DIMSTYLE · Dimensions

TXT2MTXT —Convert to multiline text

Combines single-line texts into one multiline text.

- **Command:** `TXT2MTXT` · **Global name:** `_TXT2MTXT`
- Ribbon: Annotate › Text

Combines single-line texts (and multiline texts) into one multiline text, like AutoCAD 2027 TXT2MTXT.

Use

1. Type TXT2MTXT.
2. Select the texts and press Enter.

```
Select objects:
```

Notes

- The lines are ordered from top to bottom (and from left to right at the same height).
- The new text takes the height, style, layer and rotation of the first one; its top left corner is at the leftmost start, at the cap height of the first one. The originals are erased.
- There is no settings dialog as in AutoCAD: the new text has no width (it does not wrap lines).

See also

MTEXT · TEXT

UPDATEFIELD —Update Fields

Updates the fields of the selected objects.

- **Command:** `UPDATEFIELD` · **Global name:** `_UPDATEFIELD` · **Alias:** `UPDFIELD`
- Ribbon: Annotate › Text

Evaluates the fields of the selected objects again, so that they show their current value: today's date, the new length of a line that was stretched, the file name after saving it under another name...

Usage

1. Enter UPDATEFIELD (or UPDFIELD).
2. Select the texts, multiline texts or block references with attributes that hold fields and press Enter. Type **ALL** to update those of the whole drawing.

```
Select objects:
```

3. The number of updated fields is reported.

Notes

- All the fields of the selection are updated, including the date, which does not change on its own when the drawing is opened or saved.
- In block references, the fields of their attributes are updated.
- If you select the objects before starting the command, they are updated without further prompts.
- Fields are created with FIELD; those that come in AutoCAD drawings are updated too.

See also

FIELD · FIELDDISPLAY · Text and styles

Insert: Blocks, References, Images and Geolocation

-ATTDEF —Define Attributes (command line)

Defines an attribute on the command line, like AutoCAD -ATTDEF.

- **Command:** `-ATTDEF` · **Global name:** `_-ATTDEF` · **Alias:** `-ATT`

Creates an attribute definition on the command line, with the AutoCAD 2027 -ATTDEF prompts. This is what ATTDEF does in scripts, SCRIPT and AutoLISP.

Usage

1. Enter -ATTDEF (or -ATT).
2. Change the modes you need by entering their option (each time it toggles between Y and N) and press Enter when done. The modes are kept for the next definition (like AFLAGS in AutoCAD).

```
Current attribute modes: Invisible=… Constant=… Verify=… Preset=… Lock position=… Annotative
=N Multiple lines=N. Enter an option to change [Invisible/Constant/Verify/Preset/Lock positi
on/Annotative/Multiple lines] <done>:
```

3. Enter the tag: a single word without spaces, converted to uppercase.

```
Enter attribute tag name:
```

4. Enter the prompt, the text shown when the block is inserted (not asked if the attribute is constant).

```
Enter attribute prompt:
```

5. Enter the default value (or the fixed value, if it is constant).

```
Enter default attribute value:
```

6. Specify the start point of the text (or an option), its height and its rotation angle.

```
Specify start point of text or [Justify/Style]:
```

Options

- **Invisible** (I): the value is not shown in the drawing.
- **Constant** (C): the value is fixed and is not asked on insertion.
- **Verify** (V): flags the attribute so that its value is checked on insertion.
- **Preset** (P): flags the attribute so that it takes its default value.
- **Lock position** (L): the attribute cannot be moved apart from the block (on at startup).
- **Justify** (J): the text justification (the one-point ones of TEXT).

```
Enter an option [Left/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR]:
```

- **Style** (S): the text style.

```
Enter style name or [?] <…>:
```

Notes

- The default height is the fixed height of the text style or, if it has none, the last one used (TEXTSIZE). With a fixed-height style it is not asked.
- Annotative and multiline attributes are not available yet (Annotative and Multiple lines give a warning).

See also

ATTDEF · BLOCK · BATTMAN · ATTEDIT

-ATTEDIT —Edit Attributes (command line)

Changes attributes one at a time or in the whole drawing, like AutoCAD -ATTEDIT.

- **Command:** `-ATTEDIT` · **Global name:** `_-ATTEDIT` · **Alias:** `-ATE`, `ATTE`

Changes block attributes on the command line, with the AutoCAD 2027 -ATTEDIT prompts: one at a time (value, position, height, angle, style, layer and color) or with a global edit of the values.

Usage

1. Enter -ATTEDIT (or -ATE) and choose whether to edit them one at a time.

```
Edit attributes one at a time? [Yes/No] <Y>:
```

2. In global editing, choose whether only visible attributes are changed (you select them) or all in the drawing.

```
Edit only attributes visible on screen? [Yes/No] <Y>:
```

3. Enter the block name, tag and value specifications: with wildcards (`*`, `?`, `#`, `@`, `~` and several separated by commas); Enter leaves `*`. The value specification `\` picks empty attributes.

```
Enter block name specification <…>:
```

4. Select the attributes (a click picks the nearest attribute; a window, those fully inside) and press Enter.

```
Select Attributes:
```

5. One at a time, choose what to change for each attribute; **Next** (or Enter) goes to the next one.

```
Enter an option [Value/Position/Height/Angle/Style/Layer/Color/Next] <N>:
```

6. In global editing, enter the string to change and the new one: it is changed in every value that contains it.

```
Enter string to change:
```

Options

- **Value**: Replace sets a new value; Change replaces a string of the value with another.

```
Enter type of value modification [Change/Replace] <R>:
```

- **Position**: moves the attribute text to the given point.
- **Height** and **Angle**: the text height and rotation (in degrees).
- **Style**, **Layer** and **Color**: the text style, the layer (which must exist) and the color (number, name, R,G,B, ByLayer or ByBlock).

Notes

- Attribute selection uses an approximate text box (without the font metrics).

See also

ATTEDIT · EATTEDIT · BATTMAN · ATTDEF

-ATTEXT —Extract Attributes with Template

Writes the attributes to a CDF, SDF or DXF file, like AutoCAD -ATTEXT.

- **Command:** `-ATTEXT` · **Global name:** `_-ATTEXT`

Writes the block attributes to a file with a field template, like AutoCAD 2027 -ATTEXT.

Usage

1. Enter -ATTEXT and the format. With **Objects** you select the blocks first; otherwise every block with attributes in model space is extracted.

```
Enter extraction type or enable object selection [Cdf/Sdf/Dxf/Objects] <C>:
```

2. In CDF and SDF, enter the template path (Enter: the previous one).

```
Enter template file name:
```

3. Enter the extract file (without extension: `.txt`, or `.dxx` in DXF).

```
Enter extract file name:
```

Template

One line per field: the name and its format, `C` (text) or `N` (number), with 3 digits of width and 3 of decimals. The names are the attribute tags or the block ones: `BL:NAME`, `BL:LAYER`, `BL:HANDLE`, `BL:X`, `BL:Y`, `BL:Z`, `BL:ORIENT`, `BL:XSCALE`, `BL:YSCALE`, `BL:ZSCALE`, `BL:XEXTRUDE`, `BL:YEXTRUDE`, `BL:ZEXTRUDE`, `BL:LEVEL` and `BL:NUMBER`.

BL:NAME C010000 BL:X N010003 DOOR C008000

Options

- **Cdf**: fields separated by commas; texts in single quotes and numbers with a leading space, like AutoCAD.
- **Sdf**: fixed-width columns (texts left-aligned and numbers right-aligned).
- **Dxf**: a DXF with the references and their blocks (the AutoCAD DXX only carries the references and their attributes).

Notes

- Every block that has some attribute of the template is a record; `BL:LEVEL` and `BL:NUMBER` are 1 (nested blocks are not walked).

See also

ATTEXT · EATTEXT · ATTDEF

-BEDIT —Block Editor (command line)

Opens the Block Editor asking for the block name at the command line.

- **Command:** `-BEDIT` · **Global name:** `_-BEDIT` · **Alias:** `-BE`

Opens the Block Editor asking for the block name on the command line, without a dialog box. This is the BEDIT used in scripts.

Usage

1. Enter -BEDIT (or -BE).
2. Enter the block name (it may contain spaces), or `?` to see the list of blocks that can be edited.

```
Enter block name or [?]:
```

3. Edit the block and finish with BSAVE and BCLOSE.

Notes

- The same limitations as BEDIT: external references, layouts and anonymous blocks cannot be edited.
- In a script, BCLOSE saves pending changes without asking.

See also

BEDIT · BSAVE · BCLOSE · SCRIPT · Blocks and attributes

-BLOCK —Create Block (command line)

Creates a block on the command line, like AutoCAD -BLOCK.

- **Command:** `-BLOCK` · **Global name:** `_-BLOCK` · **Alias:** `-B`

Creates a block on the command line, with the same prompts as AutoCAD 2027 -BLOCK. This is what BLOCK does in scripts, SCRIPT and AutoLISP.

Usage

1. Enter -BLOCK (or -B) and the block name. Enter `?` to list the blocks in the drawing.

```
Enter block name or [?]:
```

2. If a block with that name already exists, confirm whether to redefine it.

```
Block "…" already exists. Redefine it? [Yes/No] <No>:
```

3. Specify the base point (or an option).

```
Specify insertion base point or [Annotative/MOde]:
```

4. Select the objects and press Enter: the block is created and the objects are left according to the mode.

```
Select objects:
```

Options

- **MOde**: what to do with the selected objects. It is kept for the next blocks (when LinceCAD starts it is *Convert to block*).

```
Choose mode for converting selected objects [Convert to block/Retain/Delete] <Convert to blo
ck>:
```

- **Convert to block**: replaces the objects with an insert of the block at the base point.
- **Retain**: leaves the objects as they were.
- **Delete**: deletes the objects from the drawing.
- **Annotative**: its prompts are accepted so AutoCAD scripts keep working, but the block is not annotative yet.

```
Create annotative block [Yes/No] <Y>:
```

See also

BLOCK · INSERT · WBLOCK

-IMAGE —Image (command line)

Lists, detaches, reloads, unloads or changes the path of the images of the drawing.

- **Command:** `-IMAGE` · **Global name:** `_-IMAGE` · **Alias:** `-IM`

Manages on the command line the images attached to the drawing: lists, detaches, changes the path of, reloads or unloads them. Images are given by the name of their definition (usually the file name).

Usage

1. Enter -IMAGE.
2. Choose the option.

```
Enter image option [?/Detach/Path/Reload/Unload/Attach] <Attach>:
```

3. Enter the image names separated by commas, or * (or Enter) for all of them.

```
Enter images (names separated by commas, or * for all) <*>:
```

4. With **Path**, enter the new path or ~ to choose it in a dialog box.

```
Enter the new file path (~ to choose it):
```

Options

- **?**: lists the images with their status (Found, Not found or Unloaded), size in pixels and saved path.
- **Detach** (D): erases the images and their definition from the drawing.
- **Path** (P): changes the file path, for example when it has been moved.
- **Reload** (R): reads the file again (if it has changed) and reloads unloaded images.
- **Unload** (U): stops showing the image without removing it from the drawing; only its frame is shown.
- **Attach** (A): attaches a new image, like IMAGEATTACH. This is the default.

Notes

- The buttons of the XREF window use this command.
- All changes are undone with UNDO.

See also

IMAGEATTACH · XREF · IMAGECLIP · Raster images

-INSERT —Insert (command line)

Inserts a block or a drawing asking at the command line.

- **Command:** `-INSERT` · **Global name:** `_-INSERT` · **Alias:** `-I`
- Ribbon: Insert › Block
- Menu: Insert

Inserts a block or a drawing asking everything on the command line, without opening the Blocks palette. This is the INSERT used in scripts.

Usage

1. Enter -INSERT (or -I).
2. Enter the block name. You can also enter the path of a DWG or DXF file (`C:\drawings\door.dwg`), `name=path` to give it another name, ~ to choose the file in a dialog box or `?` to list the blocks.

```
Enter block name or [?]:
```

3. Specify the insertion point. The block follows the cursor.

```
Specify insertion point or [Scale/Rotate]:
```

4. Enter the scale factor or specify a point: its distance to the insertion point is the scale.

```
Specify scale factor <1>:
```

5. Enter the rotation angle in degrees or specify a point in the desired direction.

```
Specify rotation angle <0>:
```

6. If the block has attributes, enter the value of each one; Enter keeps the default value.

Options

- **Scale** (S): before specifying the point, sets the scale; the block already moves at that size.
- **Rotate** (R): before specifying the point, sets the rotation.

Notes

- The scale is uniform (the same in X, Y and Z) and cannot be 0. For a different scale on each axis, use the INSERT palette.
- An inserted drawing is copied as a block named after the file, with its layers, linetypes, styles and blocks. If a block with that name already exists, the existing one is used.
- The preview shows at most 2000 objects of the block.

See also

INSERT · BLOCK · WBLOCK · SCRIPT · Blocks and attributes

-PDFATTACH —Attach PDF (command line)

Attaches a page of a PDF file as an underlay, without dialog boxes.

- **Command:** `-PDFATTACH` · **Global name:** `_-PDFATTACH`

Attaches a page of a PDF as an underlay answering everything on the command line, for scripts and AutoLISP. It does the same as PDFATTACH with the same prompts.

Usage

1. Enter -PDFATTACH.
2. Enter the PDF path (in quotes if it has spaces). Enter or ~ open the file dialog.

```
Enter path to PDF file to attach or [Browse] <Browse>:
```

3. Enter the page number; ? lists them.

```
Enter page number or [?] <1>:
```

4. Specify the insertion point (the lower-left corner of the page).

```
Specify insertion point:
```

5. Enter the scale.

```
Specify scale factor <1>:
```

6. Enter the rotation angle.

```
Specify rotation <0>:
```

Example

In a script, `-PDFATTACH` followed by `C:\plans\floor.pdf`, `2`, `0,0`, `1` and `0` attaches page 2 at the origin, at scale 1 and without rotation.

Notes

- A page that does not exist, or a file that is not a readable PDF, is rejected and the prompt is repeated.

See also

PDFATTACH · -XREF · XREF · External references

-REFEDIT —Edit Reference In-Place (command line)

Edits a block in place, like AutoCAD -REFEDIT.

- **Command:** `-REFEDIT` · **Global name:** `_-REFEDIT`

Edits a block in place from the command line, like AutoCAD 2027 -REFEDIT: the same prompts as REFEDIT.

Usage

1. Enter -REFEDIT and select a block reference.

```
Select reference:
```

2. Accept the nesting level, the method (*All*) and whether attribute definitions are shown.

```
Enter object selection method [All/Nested] <All>:
```

3. Edit and finish with REFCLOSE.

See also

REFEDIT · REFCLOSE

-WBLOCK —Write Block (command line)

Saves a block, the drawing or some objects as a DWG, like AutoCAD -WBLOCK.

- **Command:** `-WBLOCK` · **Global name:** `_-WBLOCK` · **Alias:** `-W`

Saves a block, the whole drawing or some objects as a new drawing on the command line, with the AutoCAD 2027 -WBLOCK prompts. This is what WBLOCK does in scripts, SCRIPT and AutoLISP.

Usage

1. Enter -WBLOCK (or -W) and the path of the new file (without `.dwg` or `.dxf`, `.dwg` is added).

```
Enter name of output file:
```

2. If the file already exists, confirm whether to replace it.

```
A file named … already exists. Do you want to replace it? [Yes/No] <N>:
```

3. Enter the name of a block, = (the block named like the file), `*` (the whole drawing) or press Enter to define a new drawing from objects.

```
Enter name of existing block or [= (block=output file)/* (whole drawing)] <define new drawin
g>:
```

4. With Enter, specify the base point (or MOde) and select the objects.

```
Specify insertion base point or [MOde]:
```

Options

- **MOde**: what to do with the objects. It is kept for the next time and is separate from the -BLOCK one (when LinceCAD starts it is *Retain*).

```
Choose mode for converting selected objects [Convert to block/Retain/Delete] <…>:
```

See also

WBLOCK · -BLOCK · -INSERT

-XBIND —Bind Reference Symbols (command line)

Binds single symbols of an external reference, like AutoCAD -XBIND.

- **Command:** `-XBIND` · **Global name:** `_-XBIND` · **Alias:** `-XB`

Binds single symbols of an external reference to the drawing from the command line, like AutoCAD 2027 -XBIND.

Usage

1. Enter -XBIND (or -XB) and the symbol type.

```
Enter symbol type to bind [Block/Dimstyle/LAyer/LType/Style]:
```

2. Enter the dependent names (`REF|name`), separated by commas; wildcards are allowed (`REF|*`).

```
Enter dependent Layer name(s):
```

See also

XBIND · XREF

-XREF —External References (command line)

Lists, binds, detaches, unloads, reloads or changes the path of the external references.

- **Command:** `-XREF` · **Global name:** `_-XREF` · **Alias:** `-XR`

Manages external references from the command line: lists them, binds them to the drawing, detaches, unloads or reloads them, changes their path or attaches another one. It does the same as the buttons of the XREF window, and it can be used in scripts and AutoLISP.

Usage

1. Enter -XREF (or -XR).
2. Choose an option; Enter attaches another reference, like XATTACH.

```
Enter an option [?/Bind/Detach/Path/pathType/Unload/Reload/Overlay/Attach] <Attach>:
```

3. Enter the names of the references. Wildcards are accepted: `*` all of them, `?` one character, and several names separated by commas.

```
Enter xref name(s) <*>:
```

Options

- **?**: lists the references that match the name, with their type (attached or overlay), path and status.

```
Enter xref(s) to list <*>:
```

- **Bind** (B): turns the reference into a normal block of the drawing, with all its content and that of its nested references. With BINDTYPE = 0 its layers, styles and blocks are renamed `ref$0$name`; with 1 they keep their names (if one already exists, the drawing's own is used).
- **Detach** (D): removes the reference and all its inserts. It also removes PDF underlays.
- **Unload** (U) and **Reload** (R): stop showing the reference without removing it, or read it again from disk. They also work with PDF underlays.
- **Path** (P): changes the saved path of each reference; Enter or ~ opens the dialog to choose the file.

```
Xref name: …. Enter new path or <Browse>:
```

- **pathType** (T): saves the path of the loaded references as full, relative or just the file name.

```
Path type [Full/Relative/No path] <Relative>:
```

- **Attach** (A) and **Overlay** (O): attach or overlay another drawing (see XATTACH).

Notes

- Nested references are not bound, detached or unloaded on their own: they are managed from the one that contains them.
- The bound result matches what AutoCAD does: the drawing can be opened there without the reference.
- Everything can be undone with UNDO.

See also

XREF · XATTACH · XCLIP · PDFATTACH · External references

ADCENTER —DesignCenter

Browses other drawings and copies their blocks, layers, linetypes, styles and layouts.

- **Command:** `ADCENTER` · **Global name:** `_ADCENTER` · **Alias:** `ADC`, `DC`, `DCENTER`, `CONTENT`
- Ribbon: Insert › Content
- Ribbon: View › Palettes
- Menu: Insert

Opens the **DesignCenter** window, which browses other drawings and copies their blocks, layers, linetypes, text styles, dimension styles and layouts into the current one. Enter the command again to close it.

Usage

1. Enter ADCENTER (or ADC, DC) or click the button.
2. In the tree on the left, choose a drawing under **Open Drawings** or in a library folder. **Add Folder…** adds another folder to the tree (and to the Libraries tab of INSERT).
3. Choose the category at the top right; each one shows how many entries the drawing has.
4. Choose an entry and click a button:

- **Insert** (blocks only): starts inserting the block, like a click in the Blocks palette. Double-clicking the thumbnail or dragging it onto the drawing does the same.
- **Add to Tool Palette** (blocks only): adds it to the visible tool palette (TOOLPALETTES).
- **Add to Drawing**: copies the entry into the current drawing with whatever it needs (for example, a dimension style with its text style). Double-clicking a layer or a style does the same.

Notes

- The tree has **Open Drawings**, **History** (recent files) and the library folders; the search field above filters drawings by name.
- The palette can be docked to the left or right of the drawing and set to auto-hide with the buttons in its header (? ? ? ?); see The user interface.
- If the current drawing already has an entry with that name, its own is kept and a message is shown on the command line.
- Copying into the drawing is undone with UNDO in a single step.
- An open drawing that has not been saved yet cannot be added to a tool palette: the tool needs its file.
- Drawings larger than 20 MB in library folders are not browsed; insert them with **Browse…** in the Blocks palette.

See also

INSERT · TOOLPALETTES · LAYER · Blocks and attributes

ATTDEF — Define Attributes

Creates an attribute definition for a block.

- **Command:** `ATTDEF` · **Global name:** `_ATTDEF` · **Alias:** `ATT`, `DDATTDEF`
- Ribbon: Insert › Attributes
- Menu: Insert

Creates an attribute definition: a variable text that, once included in a block, asks for its value each time the block is inserted (a drawing number, a reference, a manufacturer...). It is shown with its tag until it is included in a block with BLOCK.

Usage

1. Enter ATTDEF (or ATT) or click the button.
2. In the **Attribute Definition** dialog box, choose: - **Mode**: Invisible, Constant, Verify, Preset and Lock position (the ones used last time). - **Attribute**: the tag (a single word without spaces, converted to uppercase), the prompt shown when the block is inserted and the default value. - **Insertion Point**: its coordinates or, with *Specify On-screen*, pick it after OK. - **Text Settings**: the justification, text style, height and rotation.
3. Click OK and, if you left it on-screen, specify the point.

In scripts, SCRIPT and AutoLISP there is no dialog box: everything is asked on the command line, as in -ATTDEF.

Options

- **Invisible**: the value is not shown in the drawing.
- **Constant**: the value is fixed and is not asked on insertion.
- **Verify**: flags the attribute so that its value is checked on insertion.
- **Preset**: flags the attribute so that it takes its default value.
- **Lock position**: the attribute cannot be moved apart from the block.

Notes

- The proposed height is the fixed height of the text style or, if it has none, the last one used (TEXTSIZE).
- In LinceCAD, verify and preset attributes are prompted on insertion like the others; the modes are saved for AutoCAD.
- The modes, prompt and default value can be changed later with BATTMAN.
- Annotative and multiline attributes are not available yet. The dialog box cannot be scripted in the AutoCAD console: it has a unit test.

See also

-ATTDEF · BLOCK · BATTMAN · ATTEDIT · ATTEXT · Blocks and attributes

ATTDISP —Attribute Display

Shows or hides block attributes.

- **Command:** `ATTDISP` · **Global name:** `_ATTDISP`
- Ribbon: Insert › Attributes

Shows or hides block attributes in the whole drawing, like AutoCAD ATTDISP. It changes the ATTMODE variable and redraws the blocks.

Usage

1. Enter ATTDISP.
2. Choose the visibility.

```
Enter attribute visibility setting [Normal/ON/OFF] <…>:
```

Options

- **Normal** (ATTMODE 1): each attribute is shown according to its *Invisible* mode.
- **ON** (ATTMODE 2): all are shown, also the invisible ones.
- **OFF** (ATTMODE 0): none is shown.

See also

ATTDEF · ATTEDIT · SETVAR

ATTEDIT —Edit Attributes

Changes the attribute values of a block.

- **Command:** `ATTEDIT` · **Global name:** `_ATTEDIT` · **Alias:** `ATE`, `DDATTE`
- Ribbon: Insert › Attributes

Changes on the command line the values of the attributes of an inserted block.

Usage

1. Enter ATTEDIT (or ATE). Double-clicking a block with attributes also starts it.
2. Select the block. If it was already selected, that one is used.

```
Select a block with attributes:
```

3. For each attribute, in order, enter the new value or press Enter to keep the current one.

```
Enter new value for … <…>:
```

The changes are applied when the last attribute is answered.

Notes

- If several objects are selected, the first block with attributes is edited.
- Only the value changes; position, height and modes stay as they were. To change the attribute definitions, use BATTMAN.
- To edit the values in a window, use EATTEDIT.

- No dialog box in this version: it is used from the command line.

See also

EATTEDIT · BATTMAN · ATTDEF · Blocks and attributes

ATTEXT — Extract Attributes

Extracts block attributes to a CSV file or a table.

- **Command:** `ATTEXT` · **Global name:** `_ATTEXT` · **Alias:** `DDATTEXT`
- Ribbon: Insert › Attributes
- Menu: Insert

Extracts the attribute values of blocks to a table in the drawing or to a CSV file that opens in a spreadsheet. It is useful for door, device or luminaire schedules.

Usage

1. Enter ATTEXT or click the button. To also output the other objects (count, layer, length and area), use DATAEXTRACTION.
2. Select the blocks and press Enter, or press Enter without selecting to take all those in model space.

```
Select blocks or [ALL] <ALL>:
```

3. Choose the output.

```
Output [Table/File] <Table>:
```

4. With **Table**, specify the upper-left corner of the table.

```
Upper-left corner of the table:
```

5. With **File**, enter the file path.

```
CSV file:
```

Options

- **ALL** (ALL): every block with attributes in model space.
- **Table** (T): a table titled "Block Attributes", with a header row and one row per block.
- **File** (F): a comma-separated CSV file.

Notes

- The columns are the block name, the X and Y coordinates of its insertion point and one attribute tag per column, in the order they appear. There is no wizard to choose columns: they are all output.
- If the path does not end in `.csv` or `.txt`, `.csv` is added. The file has a UTF-8 mark so that the spreadsheet reads accented characters correctly.
- The table is created with the current text height and is edited like any other table (TABLE).

See also

ATTDEF · BATTMAN · TABLE · Blocks and attributes

ATTIPEDIT — Edit Attribute In Place

Changes the value of an attribute selected in the drawing.

- **Command:** `ATTIPEDIT` · **Global name:** `_ATTIPEDIT` · **Alias:** `ATI`
- Ribbon: Insert › Attributes

Changes the value of an attribute by selecting it in the drawing, like AutoCAD ATTIPEDIT.

Usage

1. Enter ATTIPEDIT (or ATI) and click the text of the attribute.

```
Select attribute:
```

2. Enter the new value (Enter leaves it as it was).

```
Enter new value <…>:
```

Notes

- AutoCAD opens the text editor over the attribute; LinceCAD asks for the value on the command line. The attribute picked is the one nearest to the click, using an approximate text box (without the font metrics).
- To change several attributes at once, use EATTEDIT, ATTEDIT or -ATTEDIT. It cannot be scripted in the AutoCAD console (it opens the in-place editor): it has a unit test.

See also

EATTEDIT · ATTEDIT · -ATTEDIT · ATTDEF

ATTSYNC — Synchronize Attributes

Updates the attributes of a block's inserts to match its definition.

- **Command:** `ATTSYNC` · **Global name:** `_ATTSYNC`
- Ribbon: Insert › Attributes
- Menu: Insert

Updates the attributes of every insert of a block from its current definition: after changing a block's attribute definitions (with BEDIT or BLOCK), the existing inserts are brought up to date.

Usage

1. Enter ATTSYNC or click the button.
2. Select an insert of the block or enter its name.

```
Select a block or type its name:
```

The number of updated inserts is shown.

Notes

- The attributes are placed again as in the definition: position, height, style and modes.
- Values are kept by tag. A new attribute takes its default value and one that is no longer in the definition disappears from the inserts.
- Inserts in model space and in layouts are updated.
- The change is undone with UNDO.

See also

BATTMAN · ATTDEF · BEDIT · Blocks and attributes

BATTMAN — Block Attribute Manager

Changes the order, prompt, default value and modes of a block's attributes.

- **Command:** `BATTMAN` · **Global name:** `_BATTMAN`
- Ribbon: Insert › Attributes
- Menu: Insert

Opens the **Block Attribute Manager**: it changes the attribute definitions of a block (their order, prompt, default value and modes) and applies the changes to all its inserts.

Usage

1. Enter BATTMAN or click the button. If a block with attributes is selected, it opens with that block; otherwise, with the first one in the drawing.
2. Choose another block in **Block:** if needed. Below it you can see how many inserts it has.
3. Change in the table the **Prompt**, the **Default** and the **Modes**: I (Invisible), C (Constant), V (Verify) and P (Preset).
4. Click a tag to choose its row and use **Move Up**, **Move Down** or **Remove**.
5. Click **Synchronize** to apply the changes without closing, or **OK** to apply them and close.

Notes

- When applying, every insert of the block rebuilds its attributes from the definition and keeps the values it had (new attributes take the default value). This is what ATTSYNC does.
- The order of the table is the order in which INSERT asks for the values.
- The tag cannot be changed here. A constant attribute has no prompt.
- The change is undone with UNDO.

See also

ATTDEF · ATTSYNC · EATTEDIT · BEDIT · Blocks and attributes

BATTORDER — Attribute Order

Changes the order in which the block attributes are prompted (in the Block Editor).

- **Command:** `BATTORDER` · **Global name:** `_BATTORDER`

Changes the order of the attribute definitions of the block being edited, like AutoCAD BATTORDER: it is the order in which INSERT asks for their values. It only works in the Block Editor (BEDIT).

Usage

1. In the Block Editor, enter BATTORDER or click the button on the editor tab.
2. In the **Attribute Order** dialog box, pick a tag and move it with **Move Up** and **Move Down**.
3. Click OK and save the block with BSAVE.

In scripts, SCRIPT and AutoLISP there is no dialog box: enter the tags in the new order, separated by commas; the ones you leave out go after them, in their order.

```
Enter the tags in the new order (separated by commas):
```

Notes

- Existing inserts keep the order of their attributes until they are synchronized with ATTSYNC.
- AutoCAD prompts the attributes of a block made with a window in reverse order (from the last created to the first), and so does LinceCAD: this command fixes it. The dialog box cannot be scripted in the AutoCAD console (BATTORDER does not exist outside the Block Editor, which cannot be scripted either): it has a unit test.

See also

BEDIT · ATTDEF · BATTMAN · ATTSYNC

BCLOSE — Close Block Editor

Closes the Block Editor and returns to the drawing.

- **Command:** `BCLOSE` · **Global name:** `_BCLOSE` · **Alias:** `BC`

Closes the Block Editor and returns to the drawing. It can also be closed with the button on the editor's bottom bar.

Usage

1. In the Block Editor (BEDIT), enter BCLOSE or click the button.
2. If there are unsaved changes, choose whether to save them:

```
Save changes to "…"?
```

- **Yes**: saves the block, like BSAVE, and closes.
- **No**: closes without saving what changed since the last BSAVE.
- **Cancel**: stays in the editor.

Notes

- The whole editor session (every BSAVE) is undone at once with UNDO.
- In scripts there is no question: pending changes are saved.
- Outside the Block Editor, the command only shows a message.

See also

BSAVE · BSAVEAS · BEDIT · Blocks and attributes

BEDIT — Block Editor

Opens a block definition in the Block Editor.

- **Command:** `BEDIT` · **Global name:** `_BEDIT` · **Alias:** `BE`
- Ribbon: Home › Block
- Ribbon: Insert › Block
- Menu: Insert

Opens a block definition in the Block Editor. Its contents are edited as if they were a drawing, with every command, object snap and grip; when it is saved, every insert of the block is updated.

Usage

1. Enter BEDIT (or BE) or click the button. Double-clicking a block without attributes also opens it.
2. In the **Edit Block Definition** dialog box, choose the block in the list (its preview and description are shown)

and click **OK**. If a block was selected, it is already chosen.
3. Edit the block. The background changes color, the bottom bar shows "Block Editor: …"and the ribbon switches to the **Block Editor** tab.
4. Save with BSAVE or BSAVEAS and leave with BCLOSE.

Notes

- Point 0,0 in the editor is the block's base point.
- External references, layouts and anonymous blocks cannot be edited. For a dynamic block its definition is edited; its parameters and actions are kept, but new ones cannot be added.
- Inside the editor you cannot use BEDIT or PURGE, or switch between model and paper space.
- After closing, the whole editor session is undone with a single UNDO.
- Saving the file while the editor is open saves the drawing without the block's pending changes.
- In scripts, BEDIT asks for the name like -BEDIT.

See also

-BEDIT · BSAVE · BCLOSE · BLOCK · Blocks and attributes

BLOCK —Create Block

Groups objects into a reusable block.

- **Command:** `BLOCK` · **Global name:** `_BLOCK` · **Alias:** `B`, `ACADBLOCKDIALOG`, `BMAKE`, `BMOD`
- Ribbon: Home › Block
- Ribbon: Insert › Block
- Menu: Insert
- Menu: Draw

Creates a block definition from the selected objects: a named group of objects with a base point that can be inserted as many times as needed with INSERT. If the objects include attribute definitions (ATTDEF), the block asks for their values when it is inserted.

Usage

1. Enter BLOCK (or B) or click the button. Objects already selected are used as a starting point.
2. In the **Block Definition** dialog box, enter the name (or pick it from the list to redefine one) and choose: - **Base point**: its X, Y and Z coordinates or, with *Specify On-screen*, pick it in the drawing after OK. - **Objects**: with *Specify On-screen* they are selected after OK; otherwise the current selection is used. Choose whether they are **retained**, **converted to a block** (an insert at the base point) or **deleted**. - **Behavior**: *Scale uniformly* (the block can only be inserted with the same X, Y and Z scale) and *Allow exploding* (whether EXPLODE can explode it). - **Settings**: the block units and a description.
3. Click OK and, if you left them on-screen, specify the base point and select the objects.

In scripts, SCRIPT and AutoLISP there is no dialog box: everything is asked on the command line, as in -BLOCK.

Notes

- The name can be up to 255 characters long and contain spaces, but not `< > / \ " : ; ? * | , =` or the backquote.
- Redefining a block changes its contents and base point, and every insert of it is redrawn, including those nested in other blocks. A block cannot contain itself.
- Attributes are prompted in the order their definitions were created.
- External references, layouts and anonymous blocks cannot be redefined with BLOCK.
- Annotative blocks and opening the block in the editor on OK are not available yet. To change a block without

creating it again, use BEDIT. The dialog box cannot be scripted in the AutoCAD console: it has a unit test.

See also

-BLOCK · INSERT · ATTDEF · WBLOCK · BEDIT · Blocks and attributes

BLOCKREPLACE —Replace Block with Another

Changes every reference of a block to another one (Express Tools).

- **Command:** `BLOCKREPLACE` · **Global name:** `_BLOCKREPLACE`
- Ribbon: Insert › Block

Changes every reference of a block to another one, like BLOCKREPLACE from the AutoCAD Express Tools.

Usage

1. Enter BLOCKREPLACE and the name of the block to replace (`?` lists those in the drawing).

```
Enter the name of the block to replace or [?]:
```

2. Enter the name of the block that replaces it.

```
Enter the name of the block to replace it with or [?]:
```

3. Choose whether to purge the old block, which nothing uses any more.

```
Purge unreferenced items when finished? [Yes/No] <Y>:
```

Notes

- Every reference of the block changes, also those inside other blocks; each one keeps its point, scales, rotation, layer and the values of the attributes with the same tag.
- A block cannot be replaced with one that contains it. The Express Tools do not load in the AutoCAD console: it has a unit test, checked against their documentation.

See also

BREPLACE · PURGE · INSERT

BPARAMETER —Block Parameter

Adds a parameter to the dynamic block being edited.

- **Command:** `BPARAMETER` · **Global name:** `_BPARAMETER` · **Alias:** `PARAM`

Adds a parameter to the dynamic block being edited, like AutoCAD BPARAMETER. It only works in the Block Editor (BEDIT).

Usage

1. In the Block Editor, enter BPARAMETER (or PARAM) and the parameter type.

```
Enter parameter type [Alignment/Base/pOint/Linear/Polar/XY/Rotation/Flip/Visibility/looKup]
<Visibility>:
```

2. With **Visibility** it goes on as BVPARAM: the parameter point and the states with their objects.

Notes

- For now only visibility parameters are created. The other types (linear, polar, XY, rotation, flip, alignment,

lookup, point and base point) and their actions are read and work in dynamic blocks made with AutoCAD (grips and properties), but LinceCAD does not create them yet: the command says so.

See also

BVPARAM · BVSTATE · BEDIT

BREPLACE — Replace Blocks

Replaces the selected block references with another block.

- **Command:** BREPLACE · **Global name:** _BREPLACE
- Ribbon: Insert › Block

Replaces the selected block references with another block of the drawing, like AutoCAD BREPLACE.

Usage

1. Enter BREPLACE (or select the references first) and select the references to replace; press Enter.

```
Select block references to replace:
```

2. Enter the name of the block that replaces them (? lists those in the drawing).

```
Enter name of replacement block or [?]:
```

Notes

- Each reference keeps its insertion point, scales, rotation and layer. Its attributes are rebuilt from the definitions of the new block: those with the same tag keep their value and the rest take the default one.
- A reference of a dynamic block becomes a plain reference of the new block.
- In AutoCAD 2027, BREPLACE opens the Blocks palette (and asks to sign in to Autodesk): it cannot be checked in its console and has a unit test. To change every reference of a block, use BLOCKREPLACE.

See also

BLOCKREPLACE · INSERT · COUNT

BSAVE — Save Block

Saves the definition of the block being edited.

- **Command:** BSAVE · **Global name:** _BSAVE · **Alias:** BS

Saves the definition of the block being edited in the Block Editor. Every insert of the block is redrawn with the new contents, including those nested in other blocks and those of a dynamic block.

Usage

1. In the Block Editor (BEDIT), enter BSAVE or click the button on the **Block Editor** tab.
2. If the block's attributes have changed and it has inserts, you are asked whether to synchronize them:

```
The attributes of "…" changed. Synchronize its … references (ATTSYNC)?
```

The editor stays open; you can keep editing and save again.

Notes

- If the attributes are not synchronized now, it can be done later with ATTSYNC.
- The block cannot contain itself, either directly or through another block: in that case it is not saved.

- New objects in a dynamic block with visibility states are visible in every state.
- Saving the block does not save the file: do that afterwards with QSAVE or SAVEAS.
- Outside the Block Editor, the command only shows a message.

See also

BSAVEAS · BCLOSE · BEDIT · ATTSYNC · Blocks and attributes

BSAVEAS —Save Block As

Saves the edits as a new block and leaves the original unchanged.

- **Command:** BSAVEAS · **Global name:** _BSAVEAS

Saves what was edited in the Block Editor as a new block, with another name and the same base point. The original block stays as it was before editing.

Usage

1. In the Block Editor (BEDIT), enter BSAVEAS or click the button.
2. Enter the name of the new block; it must not already exist in the drawing.

```
Enter new block name:
```

The editor stays open, now with the new block.

Notes

- The editor's undo history starts again with the new block.
- The copy of a dynamic block is a regular block: it does not carry its parameters or actions.
- The name follows the same rules as in BLOCK.
- Outside the Block Editor, the command only shows a message.

See also

BSAVE · BCLOSE · BEDIT · BLOCK · Blocks and attributes

BTESTBLOCK —Test Block

Tests the block being edited, without saving it, in a test drawing.

- **Command:** BTESTBLOCK · **Global name:** _BTESTBLOCK · **Alias:** BTEST

Tests the block being edited without saving it, like AutoCAD BTESTBLOCK. It only works in the Block Editor (BEDIT).

Usage

1. In the Block Editor, enter BTESTBLOCK.
2. Another tab opens with a temporary drawing that has a single reference to the block as it is in the editor: try its grips and its Properties palette.
3. Close that tab (without saving) and go back to the editor.

Notes

- In AutoCAD the test happens in a window inside the editor; here it is a tab with a temporary file. The block of the drawing being edited does not change. It cannot be scripted in the AutoCAD console (the Block Editor does not exist there).

See also

BEDIT · BSAVE · BVPARAM

BVPARAM —Visibility Parameter

Adds a visibility parameter to the block being edited.

- **Command:** BVPARAM · **Global name:** _BVPARAM

Adds a visibility parameter to the block being edited in the Block Editor and defines its states. A visibility state shows or hides a set of the block's objects; when inserting the block, the state is chosen from a drop-down. The block becomes dynamic and AutoCAD recognizes it as such.

Usage

1. Open the block in the Block Editor (BEDIT).
2. Enter BVPARAM or click **Visibility Parameter** on the editor bar.
3. Specify the point for the visibility grip.
4. For each state: type its name and select the objects visible in that state; press Enter to finish the state.
5. With an empty name, and having defined two or more states, the visibility parameter is created.
6. Save the block with BSAVE.

Notes

- At least two states are required.
- Block objects not present in any state are hidden in all of them.
- The active state is the first one defined; it is baked into the geometry for plain readers.
- To discard the changes of an already-inserted dynamic block, use RESETBLOCK.
- Outside the Block Editor, the command only shows a message.

See also

BEDIT · BSAVE · RESETBLOCK · Blocks and attributes

BVSTATE —Visibility States

Creates, renames, deletes and orders the visibility states of the block.

- **Command:** BVSTATE · **Global name:** _BVSTATE · **Alias:** BVS

Manages the visibility states of the dynamic block being edited, like AutoCAD BVSTATE. It only works in the Block Editor of a block with a visibility parameter (BVPARAM).

Usage

1. In the Block Editor, enter BVSTATE (or BVS). In the **Visibility States** dialog box, pick a state and click **New…**, **Rename…** (the name is typed on the command line), **Delete**, **Move Up** or **Move Down**; **Close** finishes.
2. Save the block with BSAVE.

In scripts, SCRIPT and AutoLISP there is no dialog box: the option and the state are asked.

```
Enter an option [New/Rename/Delete/Up/Down] <done>:
```

Notes

- A new state starts with every object of the parameter visible. Choosing what is visible in each state inside the

editor (the AutoCAD current state) is not available yet: define them when creating the parameter. The order is the one of the Properties palette list. The dialog box cannot be scripted in the AutoCAD console: it has a unit test.

See also

BVPARAM · BPARAMETER · BEDIT

CADASTREDOWNLOAD —Download cadastre

Downloads the parcel, buildings and cadastral reference from the Spanish Cadastre.

- **Command:** `CADASTREDOWNLOAD` · **Global name:** `_CADASTREDOWNLOAD` · **Alias:** `CADASTRE`

Downloads the **parcel**, its **buildings** and the **cadastral reference** from the Spanish Cadastre (Dirección General del Catastro) and draws them in the drawing coordinate system (GEOGRAPHICLOCATION), each on its own layer.

Use

1. Type CADASTREDOWNLOAD (or CADASTRE). The **Download from the Cadastre** dialog opens with three tabs, like the Cadastre web office.
2. **Urban, by address**: pick the **Province** and the **Municipality** (the lists are requested from the Cadastre and have a search box), type the **Street name** (after three letters the list of streets appears so you can pick one, with its **Street type**) and the **Number**.
3. **Cadastral reference**: type the 14 characters of the parcel or the 20 of the property. With 20, the two check characters are verified before anything is requested.
4. **Rural, by parcel**: province, municipality, **Polygon** and **Parcel**.
5. Tick **Also draw the buildings** if you want them and click **Download**. If the address matches several parcels, the dialog lists them so you can pick one.

What it draws

Layer	Content
CATASTRO-PARCELA	The parcel, as a closed polyline
CATASTRO-EDIFICIO	The buildings on the parcel (with their courtyards)
CATASTRO-TEXTO	The cadastral reference, at the parcel reference point

Layers are created if missing. The drawing needs a coordinate system: run GEOGRAPHICLOCATION first. Data is requested in ETRS89 (EPSG:4258) and reprojected to the drawing system, so it works all over Spain, Canary Islands included.

Notes

- Source: **Dirección General del Catastro** (free public services: street directory and non-protected data query, and INSPIRE WFS for parcels and buildings). Credit it wherever you use the data.
- Usage limit: at most 30 requests per minute; the province, municipality and street lists are kept for the session so they are not requested twice.
- Errors: no connection, a reference or address the Cadastre does not know, and a service that is down are reported with their cause.
- Cadastral cartography has the capture scale of each area: it does not replace a survey.

See also

MAPSERVICE · WFS · WMS · GEOGRAPHICLOCATION

CLASSICINSERT —Insert (classic dialog)

Inserts a block or a drawing with the classic Insert dialog box.

- **Command:** `CLASSICINSERT` · **Global name:** `_CLASSICINSERT` · **Alias:** `DDINSERT`
- Ribbon: Insert › Block

Inserts a block or a drawing with the classic AutoCAD **Insert** dialog box (CLASSICINSERT), instead of the Blocks palette of INSERT.

Usage

1. Enter CLASSICINSERT (or DDINSERT). Type or pick the block name, or click **Browse…** to insert a DWG or DXF file.
2. Choose the insertion point, the scale (X, Y and Z, or uniform) and the rotation, each one fixed in the dialog box or *Specify On-screen*, and whether the block is exploded on insertion.
3. Click OK and specify on screen what you left for later (and the attribute values).

In scripts, SCRIPT and AutoLISP there is no dialog box: it is -INSERT.

Notes

- The dialog box cannot be scripted in the AutoCAD console: it has a unit test.

See also

INSERT · -INSERT · BLOCK

CONVERTIMAGE —Convert Image

Converts ECW, MrSID or JPEG 2000 images to GeoTIFF with GDAL and attaches them.

- **Command:** `CONVERTIMAGE` · **Global name:** `_CONVERTIMAGE` · **Alias:** `CONVERTIRIMAGEN`
- Ribbon: Insert › Image and tracing

Converts the attached images that LinceCAD cannot read (ECW, MrSID and JPEG 2000) to GeoTIFF and attaches the result in their place. In Spain, the CNIG's PNOA orthophotos are distributed as ECW: with this command they can be viewed and plotted.

The conversion is done by **GDAL**, installed separately as an external program: LinceCAD neither includes nor installs it.

Usage

1. Enter CONVERTIMAGE or click the button on the Insert tab. It also opens from the warning shown when a drawing with such images is opened, and from the XREF window (the image's **Convert with GDAL…** button or its shortcut menu).
2. The **Convert Images with GDAL** dialog shows the GDAL found and the drawing's images that cannot be read, all checked. Uncheck the ones you do not want to convert. In **Standalone file:** you can also convert a file that is not in the drawing (it is not attached).
3. Click **Convert**. The conversion runs in the background, with its progress and **Cancel**: you can keep working meanwhile.
4. When each image is done, its path in the drawing changes to the GeoTIFF and it is reloaded:

```
… converted to … (… MB) and attached.
```

Where GDAL is looked for

In this order: the `GDAL_HOME` variable, `gdal_translate` on the PATH, OSGeo4W (`C:\OSGeo4W\bin` and `C:\OSGeo4W64\bin`), QGIS (`C:\Program Files\QGIS …\bin`, the newest version) and the folder chosen with **Choose GDAL folder…**, which is saved in the preferences. Of those found, the first one that reads all the needed formats is used (`gdalinfo --formats`).

If there is no GDAL, the dialog says so and offers the download: **QGIS** (recommended, because its installer includes the ECW and MrSID drivers) at qgis.org/download or **OSGeo4W** at trac.osgeo.org/osgeo4w. After installing it, click **Search again**. If the GDAL found cannot read ECW, install QGIS or, in OSGeo4W, the `gdal-ecw` package.

The file created

- Tiled GeoTIFF (`TILED=YES`, `BIGTIFF=IF_SAFER`) with **DEFLATE** compression and horizontal predictor: lossless and fast to read in tiles.
- Same size in pixels and same georeferencing: the image covers exactly the same place in the drawing.
- Next to the original, with the same name and the `.tif` extension (if it already exists, "name (2).tif": nothing is ever overwritten). If that folder is not writable, in the **Output folder:** folder.
- **The original is not deleted.**

Notes

- The new path is of the same type as the image had: relative to the drawing's folder or full.
- The path change is undone with U like any other change; when saved, the drawing points to the `.tif`.
- In a script, without dialogs, CONVERTIMAGE directly converts all the images it can.
- Until it is converted, the image is shown as a frame with its name and the text "Unsupported format", different from that of a file that is not found.

See also

-IMAGE · IMAGEATTACH · GEOIMAGE · Raster images · Georeferencing

COUNT —Count Blocks

Counts the block references in an area.

- **Command:** `COUNT` · **Global name:** `_COUNT`
- Ribbon: Insert › Block

Counts the block references in an area, with the AutoCAD 2027 COUNT prompts, and writes how many there are of each block.

Usage

1. Enter COUNT and choose the area: specify its two corners or an option.

```
Specify first corner of count area or [Current area/Entire model space/Object/Polygonal] <Cu
rrent area>:
```

2. Select references of the blocks to count and press Enter, or press Enter to count them all.

```
Select target objects or [List all blocks] <List all blocks>:
```

Options

- **Current area** and **Entire model space**: the whole model space.
- **Object**: what is inside a closed object (polyline, circle...).

- **Polygonal**: what is inside a polygon; specify its points and press Enter.

Notes

- A reference is counted if its insertion point lies inside the area. Dynamic blocks count as their definition, whatever their state.
- *Current area* counts the whole model space (in AutoCAD, what is visible in the viewport). AutoCAD shows the result in the Count palette (with table and fields); LinceCAD writes it on the command line. It cannot be scripted in the AutoCAD console (it opens the palette): it has a unit test.

See also

INSERT · BREPLACE · QSELECT

DATAEXTRACTION —Data Extraction

Extracts object data (count, layer, length, area and attributes) to a table or a CSV file.

- **Command:** `DATAEXTRACTION` · **Global name:** `_DATAEXTRACTION` · **Alias:** `DX`, `EXTRACDAT`
- Ribbon: Annotate › Tables
- Menu: Insert

Extracts data from the objects in the drawing to a table or to a CSV file, like AutoCAD's data extraction with its default options: how many objects of each kind there are, their layer, length and area, and the attributes of the blocks. Useful for quantity take-offs and bills of materials.

Usage

1. Enter DATAEXTRACTION (or EXTRACDAT) or click the button.
2. Select the objects and press Enter, or press Enter without selecting to take all those in model space.

```
Select blocks or [ALL] <ALL>:
```

3. Choose the output.

```
Output [Table/File] <Table>:
```

4. With **Table**, specify the upper-left corner of the table.

```
Upper-left corner of the table:
```

5. With **File**, type the path of the file.

```
CSV file:
```

Columns

- **Count**: how many identical objects the row stands for (identical rows are combined, as in AutoCAD).
- **Name**: the block name or the object type.
- **Layer**, **Length** and **Area**: the length is that of lines, polylines and arcs and the circumference of circles; the area, that of circles and polylines.
- One attribute tag per column, with the value of each block.

Columns without any value are left out.

Notes

- There is no wizard to choose objects, properties or filters, nor a `.dxe` template: the columns above are output.
- To output only the attributes with the position of each block, use ATTEXT.
- If the path does not end in `.csv` or `.txt`, `.csv` is added. The file has a UTF-8 mark so that spreadsheets read

accented characters correctly.

See also

ATTEXT · TABLE · DATALINK · Blocks and attributes

EATTEDIT — Enhanced Attribute Editor

Edits the attributes of the selected block in a window.

- **Command:** `EATTEDIT` · **Global name:** `_EATTEDIT`
- Ribbon: Insert › Attributes
- Menu: Insert

Edits in a window, **Enhanced Attribute Editor**, the attribute values of an inserted block: one row per attribute, with its tag and its value.

Usage

1. Select a block with attributes.
2. Enter EATTEDIT or click the button. The window opens with its attributes.
3. Change the values you want and click **OK**; **Cancel** closes without changes.

Notes

- If no block with attributes is selected, the command moves to the command line and asks like ATTEDIT.
- The window only changes values. Prompts, default values and modes are changed in the definition with BATTMAN.
- The change is undone with UNDO.

See also

ATTEDIT · BATTMAN · ATTSYNC · Blocks and attributes

EATTEXT — Attribute Extraction

Extracts block attributes to a table or a CSV, choosing blocks and columns.

- **Command:** `EATTEXT` · **Global name:** `_EATTEXT`
- Ribbon: Insert › Attributes

Extracts the attributes of the model space blocks to a table in the drawing or to a CSV file, choosing the blocks and columns, like AutoCAD EATTEXT.

Usage

1. Enter EATTEXT. In the **Attribute Extraction** dialog box, tick the blocks and attributes (the Block, X and Y columns are always included).
2. Choose the output: a table in the drawing (click OK and specify its insertion point) or a CSV file.

In scripts, SCRIPT and AutoLISP there is no dialog box: it is ATTEXT.

Notes

- AutoCAD does it with a multi-page wizard (with object properties, filters and a .dxe extraction file); LinceCAD with a single dialog box and attributes only. For template formats, use -ATTEXT. The dialog box cannot be scripted in the AutoCAD console: it has a unit test.

See also

ATTEXT · -ATTEXT · BATTMAN

GEOCOORD —Geographic Coordinates

Shows a point in another coordinate system.

- **Command:** `GEOCOORD` · **Global name:** `_GEOCOORD`
- Ribbon: Insert › Geolocation
- Ribbon: Mapping › Geolocation
- Menu: Insert
- Menu: Mapping

Shows the coordinates of a point of the drawing in another coordinate system: by default, its latitude and longitude in WGS84. It works like ID, but for geographic coordinates.

Usage

1. Enter GEOCOORD or click the button.
2. Specify the point (with an object snap or by typing its coordinates).

```
Specify point or [System] <EPSG:…>:
```

The command line shows the point's coordinates in the drawing's system and in the target system.

Options

- **System** (S): changes the target system. Enter its EPSG code or its AutoCAD code.

```
EPSG code of the target system <…>:
```

Notes

- The drawing needs its coordinate system (GEOGRAPHICLOCATION); if it has none, the command warns.
- In a geographic system, latitude and longitude are shown in degrees; in a projected one, X and Y in meters.
- Any system of the catalog can be used (over 300; see the table in Georeferencing). If the target has another datum, the accuracy is that of the datum shift given there.
- With the drawing's system set, the status bar always shows the cursor's latitude and longitude.

See also

GEOGRAPHICLOCATION · ID · MAPEXPORT · Georeferencing

GEOGRAPHICLOCATION —Geographic Location

Assigns a coordinate system (EPSG code) to the drawing.

- **Command:** `GEOGRAPHICLOCATION` · **Global name:** `_GEOGRAPHICLOCATION` · **Alias:** `GEO`, `NORTH`, `NORTHDIR`
- Ribbon: Insert › Geolocation
- Ribbon: Mapping › Geolocation
- Menu: Insert
- Menu: Mapping

Assigns a coordinate system to the drawing, for example ETRS89 / UTM 30N. With it, the drawing knows which point on Earth each coordinate corresponds to: the status bar shows the cursor's latitude and longitude, and GEOIMAGE, MAPIMPORT, MAPEXPORT and GEOCOORD can reproject.

Usage

1. Enter GEOGRAPHICLOCATION (or GEO) or click the button. The **Geographic Location** dialog box opens and, at the same time, the command line asks.
2. Choose the system in the dialog box and click **OK**, or enter its code on the command line: the EPSG code (`25830`, `2154`, `31983`) or the AutoCAD code (`ETRS89.UTM-30N`, `Lambert93`).

In the dialog box, systems are grouped by country or region (click a group to expand it). To find one, type part of its EPSG code, its name or the country in **Search (EPSG code, name or country)**: `2154`, `Lambert`, `France`, `UTM zone 19S`.

```
Enter EPSG code of the coordinate system or [Remove/List] <…>:
```

Options

- **Remove** (R): removes the coordinate system from the drawing.
- **List** (L): lists the available systems, with their EPSG code, AutoCAD code and name.

Notes

- The drawing is assumed to be already drawn in coordinates of that system, as is usual in survey drawings. A drawing with local coordinates cannot be placed on a point of the map.
- It is saved the way AutoCAD saves it; if AutoCAD does not know the system, its full definition is saved.
- There are 303 systems: WGS 84 / UTM in every zone, ETRS89 (UTM 28N–38N and LAEA), the national systems of Europe (Lambert-93, PT-TM06, RD New, LV95, Lambert 72 and 2008, PL-1992, Gauss-Krüger, British National Grid, ITM…), Latin America (SIRGAS 2000, Mexico ITRF2008, POSGAR 2007, SIRGAS-Chile, MAGNA-SIRGAS…) and North Africa (Merchich, Nord Sahara 1959, Carthage, Egypt 1907, LGD2006). The full table and the accuracy of each datum shift are in Georeferencing.
- With ED50, a message tells how it is transformed to ETRS89: with the official IGN grid if it is installed or, otherwise, with 7 parameters (about 1.5 m of error).

See also

GEOCOORD · GEOIMAGE · MAPIMPORT · MAPEXPORT · Georeferencing

GEOIMAGE — Insert Georeferenced Image

Inserts an image at its real coordinates, from its world file or its GeoTIFF tags.

- **Command:** `GEOIMAGE` · **Global name:** `_GEOIMAGE`
- Ribbon: Insert › Geolocation
- Ribbon: Mapping › Geolocation

Inserts a georeferenced image (an orthophoto, a scanned map) directly at its real coordinates, with its scale and rotation, without specifying any point.

Usage

1. Enter GEOIMAGE or click the button.
2. Enter the image path, or press Enter to choose it in a dialog box.

```
Enter image file name to attach or [Browse] <Browse>:
```

The image is placed and the command shows where the georeference came from and the image's coordinate system.

Notes

- The position comes from the world file next to the image (`.tfw`, `.jgw`, `.pgw`, `.bpw`, `.tifw`, `.wld`...) or, if there is none, from the GeoTIFF tags of the TIFF itself. Without either, the command warns and inserts nothing.
- The image's coordinate system is taken from its `.prj` file or from the GeoTIFF tags.
- If the image states its system and the drawing has one (GEOGRAPHICLOCATION), the image is reprojected to the drawing's system. If that is not possible, it is inserted without reprojection and a warning is shown.
- The path is saved relative to the drawing's folder, as in IMAGEATTACH.
- For an image without a georeference, place it with GEOREFIMAGE.

See also

GEOREFIMAGE · GEOGRAPHICLOCATION · IMAGEATTACH · Georeferencing

GEOREFIMAGE —Georeference Image

Moves an image into place with 2 or 3 control points.

- **Command:** `GEOREFIMAGE` · **Global name:** `_GEOREFIMAGE`
- Ribbon: Insert › Geolocation
- Ribbon: Mapping › Geolocation

Moves an image that is already inserted to its place using two or three control points: for each one, a recognizable point on the image (a corner, a crossing) and the real position it corresponds to in the drawing.

Usage

1. Enter GEOREFIMAGE or click the button.
2. Select the image.

```
Select the image to georeference:
```

3. Specify a point on the image.

```
Specify source point … (on the image):
```

4. Specify its real position (with an object snap or by typing its coordinates).

```
Specify destination point … (real position):
```

5. Repeat for the second point. Then specify a third one or press Enter to finish with two.

```
Specify source point … (on the image) or <finish>:
```

The image is moved, rotated and scaled, and the applied scale and rotation are shown.

Notes

- With two points the fit is exact. With three, the best fit for all three is calculated (least squares).
- The image is not distorted: the scale is the same on both axes.
- The preview lines join each image point to its destination.
- The change is undone with UNDO.

See also

GEOIMAGE · IMAGEATTACH · ALIGN · Georeferencing

IMAGEADJUST —Image Adjust

Changes the brightness, contrast and fade of images.

- **Command:** `IMAGEADJUST` · **Global name:** `_IMAGEADJUST` · **Alias:** `IAD`
- Ribbon: Insert › Image and tracing

Changes the brightness, contrast and fade of one or more raster images, for example to lighten an orthophoto so that what is drawn on top reads well.

Usage

1. Enter IMAGEADJUST (or IAD) or click the button.
2. Select the images and press Enter (anything that is not an image is ignored).

```
Select images:
```

3. Choose what to adjust. The current value is shown.

```
Enter image option [Contrast/Fade/Brightness] <Brightness>:
```

4. Enter the value, from 0 to 100 (Enter keeps the current one).

```
Enter … value (0-100):
```

Options

- **Brightness** (B): 50 is the original; lower darkens and higher lightens.
- **Contrast** (C): 50 is the original; lower flattens the tones and higher spreads them apart.
- **Fade** (F): 0 is the original; higher blends the image with the background.

Notes

- Each run changes a single setting; repeat the command for the others.
- The setting is saved in each image, as in AutoCAD, and the image file does not change.
- The result is close to AutoCAD's, but the formulas are LinceCAD's own and tones may differ slightly.

See also

TRANSPARENCY · IMAGECLIP · IMAGEATTACH · Raster images

IMAGEATTACH —Attach Image

Attaches a raster image (PNG, JPEG, BMP, TIFF, GIF or WebP) to the drawing.

- **Command:** `IMAGEATTACH` · **Global name:** `_IMAGEATTACH` · **Alias:** `IAT`
- Ribbon: Insert › Image and tracing

Attaches a raster image (PNG, JPEG, BMP, TIFF, GIF or WebP) to the drawing: a scanned drawing, a photo or an orthophoto. The image is not copied into the drawing: its path is saved and the file is read when the drawing opens.

Usage

1. Enter IMAGEATTACH (or IAT) or click the button.
2. Enter the image path, or press Enter to choose it in a dialog box. Its size at scale 1 is shown.

```
Enter image file name to attach or [Browse] <Browse>:
```

3. Specify the lower-left corner of the image (Enter: the origin).

```
Specify insertion point or [Path] <0,0>:
```

4. Enter the scale or specify a point: its distance to the insertion point is the width of the image.

```
Specify scale factor <1>:
```

5. Enter the rotation angle or specify a point in that direction.

```
Specify rotation angle <0>:
```

In the interface (not in scripts, SCRIPT or AutoLISP), after choosing the image the **Attach Image** dialog box opens: its size in pixels, the path type and the insertion point, scale and rotation; whatever you leave as *Specify On-screen* is specified afterwards, with the prompts of steps 3 to 5. The dialog box cannot be scripted in the AutoCAD console: it has a unit test.

Options

- **Browse** (B): opens the dialog box to choose the file (so do ~ and Enter).
- **Path** (P): chooses how the path is saved.

```
Path type [Relative/Full] <Relative>:
```

- **Relative** (R): relative to the drawing's folder; the drawing and the image can be moved together. If the drawing has not been saved yet or the image is on another drive, the full path is saved.
- **Full** (F): the full path.

Notes

- At scale 1, if the file states its resolution and the drawing has units (INSUNITS), the image comes in at its real size; if either is missing, it is 1 unit wide.
- If the file is not found at its path when the drawing opens, a file with the same name is looked for in the drawing's folder.
- The image frame is controlled with IMAGEFRAME. Images are listed in XREF.
- For an image with a world file or GeoTIFF tags, use GEOIMAGE: it is placed at its coordinates automatically.

See also

-IMAGE · IMAGECLIP · IMAGEADJUST · GEOIMAGE · Raster images

IMAGECLIP —Image Clip

Shows only part of an image, inside or outside a boundary.

- **Command:** `IMAGECLIP` · **Global name:** `_IMAGECLIP` · **Alias:** `ICL`
- Ribbon: Insert › Image and tracing

Shows only part of a raster image: what lies inside a rectangular or polygonal boundary or, with the boundary inverted, what lies outside. The image itself does not change: the boundary can be turned off or removed later.

Usage

1. Enter IMAGECLIP (or ICL) or click the button.
2. Select the image.

```
Select image to clip:
```

3. Choose the option; Enter creates a new boundary.

```
Enter image clipping option [ON/OFF/Delete/New boundary] <New>:
```

4. Choose the boundary type.

```
Outside mode: objects outside the boundary will be hidden. Specify clipping boundary [Polygo
nal/Rectangular/Invert clip] <Rectangular>:
```

5. For a rectangular one, specify two opposite corners.

```
Specify first corner point:
```

6. For a polygonal one, specify the vertices and press Enter (or choose Close) once there are at least three.

```
Specify next point or [Close/Undo]:
```

Options

- **ON** / **OFF** (ON / OFF): turns on or off the boundary the image already has.
- **Delete** (D): removes the boundary; the whole image is shown again.
- **New** (N): draws a new boundary, which replaces any existing one.
- **Rectangular** (R) and **Polygonal** (P): the boundary type.
- **Invert** (I): switches between hiding what is outside the boundary and hiding what is inside.

```
Inside mode: objects inside the boundary will be hidden. Specify clipping boundary [Polygona
l/Rectangular/Invert clip] <Rectangular>:
```

- **Undo** (U): in a polygonal boundary, removes the last vertex.

Notes

- The boundary is saved in the image as in AutoCAD.
- The change is undone with UNDO.

See also

IMAGEATTACH · IMAGEADJUST · -IMAGE · Raster images

IMAGECROP —Crop image

Crops (frames) a raster image to a rectangle.

- **Command:** `IMAGECROP` · **Global name:** `_IMAGECROP` · **Alias:** `IREC`
- Ribbon: Insert › Image and tracing

Crops (frames) a raster image to a rectangle. What is left outside is discarded in a new PNG file, next to the original (which is not touched), and the drawing image is relinked to that file.

Usage

1. Enter IMAGECROP (or IREC) or click the button.
2. Select the image.

```
Select the image to crop:
```

3. Specify the first corner.

```
Specify first corner point:
```

4. Specify the opposite corner.

```
Specify opposite corner point:
```

Notes

- The corners are specified in the drawing; you can snap to the image's own corners and edges.
- The insertion point and size of the drawing image are adjusted to the cropped piece; the pixel size is unchanged.
- The original file is not changed: the crop goes to a new file `<name>-ed.png`.
- It is a permanent crop of the pixels; to hide without cropping the pixels use IMAGECLIP.

See also

IMAGEEDIT · IMAGECLIP · IMAGEPERSP · Raster images

IMAGEEDIT —Edit image

Edits the pixels of a raster image: invert, threshold, despeckle, rotate, mirror, colour depth and resample.

- **Command:** `IMAGEEDIT` · **Global name:** `_IMAGEEDIT` · **Alias:** `IED`
- Ribbon: Insert › Image and tracing

Edits the pixels of a raster image in the drawing. The edited image is saved to a new PNG file, next to the original (which is not touched), and the drawing image is relinked to that new file.

Usage

1. Enter IMAGEEDIT (or IED) or click the button.
2. Select the image.

```
Select the image to edit:
```

3. Choose the operation.

```
Enter the operation [Invert/Threshold/despecKle/Rotate/Mirror/Depth/reSample] <Invert>:
```

4. Depending on the operation, enter the value (threshold, degrees of rotation, axis, depth or resample factor).

Options

- **Invert** (I): negative of the image (keeps transparency).
- **Threshold** (T): black or white by luminance, at the threshold you enter (0-255).
- **despecKle** (K): cleans speckle noise with a 3×3 median.
- **Rotate** (R): rotates the image 90, 180 or 270 degrees.
- **Mirror** (M): horizontal or vertical mirror.
- **Depth** (D): colour depth (Mono, Grey or Colour).
- **reSample** (S): changes the resolution by a factor, keeping the size in the drawing.

Notes

- The original file is never changed: the edit goes to a new file `<name>-ed.png`.
- Rotate and resample change the size in pixels; the drawing frame is adjusted to the new size.
- Vectorization (raster to vector), PDF to DWG and OCR are not done here: they are a later step of the plan.

See also

IMAGECROP · IMAGEPERSP · IMAGEPAINT · IMAGEADJUST · Raster images

IMAGEPAINT —Draw on image

Draws and erases strokes on a raster image.

- **Command:** `IMAGEPAINT` · **Global name:** `_IMAGEPAINT` · **Alias:** `IPIN`
- Ribbon: Insert › Image and tracing

Draws and erases strokes on a raster image, for example to cover a blemish or mark something on a scanned plan. The stroke is burned into the pixels of a new PNG file, next to the original (which is not touched), and the drawing image is relinked to it.

Usage

1. Enter IMAGEPAINT (or IPIN) or click the button.
2. Select the image.

```
Select the image to draw on:
```

3. Trace the polyline point by point. Press Enter to burn the stroke and finish.

```
Specify next point or [Colour/Width/Erase/Undo] or <finish>:
```

Options

- **Colour** (C): stroke colour (black, White or Red).
- **Width** (W): stroke width in pixels.
- **Erase** (E): toggles erase mode (leaves the pixels transparent).

Notes

- The points are specified in the drawing; you can snap to the image's own corners and edges.
- Erase leaves transparency; choose a colour again to keep drawing.
- The original file is not changed: the stroke goes to a new file `<name>-ed.png`.

See also

IMAGEEDIT · IMAGECROP · IMAGEADJUST · Raster images

IMAGEPERSP —Correct perspective

Corrects the perspective of an image (IFIX style) with 4 points and generates its world file.

- **Command:** `IMAGEPERSP` · **Global name:** `_IMAGEPERSP` · **Alias:** `IFIX`
- Ribbon: Insert › Image and tracing

Corrects the perspective of a raster image (IFIX style): with 4 points it straightens a quadrilateral into a rectangle, for example a façade or a plan photographed at an angle. The result is saved to a new PNG file with its world file (.wld), next to the original (which is not touched), and the drawing image is relinked to it.

Usage

1. Enter IMAGEPERSP (or IFIX) or click the button.
2. Select the image.

```
Select the image whose perspective to correct:
```

3. Specify the 4 corners of what should become rectangular, in order: lower left, lower right, upper right and upper left.

```
Specify corner … of 4 (lower left, lower right, upper right, upper left):
```

Notes

- The straightened image is placed axis-aligned, with the lower-left corner at the first point.
- The world file (.wld) records the position and scale for GIS programs and for GEOIMAGE.
- The original file is not changed: the result goes to a new file `<name>-ed.png`.

See also

IMAGEEDIT · IMAGECROP · GEOIMAGE · GEOREFIMAGE · Raster images

IMPORTMAP — Import plan, orthophoto or map

Brings the cadastral plan, the PNOA orthophoto and a base map of the area you choose in one go.

- **Command:** `IMPORTMAP` · **Global name:** `_IMPORTMAP`
- Ribbon: Mapping › Map services
- Menu: Mapping

Brings, for a single area of interest, the **cadastral plan** (parcels, buildings and labels), the IGN **PNOA orthophoto** and a **base map** in one go, with their layers already named and each source credited. It gathers in one dialog what used to take several steps: CADASTREDOWNLOAD and the map layers of MAPCONNECT.

Use

1. Type IMPORTMAP (or IMPORTARPLANO). **Import plan, orthophoto or map** opens.
2. **1. Area of interest**: choose how to give the area. The default is the **Cadastral reference** (14 or 20 characters). The others: **Address** and **Polygon and parcel** (with the province and municipality from the Cadastre lists; if there are several parcels, the dialog lists them so you can pick one), **Municipality** (the CartoCiudad geocoder), **Coordinates** (geographic WGS84 or drawing), **Current view** and **Window in the drawing** (click **Pick window…** and mark two corners in the drawing).
3. **2. What to import**: **Cadastral plan** (**Parcels**, **Buildings** and, as an image from the Cadastre WMS, **Sub-parcels**), **PNOA orthophoto (IGN)** with its **Resolution** (automatic, 10, 25 or 50 cm, 1 m) and **Base map** (IGN as WMTS or WMS, MTN raster, OpenStreetMap).
4. **3. Options**: **Margin around the area**, **Maximum area size (side)**, **Maximum size of each image**, **Cadastral plan transparency (image)**, parcels as **Entities** or **Map layer (WMS)**, **Label the cadastral reference** and the **Layers:** scheme.
5. Click **Import**. Vectors are downloaded in the background in 200 m tiles (with a progress bar and **Cancel**); the images arrive by themselves, under the drawing, and show when finished. Everything is undone at once with UNDO.

What it leaves

Layer	Content
CATASTRO-PARCELA, CATASTRO-EDIFICIO, CATASTRO-TEXTO	Parcels, buildings and references (LinceCAD scheme, as CADASTREDOWNLOAD)
parcela <rc>, construccion <rc>, parcela_near <rc>, construccion_near <rc>	The same with the PV project scheme (CadFVTools): the area parcel and its surroundings
MAPA-ORTOFOTO, MAPA-BASE, MAPA-CATASTRO	The images, stacked in that order (base, orthophoto, cadastral plan)
MAPA-FUENTE	The source of the vectors; each image source is labelled next to it

The command line summarises the area, what was drawn, the images requested, the sources and the warnings. If

the drawing has no coordinate system, the one the area suggests is set (ETRS89 UTM 29, 30 or 31; WGS84 UTM 28 in the Canary Islands) or the one you pick; areas by view or window need it to be set already (GEOGRAPHICLOCATION).

Notes

- Sources: **Dirección General del Catastro** (INSPIRE WFS for parcels and buildings, WMS), **IGN** (PNOA, base map, CartoCiudad; CC BY 4.0, scne.es) and OpenStreetMap (occasional use). Credit them wherever you use the data.
- Usage limits: 30 requests per minute to the Cadastre (when reached, it waits); what is downloaded from the Cadastre is kept for 7 days and the images reuse the map cache. The area is cropped to the maximum size and each image to its pixel limit.
- The digital terrain model and LiDAR are not imported from here: use IMPORTARMDT and MDTIGN.
- If the Cadastre does not answer for some tile, you are told and what arrived is drawn: repeat the command to complete.
- Outside Spain the Cadastre and PNOA have no data.

See also

CADASTREDOWNLOAD · MAPCONNECT · ONLINEMAPS · GEOGRAPHICLOCATION

INSERT — Insert

Inserts a block or a drawing.

- **Command:** `INSERT` · **Global name:** `_INSERT` · **Alias:** `I`, `INSERTURL`
- Ribbon: Home › Block
- Ribbon: Insert › Block
- Ribbon: View › Palettes
- Menu: Insert

Inserts a block or a whole drawing. It opens the **Blocks** palette, with thumbnails of the blocks in the drawing, the most recently inserted ones and those in the library folders. Click a thumbnail (or drag it onto the drawing) and the block hangs from the cursor so you can place it.

Usage

1. Enter INSERT (or I) or click the button: the palette opens (or closes).
2. Choose the tab and, if needed, type part of the name in **Search**.
3. Click a block. If it comes from another file, its definition is copied into the drawing first.
4. Specify on screen whatever is missing, according to the palette's insertion options.

```
Specify insertion point or [Scale/Rotate]:
```

5. If the block has attributes, enter their values (Enter keeps the default value).

Tabs

- **Current Drawing**: the blocks defined in the drawing. **Browse…** inserts another DWG or DXF as a block.
- **Recent**: the last 24 inserted blocks, with their source file.
- **Libraries**: the drawings in the library folders, each one whole and with its blocks. The folder installed with the program has architectural and electrical symbols. **Add Folder…** adds another one; right-click its title to remove it.

Insertion options

- **Insertion Point**, **Scale** and **Rotation**: with **Specify On-screen** they are specified with the mouse; otherwise the typed values are used. Clear **Uniform Scale** to use different X, Y and Z scales.
- **Repeat Placement**: after placing the block it asks for the point again, until Enter or Esc.
- **Explode**: inserts the pieces of the block instead of a reference to it.

Notes

- The palette can be docked to the left or right of the drawing and set to auto-hide with the buttons in its header (? ? ? ?); see The user interface.
- If the drawing already has a block with the same name, the drawing's block is used and the other file's is not copied.
- Recent blocks, folders and tool palettes are saved in `bloques.toml`, in the program's settings folder.
- In scripts, INSERT asks on the command line like -INSERT.

See also

-INSERT · BLOCK · ADCENTER · TOOLPALETTES · Blocks and attributes

INSERTOBJ —Insert OLE Object

Not available: LinceCAD has no Windows OLE objects.

- **Command:** `INSERTOBJ` · **Global name:** `_INSERTOBJ` · **Alias:** `IO`
- Ribbon: Insert › Reference

AutoCAD uses INSERTOBJ to insert Windows OLE objects (a spreadsheet, a document). LinceCAD runs on Windows and Linux and does not support OLE: the command only explains this.

Alternatives

- Images: IMAGEATTACH. PDF drawings: PDFATTACH.
- Text from another program: PASTESPEC.
- Spreadsheets: DATALINK, which brings them into a table.

```
Insert OLE object:
```

See also

IMAGEATTACH · PDFATTACH · DATALINK

MAPCLIP —Clip map

Clips the map layer to the visible area or to a boundary.

- **Command:** `MAPCLIP` · **Global name:** `_MAPCLIP`
- Ribbon: Mapping › Map services
- Menu: Mapping

Clips the chosen map layer without losing anything: it is the clipping boundary of its image, like IMAGECLIP.

Use

1. Choose the layer in ONLINEMAPS or right-click the map in the drawing.
2. Type MAPCLIP and choose: - **View** (Enter): to the visible area. - **Manual**: to a rectangle or a polygon you pick.

- **Remove**: removes the clipping.

Notes

- A clipped layer stops refreshing by itself (the clipped part is what you want to keep); **Refresh in this area** requests it again and keeps the clipping in place.

See also

ONLINEMAPS · IMAGECLIP

MAPCONNECT —Connect to map data

Connects to a WMS, WFS, WMTS or XYZ service and adds its layers as map layers.

- **Command:** `MAPCONNECT` · **Global name:** `_MAPCONNECT`
- Ribbon: Mapping › Map services
- Menu: Mapping

Connects to a **WMS**, **WFS**, **WMTS** or **XYZ** map service and adds its layers to the drawing, like AutoCAD Map 3D **Data Connect**. Raster layers become **live map layers**: they are requested again for the visible area after panning or zooming, and are managed in ONLINEMAPS.

Use

1. Type MAPCONNECT (or click **Connect to data…** in the manager). **Connect to map data** opens.
2. Pick a connection from the list or type a new one: **Name:**, **Type:**, **URL:** and **Version:**. If the server needs it, expand **Credentials and proxy** (user, password and proxy). **Save connection** keeps it in the list.
3. Click **Connect**: the layers are read in the background (the dialog does not freeze; **Cancel** stops waiting). Errors are explained: no network, timeout, invalid XML or credentials.
4. Check the layers. For each one choose **Format** (PNG, JPG, TIF…), **Server system** (among those the server offers; by default the closest to the drawing system), **Background** (transparent or a colour) and **Style**. With several, **Combine into one layer** merges them into one layer in list order.
5. Click **Load**. The layer arrives in the background and goes **below** the drawing.

WFS: feature classes and query

- Each **class** is added as drawing **entities** on a layer named after it, with **Colour:** and **Label:** (an attribute labelled on each feature). **Show geometry** reads its fields and tells whether they are points, lines or surfaces.
- **Add with query**: only the view, by property (field, operator and value, with the server field list) or with a hand-written CQL filter. **Feature limit:** limits the features per request; if as many arrive as the limit, **Next page** requests the next ones.

Notes

- The drawing needs a coordinate system (GEOGRAPHICLOCATION). If the view lies outside the area of use of the system, or outside the server layer extent, you are warned before anything is requested.
- When you add a layer while the view lies outside it (or it declares no extent), the view is framed to the user's **country** (OPTIONS, **Country**): the layer extent inside the country and inside the system zone (a country with several zones, such as Spain, is framed in the drawing's own: mainland or Canary Islands) and the notice says so. If the country does not fit (more than 3000 km a side), 5 km around its capital.
- If the server does not offer the drawing system, another one is requested (**Server system**) and LinceCAD converts it.
- Connections and their credentials are stored in the settings (`mapas.json`; the password encrypted for your Windows user), **never in the drawing**.
- Built-in servers and terms of use: MAPSERVICE. WMS, WFS, WMTS and MAPSERVICE open this same

dialog with the type already chosen.

See also

ONLINEMAPS · MAPREFRESH · MAPCLIP · MAPSERVICE · GEOGRAPHICLOCATION

MAPEXPORT —GIS Export

Exports the selection to SHP, KML, KMZ or GeoJSON.

- **Command:** `MAPEXPORT` · **Global name:** `_MAPEXPORT`
- Ribbon: Insert › Geolocation
- Ribbon: Mapping › Geolocation
- Menu: Insert
- Menu: Mapping

Exports the selected objects to a GIS format: SHP, KML or KMZ (for Google Earth) or GeoJSON.

Usage

1. Enter MAPEXPORT or click the button.
2. Select the objects and press Enter. Objects already selected are used.

```
Select objects:
```

3. Choose the file in the dialog box or enter its path. The format comes from the extension.

```
Output file (.shp, .kml, .kmz or .geojson):
```

When it finishes, the files written are listed.

Notes

- KML, KMZ and GeoJSON are in WGS84 (latitude and longitude), so the drawing needs its coordinate system (GEOGRAPHICLOCATION).
- SHP is in the drawing's system, with its `.prj`; without a system, in the drawing's coordinates as they are. One file is written per geometry type (points, lines and polygons).
- Points, block inserts and texts are written as points; everything else as lines or, if closed, as polygons. Arcs and curves are split into straight segments.
- Each feature carries as attributes its layer, its text, the block's attributes and the fields it brought from MAPIMPORT.
- Each polygon is exported as a single ring, without holes.

See also

MAPIMPORT · GEOGRAPHICLOCATION · GEOCOORD · Georeferencing

MAPIMPORT —GIS Import

Imports SHP, KML, KMZ or GeoJSON with their attributes.

- **Command:** `MAPIMPORT` · **Global name:** `_MAPIMPORT` · **Alias:** `SHP2CAD`
- Ribbon: Insert › Geolocation
- Ribbon: Mapping › Geolocation
- Menu: Insert
- Menu: Mapping

Imports GIS data (SHP, KML, KMZ or GeoJSON files) as drawing objects, in the drawing's coordinates and with their attributes.

Usage

1. Enter MAPIMPORT (or SHP2CAD) or click the button. The **Import SHP, KML or GeoJSON** dialog box opens and, at the same time, the command line asks.
2. Choose the file with **Browse…** or enter its path. The number of features and fields is shown.

```
SHP, KML, KMZ or GeoJSON file:
```

3. Optionally, choose the field whose value gives the layer name of each feature.

```
Field for the layer (…) or [None] <None>:
```

4. Optionally, choose a block of the drawing for the points; otherwise each point is a point object.

```
Block name for the points <point object>:
```

5. In the dialog box, check the **File coordinate system** and click **OK**.

Notes

- Points, lines and polygons are reprojected to the drawing's system (GEOGRAPHICLOCATION) according to the SHP's `.prj`; KML, KMZ and GeoJSON are always in WGS84. If the drawing has no system, it takes the file's (or, if it is latitude and longitude, the UTM system of its zone).
- Polygons become closed polylines, without hatching; their holes, other closed polylines.
- Attributes are stored in each object as extended data (application `LINCE_SHP`), and MAPEXPORT writes them out again. If the points are a block, its attributes take the value of the field with the same name.
- KML styles and network links are not read. 3D objects in SHP files (multipatch) are skipped.

See also

MAPEXPORT · GEOGRAPHICLOCATION · GEOIMAGE · Georeferencing

MAPREFRESH — Refresh map

Requests the map layers again for the visible area.

- **Command:** `MAPREFRESH` · **Global name:** `_MAPREFRESH`
- Ribbon: Mapping › Map services
- Menu: Mapping

Requests the map layers again for the **visible area**, at screen resolution. It is the **Refresh in this area** of the ONLINEMAPS manager.

Use

- Type MAPREFRESH. Enter refreshes the layer chosen in the manager (or the only one); **All** refreshes all of them.

Notes

- The download runs in the background: the previous image stays visible meanwhile.
- Layers with **Automatic** checked refresh by themselves after panning or zooming.

See also

ONLINEMAPS · MAPCONNECT

MAPSERVICE —Map service

Loads web cartography (WMS, WFS, WMTS or XYZ) into the drawing.

- **Command:** `MAPSERVICE` · **Global name:** `_MAPSERVICE` · **Alias:** `WEBMAP`
- Ribbon: Mapping › Map services
- Menu: Mapping

Loads public cartography from OGC map services (WMS, WFS, WMTS or XYZ) as a background or as drawing vectors, reprojected to the drawing coordinate system (GEOGRAPHICLOCATION). It is a single command with the service as a sub-option; the direct commands WMS, WFS and WMTS also exist.

Use

1. Type MAPSERVICE (or WEBMAP) or click the button. MAPCONNECT opens with every service type.
2. Pick the connection from the list, click **Connect** to read its layers and check them (XYZ has no layers).
3. Click **Load**.

Notes

- When the dialog opens, the program reads in the background the layers (name, title, formats and coordinate systems) of every server in the list and keeps them on disk for 7 days: the layer list shows up instantly. **Connect** requests them again from the service. To download a Cadastre parcel, see CADASTREDOWNLOAD.
- WMS and WMTS/XYZ are added as **live map layers** of the visible area: they are requested again after panning or zooming and are managed in ONLINEMAPS.
- WFS brings **vectors** (points, lines and polygons with their attributes), like MAPIMPORT, in GeoJSON or simple GML.
- WMS and WMTS need the drawing to have a coordinate system (GEOGRAPHICLOCATION).

Built-in Spanish servers

Public endpoints (verified against official documentation); if one changes, save a new connection in MAPCONNECT.

- **Catastro** (Dirección General del Catastro): WMS `https://ovc.catastro.meh.es/Cartografia/WMS/ServidorWMS.aspx` (layer `Catastro`) and INSPIRE WFS of parcels `https://ovc.catastro.meh.es/INSPIRE/wfsCP.aspx`.
- **IGN / PNOA**: PNOA orthophoto via WMS `https://www.ign.es/wms-inspire/pnoa-ma` and via WMTS `https://www.ign.es/wmts/pnoa-ma`; IGN base map (WMTS) `https://www.ign.es/wmts/ign-base`; MTN raster (WMS) `https://www.ign.es/wms-inspire/mapa-raster`.
- **CartoCiudad** (IGN): INSPIRE addresses (WFS) `https://www.cartociudad.es/wfs-inspire/direcciones`.
- **Regional IDE** (example): ICGC of Catalonia (WMS) `https://geoserveis.icgc.cat/icc_mapesmultibase/utm/wms/service` (layer `topo`).
- **OpenStreetMap** (XYZ, occasional use only): `https://tile.openstreetmap.org/{z}/{x}/{y}.png`. The OpenStreetMap Foundation tiles are not meant for heavy or commercial use: LinceCAD identifies itself, caches tiles for 7 days and only downloads the tiles covering the view, as their policy requires. For regular use, choose the **IGN base map** (first in the list) or an OpenStreetMap-based map provider with your own key, added as a custom server.

Attribution. Data from these services must be credited. LinceCAD automatically adds a "Source: …"text at the bottom-left corner of what it downloads (OpenStreetMap, IGN, Cadastre, CartoCiudad, ICGC); keep it in the drawing or move the credit to the title block. OpenStreetMap data (ODbL licence) can be used in commercial drawings as long as "OpenStreetMap (openstreetmap.org/copyright)" is credited.

Adding other servers

In MAPCONNECT, type a name and the URL and click **Save connection**: it is saved and stays in the list. Any WMS/WFS/WMTS endpoint works (the service base URL, without parameters) or an XYZ template with the z, x and y placeholders in braces. Respect each service's terms of use.

To work outside Spain, Map servers by country lists the official services of Europe, Latin America and North Africa (cadastre, mapping agency, orthophoto) with their URL, layer, licence and attribution, checked with LinceCAD.

See also

WMS · WFS · WMTS · GEOGRAPHICLOCATION · MAPIMPORT · Georeferencing · Map servers by country

OCRIMAGE —Recognise text (OCR)

Recognises the text in a raster image and creates text entities at its position.

- **Command:** `OCRIMAGE` · **Global name:** `_OCRIMAGE` · **Alias:** `RECOGNIZETEXT`, `OCR`
- Ribbon: Insert › Image and tracing

Recognises (OCR) the text in a raster image already attached to the drawing and creates one text entity (TEXT) per recognised line, at its position, with the height measured from the pixels and the rotation of the line, on the current layer and text style.

The OCR engine is pure Rust (ocrs, on top of the rten inference engine), with no external binaries. It only recognises the Latin alphabet (Spanish and English) and is in preview, so it is worth reviewing the result.

Usage

1. Enter OCRIMAGE (or RECOGNIZETEXT) or click the button.
2. Select the image with the text, or press Enter to choose an image file (with FILEDIA=0 you type the path).

```
Select the image with the text to recognise or press Enter to choose a file:
```

3. Press Enter.

```
Press Enter to recognise the text in the image.
```

The recognised text is created as standard TEXT entities, which you can move, edit or delete like any other text.

Models

- The first time it is used, the OCR downloads its two models (~12 MB in total) to the configuration folder (`<configuration>/lince/ocr`). They are then reused without downloading again.
- The download is verified by size and SHA-256 checksum. Nothing is stored in the drawing.
- Offline: if the models cannot be downloaded and **tesseract** is installed, it is used as a fallback; otherwise a message is shown and no text is created.

Notes

- The text height comes from the pixel height of each line; nearly horizontal text is straightened.
- The text takes the current layer and text style. The original image is not modified.
- Vectorization (raster to vector) and PDF to DWG are separate steps of the plan.

See also

IMAGEATTACH · IMAGEEDIT · TEXT · Raster images

ONLINEMAPS —Online maps

Opens the map layer manager (on/off, transparency, order, refresh, clipping).

- **Command:** `ONLINEMAPS` · **Global name:** `_ONLINEMAPS`
- Ribbon: Mapping › Map services
- Menu: Mapping

Opens the **Online maps** palette, the map layer manager (like the AutoCAD Map 3D **Display Manager**). Each layer added with MAPCONNECT has a row; the top one is drawn on top.

In each row

- The check box turns the layer on or off; **Transparency** fades it (0–90 %).
- **? ?** move it up or down among the map layers.
- **?** Refresh in this area: requests it again for the current view. **?** Zoom to layer.
- **?** Properties…name, format, server system, background and style (it is requested again on OK).
- **?** deletes the map layer.
- **Automatic**: requested again by itself half a second after panning or zooming, only if its image no longer covers the view at enough resolution. The previous image stays until the new one arrives.
- Right-clicking the row (or the map in the drawing) offers **Clip to the visible area**, **Clip manually…** and **Remove clipping** (MAPCLIP).

Notes

- By default map layers go **below** all entities; **Above the entities** puts them on top.
- Map layers are not selected and give no object snaps: they are handled from here.
- In the DWG each layer is an ordinary attached image (the file is in the LinceCAD cache) with its connection data in extended data: AutoCAD opens it without errors; on another computer LinceCAD requests it again.
- Downloaded images are cached by area and scale: going back to a view already seen downloads nothing.

See also

MAPCONNECT · MAPREFRESH · MAPCLIP

PDFADJUST —Adjust PDF

Changes the fade, contrast or monochrome mode of PDF underlays.

- **Command:** `PDFADJUST` · **Global name:** `_PDFADJUST`
- Ribbon: Insert › Import

Changes how PDF underlays look: more or less faded, with more or less contrast, or in monochrome. A faded underlay makes what you draw on top of it easier to see.

Usage

1. Enter PDFADJUST or click the button.
2. Select one or more PDF underlays and press Enter.

```
Select PDF underlays:
```

3. Choose what to adjust; Enter, the fade.

```
Enter underlay option [Fade/Contrast/Monochrome] <Fade>:
```

4. Enter the new value.

Options

- **Fade** (F): from 0 (no fade) to 80; Enter sets 25.

```
Enter fade value (0-80) <25>:
```

- **Contrast** (C): from 0 to 100; Enter sets 20.

```
Enter contrast value (0-100) <20>:
```

- **Monochrome** (M): with Yes, the underlay is shown in grayscale instead of in color.

```
Monochrome? [Yes/No] <No>:
```

Notes

- The value is applied to all selected underlays. The one offered between < > is AutoCAD's default, not the underlay's current value.
- It is saved in the underlay as in AutoCAD. The change is undone with UNDO.

See also

PDFATTACH · PDFCLIP · PDFLAYERS · IMAGEADJUST · External references

PDFATTACH —Attach PDF

Attaches a page of a PDF file as an underlay.

- **Command:** `PDFATTACH` · **Global name:** `_PDFATTACH` · **Alias:** `PDFA`
- Ribbon: Insert › Import

Attaches a page of a PDF file as an underlay: a plan received as a PDF that serves as a background to draw on. The PDF is not copied into the drawing: its path is saved and the file is read again when the drawing is opened.

Usage

1. Enter PDFATTACH (or PDFA), or click **Attach PDF…** in the XREF window.
2. Enter the PDF path, or press Enter to choose it in a file dialog.

```
Enter path to PDF file to attach or [Browse] <Browse>:
```

3. Enter the page number; Enter, the first one. `?` lists the pages with their size in points.

```
Enter page number or [?] <1>:
```

4. Specify the insertion point, which is the lower-left corner of the page (Enter: the origin).

```
Specify insertion point:
```

5. Enter the scale, or specify a point: the distance to the insertion point is the width of the page.

```
Specify scale factor <1>:
```

6. Enter the rotation angle, or specify a point in that direction.

```
Specify rotation <0>:
```

In the interface (not in scripts, SCRIPT or AutoLISP), after choosing the PDF the **Attach PDF Underlay** dialog box opens: the page (each one drawn with its proportions and its number) and the insertion point, scale and rotation; whatever you leave as *Specify On-screen* is specified afterwards, with the prompts of steps 4 to 6. The thumbnails do not show the page contents. The dialog box cannot be scripted in the AutoCAD console: it has a unit test; -PDFATTACH always asks on the command line.

Notes

- At scale 1, the page measures in the drawing the same as on paper, in the units of INSUNITS (without units, in millimeters): a landscape A3 measures 420 × 297 in a drawing in millimeters.
- The path is saved relative to the drawing's folder if REFPATHTYPE is 1 (and the drawing has been saved); otherwise, as a full path.
- The page is turned into an image inside LinceCAD, without external programs, at about 200 dots per inch: with a lot of zoom the pixels show. That is why object snaps do not work on what is drawn in the PDF.
- The underlay frame is controlled by PDFFRAME. It is adjusted with PDFADJUST, clipped with PDFCLIP, and its layers are shown or hidden with PDFLAYERS.
- It is saved as an AutoCAD PDF underlay (PDFUNDERLAY), which AutoCAD opens the same way.

See also

-PDFATTACH · XREF · IMAGEATTACH · External references

PDFCLIP —PDF Clip

Shows only part of a PDF underlay, inside or outside a boundary.

- **Command:** `PDFCLIP` · **Global name:** `_PDFCLIP`
- Ribbon: Insert › Import

Shows only part of a PDF underlay: what lies inside a rectangular or polygonal boundary or, with the boundary inverted, what lies outside it. The PDF does not change: the boundary can be turned off or removed later.

Usage

1. Enter PDFCLIP or click the button.
2. Select the PDF underlay.

```
Select PDF underlay to clip:
```

3. Choose the option; Enter creates a new boundary, which replaces any existing one.

```
Enter PDF clipping option [ON/OFF/Delete/New boundary] <New>:
```

4. Choose the boundary type; Enter, rectangular.

```
Outside mode: objects outside the boundary will be hidden. Specify clipping boundary [Polygo
nal/Rectangular/Invert clip] <Rectangular>:
```

5. For a rectangular one, specify two opposite corners.

```
Specify first corner point:
```

6. For a polygonal one, specify the vertices and press Enter (or choose Close) when there are at least three.

```
Specify next point or [Close/Undo]:
```

Options

- **ON** / **OFF** (ON / OFF): turns on or off the boundary the underlay already has.
- **Delete** (D): removes the boundary; the whole page is shown again.
- **New** (N): draws a new boundary.
- **Rectangular** (R) and **Polygonal** (P): the boundary type.
- **Invert** (I): switches between hiding what is outside and hiding what is inside.

```
Inside mode: objects inside the boundary will be hidden. Specify clipping boundary [Polygona
l/Rectangular/Invert clip] <Rectangular>:
```

- **Undo** (U): in a polygonal boundary, removes the last vertex.

Notes

- The boundary is saved in the underlay as in AutoCAD. The change is undone with UNDO.
- For DWG external references, use XCLIP; for images, IMAGECLIP.

See also

PDFATTACH · PDFADJUST · PDFLAYERS · External references

PDFIMPORT —Import PDF

Brings a PDF's vector geometry in as drawing entities.

- **Command:** `PDFIMPORT` · **Global name:** `_PDFIMPORT` · **Alias:** `IMPORTPDF`
- Ribbon: Insert › Import

Brings a PDF's **vector geometry** into the drawing as entities, like AutoCAD's PDFIMPORT. Unlike PDFATTACH, which links the rasterized page as an underlay, this reads the PDF's vector content and rebuilds it with standard entities.

Usage

1. Type PDFIMPORT (or IMPORTPDF) or click **Import PDF**.
2. Enter the file path (or press Enter to pick it in the browser).

```
Enter the path of the PDF file to import:
```

3. If the PDF has several pages, enter which one to import.
4. Adjust the options and press Enter to import.

```
Press Enter to import or change [Vectors/Fills/Text/Images/Geometry/Scale]:
```

What is imported

- **Paths** (lines, polylines, curves and rectangles) ->lightweight polylines. Bézier curves are approximated with straight segments.
- **Fills** ->a SOLID hatch (Fills option) or, if turned off, just the outline.
- **Text** ->text entities, or polylines if you turn on "Geometry" (text as geometry).
- Embedded raster **images** ->extracted to a folder next to the PDF and linked as images (like IMAGEATTACH).

Options

- **Vectors**, **Fills**, **Text**, **Images**: turn each class on or off.
- **Geometry**: brings text in as polylines instead of text entities.
- **Scale**: import scale factor. At scale 1 the page measures what it does on paper, in drawing units (same as a PDFATTACH underlay at scale 1).

Notes

- The PDF color is copied as true color. All geometry lands on the current layer: the PDF's optional content groups (OCG layers) are not split into drawing layers.
- Non-embedded fonts are substituted with an equivalent; if the text cannot be read, use "Geometry".
- Complex fill patterns and stencil (masked) images are not imported.

See also

PDFATTACH · SVGIMPORT · IMAGEATTACH

PDFLAYERS —PDF Layers

Shows or hides the layers of a PDF underlay.

- **Command:** PDFLAYERS · **Global name:** _PDFLAYERS
- Ribbon: Insert › Import

Shows or hides the layers of a PDF underlay. Many PDFs made with CAD programs keep the layers of the original drawing (in the PDF they are called optional content groups): with this command you can turn off, for example, the dimensions or the furniture of a plan received as a PDF.

Usage

1. Enter PDFLAYERS or click the button.
2. Select the PDF underlay.

```
Select PDF underlay:
```

3. Choose the option; Enter or ? lists the PDF layers with their state.

```
Enter an option [?/ON/OFF] <?>:
```

4. With ON or OFF, enter the layer names. Wildcards are accepted (* all, ? one character), as well as several names separated by commas.

```
Enter PDF layer name(s) <*>:
```

Options

- **?**: lists the layers, each one with ON (shown) or OFF (hidden).
- **ON** (ON): turns on the given layers.
- **OFF** (OFF): turns off the given layers.

Notes

- It only changes what is shown of the underlay: the PDF file is not modified.
- If the PDF has no layers, the command says so and ends.
- The layers turned off are saved in the underlay with LinceCAD's own data: AutoCAD shows the underlay with all its layers.
- In the interface, after selecting the underlay the **Underlay Layers** dialog box opens, with a check box per PDF layer (and Show all / Hide all); in scripts, SCRIPT and AutoLISP the choice is made on the command line. The dialog box cannot be scripted in the AutoCAD console: it has a unit test.

See also

PDFATTACH · PDFADJUST · PDFCLIP · LAYER · External references

PDFSHXTEXT —PDF SHX Text

Converts the SHX-font geometry of a PDF into text (notice).

- **Command:** PDFSHXTEXT · **Global name:** _PDFSHXTEXT

Converts the SHX-font geometry of an imported PDF into text, like AutoCAD PDFSHXTEXT (minimal scope).

Usage

Enter PDFSHXTEXT and select the geometry.

Notes

- AutoCAD recognises letters that a PDF carries drawn with SHX fonts (as strokes) and turns them into text. LinceCAD does not do that recognition: whatever you select stays the same and the command line explains it.
- Real PDF text already arrives as TEXT with PDFIMPORT, and the text of a raster underlay is read with OCRIMAGE.
- INSERTPDFSHXTEXT does not exist in AutoCAD 2027.

See also

PDFIMPORT · OCRIMAGE

REFCLOSE —Close Reference Edit

Saves or discards the in-place editing of REFEDIT.

- **Command:** `REFCLOSE` · **Global name:** `_REFCLOSE`
- Ribbon: Insert › Reference

Ends the in-place editing started with REFEDIT, like AutoCAD REFCLOSE.

Usage

1. Enter REFCLOSE and choose whether to save the changes.

```
Enter option [Save/Discard reference changes] <Save>:
```

Options

- **Save**: what was edited (the block contents and whatever was drawn after REFEDIT) goes back to the block coordinates and becomes its definition; every insert of it is redrawn.
- **Discard reference changes**: what was edited is erased and the block stays as it was.

In both cases the reference being edited becomes visible again.

See also

REFEDIT · BSAVE

REFEDIT —Edit Reference In-Place

Edits a block in place, with the rest of the drawing in view.

- **Command:** `REFEDIT` · **Global name:** `_REFEDIT`
- Ribbon: Insert › Reference

Edits a block in place, with the rest of the drawing in view, like AutoCAD REFEDIT: its contents go into the drawing, where the selected reference is, and can be changed with any command. REFCLOSE saves them to the block (and every insert of it is redrawn) or discards them.

Usage

1. Enter REFEDIT and select a block reference.

```
Select reference:
```

2. Accept the nesting level (the selected block).

```
Select nesting level [Ok/Next] <Next>:
```

3. Choose the object selection method (*All*: the whole block).

```
Enter object selection method [All/Nested] <All>:
```

4. Choose whether attribute definitions are shown for editing.

```
Display attribute definitions [Yes/No] <No>:
```

5. Edit: whatever you draw while it lasts also goes into the block. Finish with REFCLOSE.

Notes

- While it lasts, the reference being edited is hidden. Undoing back to before REFEDIT cancels the editing.
- External references (open them with XOPEN), dynamic blocks and only part of the block (*Nested*) cannot be edited in place yet. The other inserts of the block are not faded.
- Avoid saving the drawing while editing: it would save the copied contents and the hidden reference.
- REFEDIT and -REFEDIT ask the same on the command line (AutoCAD opens the Reference Edit dialog box first).

See also

REFCLOSE · BEDIT · XOPEN

REFSET — Edit Reference Working Set

Adds or removes objects of the REFEDIT working set.

- **Command:** REFSET · **Global name:** _REFSET

Adds or removes objects of the working set of an in-place edit, like AutoCAD REFSET.

Usage

1. Start with REFEDIT.
2. Enter REFSET and choose Add or Remove.
3. Select the objects and press Enter.
4. Finish with REFCLOSE.

Options

- **Add**: the selected drawing objects go into the edit (they are copied and leave the drawing); saving with REFCLOSE puts them into the block. With Discard they go back to the drawing. It also puts back into the edit those that had been removed.
- **Remove**: the selected objects of the edit stop belonging to the block: on close they stay in the drawing as ordinary objects.

It is only used during an in-place edit; otherwise the command line says so.

See also

REFEDIT · REFCLOSE

RESETBLOCK — Reset Block

Restores the selected dynamic blocks to the values of their definition.

- **Command:** `RESETBLOCK` · **Global name:** `_RESETBLOCK` · **Alias:** `RSB`
- Ribbon: Insert › Block

Returns the selected dynamic blocks to the values of their definition: they lose the changes made to their custom properties (visibility state, flip, distances, angles, lookup values...).

Usage

1. Enter RESETBLOCK (or RSB) or click the button. Objects already selected are used.
2. Select the blocks and press Enter.

```
Select objects:
```

Notes

- Only dynamic blocks change; if none was selected, a message is shown: Not a dynamic block.
- The blocks end up as if just inserted: with the definition's geometry, at the same place and with the same scale and rotation.
- The custom properties of a dynamic block are changed with its cyan grips or in the **Custom** section of PROPERTIES.
- The change is undone with UNDO.

See also

BEDIT · PROPERTIES · INSERT · Blocks and attributes

SVGIMPORT — Import SVG

Brings an SVG file in as drawing geometry.

- **Command:** `SVGIMPORT` · **Global name:** `_SVGIMPORT`
- Ribbon: Insert › Import

Brings an SVG file into the drawing as geometry. Each SVG outline becomes a polyline on the current layer.

Usage

1. Type SVGIMPORT or click **Import SVG**.
2. Enter the file path (or press Enter to pick it in the browser).

```
Enter the path of the SVG file to import:
```

Notes

- Outlines are imported: lines, polylines and curves (Bézier curves are approximated with straight segments), and text flattened to outlines. Fills and embedded raster images are not imported.
- Each outline's stroke color is copied as the object's true color.
- The SVG Y axis points down; on import it is flipped so the drawing comes out upright. SVG units map 1:1 to drawing units; scale the result if you need to.

See also

INSERT · IMAGEATTACH · PDFATTACH

TOOLPALETTES —Tool Palettes

Palettes with blocks ready to insert with their scale, rotation and layer.

- **Command:** `TOOLPALETTES` · **Global name:** `_TOOLPALETTES` · **Alias:** `TP`
- Ribbon: Insert › Content
- Ribbon: View › Palettes
- Ribbon: Manage › Customization
- Menu: Insert

Opens the **Tool Palettes** window: palettes with blocks ready to insert, each one with its own scale, rotation and layer. Enter the command again to close it.

Usage

1. Enter TOOLPALETTES (or TP) or click the button. The first time, one palette is created for each drawing in the installed library, with all its blocks.
2. Choose the palette in the tabs at the top.
3. Click a tool and specify the insertion point; the scale, rotation and layer are those of the tool.

```
Specify insertion point or [Scale/Rotate]:
```

Organizing the palettes

- **+** (New Palette) creates an empty palette; **Rename** and **Delete Palette** act on the visible one.
- To add tools, drag blocks from the Blocks palette (INSERT) or from ADCENTER and drop them on the palette.
- Right-click a tool to change its **Scale**, **Rotation** and **Layer** (empty: the current layer), or to remove it with **Delete Tool**.

Notes

- The palette can be docked to the left or right of the drawing and set to auto-hide with the buttons in its header (? ? ? ?); see The user interface.
- If the tool's layer does not exist in the drawing, it is created on insertion.
- Repeat placement and Explode from the Blocks palette do not apply to tools.
- Palettes are saved in `bloques.toml`, in the program's settings folder; each tool remembers the file of its block.

See also

INSERT · ADCENTER · -INSERT · Blocks and attributes

TRANSPARENCY —Transparency

Turns the transparent pixels of images on or off.

- **Command:** `TRANSPARENCY` · **Global name:** `_TRANSPARENCY`
- Ribbon: Insert › Image and tracing

Turns transparency of raster images on or off: with it, the transparent pixels of the file (for example, the background of a PNG) let the objects underneath show through.

Usage

1. Enter TRANSPARENCY or click the button.
2. Select the images and press Enter.

```
Select images:
```

3. Choose the mode.

```
Enter transparency mode [ON/OFF] <OFF>:
```

Options

- **ON** (ON): the transparent pixels of the file are shown as transparent.
- **OFF** (OFF): the image is shown opaque. This is the default.

Notes

- It only affects images whose file has a transparency channel, such as PNG, GIF or WebP.
- The mode is saved in each image, as in AutoCAD.
- To let the whole image show what is underneath, increase its fade with IMAGEADJUST.
- The transparency of objects and layers (0 to 90 %, group code 440) is a different property: it is changed in the PROPERTIES palette, in the layer manager and with CETRANSPARENCY, and shown or hidden with the TRANSP button on the status bar. See Layers and properties.

See also

IMAGEADJUST · IMAGEATTACH · IMAGECLIP · Raster images

VECTORIZAR —Vectorize image

Turns a monochrome/scanned image into drawing entities.

- **Command:** `VECTORIZAR` · **Global name:** `_VECTORIZE`
- Ribbon: Insert › Image and tracing

Turns a monochrome or scanned image, already attached with IMAGEADJUNTAR, into drawing entities: polylines, circles, arcs and lines. This is the RASTOOLS raster-to-vector step.

Usage

1. Type VECTORIZAR or click **Vectorize image**.
2. Select the image in the drawing, or press Enter to choose an image file (it is vectorized at the origin, 1 pixel = 1 unit; with FILEDIA=0 you type the path).

```
Select the image to vectorize or press Enter to choose a file:
```

3. Adjust the parameters or press Enter to run with the current ones.

```
Set [Threshold/Fit/Minimum/Centerline/Join] or <Run> (centerline: …, join: …):
```

Parameters

- **Threshold** —binarization cutoff (0-255): a pixel darker than the threshold is ink. Enter the binarization threshold (0-255) <…>:
- **Fit** —fit tolerance in pixels: how far an outline may stray and still count as a line, circle or arc, and how much polylines are simplified. Enter the fit tolerance in pixels <…>:
- **Minimum** —drops chains shorter than this size in pixels. Enter the minimum size in pixels <…>:
- **Centerline** —thins strokes to one pixel (skeleton) and follows their center instead of tracing the outline; for scanned line drawings with thick strokes.
- **Join** —joins broken strokes whose endpoints nearly touch.

Notes

- Entities are created on the current layer and color (like SVGIMPORT); set LAYER and COLOR first if you want others.
- Outlines are recognised by least-squares fit: a near-circular outline yields a circle or an arc, a near-straight one a line, and the rest a polyline. These are standard entities that audit with no errors.
- It does not OCR text or import PDF (PDFIMPORT): those are separate raster steps in the plan.

See also

IMAGEATTACH · IMAGEEDIT · SVGIMPORT

WBLOCK — Write Block

Writes a block to a DWG or DXF drawing file.

- **Command:** `WBLOCK` · **Global name:** `_WBLOCK` · **Alias:** `W`, `ACADWBLOCKDIALOG`
- Ribbon: Insert › Block
- Menu: Insert

Saves a block, the whole drawing or some objects as a new DWG or DXF drawing, to use it in other drawings or in a library folder.

Usage

1. Enter WBLOCK (or W) or click the button. Objects already selected are used as a starting point.
2. In the **Write Block** dialog box, choose the source: - **Block**: one of the blocks of the drawing. - **Entire drawing**: the whole model space. - **Objects**: the base point (coordinates or *Specify On-screen*) and the objects (the selected ones or, with *Specify On-screen*, those you select after OK), and whether they are **retained**, **converted to a block** (a block of the drawing named like the file) or **deleted**.
3. Enter the file name and path (or browse with **...**) and click OK.

In scripts, SCRIPT and AutoLISP there is no dialog box: everything is asked on the command line, as in -WBLOCK .

Notes

- The contents go to the model space of the new drawing, with the layers, linetypes, styles and nested blocks they use. The base point becomes the drawing's base point (INSBASE).
- The file is saved in the version of the current drawing (DWG 2018 if the current one is older than 2000). If it already exists, it is replaced and a `.bak` copy is left.
- As in AutoCAD, the file only carries the blocks, layers, linetypes and styles it uses (the rest is purged, as with PURGE).
- To insert the file into another drawing, enter its path in -INSERT or use **Browse...** in the INSERT palette. The dialog box cannot be scripted in the AutoCAD console: it has a unit test.

See also

-WBLOCK · BLOCK · INSERT · ADCENTER · Blocks and attributes

WFS — WFS vectors

Imports entities from a WFS service (as GeoJSON).

- **Command:** `WFS` · **Global name:** `_WFS`
- Ribbon: Mapping › Map services

- Menu: Mapping

Imports entities (vectors) from a **WFS** (Web Feature Service) as drawing objects with their attributes, reprojected to the drawing coordinate system. Like MAPIMPORT, but from the web.

Use

- Type WFS or click the button: MAPCONNECT opens with the WFS servers. Pick the connection, click **Connect**, check the class and click **Load**. Each class goes to a layer named after it, with colour and label; **Add with query** filters by view, by property or with CQL.

Notes

- Reads **GeoJSON** (with whatever format name the server announces: `application/json`, `geojson`…) and, if the server only gives **GML** (Spanish Cadastre INSPIRE, CartoCiudad, IGN), simple GML 3.2/3.1/2: points, lines and polygons with holes and their attributes. With GML the box is requested in a system the server accepts.
- With a CQL filter and the view at once, the box goes inside the CQL (GeoServer does not accept both separately).
- If as many features as the limit arrive, or the server says there are more, **Next page** brings the next ones (some servers stop earlier, IDENA at 1000).
- Built-in servers: MAPSERVICE.

See also

MAPCONNECT · MAPIMPORT · WMS · GEOGRAPHICLOCATION

WMS — WMS map

Inserts a WMS map as a georeferenced image.

- **Command:** `WMS` · **Global name:** `_WMS`
- Ribbon: Mapping › Map services
- Menu: Mapping

Adds a map from a **WMS** (Web Map Service) as a **live map layer**: a georeferenced image of the visible area, reprojected to the drawing system, requested again after panning or zooming.

Use

- Type WMS or click the button: MAPCONNECT opens with the WMS servers. Pick the connection, click **Connect**, check the layers, choose format, server system, background and style, and click **Load**.

Notes

- The layer is managed in ONLINEMAPS: on/off, transparency, order, refresh and clipping (MAPCLIP).
- If the server does not offer the drawing system (Catastro has no 25828, Canary Islands), another one it offers is requested and converted. The background is transparent (PNG) by default, so the orthophoto below shows.
- The drawing needs a coordinate system (GEOGRAPHICLOCATION). Built-in servers: MAPSERVICE.

See also

MAPCONNECT · ONLINEMAPS · WMTS · WFS · GEOGRAPHICLOCATION

WMTS —WMTS tiles

Inserts WMTS/XYZ tiles (Web Mercator) stitched as an image.

- **Command:** `WMTS` · **Global name:** `_WMTS`
- Ribbon: Mapping › Map services
- Menu: Mapping

Adds **WMTS** or **XYZ** tiles (Web Mercator, like OpenStreetMap) as a **live map layer**: the tiles covering the view are stitched into an image reprojected to the drawing system and requested again after panning or zooming.

Use

- Type WMTS or click the button: MAPCONNECT opens with the WMTS and XYZ servers. Pick the connection, for WMTS click **Connect** and check the layer (XYZ has none), and click **Load**.

Notes

- WMTS via `GetTile` with the GoogleMapsCompatible set; XYZ with a `{z}/{x}/{y}` template. Tiles are cached for 7 days. They are managed in ONLINEMAPS.

See also

MAPCONNECT · ONLINEMAPS · WMS · MAPSERVICE

XATTACH —Attach DWG

Attaches a DWG or DXF drawing as an external reference.

- **Command:** `XATTACH` · **Global name:** `_XATTACH` · **Alias:** `XA`
- Ribbon: Insert › Reference

Attaches another DWG or DXF drawing as an external reference: it is shown in place inside the current drawing, but it is not copied. The drawing only stores the file path and reads the file again each time it is opened.

Usage

1. Enter XATTACH (or XA), or click **Attach DWG…** in the XREF window.
2. Enter the file path, or press Enter to choose it in a file dialog.

```
Enter name of file to attach or [Browse] <Browse>:
```

3. Specify the insertion point (Enter: the origin).

```
Specify insertion point or [Overlay/pathType]:
```

4. Enter the X scale; Enter keeps 1.

```
Enter X scale factor <1>:
```

5. Enter the Y scale; Enter uses the same as X.

```
Enter Y scale factor <use X scale factor>:
```

6. Enter the rotation angle, or specify a point in that direction.

```
Specify rotation angle <0>:
```

In the interface (not in scripts, SCRIPT or AutoLISP), after choosing the file the **Attach External Reference** dialog box opens: the reference type (attachment or overlay), the path type and the insertion point, X, Y and Z scale and rotation; whatever you leave as *Specify On-screen* is specified afterwards, with the prompts of steps 3 to 6. The

dialog box cannot be scripted in the AutoCAD console: it has a unit test.

Options

- **Overlay** (O): the reference is overlaid instead of attached. An overlay is not shown in the drawings that attach this one. The prompt then offers **Attach** (A) to switch back.

```
Specify insertion point or [Attach/pathType]:
```

- **pathType** (T): chooses how the file path is saved. Enter keeps the value of REFPATHTYPE.

```
Path type [Full/Relative/No path] <Relative>:
```

- **Full** (F), **Relative** (R) or **No path** (N): the full path, a path relative to the drawing's folder, or just the file name (looked for in the drawing's folder). If the drawing has not been saved yet or the file is on another drive, the relative path is saved as a full one.

Notes

- The layers of the attached drawing appear as `reference|layer`. Their color or visibility changes are kept when it is reloaded if VISRETAIN is 1.
- If the reference is already defined in the drawing, the same one is inserted again. A file that is already a normal block of the drawing is not attached, nor one that attaches (directly or indirectly) the current drawing: that would be a circular reference.
- Its content is drawn somewhat faded, according to XFADECTL, to tell it apart from your own objects.
- For a PDF, use PDFATTACH; for an image, IMAGEATTACH.

See also

XREF · -XREF · XCLIP · INSERT · External references

XBIND —Bind Reference Symbols

Binds layers, linetypes, styles and blocks of an external reference to the drawing.

- **Command:** `XBIND` · **Global name:** `_XBIND` · **Alias:** `XB`
- Ribbon: Insert › Reference

Binds single symbols of an external reference (blocks, dimension styles, layers, linetypes and text styles) to the drawing, like AutoCAD XBIND: the symbol `REF|name` is renamed `REF$0$name` and no longer depends on the reference, which stays attached. To bind the whole reference, use Bind in XREF.

Usage

1. Enter XBIND (or XB). In the **Xbind** dialog box, tick the symbols of each type and click OK.

In scripts, SCRIPT and AutoLISP there is no dialog box: it asks as in -XBIND.

Notes

- A layer also binds its dependent linetype. If the name `REF$0$name` already exists, `$1$`, `$2$`... are used.
- When the reference is reloaded, its dependent symbols appear again as `REF|name`. The dialog box cannot be scripted in the AutoCAD console: it has a unit test.

See also

-XBIND · XREF · XATTACH

XCLIP —Clip

Shows only part of an external reference or a block, inside or outside a boundary.

- **Command:** `XCLIP` · **Global name:** `_XCLIP` · **Alias:** `XC`
- Ribbon: Insert › Reference

Shows only part of an external reference or a block: what lies inside a boundary or, with the boundary inverted, what lies outside it. The reference itself does not change: the boundary can be turned off or removed later.

Usage

1. Enter XCLIP (or XC) or click the button.
2. Select the external references or blocks and press Enter.

```
Select objects:
```

3. Choose the option; Enter creates a new boundary.

```
Enter clipping option [ON/OFF/Clipdepth/Delete/Polyline/New boundary] <New>:
```

4. If any of them was already clipped, confirm that its boundary is replaced.

```
Delete old boundary(s)? [Yes/No] <Yes>:
```

5. Choose the boundary type; Enter, rectangular.

```
Outside mode: objects outside the boundary will be hidden. Specify clipping boundary [Select
 polyline/Polygonal/Rectangular/Invert clip] <Rectangular>:
```

6. For a rectangular one, specify two opposite corners; for a polygonal one, the vertices and Enter (or Close) when there are at least three.

```
Specify first corner point:
```

Options

- **New** (N): draws a new boundary.
- **ON** / **OFF** (ON / OFF): turns on or off the boundary they already have.
- **Delete** (D): removes the boundary; they are shown whole again.
- **Polyline** (P): creates a polyline with the shape of the boundary of each clipped object.
- **Clipdepth** (C): the front and back planes of the clips: a point (its Z) or **Distance** from the boundary; **Remove** removes the plane.

```
Specify front clip point or [Distance/Remove]:
```

- **Select** (S): uses a closed polyline with straight segments as the boundary.

```
Select polyline:
```

- **Polygonal** (P) and **Rectangular** (R): the type of boundary to draw.
- **Invert** (I): switches between hiding what is outside and hiding what is inside.

```
Inside mode: objects inside the boundary will be hidden. Specify clipping boundary [Select p
olyline/Polygonal/Rectangular/Invert clip] <Rectangular>:
```

Notes

- The boundary is shown if XCLIPFRAME is not 0, on the layer and in the color of the insert.
- It is saved as in AutoCAD (a spatial filter on the insert): AutoCAD shows it the same way.
- **Clipdepth** saves the front and back clipping planes, but the plan view does not clip by Z (AutoCAD does, in

3D).
- For PDF underlays, use PDFCLIP; for images, IMAGECLIP.

See also

XATTACH · XREF · INSERT · External references

XOPEN —Open External Reference

Opens the drawing of an external reference in another tab.

- **Command:** XOPEN · **Global name:** _XOPEN
- Ribbon: Insert › Reference

Opens the drawing of an external reference in another tab, like AutoCAD XOPEN.

Usage

1. Enter XOPEN and select an external reference.

```
Select Xref:
```

Notes

- If the drawing is already open, its tab becomes active. After changing and saving it, go back to the host and reload the reference in XREF.
- It cannot be scripted in the AutoCAD console (it opens another drawing): it has a unit test.

See also

XREF · XATTACH · REFEDIT

XREF —External References

Lists the external references of the drawing.

- **Command:** XREF · **Global name:** _XREF · **Alias:** XR
- Ribbon: Insert › Reference
- Ribbon: View › Palettes

Opens (or closes) the **External References** palette: all the external references of the drawing (DWG or DXF drawings, PDF underlays and raster images) with their status, and the buttons to attach, reload, unload, detach, bind or change the path of each one.

Usage

1. Enter XREF (or XR) or click the button.
2. At the top, **Attach DWG…** opens XATTACH, **Attach PDF…** opens PDFATTACH and **Reload All References** reads all of them again from disk.
3. Below, the tree of DWG references: each one with its nested references hanging from it, and the columns **Reference Name**, **Status**, **Type** (Attach or Overlay), **Size**, **Date**, **Saved Path** and **Found At** (the file that was read).
4. The buttons of each row: **Reload**, **Unload**, **Detach**, **Bind** and **Change Path…** Nested references are managed from the one that contains them.
5. Further down, **PDF Underlays** (with their page) and **Raster images**, with their own buttons.

Reference status

- **Loaded**: it has been read and is drawn at each insert.
- **Unloaded**: it is still in the drawing, but it is not read or drawn.
- **Not Found**: the file is neither at the saved path nor in the drawing's folder.
- **Unreferenced**: it is defined, but not inserted anywhere.
- **Needs reloading**: the file has changed since it was read.
- **Circular reference: not loaded**, **Network path: not loaded** or **Read error**: it has not been loaded for that reason.

Notes

- The palette can be docked to the left or right of the drawing and set to auto-hide with the buttons in its header (? ? ? ?); see The user interface.
- The buttons run -XREF, -IMAGE or the corresponding command, so everything can be undone with UNDO.
- When the file of a loaded reference changes on disk, the status bar warns about it (**External references modified: ..**) with a **Reload** button.
- The date is shown in UTC.
- The content of references is never saved in this drawing: when it is saved they remain references, except those that are bound.

See also

XATTACH · -XREF · XCLIP · PDFATTACH · IMAGEATTACH · External references

Layers

-LAYER —Layers (command line)

Manages layers from the command line.

- **Command:** -LAYER · **Global name:** _-LAYER · **Alias:** -LA

Manages layers from the command line, without windows: useful in scripts and in LISP. It does the same as the LAYER: create layers, set the current one, rename them and change their properties.

Usage

1. Enter -LAYER (or -LA).

```
Enter an option [?/Make/Set/New/Rename/ON/OFF/Color/Ltype/LWeight/TRansparency/MATerial/Plot
/Freeze/Thaw/LOck/Unlock/stAte/Description/rEconcile]:
```

2. Choose an option. Options that change a property ask for the value first and then for the layers:

```
Enter name list of layer(s) (wildcards allowed) <…>:
```

3. After each option the first prompt comes back. Press Enter to finish.

Layer lists accept several names separated by commas and wildcards (`*`, `?`, `#`, `@`, `~`). Pressing Enter without typing anything applies the option to the current layer.

Options

- **?**: lists the layers with their status, color and linetype.
- **Make** (M): makes a layer current; if it does not exist, it is created.
- **Set** (S): makes an existing layer current.
- **New** (N): creates layers without making them current.

```
Enter name list for new layer(s) (comma-separated):
```

- **Rename** (R): renames a layer.
- **ON** / **OFF**: turns layers on or off.
- **Color** (C): a number from 1 to 255, a standard color name (red, yellow...) or R,G,B.

```
Enter color (number from 1 to 255, standard name or R,G,B):
```

- **Ltype** (L): a loaded linetype (Enter = Continuous).
- **LWeight** (LW): a standard lineweight in millimeters or **Default**.
- **TRansparency** (TR): the layer transparency, from 0 (opaque) to 90 %.

```
Enter transparency value (0-90):
```

- **Plot** (P): **Plot** or **No plot**.
- **Freeze** (F) / **Thaw** (T): freezes or thaws layers.
- **LOck** (LO) / **Unlock** (U): locks or unlocks layers.
- **stAte** (A): layer states.

```
Enter a layer state option [?/Save/Restore/Delete]:
```

Save saves the current setup under a name, **Restore** brings it back, **Delete** deletes it and **?** lists the saved states.

- **Description** (D): sets the description of the layers.

Notes

- The current layer cannot be frozen; with wildcards, a warning is shown and the other layers are changed.
- Options are typed in the language of the interface (in Spanish: Establ, Def, Inut...).
- Every option can be undone with UNDO and is remembered by LAYERP.

See also

LAYER · LAYERSTATE · SCRIPT · Layers

CHECKSTANDARDS —Check Standards

Checks layers, linetypes and styles against the associated standards.

- **Command:** `CHECKSTANDARDS` · **Global name:** `_CHECKSTANDARDS` · **Alias:** `CHK`
- Ribbon: Home › Layers

Checks the drawing against the standards files associated with STANDARDS: layers, linetypes, text styles and dimension styles. It lists what is not in the standard and what has the same name with other properties, and offers to fix the latter by copying the properties of the standard.

Usage

1. Enter CHECKSTANDARDS. The report of each standard appears.
2. If something can be fixed:

```
Fix the properties that differ from the standard? [Yes/No] <N>:
```

Notes

- What is not in the standard is only listed: it is not renamed or replaced.
- The fix can be undone with UNDO.

See also

STANDARDS · LAYTRANS · Layers

COPYTOLAYER —Copy to Layer

Copies the selected objects to another layer.

- **Command:** `COPYTOLAYER` · **Global name:** `_COPYTOLAYER`
- Ribbon: Home › Layers

Copies the selected objects to another layer: the copies go on the destination layer and the originals stay as they were. The copies can be left in place or moved.

Usage

1. Enter COPYTOLAYER.

```
Select objects to copy:
```

2. Select the objects and press Enter.

```
Select object on destination layer or [Name] <Name>:
```

3. Select an object on the destination layer, or press Enter and type the layer name.

```
… object(s) copied and placed on layer "…".
```

4. Specify the base point and the second point to move the copies, or press Enter to leave them in place.

```
Specify base point or [Displacement/eXit] <eXit>:
```

Options

- **Name** (N): type the name of the destination layer.
- **Displacement** (D): type the displacement of the copies instead of two points.

```
Specify displacement <0.0000, 0.0000, 0.0000>:
```

- **eXit**: leaves the copies on top of the originals.

See also

LAYCUR · LAYMCH · COPY · Layers

LAYCUR —Change to Current Layer

Moves the selected objects to the current layer.

- **Command:** LAYCUR · **Global name:** _LAYCUR
- Ribbon: Home › Layers

Moves the selected objects to the current layer, without changing their other properties. Use it to gather on the working layer objects drawn on another one.

Usage

1. Enter LAYCUR.

```
Select objects to be changed to the current layer:
```

2. Select the objects and press Enter. The program says how many changed.

```
… object(s) changed to layer "…" (the current layer).
```

Notes

- If objects are selected before the command starts, those are the ones changed, with no more questions.
- Objects on locked layers cannot be selected.

See also

LAYMCUR · LAYMCH · COPYTOLAYER · Layers

LAYDEL —Delete

Deletes some layers with all their objects.

- **Command:** LAYDEL · **Global name:** _LAYDEL
- Ribbon: Home › Layers

Deletes layers together with all their objects. Unlike the **Delete Layer** button of the LAYER, which only deletes empty layers, this command first erases whatever is on them.

Usage

1. Enter LAYDEL or click **Delete** on the ribbon.

```
Select object on layer to delete or [Name]:
```

2. Select objects on the layers to delete; the command line shows the chosen layers. Press Enter.
3. Confirm. The default answer is No.

```
… objects and layers … will be deleted. Do you wish to continue? [Yes/No] <No>:
```

Options

- **Name** (N): type the layer names instead of selecting objects (wildcards are accepted).

```
Enter names of the layers to delete:
```

- **Yes** (Y) / **No** (N): deletes or cancels.

Notes

- Layer 0, Defpoints, the current layer and xref layers cannot be deleted.
- Layout viewports on those layers are not erased: they are moved to layer 0.
- It can be undone with UNDO.

See also

LAYMRG · PURGE · LAYER · Layers

LAYER —Layers

Opens the Layer Properties Manager.

- **Command:** `LAYER` · **Global name:** `_LAYER` · **Alias:** `LA`, `DDLMODES`
- Ribbon: Home › Layers
- Ribbon: View › Palettes
- Menu: View
- Menu: Format

Opens and closes the **Layer Properties Manager**, a palette that lists every layer of the drawing in a table. From it you create, delete and rename layers, set the current layer and change layer properties. To do the same from the command line (scripts, LISP), use -LAYER.

Usage

1. Enter LAYER (or LA) or click **Layers** on the ribbon. Run it again to close the window.
2. Click a row to select that layer; Ctrl+click adds or removes rows and Shift+click selects a range.
3. Click the icons or drop-down lists of the row. If the row is one of the selected ones, the change applies to all selected layers.

Columns

- **Status**: marks the current layer. Double-click to make the layer current.
- **Name**: double-click to type a new name (Enter confirms, Esc cancels).
- **On**: light bulb. A layer that is off is neither displayed nor plotted.
- **Freeze**: sun. A frozen layer is not displayed, not plotted and not regenerated. The current layer cannot be frozen.
- **VP Freeze**: only shown inside a layout viewport; freezes the layer in that viewport only.

- **Lock**: padlock. Objects on a locked layer are visible but cannot be selected or modified.
- **Color**, **Linetype**, **Lineweight**: drop-down lists. The linetype list shows the loaded linetypes and, at the end, **Load…** opens LINETYPE.
- **Transparency**: from 0 (opaque) to 90 %, in steps of 10. Objects on the layer with ByLayer transparency take it; on screen it shows while the TRANSP button on the status bar is on.
- **Plot**: printer; when crossed out, the layer is not plotted.
- **New VP Freeze**: the layer will be frozen in viewports created from now on.
- **Description**: free text; double-click to type it.

Click a column heading to sort by it; another click reverses the order.

Buttons and filters

- **New Layer**: creates a layer with the properties of the selected one and leaves the name ready to type.
- **New Layer VP Frozen in All Viewports**: the same, but frozen in all layout viewports.
- **Delete Layer**: does not delete layer 0, Defpoints, the current layer, xref layers or layers that contain objects (to delete a layer together with its objects, use LAYDEL).
- **Set Current**, **New Property Filter…**, **New Group Filter** and **Layer States…** (opens LAYERSTATE).
- The search box shows only the layers whose name contains the text; wildcards are accepted (`*`, `?`, `#`, `@`, `~` and commas).
- The **Filters** tree chooses which layers are listed. The shortcut menu of a filter has **Properties…**, **Add Selected Layers**, **Remove Selected Layers** and **Delete Filter**; **Invert filter** lists the layers that do not match it.
- The shortcut menu of a row: **Set Current**, **Rename**, **Delete Layer**, **Select All**, **Invert Selection** and **Add to Group Filter**.

Notes

- The palette can be docked to the left or right of the drawing and set to auto-hide with the buttons in its header (? ? ? ?); see The user interface.
- Every change can be undone with UNDO, and LAYERP restores the layers as they were before the last change.
- Layers cannot be changed while a command is running: finish or cancel it first.
- The layer control on the ribbon (and on the layers toolbar of the Classic workspace) shows the current layer: open it to change the current layer with one click or to turn on, freeze or lock any layer with its icons.
- Layer filters and layer states are saved in the drawing.

See also

-LAYER · LAYERSTATE · LAYERP · LAYMCUR · LAYISO · Layers

LAYERP — Previous

Undoes the last change to layer settings.

- **Command:** LAYERP · **Global name:** _LAYERP
- Ribbon: Home › Layers

Undoes the last change to layer settings: the layers return to how they were before (on, frozen, locked, color, linetype, lineweight...). Objects are not touched: an object that was moved or put on another layer stays as it is.

Usage

1. Enter LAYERP or click **Previous** on the ribbon.
2. The layers return to their previous settings and the command line confirms it. Repeat to go further back.

Notes

- Up to 50 changes are remembered, made in the LAYER, in the layer control or with the layer commands (LAYOFF, LAYISO, -LAYER...).
- It does not undo renamed layers or bring back deleted layers: use UNDO for that.
- When there is nothing left to undo, it says so. It does not work while a command is running: finish it first.
- LAYERP itself can be undone with UNDO.

See also

LAYER · LAYERSTATE · UNDO · Layers

LAYERSTATE — Layer States

Opens the Layer States Manager.

- **Command:** LAYERSTATE · **Global name:** _LAYERSTATE · **Alias:** LAS, LMAN
- Ribbon: Home › Layers
- Menu: Format

Opens the **Layer States Manager**. A layer state saves under a name how all layers are set (on, frozen, locked, color, linetype, lineweight...) so that you can return to that setup later. Layer states are saved in the drawing.

Usage

1. Enter LAYERSTATE (or LAS) or click **Layer States…** in the LAYER.
2. To save the current setup, click **New…**, type the **State name** and, optionally, a **Description**, and click **Save**. A state with the same name is replaced.
3. To return to a state, choose it in the list and click **Restore** (or double-click it).

Controls

- **Delete**: deletes the chosen state; the layers do not change.
- **Layer properties to restore:** check boxes that decide what is restored from the chosen state: **On/Off**, **Frozen/Thawed**, **Locked/Unlocked**, **Plot/No Plot**, **New VP Frozen**, **Color**, **Linetype**, **Lineweight** and **Transparency**.

Notes

- Restoring a state can be undone with UNDO or with LAYERP.
- From the command line, the **stAte** option of -LAYER saves, restores and deletes states.
- Layer states are stored in the drawing in the same place as AutoCAD's.

See also

LAYER · -LAYER · LAYERP · Layers

LAYFRZ — Freeze

Freezes the layer of the selected object.

- **Command:** LAYFRZ · **Global name:** _LAYFRZ
- Ribbon: Home › Layers

Freezes the layer of the objects you select. A frozen layer is neither displayed nor plotted, like a layer that is off, and it is also not regenerated, which makes large drawings lighter. The command keeps asking for objects until

you press Enter.

Usage

1. Enter LAYFRZ or click **Freeze** on the ribbon.

```
Select an object on the layer to be frozen or [Settings/Undo]:
```

2. Select an object: its layer is frozen. Repeat with other objects.
3. Press Enter to finish.

Options

- **Settings** (S): inside a layout viewport, whether the layer is frozen in that viewport only or in the whole drawing.

```
In paper space viewport use [Vpfreeze/Freeze] <Vpfreeze>:
```

- **Undo** (U): thaws the last layer frozen in this command.

Notes

- If objects are selected before the command starts, their layers are frozen and the command ends.
- The current layer cannot be frozen: a warning is shown and it does not change.
- The layer is frozen in the whole drawing. To freeze it in one layout viewport only, use the **VP Freeze** column of the LAYER.

See also

LAYTHW · LAYOFF · LAYER · Layers

LAYISO — Isolate

Leaves only the layers of the selected objects visible.

- **Command:** `LAYISO` · **Global name:** `_LAYISO`
- Ribbon: Home › Layers

Keeps only the layers of the selected objects visible: all other layers are turned off. Useful to work on one part of the drawing without the rest getting in the way. LAYUNISO returns the layers to how they were.

Usage

1. Enter LAYISO or click **Isolate** on the ribbon.

```
Select objects on the layer(s) to be isolated or [Settings]:
```

2. Select one or more objects and press Enter. Their layers stay on and all others are turned off.

Options

- **Settings** (S): what happens to the other layers: turned off (and, inside a viewport, frozen in it only) or locked and faded.

```
Enter setting for layers not isolated [Off/Lock and fade] <Off>:
Enter fade value (0-90) <…>:
```

Notes

- If objects are selected before the command starts, their layers are isolated without asking.
- If the current layer is not one of the isolated layers, the first of them becomes current.
- Layers are turned off, not frozen or locked. If you isolate several times in a row, LAYUNISO returns to the

setup before the first isolation.

- To isolate individual objects instead of whole layers, use ISOLATEOBJECTS.

See also

LAYUNISO · LAYOFF · ISOLATEOBJECTS · Layers

LAYLCK — Lock

Locks the layer of the selected object.

- **Command:** `LAYLCK` · **Global name:** `_LAYLCK`
- Ribbon: Home › Layers

Locks the layer of the selected object. Its objects are still displayed, plotted and usable for object snaps, but they cannot be selected or modified. This is how you protect a base plan or a reference drawing while drawing on top of it.

Usage

1. Enter LAYLCK or click **Lock** on the ribbon.

```
Select an object on the layer to be locked:
```

2. Select an object. Its layer is locked and the command ends.

Notes

- If objects are selected before the command starts, their layers are locked.
- You can draw on a locked layer if it is the current layer.

See also

LAYULK · LAYER · Layers

LAYMCH — Match Layer

Moves the selected objects to the layer of another object.

- **Command:** `LAYMCH` · **Global name:** `_LAYMCH`
- Ribbon: Home › Layers

Moves the selected objects to the layer of another object (the destination layer). Use it to fix objects drawn on the wrong layer without having to look up the name of the right one.

Usage

1. Enter LAYMCH or click **Match Layer** on the ribbon.

```
Select objects to be changed:
```

2. Select the objects to change and press Enter.

```
Select object on target layer or [Name]:
```

3. Select an object on the destination layer. The objects change layer and the command ends.

Options

- **Name** (N): type the name of the destination layer instead of selecting an object.

```
Enter target layer name:
```

Notes

- If objects are selected before the command starts, those are the ones changed and the command goes straight to the destination layer.
- Objects on locked layers cannot be selected for changing; the destination object may be on a locked layer.
- To copy color, linetype and lineweight as well, use MATCHPROP.

See also

MATCHPROP · LAYMCUR · LAYMRG · Layers

LAYMCUR — Make Current

Makes the layer of the selected object current.

- **Command:** `LAYMCUR` · **Global name:** `_LAYMCUR`
- Ribbon: Home › Layers

Makes the layer of an object current: whatever you draw next goes on that layer. It is quicker than looking for the layer by name when an object on it is in view.

Usage

1. Enter LAYMCUR or click **Make Current** on the ribbon.

```
Select object whose layer will become current:
```

2. Select an object. Its layer becomes the current layer and the command ends.

Notes

- If objects are selected before the command starts, the layer of the first one is used without asking.
- A frozen layer cannot be the current layer.
- The change can be undone with UNDO or with LAYERP.

See also

LAYMCH · LAYER · Layers

LAYMRG — Merge

Moves all the objects of some layers to another one and deletes the former.

- **Command:** `LAYMRG` · **Global name:** `_LAYMRG`
- Ribbon: Home › Layers

Merges layers: moves all objects of some layers to another one (the destination layer) and deletes the first ones. Use it to clean up drawings with duplicated or badly named layers.

Usage

1. Enter LAYMRG or click **Merge** on the ribbon.

```
Select object on layer to merge or [Name]:
```

2. Select objects on the layers to merge; the command line shows the chosen layers. Press Enter.

```
Select object on target layer or [Name]:
```

3. Select an object on the destination layer.
4. Confirm. The default answer is No.

```
… objects will move to layer … and layers … will be deleted. Do you wish to continue? [Yes/No] <No>:
```

Options

- **Name** (N): type the layer names instead of selecting objects (wildcards are accepted). At the destination prompt, the same option asks for its name.

```
Enter names of the layers to merge:
```

- **Yes** (Y) / **No** (N): merges or cancels.

Notes

- Layer 0, Defpoints and xref layers cannot be merged, and the destination cannot be one of the layers being merged.
- If the current layer is one of the merged layers, the destination layer becomes current.
- It can be undone with UNDO.

See also

LAYDEL · LAYMCH · PURGE · Layers

LAYOFF —Off

Turns off the layer of the selected object.

- **Command:** LAYOFF · **Global name:** _LAYOFF
- Ribbon: Home › Layers

Turns off the layer of the objects you select: their objects are no longer displayed or plotted until the layer is turned on again. The command keeps asking for objects, so you can turn off several layers in a row.

Usage

1. Enter LAYOFF or click **Off** on the ribbon.

```
Select an object on the layer to be turned off or [Settings/Undo]:
```

2. Select an object: its layer is turned off. Repeat with other objects.
3. Press Enter to finish.

Options

- **Settings** (S): inside a layout viewport, whether the layer is frozen in that viewport only or turned off in the whole drawing.

```
In paper space viewport use [Vpfreeze/Off] <Vpfreeze>:
```

- **Undo** (U): turns back on the last layer turned off in this command.

Notes

- If objects are selected before the command starts, their layers are turned off and the command ends.
- The current layer can be turned off: whatever you draw on it will not be seen until it is turned on.

- To turn them all back on, use LAYON; for a single layer, the LAYER or the layer control.

See also

LAYON · LAYFRZ · LAYISO · LAYERP · Layers

LAYON —Turn All Layers On

Turns on all the layers.

- **Command:** `LAYON` · **Global name:** `_LAYON`
- Ribbon: Home › Layers

Turns on all layers of the drawing at once. There are no prompts.

Usage

1. Enter LAYON or click **Turn All Layers On** on the ribbon.

Notes

- Frozen layers stay frozen: use LAYTHW for that.
- It can be undone with UNDO or with LAYERP.

See also

LAYOFF · LAYTHW · LAYUNISO · Layers

LAYTHW —Thaw All Layers

Thaws all the layers.

- **Command:** `LAYTHW` · **Global name:** `_LAYTHW`
- Ribbon: Home › Layers

Thaws all frozen layers of the drawing at once. There are no prompts.

Usage

1. Enter LAYTHW or click **Thaw All Layers** on the ribbon.

Notes

- Layers that are off stay off: use LAYON for that.
- Layers frozen in individual layout viewports are not changed.
- It can be undone with UNDO or with LAYERP.

See also

LAYFRZ · LAYON · Layers

LAYTRANS —Layer Translator

Translates the layers of the drawing to those of another drawing or standard.

- **Command:** `LAYTRANS` · **Global name:** `_LAYTRANS`
- Ribbon: Home › Layers

Translates the layers of the drawing to those of another drawing or standard: objects move to the target layer, which takes the color, linetype and lineweight of the reference. LinceCAD does it on the command line, without the Layer Translator dialog box.

Usage

1. Enter LAYTRANS.

```
Enter the drawing or standard with the target layers:
```

2. Type `source=target` mappings; the source accepts wildcards (`WAL*=A-WALL`). **Map same** applies to the layers that already have the same name as in the reference.

```
Enter source=target, Map same or <Translate>:
```

3. Enter (or **Translate**) translates: the source layer is deleted if it ends up empty (never 0 or the current one).

Notes

- The result can be undone with UNDO.
- It does not save the mapping file and does not change the linetype if the reference uses one that is not loaded.

See also

STANDARDS · CHECKSTANDARDS · LAYMRG · Layers

LAYULK —Unlock

Unlocks the layer of the selected object.

- **Command:** `LAYULK` · **Global name:** `_LAYULK`
- Ribbon: Home › Layers

Unlocks the layer of the selected object: its objects can be selected and modified again.

Usage

1. Enter LAYULK or click **Unlock** on the ribbon.

```
Select an object on the layer to be unlocked:
```

2. Select an object on the locked layer. The layer is unlocked and the command ends.

Notes

- If objects are selected before the command starts, their layers are unlocked.
- You can also unlock a layer by clicking the padlock in the LAYER or in the layer control.

See also

LAYLCK · LAYER · Layers

LAYUNISO —Unisolate

Undoes the last LAYISO.

- **Command:** `LAYUNISO` · **Global name:** `_LAYUNISO`
- Ribbon: Home › Layers

Undoes the last LAYISO: turns layers on and off as they were before they were isolated.

Usage

1. Enter LAYUNISO or click **Unisolate** on the ribbon. There are no prompts.

Notes

- Only the on/off setting of the layers is restored; changes to color, linetype or other properties made in the meantime are kept.
- If no layers are isolated, it says so and nothing changes.
- If the current layer has ended up frozen, layer 0 becomes current.

See also

LAYISO · LAYON · LAYERP · Layers

LAYVPI —Isolate to Viewport

Freezes the selected layers in every viewport except the current one.

- **Command:** `LAYVPI` · **Global name:** `_LAYVPI`
- Ribbon: Home › Layers

Freezes the layers of the selected objects in every viewport of the layout except the current one, so that those objects are only seen in it.

Usage

1. In a layout, go into a viewport (MSPACE) and enter LAYVPI.

```
Select objects on the layer to be isolated in viewport or [Undo]:
```

2. Select objects. Their layers are frozen in the other viewports at once; keep selecting or press Enter.

```
Layers … frozen in the other viewports.
```

Options

- **Undo** (U): thaws again the layers of the last selection.

Notes

- If you are not inside a viewport, the program says so.

```
You must be in a layout, inside a viewport (MSPACE).
```

See also

VPLAYER · LAYISO · LAYER · Layers

LAYWALK —Layer Walk

Shows only the checked layers to see what each one holds.

- **Command:** `LAYWALK` · **Global name:** `_LAYWALK`
- Ribbon: Home › Layers

Shows only the layers you check, to see what is on each one. The others are turned off while the dialog box is open.

Usage

1. Enter LAYWALK.

```
Layer Walk: use the dialog box; Enter closes it.
```

2. In the dialog box, type a filter (* and ? are accepted) and check the layers you want to see; the number of objects of each appears in brackets. With nothing checked, all are shown.
3. **Purge layer** deletes the checked layer if it is empty (not 0 or the current one).
4. Click **Close** or press Enter: the layers go back to how they were before opening the dialog box, except the purged ones.

See also

LAYER · LAYISO · Layers

SETBYLAYER — Set to ByLayer

Sets the color, linetype, lineweight and other properties of the objects to ByLayer.

- **Command:** SETBYLAYER · **Global name:** _SETBYLAYER
- Ribbon: Home › Layers

Sets the properties of the selected objects to ByLayer, so that they take those of their layer: color, linetype, lineweight, material, plot style and transparency, according to SETBYLAYERMODE.

Usage

1. Enter SETBYLAYER. The program says which properties it will change.

```
Current active settings: …
```

2. Select the objects and press Enter.

```
Select objects or [Settings]:
```

3. Say whether to change the ByBlock properties too.

```
Change ByBlock to ByLayer? [Yes/No] <Yes>:
```

4. Say whether to change the objects inside the selected blocks too (their definition changes).

```
Include blocks? [Yes/No] <Yes>:
```

Options

- **Settings** (S): changes SETBYLAYERMODE, the sum of the properties set to ByLayer.

```
Enter new value for SETBYLAYERMODE (1 color, 2 linetype, 4 lineweight, 8 material, 16 plot s
tyle, 128 transparency) <…>:
```

Notes

- With "Include blocks" every reference of the block changes, because its definition does.

See also

MATCHPROP · PROPERTIES · Layers

STANDARDS —Standards

Associates standards files (.dws) with the drawing.

- **Command:** `STANDARDS` · **Global name:** `_STANDARDS` · **Alias:** `STA`
- Ribbon: Home › Layers

Associates standards files with the drawing so that CHECKSTANDARDS can check it later. A `.dws` file or any drawing (`.dwg`, `.dxf`, `.dwt`) will do.

Usage

1. Enter STANDARDS.

```
Enter an option [Associate/Remove/List] <List>:
```

2. **Associate**: give the file; it is checked that it opens.

```
Enter the standards file (.dws or drawing):
```

3. **Remove**: give the number in the list or the path.
4. **List** (Enter): writes the associated standards.

Notes

- The list travels with the drawing as the `LinceNormas` custom property (see DWGPROPS); it is not AutoCAD's standards dictionary, which is not read.

See also

CHECKSTANDARDS · LAYTRANS · Layers

VPLAYER —Viewport Layers

Freezes and thaws layers in some viewports, from the command line.

- **Command:** `VPLAYER` · **Global name:** `_VPLAYER`
- Ribbon: Home › Layers

Freezes and thaws layers in some viewports of the layout, from the command line: the same as the "VP Freeze" column of the layer manager, but in several viewports at once.

Usage

1. In a layout, enter VPLAYER.

```
Enter an option [?/Color/Ltype/LWeight/TRansparency/Freeze/Thaw/Reset/remOveoverrides/Newfrz
/Vpvisdflt]:
```

2. Choose an option, type the layer names (wildcards allowed; Enter to select objects on them) and choose the viewports.

```
Specify viewports [All/Select/Current/eXcept current] <Current>:
```

3. Go on with another option or press Enter to finish.

Options

- **?**: lists the layers frozen in the current viewport.
- **Freeze** (F): freezes the layers in the viewports.

- **Thaw** (T) and **Reset** (R): thaw them.
- **Newfrz** (N): creates new layers frozen in every viewport.
- **Vpvisdflt** (V): sets whether the layer starts frozen in new viewports.

```
Enter a viewport visibility option [Frozen/Thawed] <Thawed>:
```

- Viewports: **All** those of the layout, **Select** the ones you select, **Current** the current one and **eXcept current** all but the current one.

Notes

- Color, Ltype, LWeight, TRansparency and Removeoverrides (per-viewport property overrides) are not available yet.

```
Per-viewport color, linetype, lineweight and transparency overrides are not available yet.
```

See also

LAYVPI · LAYER · MVIEW · Layers

Properties

-COLOR —Color (command line)

Sets the color of new objects from the command line.

- **Command:** `-COLOR` · **Global name:** `_-COLOR` · **Alias:** `-COL`

Sets the color of new objects from the command line, like COLOR without the palette. Useful in scripts and AutoLISP.

Usage

1. Enter -COLOR. The program says the current color.

```
Enter default object color [Truecolor] <BYLAYER>:
```

2. Type ByLayer, ByBlock, an index number (1 to 255) or a standard name (red, yellow, green, cyan, blue, magenta, white).

Options

- **Truecolor** (T): a true color, with its red, green and blue components.

```
True Color (R,G,B):
```

Notes

- It is the CECOLOR variable. Color books (AutoCAD's Colorbook) are not available.

See also

COLOR · -LWEIGHT · MATCHPROP

-LWEIGHT —Lineweight (command line)

Sets the lineweight of new objects from the command line.

- **Command:** `-LWEIGHT` · **Global name:** `_-LWEIGHT` · **Alias:** `-LW`

Sets the lineweight of new objects from the command line, like LWEIGHT without the palette. Useful in scripts and AutoLISP.

Usage

1. Enter -LWEIGHT. The program says the current lineweight.

```
Enter default lineweight for new objects or [?]:
```

2. Type ByLayer, ByBlock, Default or one of the standard lineweights in millimetres (0.25, 0.35…).

Options

- **?**: lists the standard lineweights.

```
Standard lineweights (mm): …
```

Notes

- It is the CELWEIGHT variable. A non-standard lineweight is rejected.

```
Invalid lineweight: enter one of the standard lineweights in mm (e.g. 0.25) or Default.
```

See also

LWEIGHT · -COLOR · LWDISPLAY

COLOR —Color

Sets the color of new objects.

- **Command:** `COLOR` · **Global name:** `_COLOR` · **Alias:** `COL`, `COLOUR`, `DDCOLOR`
- Ribbon: Home › Properties
- Menu: Format

Sets the color of new objects with the **Select Color** dialog box, as in AutoCAD. From the command line use -COLOR.

Usage

1. Enter COLOR (or COL) or click **Color** on the ribbon.
2. In the dialog box, choose the tab and the color:

- **Index Color**: **ByLayer** (each object takes its layer color, the usual choice), **ByBlock** (it takes the color of the block it is inserted in), the 7 standard colors and the 255 of **Index color (ACI)**.
- **True color**: hue, saturation and luminance (HSL) or red, green and blue (RGB).
- **Color Books**: named colors from a book; it includes an extract of RAL Classic (AutoCAD .acb books are not read).

3. Click **OK**.

Notes

- The PROPERTIES palette still lets you change the current color and that of the selected objects.
- The current color is stored in the CECOLOR variable.

See also

PROPERTIES · LINETYPE · LWEIGHT · LAYER

LINETYPE —Linetype

Loads and sets linetypes.

- **Command:** `LINETYPE` · **Global name:** `_LINETYPE` · **Alias:** `LT`, `LTYPE`, `DDLTYPE`
- Ribbon: Home › Properties
- Menu: Format

Opens the **Linetype Manager**: loads linetypes into the drawing, sets the current one (used by new objects), deletes unused ones and sets the pattern scales.

Usage

1. Enter LINETYPE (or LT) or click **Linetype** on the ribbon.
2. Choose a linetype in the list and click **Set Current** so that new objects use it.

3. To add linetypes, click **Load...**, check the ones you want and click **OK**.

Controls

- **Load...**: shows **Load or Reload Linetypes** with the linetypes of the LinceCAD library that are not in the drawing yet (DASHED, HIDDEN, CENTER, DASHDOT, DOT, PHANTOM, DIVIDE and BORDER). **File...** reads the linetypes of a `.lin` file instead.
- **Delete**: deletes the chosen linetype unless it is fixed (ByLayer, ByBlock, Continuous), current or used by an object or a layer.
- **Global scale factor** (LTSCALE): multiplies the size of the dashes of all objects.
- **Current object scale** (CELTSCALE): linetype scale of new objects.
- **Apply scales**: applies the values typed in the scale boxes.

Notes

- The patterns of the built-in library are in millimeters. Complex linetypes (with text or shapes) in a `.lin` file are skipped.
- Linetypes already in the drawing are not loaded again.
- The linetype of a layer is changed in the LAYER; that of some objects, in the PROPERTIES palette.

See also

PROPERTIES · COLOR · LWEIGHT · CELTYPE

LWEIGHT —Lineweight

Sets the lineweight of new objects.

- **Command:** `LWEIGHT` · **Global name:** `_LWEIGHT` · **Alias:** `LW`, `LINEWEIGHT`
- Ribbon: Home › Properties
- Menu: Format

Sets the lineweight of new objects with the **Lineweight Settings** dialog box, as in AutoCAD. From the command line use -LWEIGHT.

Usage

1. Enter LWEIGHT (or LW) or click **Lineweight** on the ribbon.
2. In the dialog box, choose one of the **Lineweights:**: **ByLayer**, **ByBlock**, **Default** or one of the standard lineweights.
3. Adjust, if you wish:

- **Units for Listing:** Millimeters (mm) or Inches (in) (LWUNITS variable).
- **Display Lineweight**: draws the lineweight on screen (LWDISPLAY variable, like LWDISPLAY).
- **Default:** the lineweight given to what is "default" (LWDEFAULT); plotting uses it too.

4. Click **OK**.

Notes

- There is no "Adjust Display Scale": the lineweight is drawn in pixels and has no scale of its own.
- The PROPERTIES palette changes the lineweight of the selected objects.

See also

PROPERTIES · COLOR · LINETYPE · LWDISPLAY

MATCHPROP —Match Properties

Copies the properties of one object to others.

- **Command:** `MATCHPROP` · **Global name:** `_MATCHPROP` · **Alias:** `MA`, `PAINTER`
- Ribbon: Home › Properties
- Menu: Modify

Copies the properties of one object (the source) to others: color, layer, linetype, linetype scale, lineweight, transparency, thickness and plot style, and the type-specific ones (dimension style, text style, hatch, polyline). It is the quick way to make some objects look like another one that is already right.

Usage

1. Enter MATCHPROP (or MA) or click **Match Properties** on the ribbon.

```
Select source object:
```

2. Select the source object.

```
Select destination object(s) or [Settings]:
```

3. Select the destination objects; you can select several times.
4. Press Enter: the destination objects take the properties of the source and the command ends.

Notes

- If a single object is selected before the command starts, it is used as the source.
- The **Settings** option chooses which properties are matched: it opens the **Property Settings** dialog box and also asks for them at the command line (the number of each property turns it on or off; Enter ends), for scripts. It applies to the whole session.

```
Properties to match (…); enter a number to change it or Enter:
```

- Special properties only pass between objects of the same kind (dimension to dimension, text or multiline text to text, hatch to hatch, polyline to polyline). Those of viewports, tables and multileaders are not available yet, nor material or shadow.
- It can be undone with UNDO.

See also

PROPERTIES · LAYMCH · Editing

PROPERTIES —Properties

Displays and edits the properties of the selection.

- **Command:** `PROPERTIES` · **Global name:** `_PROPERTIES` · **Alias:** `PR`, `CH`, `MO`, `PROPS`, `DDCHPROP`, `DDMODIFY` · **Shortcut:** Ctrl+1
- Ribbon: Home › Properties
- Ribbon: View › Palettes
- Menu: View
- Menu: Modify
- Toolbar: Standard

Opens and closes the **Properties** palette, which shows the properties of the selected objects and lets you change them by typing or choosing from a list. With nothing selected, it shows the current properties of the drawing, the

ones that new objects will take.

Usage

1. Enter PROPERTIES (or PR), press Ctrl+1 or click **Properties** on the ribbon. Doing it again closes the palette.
2. Select objects, before or with the palette open: the palette updates by itself.
3. Change a value: in text boxes, type and press Enter (or leave the box); in drop-down lists, choose.

The palette

- **Top drop-down list**: the selected objects grouped by type, with their count (**All (…)** when there are several types). Choose a type to see and change only those objects. With nothing selected it reads **No selection**.
- **General** (or **General (current drawing)** with no selection): **Color**, **Layer**, **Linetype**, **Linetype scale**, **Lineweight** and **Transparency**.
- Sections depending on the type: **Geometry** (coordinates, radius, length, area...), **Text**, **Dimension** and **Overrides**, **Table** and **Cell**, **Misc**.
- For a polyline, **Vertex** has < and > buttons to step through the vertices and change their coordinates.
- For a block: **Attributes** (read-only; change them with EATTEDIT) and, for a dynamic block, **Custom** with its parameters.
- For texts and dimensions: **Annotative** (Yes or No) makes the object annotative with the current annotation scale or turns it back to normal, and **Annotative scales** shows the scales it supports (read-only; change them with OBJECTSCALE).
- Click the title of a section to collapse or expand it.

Notes

- The palette can be docked to the left or right of the drawing and set to auto-hide with the buttons in its header (? ? ? ?); see The user interface.
- When the selected objects have different values, ***VARIES*** is shown; what you type is applied to all of them.
- Calculated values (deltas, length, angle of a line, area...) are dimmed: they cannot be typed.
- Lengths and coordinates use the drawing units (UNITS).
- The color list offers ByLayer, ByBlock, the 7 standard colors, the 255 index colors and a **True color** applied with **Apply**.
- **Transparency** is ByLayer, ByBlock or 0 to 90 %, in steps of 10. With nothing selected it is the one for new objects (CETRANSPARENCY). It is saved in DWG files from version 2004.
- Double-clicking an object with no command running selects it and opens the palette (for texts, hatches, tables and blocks it opens their editor instead). Every change can be undone with UNDO.

See also

MATCHPROP · COLOR · LINETYPE · LWEIGHT · LIST · Editing

View

-LAYOUT — Layout (command line)

Creates, copies, deletes, renames and sets layouts from the command line.

- **Command:** `-LAYOUT` · **Global name:** `_-LAYOUT` · **Alias:** `LO`

Command-line version of LAYOUT, with the same options.

Usage

```
Enter layout option [Copy/Delete/New/Template/Rename/SAveas/Set/?] <Set>:
```

See also

LAYOUT

-VIEW — Preset Views

Sets a preset orthographic or isometric view.

- **Command:** `-VIEW` · **Global name:** `_-VIEW` · **Alias:** `-V`
- Ribbon: View › 3D Views
- Menu: View

Sets one of the AutoCAD preset orthographic or isometric views, with the target at the origin and zoom extents.

Usage

1. Enter -VIEW (or -V).

```
Enter an option [?/Save/Restore/Delete] or a preset view [Top/Bottom/Left/Right/Front/BAck/S
W iso/SE iso/NE iso/NW iso]:
```

2. Choose the view.

Options

- **Top**, **Bottom**, **Left**, **Right**, **Front** and **Back**: looking along an axis.
- **SW Isometric**, **SE Isometric**, **NE Isometric** and **NW Isometric**: isometric views from above.
- In scripts the AutoCAD global options also work: `_TOP`, `_BOTTOM`, `_LEFT`, `_RIGHT`, `_FRONT`, `_BACK`, `_SWISO`, `_SEISO`, `_NEISO` and `_NWISO`.

Notes

- The same views are in the list on the View tab, in the control at the top left corner of 3D views and on the view cube (faces, edges and corners; the house is the SW isometric).
- It does not change the UCS. AutoCAD named views cannot be created or restored.

See also

3DORBIT · VPOINT · PLAN · VSCURRENT

-VPORTS —Viewports (command line)

In a layout, creates viewports like MVIEW.

- **Command:** `-VPORTS` · **Global name:** `_-VPORTS`

In a layout, creates viewports like MVIEW.

Usage

```
Specify corner of viewport or [ON/OFF/Fit/Lock/Object/Polygonal/2/3/4] <Fit>:
```

3DCORBIT —Continuous Orbit

When the drag is released, the view keeps rotating.

- **Command:** `3DCORBIT` · **Global name:** `_3DCORBIT`
- Ribbon: View › 3D Views

Continuous orbit: drag and release; the view keeps rotating with the speed and direction of the last drag.

Usage

1. Enter 3DCORBIT or click the button on the View tab.

```
Drag to rotate the view; Enter or Esc to end:
```

2. Drag with the left button and release it while moving.
3. A click stops the rotation; Enter or Esc end the command.

See also

3DORBIT · 3DFORBIT

3DFORBIT —Free Orbit

Rotates the view freely around the screen axes.

- **Command:** `3DFORBIT` · **Global name:** `_3DFORBIT`
- Ribbon: View › 3D Views

Free orbit: like 3DORBIT, but dragging rotates the view around the screen axes, so the view may end up twisted (the Z axis no longer has to look vertical).

Usage

1. Enter 3DFORBIT or click the button on the View tab.

```
Drag to rotate the view; Enter or Esc to end:
```

2. Drag with the left button: horizontally it rotates around the vertical screen axis and vertically around the horizontal one.
3. Press Enter or Esc to end.

Notes

- The view twist is saved as VIEWTWIST.

See also

3DORBIT · 3DCORBIT · PLAN

3DORBIT —3D Orbit

Rotates the view around the target by dragging with the mouse (the Z axis stays vertical).

- **Command:** `3DORBIT` · **Global name:** `_3DORBIT` · **Alias:** `3DO`, `ORBIT`
- Ribbon: View › 3D Views
- Menu: View

Rotates the view around its target by dragging with the mouse, like AutoCAD 3DORBIT. The Z axis of the drawing stays vertical on screen.

Usage

1. Enter 3DORBIT (or 3DO) or click the button on the View tab.

```
Drag to rotate the view; Enter or Esc to end:
```

2. Drag with the left button: horizontally the view rotates around the Z axis; vertically it tilts.
3. Press Enter or Esc to end.

Notes

- With no active command, Shift + dragging with the middle button does the same orbit (as in AutoCAD).
- From the plan view, the target is the point at the center of the screen; in a 3D view, the center of the screen at half the depth of the drawing.
- The view is saved in the drawing (VIEWDIR, TARGET and VIEWTWIST of the current viewport) and AutoCAD opens it the same way.
- To go back to the plan view, PLAN.

See also

3DFORBIT · 3DCORBIT · -VIEW · VSCURRENT · PLAN

CHSPACE —Change Space

Moves objects from paper space to model space, or back, keeping their appearance.

- **Command:** `CHSPACE` · **Global name:** `_CHSPACE`

Moves objects from paper space to model space, or back, keeping their appearance.

Usage

1. In a layout, type CHSPACE and select the paper space objects.

```
Select objects:
```

2. Select the viewport through which they will be seen in model space.

```
Select the DESTINATION viewport:
```

Notes

- The objects are scaled with the viewport scale (and rotated with its twist) so they look the same as before, and the command reports the factor applied.

- Inside a viewport (MSPACE), CHSPACE moves the model objects you select to paper space.
- Viewports are not moved. On the Model tab the command does nothing.

See also

MSPACE · PSPACE · VPSYNC · Layouts

CLEANSCREENOFF —Clean Screen Off

Brings back the ribbon, toolbars and palettes after CLEANSCREENON.

- **Command:** `CLEANSCREENOFF` · **Global name:** `_CLEANSCREENOFF`

Leaves the clean screen and brings back the ribbon, toolbars and palettes, like AutoCAD CLEANSCREENOFF.

Usage

Enter CLEANSCREENOFF.

Notes

- If the clean screen was not on, it only shows the palettes again.

See also

CLEANSCREENON · SHOWPALETTES

CLEANSCREENON —Clean Screen

Hides the ribbon, toolbars, file tabs and palettes.

- **Command:** `CLEANSCREENON` · **Global name:** `_CLEANSCREENON`

Leaves only the drawing area, the status bar and the command line, like AutoCAD CLEANSCREENON.

Usage

Enter CLEANSCREENON. The ribbon (or the menus and toolbars of the Classic workspace), the file tabs and the palettes disappear.

Notes

- Commands are still typed on the command line. To go back, enter CLEANSCREENOFF.
- It is not stored with the workspace: after a restart the screen is shown complete again.

See also

CLEANSCREENOFF · HIDEPALETTES

DDPTOVISTA —Punto de vista por ángulos

Fija la dirección de la vista con dos ángulos.

- **Comando:** `DDPTOVISTA` · **Nombre global:** `_DDVPOINT` · **Alias:** `DDVP`
- Cinta: Vista › Vistas 3D

Fija la dirección de la vista con dos ángulos, como el cuadro de punto de vista predefinido (DDVPOINT) de AutoCAD, y hace zoom extensión.

Uso

1. Escriba DDPTOVISTA (o DDVP).

```
Indique el ángulo en el plano XY desde el eje X:
```

2. Indique el ángulo en el plano XY desde el eje X (en grados).

```
Indique el ángulo desde el plano XY:
```

3. Indique el ángulo desde el plano XY (la altura del observador).

Notas

- Con 270 y 90 se mira desde arriba (planta); con 270 y 0, de frente.
- Es otra forma de VPOINT: el objetivo pasa al origen y la dirección se guarda como VIEWDIR.
- Sin cuadro de diálogo en esta versión: se usa desde la línea de comandos.

Véase también

VPOINT · -VIEW · 3DORBIT · PLAN

LUZDISTANTE —Luz distante

Crea una luz distante de rayos paralelos, como el sol.

- **Comando:** `LUZDISTANTE` · **Nombre global:** `_DISTANTLIGHT`
- Cinta: Vista › Render

Crea una luz distante de rayos paralelos, como DISTANTLIGHT de AutoCAD (y como el sol).

Uso

1. Escriba LUZDISTANTE.

```
Indique el primer punto de la dirección:
```

2. Indique dos puntos: la dirección de los rayos va del primero al segundo.

Notas

- Se guarda como entidad LIGHT (AUDIT 0). Para la luz del sol según la geolocalización, use Sol.

Véase también

POINTLIGHT · SPOTLIGHT · SUN · RENDER

HIDEOBJECTS —Hide Objects

Hides the selected objects.

- **Command:** `HIDEOBJECTS` · **Global name:** `_HIDEOBJECTS`
- Ribbon: View › Isolate
- Menu: View

Hides the selected objects until isolation ends. It is the opposite of ISOLATEOBJECTS: here you select what is in the way.

Usage

1. Enter HIDEOBJECTS or choose **Hide Objects** on the **ISOLATE** button of the status bar.
2. Select the objects you want to hide and press Enter. If they were already selected before starting the command, they are hidden without asking.

```
Select objects:
```

3. To see them again, use UNISOLATEOBJECTS or **End Object Isolation** on the same button.

Notes

- It can be used several times in a row: each time more objects are hidden, and UNISOLATEOBJECTS shows them all.
- It does not change the drawing: nothing is saved or undone and the visibility of layers is not touched.
- Hidden objects cannot be seen, selected or used with object snaps.
- Known limit: hidden objects show again if the whole view is rebuilt (for example, when changing the color of a layer or switching to a layout).

See also

ISOLATEOBJECTS · UNISOLATEOBJECTS · LAYOFF · Layers

HIDEPALETTES — Hide Palettes

Hides all palettes, docked or floating.

- **Command:** `HIDEPALETTES` · **Global name:** `_HIDEPALETTES` · **Alias:** `POFF`

Hides all open palettes, like AutoCAD HIDEPALETTES.

Usage

Enter HIDEPALETTES. The docked and floating palettes disappear and the drawing area gets the room back.

Notes

- The palettes stay open, with their position and width: SHOWPALETTES brings them back.
- It does not hide the ribbon or the toolbars: for that, CLEANSCREENON.

See also

SHOWPALETTES · CLEANSCREENON

ISOLATEOBJECTS — Isolate Objects

Hides everything except the selected objects.

- **Command:** `ISOLATEOBJECTS` · **Global name:** `_ISOLATEOBJECTS` · **Alias:** `ISOLATE`
- Ribbon: View › Isolate
- Menu: View

Leaves only the selected objects in view: everything else in the same space (model or layout) is hidden until isolation ends. It helps to work on a part of a busy drawing without changing layers.

Usage

1. Enter ISOLATEOBJECTS or choose **Isolate Objects** on the **ISOLATE** button of the status bar.

2. Select the objects you want to keep seeing and press Enter. If they were already selected before starting the command, they are isolated without asking.

```
Select objects:
```

3. To see everything again, use UNISOLATEOBJECTS or **End Object Isolation** on the same button.

Notes

- It does not change the drawing: nothing is saved or undone and the visibility of layers is not touched.
- Hidden objects cannot be seen, selected or used with object snaps.
- The **ISOLATE** button of the status bar is lit while there are hidden objects.
- Known limit: hidden objects show again if the whole view is rebuilt (for example, when changing the color of a layer or switching to a layout).

See also

HIDEOBJECTS · UNISOLATEOBJECTS · LAYISO · Layers

LAYOUT —Layout

Creates, copies, deletes, renames and sets layouts.

- **Command:** `LAYOUT` · **Global name:** `_LAYOUT`
- Ribbon: View › Layout
- Menu: View

Creates, copies, deletes, renames and sets layouts, like AutoCAD's -LAYOUT. Each layout is stored as in AutoCAD: a LAYOUT object (with its page setup and tab order) and its `*Paper_Space…` block.

Usage

1. Type LAYOUT.

```
Enter layout option [Copy/Delete/New/Template/Rename/SAveas/Set/?] <Set>:
```

Options

- **New**: new layout with the default page setup and, with LAYOUTCREATEVIEWPORT set to 1, a viewport showing the model extents.
- **Copy**: copies a layout with its drawing and viewports, placed after it.
- **Delete**: deletes one or more (comma separated). Deleting all of them creates an empty one.
- **Template**: inserts layouts from a DWT, DWG or DXF (`*` for all).
- **Rename**, **SAveas** (to a `.dwt` template), **Set** (make it current) and **?**.

Notes

- The layout tab context menu does the same; drag the tabs to reorder, double-click to rename, **+** for a new one.

See also

MVIEW · PAGESETUP · PLOT

LAYOUTWIZARD —Layout Wizard

Creates a layout asking for name, paper, orientation and viewports.

- **Command:** `LAYOUTWIZARD` · **Global name:** `_LAYOUTWIZARD`

Creates a new layout asking for its name, the paper, the orientation and the viewports.

Usage

```
Enter the name of the new layout <Layout>:
Enter the paper size (? for the list) <…>:
Enter the orientation [Landscape/Portrait] <Landscape>:
Enter the viewports [None/One/2/3/4] <One>:
```

Notes

- The viewports (one, or 2, 3 or 4 spread out) fill the printable area with a 5 mm margin and show the model extents.
- There is no title block and it does not ask for the printer: use PAGESETUP to change it.

See also

LAYOUT · MVIEW · PAGESETUP

MATBROWSER —Materials

Assigns a library material to the selected objects or to the current layer.

- **Command:** `MATBROWSER` · **Global name:** `_MATBROWSER` · **Alias:** `MAT`, `RMAT`
- Ribbon: View › Render

Assigns a library material (or one created with a texture image) to the selected objects or to the current layer, like AutoCAD's materials browser.

Usage

1. Enter MATERIALES (or MAT).

```
Specify the material [?/Create] <…>:
```

2. Enter the material name (`?` lists the library; a material already created in the drawing also works). Enter uses **Global**. Or choose **Create** to make (or change) a textured material: > Name of the new material (or of an existing one, to change it): > Texture image (path, or Madera/Ladrillo/Baldosa/Césped/Agua) or Enter for none: > Size of one texture repeat, in drawing units <1>:

The image is the path of a PNG, JPEG or BMP, or the name of a built-in texture (Madera = wood, Ladrillo = brick, Baldosa = tile, Césped = grass, Agua = water). The size is what one repeat of the image measures in the drawing (e.g. `2` = the image covers 2 × 2 units). Enter at the image prompt creates a plain material.

3. Select the objects; Enter without selecting anything assigns it to the current layer.

Library

- Global, Rojo mate, Metal pulido, Cristal, Plástico azul: basic diffuse, specular, gloss and opacity.
- With a **built-in texture**: Madera (wood flooring), Ladrillo (brick), Baldosa (tile), Césped (grass) and Agua (water, glossy: it reflects the sky in RENDER). The images are generated the first time in the configuration folder (`%APPDATA%\lince\texturas`). Their repeat size is in metres and is converted with the drawing units

(INSUNITS; unitless, 1 = 1 m).

Notes

- The material is stored as a MATERIAL object in the ACAD_MATERIAL dictionary, with its named entry, as in AutoCAD (AUDIT 0). The texture goes in the diffuse map (file source, with its path and scale), which AutoCAD also reads.
- If the drawing is opened on another computer and the image path does not exist, an image with the same name is looked up in the textures folder.
- Assigned per object (entity `material_flags`/handle) or per layer (the layer's `material` field). The diffuse colour tints the face in the shaded styles; the texture shows in Render.
- No dialog box in this version: it is used from the command line.

See also

RENDER · SUN · VSCURRENT

MSPACE —Model Space

Enters model space through a viewport.

- **Command:** `MSPACE` · **Global name:** `_MSPACE` · **Alias:** `MS`

In a layout, enters model space through a layout viewport: from then on you draw, edit and zoom in the model seen inside the viewport, not on the sheet.

Usage

1. With a layout active, enter MSPACE (or MS) or click **Model Space** on the ribbon.
2. The first viewport of the layout is entered. To go back to the sheet, use PSPACE.

Notes

- To enter a particular viewport, double-click inside it.
- On the Model tab it does nothing: the command line says so.
- Inside a viewport, the LAYER shows the **VP Freeze** column to freeze layers in that viewport only.
- Entering or leaving cancels the running command and clears the selection.

See also

PSPACE · Layouts

MVIEW —Viewports

Creates layout viewports and turns them on, off or locks them.

- **Command:** `MVIEW` · **Global name:** `_MVIEW` · **Alias:** `MV`
- Ribbon: View › Layout
- Menu: View

Creates viewports in the current layout and turns them on, off or locks them. Each new viewport shows the model extents.

Usage

```
Specify corner of viewport or [ON/OFF/Fit/Lock/Object/Polygonal/2/3/4] <Fit>:
```

Options

- **Fit**, **2 / 3 / 4**, **Object** (non-rectangular viewport from a closed object), **Polygonal**, **ON / OFF** and **Lock**.
- **Shadeplot**: how the shading of the selected viewports is plotted: As displayed, Wireframe, Hidden (removes hidden lines when plotting), Visual styles or Rendered.
- **Restore**: creates the viewports with the layout of a model viewport configuration (`*Active` or a named one; `?` lists them), spread over two corners or the printable area.
- **Named**: a viewport showing a named view of the drawing.

Notes

- Inside a viewport, zoom and pan change that viewport's view and the sheet stays put; `ZOOM 1/100XP`, the status bar scale list and the PROPERTIES palette set its scale.

See also

VPCLIP · VPMAX

PAN —Pan

Moves the view without changing the scale.

- **Command:** `PAN` · **Global name:** `_PAN` · **Alias:** `P`
- Ribbon: View › Navigate
- Menu: View

Will move the view of the drawing without changing the scale, by dragging with the mouse. As a command it is not available yet in this version.

Usage

Meanwhile, you can pan with the mouse at any time, also while another command is running:

1. Press the middle mouse button (the wheel) and drag without releasing it: the view moves with the cursor.

Notes

- The wheel zooms in and out at the cursor, and a middle double-click zooms to the whole drawing (ZOOMEXTENTS).

See also

ZOOM · ZOOMEXTENTS · User interface

PLAN —Plan View

Displays an orthographic view of the XY plane of a UCS.

- **Command:** `PLAN` · **Global name:** `_PLAN`
- Ribbon: View › Coordinates
- Menu: View

Shows the plan view of a UCS: it rotates the view so that the X axis of that UCS is horizontal on screen and does a zoom extents. With a UCS aligned with a rotated building, the building is shown straight.

Usage

1. Enter PLAN or click the button.
2. Choose which UCS; Enter, the current one.

```
Enter an option [Current ucs/Ucs/World] <Current>:
```

Options

- **Current ucs** (C): the plan of the current UCS.
- **Ucs** (U): that of a named UCS, whose name you type next.

```
Enter name of UCS to restore or [?]:
```

- **World** (W): that of the world system: the view is no longer rotated.

Notes

- PLAN does not change the current UCS or move any object: it only rotates the view.
- The rotation is saved in the drawing as in AutoCAD (VIEWTWIST of the current viewport), and AutoCAD opens the drawing with the view rotated the same way.
- With the view rotated, the selection window takes the world box of its two corners.

See also

UCS · UCSMAN · ZOOM · Coordinates and data entry

LUZPUNTUAL —Luz puntual

Crea una luz puntual que irradia en todas direcciones.

- **Comando:** `LUZPUNTUAL` · **Nombre global:** `_POINTLIGHT` · **Alias:** `PUNTOLIBRE`
- Cinta: Vista › Render

Crea una luz puntual que irradia en todas direcciones, como POINTLIGHT de AutoCAD.

Uso

1. Escriba LUZPUNTUAL.

```
Indique la posición de la luz:
```

2. Indique la posición de la luz.

Notas

- Se guarda como entidad LIGHT del DWG (AUDIT 0). El Render la usa para iluminar con sombras.

Véase también

SPOTLIGHT · DISTANTLIGHT · SUN · RENDER

PSPACE —Paper Space

Returns to the paper space of the layout.

- **Command:** PSPACE · **Global name:** _PSPACE · **Alias:** PS

Leaves the layout viewport you are working in and returns to the paper space of the layout: you draw and edit on

the sheet again.

Usage

1. Inside a layout viewport, enter PSPACE (or PS) or click **Paper Space** on the ribbon.

Notes

- You can also leave by double-clicking outside the viewport.
- On the Model tab it does nothing: the command line says so.
- Leaving cancels the running command and clears the selection.

See also

MSPACE · Layouts

REDRAW — Redraw

Repaints the current view.

- **Command:** `REDRAW` · **Global name:** `_REDRAW` · **Alias:** `R`

Repaints the current view without recalculating the geometry.

Usage

Enter REDRAW. There are no further prompts.

Notes

- LinceCAD paints the whole scene on every frame: REDRAW just asks for the next one, and exists so that scripts and AutoCAD habits keep working.
- To redo the tessellation of curves and texts use REGEN.

See also

REDRAWALL · REGEN · REGENALL

REDRAWALL — Redraw All

Repaints all viewports.

- **Command:** `REDRAWALL` · **Global name:** `_REDRAWALL` · **Alias:** `RA`

Repaints all viewports, like AutoCAD REDRAWALL.

Usage

Enter REDRAWALL. There are no further prompts.

Notes

- The model and layout viewports are all painted on every frame, so it is the same as REDRAW.

See also

REDRAW · REGENALL

REGEN — Regen

Recalculates the geometry on screen.

- **Command:** `REGEN` · **Global name:** `_REGEN` · **Alias:** `RE`
- Ribbon: View › Navigate
- Menu: View

Recomputes the geometry displayed on screen (the segments that curves and texts are divided into) for the whole drawing, or for the space being shown (model or layout).

Usage

Enter REGEN or choose it in the View menu or on the ribbon (View › Navigate). The command line says "Regenerating drawing.".

Notes

- LinceCAD regenerates only what is needed, automatically: when objects, layers, styles or the linetype scale change, the screen is updated without being asked. DRAWORDER regenerates when it finishes so that the new order shows.
- Use it after changing variables that AutoCAD only applies on regeneration, or if something looks out of date after undoing a DRAWORDER.

See also

ZOOM · ZOOMEXTENTS · DRAWORDER · User interface

REGENALL — Regenerate All

Regenerates the drawing and all viewports.

- **Command:** `REGENALL` · **Global name:** `_REGENALL` · **Alias:** `REA`

Regenerates the drawing and all its viewports, like AutoCAD REGENALL.

Usage

Enter REGENALL. The command line says "Regenerating drawing.".

Notes

- It does the same as REGEN: it redoes the tessellation of the space being shown and of the layout viewports.

See also

REGEN · REDRAWALL

RENDER — Render

Shades the current 3D view into a PNG image with lights, shadows and ambient occlusion.

- **Command:** `RENDER` · **Global name:** `_RENDER` · **Alias:** `RR`
- Ribbon: View › Render

Shades the current 3D view into a photorealistic PNG image, with lighting, shadows, ambient occlusion, reflections and textures, like AutoCAD's RENDER.

Usage

1. Enter RENDER (or RR).

```
Specify the PNG file <…> or [Size/Background/Ground] (…×… image):
```

2. Specify the output PNG file (Enter uses `render.png`), or first choose an option: - **Size**: image width and height in pixels, `1920,1080` (also `1920x1080`). Default 1920 × 1080. > Image size in pixels (width,height) <…,…>: **Background**: **Sky** (gradient from horizon to zenith, the default), **Light** (very light grey) or **Dark**. > Background [Sky/Light/Dark] <current>: - **Ground**: an infinite ground below the scene (at its lowest Z) that receives shadows and fades into the horizon in the distance. Off by default. > Ground below the scene, with shadows? [Yes/No] <…>:

The options are remembered during the session.

Notes

- It renders **the current 3D view**: the same direction, target and framing as the window (zoom and pan included) and, if the view is in **perspective** (PERSPECTIVE = 1), in perspective with the same lens. If the image has a different aspect ratio than the window, at least everything visible in the window is shown.
- It lights with the defined lights (POINTLIGHT, SPOTLIGHT, DISTANTLIGHT) and the SUN; with none, a default light from the viewer.
- CPU ray tracing (BVH accelerated) with 4 rays per pixel (antialiasing): shadows, ambient occlusion, specular highlights and mirror reflections on glossy materials (water and glass reflect the sky). Everything is computed per pixel, so shadows and textures also show inside large faces.
- The material (MATBROWSER) tints each face and, if it has a **diffuse map** (image), applies it as a texture with a planar box projection (by face orientation) and the material's scale.
- There is no transparency or refraction: glass looks like a tinted mirror.
- It sets the Realistic visual style in the window (the GPU 3D viewer shows the shaded scene live, without shadows or reflections: those are only in the image).

See also

RENDERWIN · MATBROWSER · SUN · VSCURRENT

RENDERV —Ventana de render

Como RENDER: sombrea la vista 3D actual en una imagen PNG.

- **Comando:** `RENDERV` · **Nombre global:** `_RENDERWIN` · **Alias:** `VENTANARENDER`

Como RENDER: sombrea la vista 3D actual en una imagen PNG. Equivale a RENDERWIN de AutoCAD.

Uso

1. Escriba RENDERV.

```
Indique el archivo PNG <…> o [Tamaño/Fondo/Suelo] (imagen de …×…):
```

2. Indique el archivo PNG de salida.

Véase también

RENDER · MATBROWSER · SUN

SHOWPALETTES —Show Palettes

Shows the palettes hidden with HIDEPALETTES again.

- **Command:** `SHOWPALETTES` · **Global name:** `_SHOWPALETTES` · **Alias:** `PON`

Shows again the palettes hidden with HIDEPALETTES, like AutoCAD SHOWPALETTES.

Usage

Enter SHOWPALETTES. The palettes return to their place.

Notes

- It also brings back those hidden by CLEANSCREENON, but not the ribbon: use CLEANSCREENOFF.

See also

HIDEPALETTES · CLEANSCREENOFF

FOCO —Foco

Crea un foco (cono de luz) desde un punto hacia un objetivo.

- **Comando:** `FOCO` · **Nombre global:** `_SPOTLIGHT`
- Cinta: Vista › Render

Crea un foco (cono de luz) desde un punto hacia un objetivo, como SPOTLIGHT de AutoCAD.

Uso

1. Escriba FOCO.

```
Indique el origen del foco:
```

2. Indique el origen y luego el objetivo del foco.

Notas

- El haz (hotspot) y la caída (falloff) toman 45° y 50° por defecto. Se guarda como entidad LIGHT (AUDIT 0).

Véase también

POINTLIGHT · DISTANTLIGHT · SUN · RENDER

SOL —Sol

Enciende o apaga el sol, que usa la geolocalización del dibujo.

- **Comando:** `SOL` · **Nombre global:** `_SUN`
- Cinta: Vista › Render

Enciende o apaga el sol, cuya posición se calcula con la geolocalización del dibujo, como SUN de AutoCAD.

Uso

1. Escriba SOL.

```
Indique opción del sol [Activar/Desactivar/Geolocalizar] <Activar>:
```

2. **Activar** (Intro) enciende el sol, **Desactivar** lo apaga y **Geolocalizar** muestra la latitud y la longitud actuales.

Notas

- La dirección del sol se obtiene de la latitud y la longitud del dibujo (geolocalización) y la fecha, con el algoritmo PSA.
- El Render usa el sol como luz distante.

Véase también

RENDER · DISTANTLIGHT · MATBROWSER

TEXTSCR —Text Window

Shows the text window with the command line history.

- **Command:** `TEXTSCR` · **Global name:** `_TEXTSCR`

Shows the text window with the command line history, like AutoCAD TEXTSCR.

Usage

Enter TEXTSCR (or press F2). The text window opens, with everything typed and answered on the command line.

Notes

- The window can be moved and resized, and is closed with its cross or with F2.
- AutoCAD also has GRAPHSCR to go back to the graphics screen; here closing the window is enough.

See also

Interfaz

UNISOLATEOBJECTS —End Object Isolation

Shows the isolated or hidden objects again.

- **Command:** `UNISOLATEOBJECTS` · **Global name:** `_UNISOLATEOBJECTS` · **Alias:** `UNHIDE`, `UNISOLATE`
- Ribbon: View › Isolate
- Menu: View

Ends object isolation: shows again every object hidden by ISOLATEOBJECTS and HIDEOBJECTS.

Usage

1. Enter UNISOLATEOBJECTS or choose **End Object Isolation** on the **ISOLATE** button of the status bar.
2. It asks nothing: every hidden object shows again and the status bar button goes off.

Notes

- The button option is only available while there are hidden objects.
- It does not change the drawing or the visibility of layers; for layers isolated with LAYISO use LAYUNISO.

See also

ISOLATEOBJECTS · HIDEOBJECTS · LAYUNISO

VISTA —Vistas con nombre

Guarda y restituye vistas con nombre.

- **Comando:** `VISTA` · **Nombre global:** `_VIEW` · **Alias:** `V`, `VI`, `DDVIEW`
- Cinta: Vista › Vistas 3D

Abre el administrador de vistas con nombre: guardar la vista actual, restituir o suprimir una guardada, y las vistas predefinidas, como VISTA (_VIEW) de AutoCAD. Las vistas se guardan en la tabla VIEW del dibujo.

Uso

1. Escriba VISTA (o V).
2. En el cuadro, pulse una vista predefinida, elija una vista con nombre y **Restituir**, escriba un nombre y **Guardar actual**, o suprima la elegida.

Notas

- Al guardar se recuerdan la dirección, el objetivo, el encuadre (centro y altura), el giro, el estilo visual y el SCP. Al restituir se recupera el encuadre exacto (no un zoom extensión).
- AutoCAD abre el dibujo con las mismas vistas con nombre.
- La versión de la línea de comandos es -VIEW.

Véase también

-VIEW · VPOINT · DDVPOINT · 3DORBIT

VPCLIP —Clip Viewport

Clips a viewport with a closed object or a polygon.

- **Command:** `VPCLIP` · **Global name:** `_VPCLIP`
- Ribbon: View › Layout
- Menu: View

Clips a viewport with a closed object or a polygon, or removes the clipping.

Usage

```
Select viewport to clip:
Select clipping object or [Polygonal/Delete] <Polygonal>:
```

See also

MVIEW

VPMAX —Maximize Viewport

Expands the current viewport to the whole drawing area to edit the model.

- **Command:** `VPMAX` · **Global name:** `_VPMAX`
- Ribbon: View › Layout
- Menu: View

Expands the current viewport to the whole drawing area, with a red border. VPMIN returns to the layout.

VPMIN —Minimize Viewport

Returns to the layout from a maximized viewport.

- **Command:** `VPMIN` · **Global name:** `_VPMIN`
- Ribbon: View › Layout
- Menu: View

Returns to the layout from a viewport maximized with VPMAX.

VPOINT —Viewpoint

Sets the viewing direction with a vector.

- **Command:** `VPOINT` · **Global name:** `_VPOINT` · **Alias:** `-VP`, `VP`
- Ribbon: View › 3D Views

Sets the viewing direction with a vector from the target to the viewer, like AutoCAD VPOINT, and zooms to extents.

Usage

1. Enter VPOINT (or -VP).

```
Specify a view point (x,y,z):
```

2. Type the vector: `0,0,1` is the plan view, `0,-1,0` the front view and `-1,-1,1` the SW isometric.

Notes

- The vector is saved as VIEWDIR; the target moves to the origin.

See also

-VIEW · 3DORBIT · PLAN

VPORTS —Viewports

Dialog to split the model into viewports or create viewports in a layout.

- **Command:** `VPORTS` · **Global name:** `_VPORTS` · **Alias:** `VIEWPORTS`

Splits the drawing area of the model into viewports, or creates viewports in a layout, choosing a standard configuration.

Usage

1. Type VPORTS: the dialog opens.
2. Choose a configuration (Single, two, three or four viewports) and click OK.

Notes

- On the Model tab it splits the drawing area, like -VPORTS; Single returns to one viewport.
- In a layout it creates the viewports spread over the printable area, like the Fit option of MVIEW.
- Without the dialog (scripts, FILEDIA=0) use -VPORTS or MVIEW.

See also

MVIEW · -VPORTS · VPCLIP · Layouts

VPSYNC —Synchronize Viewports

Matches the view and scale of several viewports to those of one.

- **Command:** `VPSYNC` · **Global name:** `_VPSYNC`

Matches the view and scale of several viewports to those of a reference viewport (AutoCAD Express tool).

Usage

1. In a layout, type VPSYNC.

```
Select the reference viewport:
```

2. Select the leading viewport and then the ones to synchronize; Enter ends.

```
Select the viewports to synchronize:
```

Notes

- The synchronized viewports take the viewpoint, direction, twist and scale of the reference viewport (each keeps its own size on the sheet).
- Viewports with a locked display (MVIEW Lock) do not change.

See also

MVIEW · VPCLIP · CHSPACE · Layouts

VSCURRENT —Visual Styles

Changes the visual style of the current viewport.

- **Command:** `VSCURRENT` · **Global name:** `_VSCURRENT` · **Alias:** `VS`
- Ribbon: View › 3D Views
- Menu: View

Changes the visual style of the current viewport, like AutoCAD VSCURRENT.

Usage

1. Enter VSCURRENT (or VS), click the button on the View tab or choose the style in its list.

```
Enter an option [2dwireframe/Wireframe/Hidden/Realistic/Shaded/shaded with Edges/X-ray] <cur
rent>:
```

2. Choose the style; Enter keeps the current one.

Options

- **2D Wireframe** (2): the usual drawing; in plan view it is drawn exactly as in 2D.
- **Wireframe** (W): all lines and edges, hiding nothing.
- **Hidden** (H): faces hide what is behind them and take the background color, with their edges.
- **Realistic** (R): lit faces with highlights, without the face edges.
- **Shaded** (S): smoothly lit faces, without the face edges.
- **Shaded with edges** (E): like shaded, with the edges.
- **X-Ray** (X): translucent faces and all edges.
- In scripts the AutoCAD abbreviations with _ also work (`_2`, `_W`, `_H`, `_R`, `_S`, `_E`, `_X`).

Notes

- The style is saved in the viewport with its VISUALSTYLE object (created with the AutoCAD name if the drawing does not have it) and AutoCAD opens the drawing with the same style.
- Lines, arcs, texts and hatches show in every style; 3D faces, meshes and the walls of objects with thickness are filled in the styles with faces. The color of the assigned material (of the object or its layer) tints the faces; the light is fixed, from the viewer (the drawing lights are used by RENDER).
- Without a GPU (LINCE_GPU=0) 3D views are drawn as wireframe.

See also

3DORBIT · -VIEW · PLAN

ZOOM —Zoom

Enlarges or reduces the view.

- **Command:** `ZOOM` · **Global name:** `_ZOOM` · **Alias:** `Z`
- Ribbon: View › Navigate
- Menu: View

Enlarges or reduces the view of the drawing without changing the real size of the objects. The mouse wheel, which zooms in and out around the cursor, is usually more convenient; the command is for an exact window, a scale factor or going back to the previous view.

Usage

1. Enter ZOOM (or Z) or click **Zoom** on the ribbon.

```
Specify corner of window, enter a scale factor (nX) or [All/Extents/Previous/Scale/Window] <
real time>:
```

2. Specify two opposite corners to zoom into that area, type a factor or choose an option.

```
Specify opposite corner:
```

3. The command ends after changing the view. Enter on its own ends it without changing the view.

Options

- **All** (A) and **Extents** (E): show the whole drawing, like ZOOMEXTENTS.
- **Previous** (P): returns to the view before the last ZOOM; it can be repeated.
- **Scale** (S): asks for the factor. `2` or `2X` zooms in to twice the size and `0.5` zooms out to half, always relative to the current view.

```
Enter a scale factor (nX):
```

- **Window** (W): asks for the two corners of the area to show.

```
Specify first corner:
```

Notes

- Mouse wheel: zooms in or out at the cursor. Dragging with the middle button: pan. Middle double-click: extents.
- **Previous** only remembers views changed with ZOOM, not those changed with the wheel or by panning.
- **All** does not take drawing limits into account: it does the same as **Extents**.

See also

ZOOMEXTENTS · PAN · User interface

ZOOMEXTENTS —Extents

Displays the whole drawing on screen.

- **Command:** `ZOOMEXTENTS` · **Global name:** `_ZOOMEXTENTS`
- Ribbon: View › Navigate
- Menu: View
- Toolbar: Standard

Zooms to the whole drawing: enlarges or reduces the view so that all objects fit on the screen. It is the same as the **Extents** option of ZOOM.

Usage

1. Enter ZOOMEXTENTS or click **Extents** on the ribbon. There are no prompts.

Notes

- It can also be done by double-clicking the middle mouse button (the wheel), and from the shortcut menu of the drawing area when no command is running.
- Use it to get the drawing back when the view has wandered away from the objects.

See also

ZOOM · PAN · User interface

3D modeling

BOX —Box

Creates a solid box.

- **Command:** `BOX` · **Global name:** `_BOX`

Creates a solid box (cuboid) as an ACIS 3DSOLID, just like the ones AutoCAD writes.

Usage

1. Enter BOX (CAJA) or click **Box**.

```
First base corner:
```

2. Pick the first base corner and the opposite corner (they set the length and width).

```
Height:
```

3. Type the height (or pick it as a distance). The box is created.

Notes

- The base is at the Z of the first point if you type `x,y,z` (e.g. `0,0,3`) and otherwise at the current elevation (ELEV); the box grows towards +Z. An opposite corner with another Z (`10,5,6`) gives the height without asking for it. The same holds for the base point of WEDGE, CYLINDER, CONE, SPHERE, TORUS and PYRAMID.
- The solid is an exact ACIS 3DSOLID: AutoCAD audits it with no errors and MASSPROP reports the correct volume.

See also

WEDGE · CYLINDER · SPHERE · MESH

CONE —Cone

Creates a solid cone.

- **Command:** `CONE` · **Global name:** `_CONE`

Creates a solid right cone as an ACIS 3DSOLID.

Usage

1. Enter CONE (CONO) or click **Cone**.

```
Base center:
```

2. Pick the base center.

```
Radius:
```

3. Type the base radius.

```
Height:
```

4. Type the height (the apex is over the base center).

Notes

- The side surface is an exact cone; MASSPROP reports the correct volume.

See also

CYLINDER · SPHERE · PYRAMID

CONVTOMESH —Convert to mesh

Converts solids into meshes.

- **Command:** `CONVTOMESH` · **Global name:** `_CONVTOMESH` · **Alias:** `SMOOTH`

Converts 3D solids (or regions) into meshes.

Usage

1. Enter CONVTOMESH (CONVAMALLA).

```
Select objects:
```

2. Select the solids and press Enter. Each one is tessellated and replaced by a mesh.

See also

CONVTOSOLID · MESH

CONVTOSOLID —Convert to solid

Converts meshes into faceted solids.

- **Command:** `CONVTOSOLID` · **Global name:** `_CONVTOSOLID`

Converts meshes into faceted (flat-faced) 3D solids.

Usage

1. Enter CONVTOSOLID (CONVASOLIDO).

```
Select objects:
```

2. Select the meshes and press Enter. Each closed mesh becomes a faceted 3DSOLID.

Notes

- The mesh must be a closed surface; otherwise it is not converted.

See also

CONVTOMESH · MESH

CYLINDER —Cylinder

Creates a solid cylinder.

- **Command:** `CYLINDER` · **Global name:** `_CYLINDER` · **Alias:** `CYL`

Creates a solid right cylinder as an ACIS 3DSOLID.

Usage

1. Enter CYLINDER (CILINDRO) or click **Cylinder**.

```
Base center:
```

2. Pick the base center.

```
Radius:
```

3. Type the radius.

```
Height:
```

4. Type the height.

See also

CONE · SPHERE · BOX

MESH —Mesh

Creates a mesh primitive (box, cone, sphere…).

- **Command:** `MESH` · **Global name:** `_MESH`

Creates a subdivision (SubD) mesh primitive: box, cone, cylinder, wedge, sphere, torus or pyramid.

Usage

1. Enter MESH (MALLA) or click **Mesh**.

```
Mesh shape [Box/COne/CYlinder/Wedge/Sphere/Torus/Pyramid]:
```

2. Choose the shape and enter its data just like the matching solid primitive.

Notes

- The mesh is obtained by tessellating the solid primitive; the 3D viewer draws it and it can be smoothed.
- Raise its smoothness with MESHSMOOTHMORE and, with MESHREFINE, make the smoothed shape the mesh itself.

See also

BOX · CONVTOMESH · CONVTOSOLID · MESHSMOOTHMORE

MESHREFINE —Refine mesh

Refines the selected meshes.

- **Command:** `MESHREFINE` · **Global name:** `_MESHREFINE` · **Alias:** `REFINE`

Refines the selected meshes, as AutoCAD does: the smoothed shape becomes the mesh, with more faces, and its smoothness level goes back to 0. Smooth them first with MESHSMOOTHMORE; meshes that are not smoothed do not change.

Usage

1. Enter MESHREFINE (REFINARMALLA).

```
Select objects:
```

2. Select the meshes and press Enter.

See also

MESHSMOOTHMORE · MESH

MESHSMOOTHMORE —Smooth more

Increases mesh smoothness.

- **Command:** `MESHSMOOTHMORE` · **Global name:** `_MESHSMOOTHMORE` · **Alias:** `MORE`

Raises the smoothness (subdivision) of the selected meshes by one level.

Usage

1. Enter MESHSMOOTHMORE (SUAVIZARMAS).

```
Select objects:
```

2. Select the meshes and press Enter. Their smoothness level goes up by one (up to a maximum of 4).

See also

MESHREFINE · MESH

POLYSOLID —Polysolid

Creates a wall-shaped solid along a path.

- **Command:** `POLYSOLID` · **Global name:** `_POLYSOLID` · **Alias:** `PSOLID`

Creates a wall-shaped solid (rectangular section) along a path, as an ACIS 3DSOLID.

Usage

1. Enter POLYSOLID (POLISÓLIDO) or click **Polysolid**.

```
Start point or [Height/Width]:
```

2. Pick the path points. Use **Height** and **Width** to change the section before you start.

```
Next point or [Height/Width] (Enter to finish):
```

3. Press Enter to finish: the extruded solid is created.

Notes

- The section is constant (width x height) and the path is followed with miter joints.
- Open path only; at very sharp corners the band self-intersects and no solid is created.

See also

BOX · REGION

PYRAMID — Pyramid

Creates a solid pyramid with a square base.

- **Command:** PYRAMID · **Global name:** _PYRAMID · **Alias:** PYR

Creates a solid pyramid with a square base as an ACIS 3DSOLID.

Usage

1. Enter PYRAMID (PIRÁMIDE) or click **Pyramid**.

```
Base center:
```

2. Pick the base center.

```
Base side:
```

3. Type the side of the square base.

```
Height:
```

4. Type the height (the apex is over the center).

Notes

- The base is a square aligned with the UCS X and Y axes. This is a simplification of AutoCAD, which allows polygons with several sides and inscribed or circumscribed radii.

See also

CONE · BOX · WEDGE

SPHERE — Sphere

Creates a solid sphere.

- **Command:** SPHERE · **Global name:** _SPHERE

Creates a solid sphere as an ACIS 3DSOLID.

Usage

1. Enter SPHERE (ESFERA) or click **Sphere**.

```
Center:
```

2. Pick the center.

```
Radius:
```

3. Type the radius.

Notes

- The sphere is an exact surface (a single ACIS face); AutoCAD audits it with no errors.

See also

CYLINDER · CONE · TORUS

TORUS —Torus

Creates a solid torus.

- **Command:** `TORUS` · **Global name:** `_TORUS` · **Alias:** `TOR`

Creates a solid torus as an ACIS 3DSOLID, with its axis along Z.

Usage

1. Enter TORUS (TOROIDE) or click **Torus**.

```
Center:
```

2. Pick the center.

```
Torus radius:
```

3. Type the torus radius (center to tube center).

```
Tube radius:
```

4. Type the tube radius.

Notes

- The surface is an exact torus (a single ACIS face).

See also

SPHERE · CYLINDER

VIEWBASE —Base view

Creates the drawing view of 3D solids and its projected views.

- **Command:** `VIEWBASE` · **Global name:** `_VIEWBASE`

Creates in the current space (normally a layout) the **drawing view** of the 3D solids of the model and then its **projected views**, like AutoCAD's VIEWBASE.

Usage

1. Type VIEWBASE. Select the solids and press Enter (Enter with no selection takes all 3D solids of the model).

```
Base view orientation [Top/Bottom/Front/Back/Left/Right/SW/SE/NE/NW/Current] <Front>:
```

2. Give the scale (`2`, `1:2`) and the style: **Visible** (visible lines only), **Hidden** (visible and hidden lines; the hidden ones go on the `MD_Hidden` layer with a dashed linetype) or **Shaded**.
3. Specify where the base view goes.
4. Each point you specify afterwards creates a **projected view** on the matching side of the base (third-angle projection): above, the top view; to the right, the right view, and so on. Enter finishes.

Result

- Each view is a reference to an anonymous block with the lines on the `MD_Visible`, `MD_Hidden`, `MD_Hatch` and `MD_Shaded` layers.
- Its XDATA (`LINCE_VISTA3D`) keeps the solids, the direction, the scale and the parent view, to derive sections and details with VIEWSECTION and VIEWDETAIL.
- AutoCAD opens the drawing with the view as an ordinary block (AUDIT 0).

Notes

- Views are **not updated** when the solid changes: create them again.
- No silhouette edges on curved surfaces and no tangent lines; the shading is a simple gray one.
- See also FLATSHOT.

VIEWDETAIL —Detail view

Creates an enlarged view of a circular or rectangular area of a drawing view.

- **Command:** `VIEWDETAIL` · **Global name:** `_VIEWDETAIL`

Creates a **detail view**: a circular or rectangular area of a drawing view, enlarged to its own scale, like AutoCAD's VIEWDETAIL.

Usage

1. Type VERDETALLE and pick the parent view.

```
Boundary shape [Circular/Rectangular] <Circular>:
```

2. Specify the center and a point on the edge (circle) or the two corners (rectangle).
3. Give the detail scale (twice the parent's by default) and place the view.

Result

- The detail clips the parent's lines to the area (including a section's hatching) and enlarges them.
- The boundary is drawn in the parent and in the detail, with the letter and the scale title.

VIEWSECTION —Section view

Creates a section view of a drawing view along a cutting line.

- **Command:** `VIEWSECTION` · **Global name:** `_VIEWSECTION`

Creates a **section view** of a drawing view, like AutoCAD's VIEWSECTION.

Usage

1. Type VERSECCION and pick the parent view (a view created with VIEWBASE).
2. Specify the two points of the **cutting line**.

```
Specify the location of the section view (the side sets the direction of sight):
```

3. The side of the line where you place the view is the **direction of sight**: what lies behind the cutting plane is kept.

Result

- The view is aligned with the parent, with the cut area **hatched** (45-degree lines on `MD_Hatch`).
- In the parent the cutting line is drawn with its arrows and letter (A, B...), and under the section its title and scale.

Notes

- The parent must be a base, projected or detail view.
- The hatching is made of lines, not an associative HATCH object.

WEDGE — Wedge

Creates a solid wedge.

- **Command:** `WEDGE` · **Global name:** `_WEDGE` · **Alias:** `WE`

Creates a solid wedge (a box cut along a diagonal) as an ACIS 3DSOLID.

Usage

1. Enter WEDGE (CUÑA) or click **Wedge**.

```
First base corner:
```

2. Pick two opposite base corners (length and width).

```
Height:
```

3. Type the height. The tall edge is over the first corner and the slope drops towards the opposite side.

Notes

- The base sits on Z = 0 of the current UCS.

See also

BOX · PYRAMID · MESH

Topography: terrain model and contours

CONTOUR —Contours

Generates the contour lines of a terrain or surface.

- **Command:** `CONTOUR` · **Global name:** `_CONTOUR` · **Alias:** `CN`, `ISOLINES`
- Ribbon: Topography › Terrain

Generates the **contour lines** of a terrain or surface at a given interval, with index contours every N on their own layer, optional **smoothing** and **elevation labels aligned** with the contour, which is broken where the text goes.

Usage

1. Enter CONTOUR (CURVASNIVEL) or click **Contours**.

```
Select the terrain points (points, blocks or a surface):
```

2. Select the terrain points or a surface (SURFACE) and press Enter. The **Contours** dialog opens; you can also go on at the command line.

```
Contour interval <…>:
```

3. Give the contour interval (elevation step between minor contours).

```
Index contour every (0 for none) <…>:
```

4. Give how often a contour is an index contour (0 for none).

```
Smoothing [None/Polyline/Spline] <…>:
```

5. Choose the smoothing: **None** (contours follow the triangles), **Polyline** (polyline with more vertices) or **Spline** (fit spline).

```
Distance between index contour labels (0 no labels) <…>:
```

6. Give the distance between labels on index contours (0 no labels). The contours are drawn.

Notes

- Contours are polylines (or splines) at their elevation: minor ones on layer `CN_FINA`, index ones on `CN_MAESTRA`, with their labels.
- The algorithm interpolates where each level crosses the triangle edges and chains the segments; contours around a peak close on themselves.
- **Breaklines**: if the surface has breaklines, the contours fold along them, also when smoothed.
- **Smoothing never crosses neighbouring contours**: instead of smoothing each contour on its own, the surface is refined (each triangle split into 16 with elevations that follow the slopes) and all contours come from that one surface. Next to breaklines the surface stays planar.
- Labels are spread evenly along the contour and always read from left to right. The contour is broken (not masked), so it looks the same in any program. To label by hand, CONTOURLABEL.
- The dialog also sets the label text height.

See also

SURFACE · CONTOURLABEL · SPOTELEVATION

CONTOURLABEL —Label contour

Labels the elevation of a contour at the given point.

- **Command:** `CONTOURLABEL` · **Global name:** `_CONTOURLABEL` · **Alias:** `CLB`
- Ribbon: Topography › Terrain

Places an **elevation label** on a contour by hand, aligned with it, and **breaks the contour** where the text goes.

Usage

1. Enter CONTOURLABEL (ROTULARCURVA) or click **Label contour**.

```
Pick a point on the contour:
```

2. Pick a point on or next to the contour. The nearest contour is labelled. Repeat with other contours; Enter ends.

Notes

- It works with the contours from CONTOUR and with any polyline or fit spline at its elevation: the label is that elevation.
- The text height is that of the labels of the last CONTOUR (otherwise, the current text height).
- The contour is split into two pieces that keep its layer, colour and linetype.

See also

CONTOUR · SPOTELEVATION

COORDTABLE —Coordinate table

Creates a table with the number, X, Y, Z and code of the selected points or vertices.

- **Command:** `COORDTABLE` · **Global name:** `_COORDTABLE`
- Ribbon: Topography › Terrain data

Creates a **coordinate table**: a table (AutoCAD TABLE object) with the number, X, Y, Z and, if any, the code of the selected points, blocks or polyline vertices. It is meant for setting-out and control-point drawings. It can label the numbers in the drawing and write the same rows to a CSV file.

Usage

1. Enter COORDTABLE or click **Coordinate table** in the **Terrain data** panel of the Topography tab.

```
Select objects:
```

2. Select points, blocks with attributes or polylines (2D, 3D or lightweight) and press Enter.
3. In the **Coordinate table** dialog choose the format and, optionally, the CSV file, and click **OK**.
4. Specify the upper-left corner of the table.

```
Specify insertion point:
```

What each row shows

- **Point object**: a running number from the **First number** and its coordinates.
- **Block with attributes**: the number, code and elevation from its attributes if it has them (tags such as NUMBER, POINT or ID; CODE or DESC; Z or ELEV). Without a number it is numbered like a point; the position is the insertion point.
- **Polyline**: one numbered row per vertex; the elevation is the polyline elevation (or the vertex Z in a 3D

polyline).

The **Code** column only appears when some row has a code.

Options

- **Decimals**: 0 to 8. The table uses the decimal separator of the language; the CSV always uses a point.
- **Order**: Selection (the order in which objects were selected), Number, X or Y.
- **First number**: the first number for the points and vertices that do not carry one.
- **Label the numbers in the drawing**: writes each number next to its point or vertex, on layer COORD_NUM with the current text height. Blocks already show their own number and are not labelled.
- **Export to CSV**: path of the CSV file (empty: none).

Command line

While the dialog is open the same options can be typed; Enter accepts.

```
Table options [Decimals/Order/Start/Label/Csv] <Accept>:
```

Decimals (D), Order (O), Start (S), Label (L, toggles yes and no) and Csv (C, file path).

Notes

- The result is an ordinary table: edit it with TABLEDIT and export it with TABLEEXPORT. It does not update when the points move: create it again.
- AutoCAD audits the table with no errors in DWG 2018 and 2000.

See also

TABLE · EXPORTPOINTS · IMPORTPOINTS · LANDXMLOUT

EDITSURFACE —Edit surface

Edits a terrain model: edges, points, triangles and constraints.

- **Command:** `EDITSURFACE` · **Global name:** `_EDITSURFACE` · **Alias:** `EDS`
- Ribbon: Topography › Terrain

Edits a **digital terrain model** created with SURFACE: swap edges, add or delete points, delete triangles and add breaklines, boundaries or holes. After each change the surface is rebuilt and stored in the same mesh.

Usage

1. Enter EDITSURFACE (EDITARMDT) or click **Edit surface**.

```
Select surface:
```

2. Select the surface.

```
Option [Swap edge/Add point/Delete point/Erase triangle/Breakline/bOundary/Hole/Length/eXit]
 <eXit>:
```

3. Choose what to do. After each change you return to the options (Enter, eXit): - **Swap edge**: pick a point next to the edge; it becomes the other diagonal of its two triangles. - **Add point**: pick the point and its elevation (or type x,y,z); the default is the interpolated elevation. - **Delete point**: the vertex nearest to the point given. - **Erase triangle** (E or T): deletes the triangle that contains the point (leaving a hole). - **Breakline**, **bOundary**, **Hole**: select polylines, as in SURFACE. - **Length**: the maximum edge length on the border.

Notes

- The surface is rebuilt from its data (points and constraints): the constrained Delaunay triangulation is unique, so the result is the same as retriangulating only the area touched.
- A swapped edge is stored as a forced edge: it is kept in later edits and can be swapped back. Breakline, boundary and hole edges cannot be swapped.
- Deleted triangles are remembered by one of their points and stay deleted after other edits.
- A polyface mesh not created with SURFACE takes its vertices as points and its border as boundary and holes (no boundary if the border is convex); editing it triangulates it again by Delaunay.
- UNDO undoes it in one step.

See also

SURFACE · CONTOUR · SPOTELEVATION

EXPORTPOINTS —Export points

Writes the selected points to CSV, stake-out ASCII or Leica GSI.

- **Command:** `EXPORTPOINTS` · **Global name:** `_EXPORTPOINTS`
- Ribbon: Topography › Points
- Menu: Survey

Writes the selected **survey points** and point objects to a file for **stake-out** with the total station or for another program: CSV, ASCII with the column order of your choice, or Leica GSI.

Usage

1. Enter EXPORTPOINTS or click **Export points** on the Survey tab.

```
Select objects:
```

2. Select the points and press Enter.

```
Destination file (.csv, .txt or .gsi):
```

3. Choose the file and its type: CSV (.csv), stake-out ASCII (.txt) or Leica GSI (.gsi).

```
Column order <…>:
```

4. For ASCII, give the column order (P number, N northing, E easting, Z elevation, D code) and the separator.

```
GSI format [Gsi8/gsI16] <gsI16>:
```

5. For GSI, choose GSI-8 or GSI-16.

Notes

- The CSV has one row per point with number, X, Y, Z, R, G, B (if the point has a colour) and code.
- ASCII and GSI use 3 decimals (millimetres) and a decimal point. GSI gets words 11 (number), 71 (code: its first word), 81, 82 and 83.
- GSI-8 holds 8 digits: UTM coordinates do not fit in millimetres and the command warns you; use GSI-16.
- Point objects, which have no number, are numbered in the order they were created.

See also

IMPORTPOINTS · RENUMBERPOINTS · POINTCLOUD

IGNDEM — IGN DEM

Downloads the DEM of the current view from the IGN WCS service (Spain).

- **Command:** `IGNDEM` · **Global name:** `_IGNDEM`
- Ribbon: Topography › Terrain data

Downloads the digital elevation model of the area shown on screen from the **IGN elevation WCS service** (Spain) and creates it as a surface, like IMPORTDEM. There is no need to download or unzip sheets.

Usage

1. Give the drawing its coordinate system with GEOGRAPHICLOCATION and pan to the area.
2. Enter IGNDEM or click **IGN DEM** in the **Terrain data** panel of the Topography tab.

```
Resolution in metres [5/25/200/500/1000/Auto] <Auto>:
```

3. Press Enter for the automatic resolution or choose the DEM grid spacing (5, 25, 200, 500 or 1000 m).

Notes

- **Auto**: the finest grid with which the view stays below about 250,000 cells; it is then subsampled to about 30,000 points. For the 5 m DEM, zoom in.
- The area is requested in ETRS89 / UTM 30 (EPSG 25830; in the Canary Islands, REGCAN95 / UTM 28), clipped to the view and reprojected to the drawing coordinate system, with a warning when the datum changes.
- It needs a connection. The answer is kept in the map cache for seven days, like the tiles.
- The source ("Instituto Geográfico Nacional", CC BY 4.0) is labelled in the lower-left corner of the view, as the licence requires.
- Service: `https://servicios.idee.es/wcs-inspire/mdt`. It only covers Spain; for other countries, see Map servers by country.

See also

IMPORTDEM · CONTOUR · MAPSERVICE · GEOGRAPHICLOCATION

IMPORTDEM — Import DEM

Imports a digital elevation model (ASCII Grid or GeoTIFF) as a surface.

- **Command:** `IMPORTDEM` · **Global name:** `_IMPORTDEM`
- Ribbon: Topography › Terrain data

Imports a **gridded digital elevation model** (raster) and creates it as a surface, like SURFACE: a mesh of triangular faces on which CONTOUR works. It reads the DEMs of the Spanish CNIG download centre (MDT05, MDT25 and MDT200) and any other in **ESRI ASCII Grid** (`.asc`) or **GeoTIFF** (`.tif`).

Usage

1. Enter IMPORTDEM or click **Import DEM** in the **Terrain data** panel of the Topography tab.

```
DEM file (.asc or .tif):
```

2. Choose the file. The **Import DEM** dialog opens with the grid size and the coordinate system of the file.
3. Check the **Source coordinate system**, choose the clip and the subsampling and click **OK**.
4. With **Window**, specify the two corners; with **Boundary**, select a closed polyline.

```
Select a closed polyline as boundary:
```

Options

- **Source coordinate system**: the one stated by the file is proposed (GeoTIFF EPSG GeoKey, the `.prj` of the ASCII Grid or the zone in the CNIG sheet name, such as `…_ETRS89_HU30_…`). If the drawing has no coordinate system it adopts it; if it has another one, the data are reprojected to it, with a warning when the datum changes (for example, from ETRS89 to ED50).
- **Clip and subsample**: All (the whole sheet), Window or Boundary, in drawing coordinates.
- **One cell of every**: takes one cell of every N in rows and columns. With 0 it is chosen automatically for about 30,000 points.

Command line

After the file, the same questions as in the dialog are asked; Enter accepts the proposed value.

```
Source EPSG <…>:
```

```
Subsample, one of every N cells [Auto] <Auto>:
```

```
Clip [All/Window/Boundary] <All>:
```

Notes

- The Spanish IGN and CNIG DEMs are in **ETRS89 / UTM**, in the zone of the sheet (EPSG 25829, 25830 or 25831); the Canary Islands, in REGCAN95 / UTM 28. If the drawing is in ED50 (EPSG 23030…), the datum change uses the official grid when installed (see Georeferencing).
- A DWG face mesh holds at most 32,767 vertices: if the requested subsampling gives more, it is increased with a warning.
- The centre of each cell is used; cells without data (NODATA) are skipped.
- Where to download official DEMs for Spain and other countries: Map servers by country.

See also

IGNDEM · SURFACE · CONTOUR · LANDXMLIN · GEOGRAPHICLOCATION

IMPORTPOINTS — Import points

Imports a survey (CSV/ASCII, Leica GSI, Sokkia SDR, Topcon, Trimble JobXML or RW5) as labelled points or point objects.

- **Command:** `IMPORTPOINTS` · **Global name:** `_IMPORTPOINTS`
- Ribbon: Topography › Points
- Menu: Survey

Imports a total station or GNSS **survey** and creates **survey points** in the drawing with their number, elevation and code labelled, or **point objects** with their elevation. If asked, it also draws the lines and symbols from the field codes (PROCESSLINEWORK).

Usage

1. Enter IMPORTPOINTS or click **Import points** on the Survey tab.

```
Points file (CSV, TXT, GSI, SDR, JobXML, RW5):
```

2. Choose the file with Browse or enter its path. The format is recognised from the extension and the content.

```
Source EPSG or [Format/Order/Type/Codes] <none>:
```

3. In the dialog, check the format and, for ASCII, the column order and the separator. Choose **Create as** (survey points or point objects), the label style, the layer by code and whether to draw by codes. Enter the source

EPSG if needed and click OK.

On the command line (scripts), the Format, Order, Type and Codes options change the same as the dialog, and Enter imports.

Supported formats

The point coordinates are read (name, X or easting, Y or northing, Z and code or description):

Format	What is read
ASCII (CSV, TXT…)	Columns by letters: P number, N northing (Y), E easting (X), Z elevation, D code and - to skip one (PNEZD, PENZD, NEZ, PXYZD…), or by header. Comma, semicolon, tab or space separator, and dot or comma decimal. With spaces, the last D column takes the rest of the line. With no header or order: number, X, Y, Z, code.
Leica GSI-8 and GSI-16	Words 11 (number), 81, 82 and 83 (E, N, Z, in mm, 1/10 mm, 1/100 mm or feet) and 71 (code). A code block 41 gives the code of the following points.
Sokkia SDR33 / SDR2x	Coordinate records 08, with 4 or 16-character names and 10 or 16-character numbers. The N-E or E-N order is taken from record 00.
Topcon GTS	ASCII coordinate dump: number, N, E, Z and code separated by commas.
Trimble JobXML (.jxl)	The reduced coordinates in `Reductions`; if there are none, the non-deleted `PointRecord` entries with grid coordinates.
Carlson/TDS RW5	Stored points (SP) and side shots (SS, TR, BD, FD) reduced from the station (OC), the backsight (BK) and the instrument and rod heights (LS). Angles in DDD.MMSS or gons.

Not read: raw observations in GSI, SDR and JobXML (only their coordinates), RW5 GPS records (latitude and longitude) and code attributes.

Notes

- A **survey point** is a `PUNTO_TOPO` block with the NUMERO, COTA and CODIGO attributes: it looks the same in AutoCAD and in any program that reads DWG. The elevation is the Z of the insertion point, so SURFACE and CONTOUR use it as data.
- With **layer by code**, each point goes to layer `TOPO_PUNTOS_<code>` (without trailing digits: BOR1 and BOR2 go to `TOPO_PUNTOS_BOR`), with the code's colour in the table (SURVEYCODES).
- Point objects take the current layer and colour or, if the CSV carries RGB, each point's colour.
- If the source EPSG is given, points are reprojected to the drawing's system (GEOGRAPHICLOCATION); if the drawing has no system, it adopts that one.

See also

EXPORTPOINTS · PROCESSLINEWORK · POINTLABELSTYLE · SURVEYCODES · POINTCLOUD · SURFACE

LANDXMLIN —Import LandXML

Imports the surfaces and points of a LandXML file.

- **Command:** `LANDXMLIN` · **Global name:** `_LANDXMLIN`
- Ribbon: Topography › Terrain data

Imports a **LandXML 1.2** file, the exchange format of Civil 3D, MDT/TcpMDT, BricsCAD Civil and many field data collectors: each **TIN surface** is created as a surface (like SURFACE) and each **COGO point** (CgPoint) as a point object.

Usage

1. Enter LANDXMLIN or click **Import LandXML** in the **Terrain data** panel of the Topography tab.

```
LandXML file:
```

2. Choose the `.xml` file. Surfaces and points are created on the current layer.

Notes

- LandXML stores coordinates as northing, easting, elevation; LinceCAD places them as X = easting, Y = northing.
- If the file states its coordinate system (`epsgCode`) and the drawing has none, the drawing adopts it; if the drawing has another, the data are reprojected.
- Faces marked invisible (outside the boundary) are not imported; four-vertex faces are split into two triangles.
- A DWG face mesh holds at most 32,767 vertices: a larger surface is skipped with a warning.
- Surfaces and points are imported; alignments, profiles and parcels are ignored for now. Points arrive as point objects, without their name or code.
- Metric units are assumed.

See also

LANDXMLOUT · IMPORTDEM · SURFACE · CONTOUR · IMPORTPOINTS

LANDXMLOUT —Export LandXML

Exports the selected surfaces and points to LandXML.

- **Command:** `LANDXMLOUT` · **Global name:** `_LANDXMLOUT`
- Ribbon: Topography › Terrain data

Exports the selected surfaces and points to **LandXML 1.2**, to take them to Civil 3D, MDT/TcpMDT, BricsCAD Civil or a field data collector.

Usage

1. Enter LANDXMLOUT or click **Export LandXML** in the **Terrain data** panel of the Topography tab.

```
Select objects:
```

2. Select the surfaces created with SURFACE, IMPORTDEM or LANDXMLIN, and the points or blocks, and press Enter.

```
Target LandXML file:
```

3. Enter the target `.xml` file.

What is written

- Each selected **face mesh**, as a TIN surface (`Surface`, with its `Pnts` and `Faces`).
- Each **point object**, as a CgPoint numbered from 1; each **block**, with the number and code from its attributes if it has them (as in COORDTABLE).
- The drawing coordinate system (`epsgCode`), if it has one, and metric units.

Notes

- Coordinates are written in LandXML order (northing, easting, elevation) with four decimals.
- Alignments are not exported for now.

See also

LANDXMLIN · EXPORTPOINTS · COORDTABLE · SURFACE

MOVEPOINTLABEL —Move labels

Moves the labels of survey points without moving the points.

- **Command:** `MOVEPOINTLABEL` · **Global name:** `_MOVEPOINTLABEL` · **Alias:** `MLABEL`
- Ribbon: Topography › Points
- Menu: Survey

Moves the **labels** of survey points (number, elevation and code) **without moving the point**, so that they do not overlap other lines or texts on the plan.

Usage

1. Enter MOVEPOINTLABEL or click **Move labels** on the Survey tab.

```
Select objects:
```

2. Select one or more points and press Enter.

```
Specify base point or [Reset]:
```

3. Specify the base point of the displacement, or choose **Reset** to put the labels back where the default label style places them.

```
Specify second point:
```

4. Specify the second point: the labels move by that distance.

Notes

- The insertion point (with its elevation) does not change, so the TIN and the contours are not affected.
- The labels are attributes: AutoCAD shows them in the same position.

See also

POINTLABELSTYLE · IMPORTPOINTS

POINTCLOUD —Import point cloud

Imports a LAS or LAZ point cloud (LiDAR).

- **Command:** `POINTCLOUD` · **Global name:** `_POINTCLOUD`
- Ribbon: Topography › Points
- Menu: Survey

Imports a LiDAR point cloud (LAS or LAZ files) and shows it in the drawing as point objects, in the drawing's coordinates.

Usage

1. Enter POINTCLOUD or click the button on the Survey tab. The dialog box opens and the command line asks at the same time.
2. Choose the LAS or LAZ file with Browse or enter its path. The number of points and how many remain after subsampling are shown.
3. Choose the point colour (RGB, intensity, class or elevation) and, if needed, the file's coordinate system.

4. Click OK.

Notes

- Reading is pure Rust (the `las` crate with LASzip `laz`), with no C components.
- Large clouds are subsampled to a manageable number (maximum points configurable in the dialog): 1 of every N points, spread across the whole file, is kept.
- If the file carries its coordinate system (WKT) it is reprojected to the drawing's (GEOGRAPHICLOCATION); if not, you can enter it by hand.
- Each point is an ordinary point object: the 2D and 3D viewers draw it and it is saved in the DWG. The massive engine (billions of points) is out of scope in this version.

See also

IMPORTPOINTS · EXPORTPOINTS · GEOGRAPHICLOCATION · Georeferencing

POINTLABELSTYLE —Label style

Creates and edits the point label styles and applies them.

- **Command:** `POINTLABELSTYLE` · **Global name:** `_POINTLABELSTYLE` · **Alias:** `LABELSTYLE`
- Ribbon: Topography › Points
- Menu: Survey

Creates and edits the **label styles** of survey points and applies one to the points.

Usage

1. Enter POINTLABELSTYLE or click **Label style** on the Survey tab. If points are selected, the style is applied to them; if not, to every point in the drawing.
2. Choose a style from the list or click **New** to create a copy of the current one.
3. Change what is **shown** (number, elevation, code), the label **height** (also the size of the point mark), the elevation **decimals** and the label **position** (right, left, above or below).
4. Click **OK** to save the styles and apply the chosen one, or **Apply** to see it without closing the dialog.

On the command line only the style is chosen:

```
Label style to apply <…>:
```

Notes

- Styles are saved in `estilos_rotulo_topo.txt`, in the user data folder. The first one is the default for IMPORTPOINTS.
- What is not shown stays as an invisible attribute: the value is still in the drawing (for EXPORTPOINTS, for example).
- When a style is applied, the elevation is written again from the point's Z with the style's decimals, and the labels go back to their position.

See also

MOVEPOINTLABEL · IMPORTPOINTS · RENUMBERPOINTS

PROCESSLINEWORK —Draw by codes

Joins points with the same code into polylines and inserts the blocks, following the code table.

- **Command:** `PROCESSLINEWORK` · **Global name:** `_PROCESSLINEWORK` · **Alias:** `LINEWORK`
- Ribbon: Topography › Points
- Menu: Survey

Draws by codes: joins the survey points with the same line code into 3D or 2D polylines and inserts the symbol of block codes, following the code table (SURVEYCODES).

Usage

1. Enter PROCESSLINEWORK (alias LINEWORK) or click **Draw by codes** on the Survey tab.

```
Select the terrain points (points, blocks or a surface):
```

2. Select the points and press Enter.

It is also done on import, with the **Draw by codes** check box of IMPORTPOINTS.

Notes

- Points are taken in the order they were created, which is the order of the field file.
- Each line goes on its code's layer and colour. 3D lines pass through the elevation of each point (arcs are split into segments of 5° at most); 2D lines keep arcs as polyline arcs, at elevation 0.
- The start, end, close, arc and offset modifiers are explained in SURVEYCODES.
- At the end you are told how many lines and blocks were drawn and which codes are not in the table.
- 3D polylines can be used as TIN breaklines.

See also

SURVEYCODES · IMPORTPOINTS · SURFACE

PROFILEVIEW —Profile view

Draws the longitudinal profile of an alignment over the terrain and, optionally, the design grade.

- **Command:** `PROFILEVIEW` · **Global name:** `_PROFILEVIEW` · **Alias:** `PROFILE`
- Ribbon: Topography › Analysis

Draws the **longitudinal profile** of an alignment over the terrain surface and, optionally, over a second design surface (grade), with an exaggerated vertical scale, the elevation grid, the stations and the **data band**.

Usage

1. Enter PROFILEVIEW (PERFIL) or click **Profile view** on the Survey tab, Analysis panel.

```
Select the alignment (polyline or line):
```

2. Select the alignment: a polyline (arcs are followed in short segments), a line, an arc or a 3D polyline.

```
Select the terrain surface (TIN):
```

3. Select the terrain surface, a mesh created with SURFACE.

```
Select the design surface (grade) or [None] <None>:
```

4. Select the design surface or press Enter for the terrain only.

```
Enter an option [Horizontal/Vertical/Station] or press Enter to draw <H 1:…, V 1:…, station every … m>:
```

5. The **Profile view** dialog opens with the scales and the station interval. Click **OK** (or press Enter on the command line to accept the values).

```
Specify the profile origin (lower-left corner of the grid):
```

6. Specify the origin: the lower-left corner of the grid. The data band goes below it.

Options

- **Horizontal** (H): horizontal scale denominator, 1:1000 by default. Texts are 2.5 mm high at this scale.
- **Vertical** (V): vertical scale denominator, 1:100 by default. The vertical is exaggerated H/V times (×10 by default).
- **Station** (S): station interval, 20 m by default. There is a vertical grid line and a data band column at each station and at the end of the alignment.

What it draws

Layer	Contents
`PERFIL_CUADRICULA`	Frame, elevation lines (about 10 mm on paper) with their labels, station lines and title
`PERFIL_TERRENO`	Terrain profile
`PERFIL_RASANTE`	Design surface profile
`PERFIL_GUITARRA`	Data band: station, existing ground, design grade and difference (grade ? ground), and the datum

Notes

- The profile is exact: each alignment segment is cut with the TIN triangles and the elevation is interpolated inside each one. Where the alignment leaves the surface the profile breaks and the data band shows "-".
- Everything is plain DWG lines, polylines and texts. Each object stores in its extended data (XDATA `LINCE_PERFIL`) the alignment, the surfaces and the settings: if the TIN or the alignment changes, PROFILEVIEWUPDATE redraws it.
- The difference is positive in fill (grade above the ground) and negative in cut.

See also

PROFILEVIEWUPDATE · SURFACEVOLUME · SURFACE · CONTOUR

PROFILEVIEWUPDATE —Update profile

Redraws a profile view with the current alignment and surfaces.

- **Command:** `PROFILEVIEWUPDATE` · **Global name:** `_PROFILEVIEWUPDATE` · **Alias:** `PVUPDATE`
- Ribbon: Topography › Analysis

Redraws one or more **profile views** created with PROFILEVIEW, reading the alignment and the surfaces as they are now. Use it after changing the TIN, the design grade or the alignment.

Usage

1. Enter PROFILEVIEWUPDATE (ACTUALIZARPERFIL) or click **Update profile**.

```
Select objects of the profiles to update:
```

2. Select any object of each profile (a grid line, a data band text…) and press Enter. If objects were already selected, they are updated without asking.

Notes

- All the objects of the profile are erased and drawn again at the same origin, with the same scales and stations, stored in their extended data (XDATA `LINCE_PERFIL`).
- Manual changes made to the profile objects are lost when updating.
- If the alignment or one of the surfaces was erased, the profile cannot be rebuilt and the reason is reported; if the TIN was created again as a new object, draw the profile again with PROFILEVIEW.

See also

PROFILEVIEW · SURFACE

RENUMBERPOINTS — Renumber points

Numbers the selected survey points from a start number with an increment.

- **Command:** `RENUMBERPOINTS` · **Global name:** `_RENUMBERPOINTS` · **Alias:** `RENUMPT`
- Ribbon: Topography › Points
- Menu: Survey

Changes the **number** of the selected survey points: numbers them from a start number with an increment.

Usage

1. Enter RENUMBERPOINTS or click **Renumber points** on the Survey tab.

```
Select objects:
```

2. Select the points and press Enter.

```
Start number <1>:
```

3. Give the start number. It may have a prefix and leading zeros: `E100`, `P001`.

```
Increment <1>:
```

4. Give the increment (it may be negative).

Notes

- Points are numbered in the order of their current number (first by prefix, then by digits), which keeps the survey order.
- If any new number already belongs to a point that was not selected, nothing is changed and you are warned.

See also

ZOOMTOPOINT · IMPORTPOINTS · POINTLABELSTYLE

SPOTELEVATION — Spot elevation

Queries the terrain elevation at a point, interpolated from the surface.

- **Command:** `SPOTELEVATION` · **Global name:** `_SPOTELEVATION` · **Alias:** `SPOTZ`
- Ribbon: Topography › Terrain

Queries the **terrain elevation at a point**, interpolated in the triangle of the surface that contains it, and optionally **labels** it.

Usage

1. Enter SPOTELEVATION (COTAMDT) or click **Spot elevation**.

```
Select surface:
```

2. Select the surface (SURFACE).

```
Pick a point or [Label] (label: …) <eXit>:
```

3. Pick the point: its elevation is shown at the command line, as with ID. With the **Label** option (L) before the point, a point at that elevation and a text with it are also drawn. Enter repeats the command.

Notes

- Interpolation is linear within the triangle: it respects the breaklines of the surface.
- Outside the surface (or in a hole) there is no elevation.

See also

SURFACE · EDITSURFACE · CONTOURLABEL

SURFACE —Surface (TIN)

Builds the digital terrain model by triangulating the selected points.

- **Command:** `SURFACE` · **Global name:** `_SURFACE` · **Alias:** `TIN`, `MDT`
- Ribbon: Topography › Terrain

Builds the **digital terrain model (TIN)** by Delaunay-triangulating points with elevation, with **breaklines** forced as edges, an **outer boundary**, **holes** and a **maximum edge length**. The surface is stored as a 3D face mesh that the viewer draws and that the contour lines are generated from.

Usage

1. Enter SURFACE (MDT) or click **Surface (TIN)**.

```
Select the terrain points (points, blocks or a surface):
```

2. Select the entities with elevation: points, blocks (their insertion point is used) or an existing surface. Press Enter.

```
Select breaklines <none>:
```

3. Select the breaklines: 3D polylines, 2D polylines (at their elevation) or lines. Their segments become mesh edges: the surface folds along them (top and toe of a slope, edge of a wall, ditch line). Enter, none.

```
Select outer boundary <none>:
```

4. Select the outer boundary (closed polyline): triangles outside it are discarded. Enter, the whole hull of the points.

```
Select holes <none>:
```

5. Select the holes (closed polylines): areas with no surface inside the boundary, such as a building or a pond.

```
Maximum edge length (0 no limit) <…>:
```

6. Give the maximum edge length: border triangles with an exposed edge longer than that are removed, so distant points are not joined. 0, no limit.

Notes

- The Z of each entity is its elevation; points that coincide in plan are merged. Where two breaklines cross, a vertex is created with the elevation of the second one.
- A breakline whose vertices are all at elevation 0 is taken as a **breakline without elevation**: it forces the edge and takes the terrain elevations. The boundary and holes also take the terrain elevation.
- The mesh is a PolyfaceMesh that any DWG program reads. A model with more than 32,767 vertices or faces (the format limit) is stored as several meshes, in strips, which are selected and edited as one.
- The constraints are stored with the mesh (in its extension dictionary, record `LINCE_MDT`): they survive saving and opening the DWG and allow editing it with EDITSURFACE.
- For contours, select this surface in CONTOUR: its breaklines are respected.

See also

EDITSURFACE · CONTOUR · SPOTELEVATION · POINT

SURFACEANALYSIS —Surface analysis

Colors a surface by slope, elevation bands or aspect, with a legend.

- **Command:** `SURFACEANALYSIS` · **Global name:** `_SURFACEANALYSIS` · **Alias:** `SLOPEMAP`
- Ribbon: Topography › Analysis

Colors a surface (TIN) by **slope**, by **elevation bands** (hypsometric map) or by **aspect** in 8 directions, and draws the **legend** with the area of each class.

Usage

1. Enter SURFACEANALYSIS (ANALISISTERRENO) or click **Surface analysis** on the Survey tab, Analysis panel.

```
Select the terrain surface (TIN):
```

2. Select the surface (a SURFACE mesh).

```
Analysis type [Slopes/Elevations/Aspects] <…>:
```

3. The **Surface analysis** dialog opens: choose the map type, the slope unit and the range limits; the legend updates at once. Click **OK**. On the command line, the same questions:

```
Slope unit [Percent/Degrees] <…>:
Range limits, separated by spaces <…>:
Specify the legend point or press Enter to place it on the right:
```

4. Specify where the legend goes or press Enter to place it to the right of the terrain.

Map types

Type	Classifies	Default ranges	Layer
Slopes	the slope of each triangle, in % or degrees	3, 6, 12, 25 and 50 % (2, 5, 10, 20 and 30°)	`MDT_PENDIENTES`
Elevations	the elevation: each triangle is split exactly at the limit elevations	6 round-step bands between the lowest and highest elevation	`MDT_ELEVACIONES`
Aspects	where each triangle faces downhill: N, NE, E, SE, S, SW, W, NW, and Flat	(fixed)	`MDT_ORIENTACIONES`

Notes

- Limits are typed separated by spaces ("3 6 12 25 50"); **Automatic** restores the defaults. With *n* limits there are *n* + 1 classes: "< a", "a – b"…">=z".
- Each class is filled with SOLID objects of its color (RGB colors, stored as the nearest ACI in DWG 2000); the legend goes on the same layer as the map, so both can be turned off together.
- The slope ramp goes from green (flat) to red (steep); the hypsometric one from green (low) to brown and white (high).

See also

SURFACE · CONTOUR · SURFACEVOLUME

SURFACEVOLUME — Earthwork volume

Computes cut and fill volumes between two surfaces or between a surface and an elevation.

- **Command:** `SURFACEVOLUME` · **Global name:** `_SURFACEVOLUME` · **Alias:** `VOLUMES`, `CUTFILL`
- Ribbon: Topography › Analysis

Computes **cut and fill** volumes between the terrain surface and a design surface, or between the terrain and a **reference elevation**, inside an optional boundary. It draws the results table as a TABLE, the **cut and fill map** with its legend and, if asked, saves the report as CSV.

Usage

1. Enter SURFACEVOLUME (CUBICACION) or click **Earthwork volume** on the Survey tab, Analysis panel.

```
Select the terrain surface (TIN):
```

2. Select the terrain surface (a SURFACE mesh).

```
Select the design surface or [Elevation]:
```

3. Select the design surface, or enter **E** to compare with an elevation.

```
Reference elevation:
```

4. With Elevation: give the reference elevation (horizontal plane).

```
Select the boundary (closed polyline) or [None] <None>:
```

5. Select a closed polyline that bounds the area, or press Enter to compute wherever both surfaces overlap.

```
Enter an option [Method/MAp/Csv] or press Enter to compute <…>:
```

6. The **Earthwork volume** dialog opens: method, map and CSV. Click **OK** (or press Enter on the command line).

```
Specify the table insertion point:
```

7. Specify where the results table goes.

Options

- **Method** (M): **Prisms** (default) or **Grid**, with the cell size.
- **Map** (MA): draw the cut and fill map or not (yes by default).
- **Csv** (C): file where the report (item, value and unit) is saved. With FILEDIA=1 it is chosen with the Windows dialog.

Results

- **Cut**: volume where the terrain is above the design (to be excavated). **Fill**: where it is below (to be filled).

- **Net** = cut ? fill: positive when there is surplus earth.
- **Areas** in cut, in fill and the total compared, and **balance** = fill / cut, in %.
- The table goes on layer `CUB_INFORME`; the map and its legend on `CUB_CORTE_RELLENO`, in three shades of blue (fill) and three of red (cut) by depth.

Accuracy of the methods

- **Prisms**: overlays both triangulations. Each terrain triangle is clipped with each design triangle (and with the boundary); in each piece both surfaces are planar, the elevation difference is linear and it is split along its zero line. The volume on each side is its area times the difference at its centroid. It is **exact** for triangulated surfaces, with no error but rounding: in the test cases (pyramid, prism, sloped plane against an elevation, L-shaped boundary) the measured error is below 10??, far below the required 0.01 %.
- **Grid**: adds difference × cell area at the center of each cell. It is approximate: it errs in the cells that cross the boundary, the triangle edges or the zero line. The error decreases with the cell size (in the test pyramid, below 0.1 % with 5 cm cells); it is useful as a cross-check and for very large surfaces.

See also

SURFACE · SURFACEANALYSIS · PROFILEVIEW · TABLE

SURVEYCODES —Code table

Edits the field code table: line, point or block, with its layer and colour.

- **Command:** `SURVEYCODES` · **Global name:** `_SURVEYCODES` · **Alias:** `CODETABLE`
- Ribbon: Topography › Points
- Menu: Survey

Edits the **field code table**: what is drawn for each code the surveyor writes in the field book or the data collector.

Usage

1. Enter SURVEYCODES or click **Code table** on the Survey tab.
2. In the dialog, each row is a code with its **type** (line, point or block), its **layer**, its **colour**, the **block** and its **scale** (for block codes), whether the line is **3D**, and a description. Add codes with **Add code** and remove them with **Remove**; **Default values** restores the initial table.
3. Click **OK** to save it.

On the command line:

```
Code table … (… codes) [Reset] <Close>:
```

Field coding

The description of each point is the code followed, separated by spaces, by its modifiers:

Modifier	What it does
`I` or `S`	Starts a new line at that point.
`F` or `E`	Ends the line at that point.
`C`	Ends the line and closes it to its first point.
`A`	The point is the middle of an arc through the previous point, this one and the next.
`P<d>`	Also draws an offset line at distance d (positive to the left of the direction of travel), for example `P0.5`.

A code with trailing digits (`BOR1`, `BOR2`) uses the row of the code without them (`BOR`), but each one is a separate

line: this is how two kerbs are surveyed at the same time.

Notes

- The table is the text file `codigos_topo.txt` in the user data folder, with one row per code: `codigo;tipo;capa;color;bloque;3d;escala;descripcion`. It can be edited by hand; Enter on the command line creates it with the default values if it does not exist yet.
- The TOPO_ARBOL, TOPO_FAROLA, TOPO_POSTE and TOPO_ARQUETA blocks are built in and are created in the drawing when used; any other block must be defined in the drawing.

See also

PROCESSLINEWORK · IMPORTPOINTS

ZOOMTOPOINT — Zoom to point

Finds a survey point by its number and zooms the view to it.

- **Command:** `ZOOMTOPOINT` · **Global name:** `_ZOOMTOPOINT` · **Alias:** `FINDPOINT`, `ZTP`
- Ribbon: Topography › Points
- Menu: Survey

Finds a **survey point by its number**, shows its coordinates and code and zooms the view to it.

Usage

1. Enter ZOOMTOPOINT or click **Zoom to point** on the Survey tab.

```
Point number:
```

2. Enter the point number (case-insensitive). The command line shows X, Y, Z and the code, and the view is centred on the point.

Notes

- If no point has that number, you are warned and asked again.

See also

RENUMBERPOINTS · ZOOM · ID

Utilities and Inquiry

-HYPERLINK —Hyperlink (command line)

Attaches a file or a URL to objects of the drawing.

- **Command:** `-HYPERLINK` · **Global name:** `_-HYPERLINK`

Command-line version of HYPERLINK: Insert or Remove the hyperlink of the selected objects.

See also

HYPERLINK

-PURGE —Purge (command line)

Purges unused items from the command line.

- **Command:** `-PURGE` · **Global name:** `_-PURGE` · **Alias:** `-PU`

Command-line version of PURGE: removes the unused items of one type, by name and optionally confirming each one. It is the way to purge from a script (SCRIPT).

Usage

1. Enter -PURGE (or -PU).
2. Choose the type of items.

```
Enter type of unused objects to purge [Blocks/Dimstyles/LAyers/LTypes/MUltileaderstyles/Plot styles/textSTyles/SHapes/Mlinestyles/Tablestyles/Regapps/Zero-length geometry/Empty text objects/All]:
```

3. Enter the names you want to purge. The wildcards `*` (any text) and `?` (one character) and several patterns separated by commas are accepted; Enter purges all items of the type.

```
Enter name(s) to purge <*>:
```

4. Say whether you want to confirm each item. With Yes, the program asks about each one; with No, it removes them all.

```
Verify each name to be purged? [Yes/No] <Y>:
Purge … "…"? [Yes/No] <N>:
```

Options

Option	Type
Blocks (B)	Blocks
LAyers (LA)	Layers
LTypes (LT)	Linetypes
textSTyles (ST)	Text styles
SHapes (SH)	Shapes
Dimstyles (D)	Dimension styles
MUltileaderstyles (MU)	Multileader styles
Tablestyles (T)	Table styles

Mlinestyles (M)	Multiline styles
Plotstyles (P)	Plot styles
Regapps (R)	Registered applications
Zero-length (Z)	Zero-length geometry
Empty (E)	Empty text objects
All (A)	All named types

Notes

- Zero-length and Empty ask for no names and no confirmation: they remove everything they find.
- All does not include zero-length geometry or empty text, and also purges nested items (what becomes unused when others are removed).
- The same safety rules as in PURGE apply. The result is undone with UNDO.

See also

PURGE · SCRIPT · -OVERKILL

-RENAME —Rename (command line)

Renames named objects from the command line.

- **Command:** `-RENAME` · **Global name:** `_-RENAME` · **Alias:** `-REN`

Changes the name of a named object from the command line, like RENAME without the dialog. Useful in scripts and AutoLISP.

Usage

1. Enter -RENAME (or -REN).
2. Choose the object type.

```
Enter object type to rename [Block/Dimstyle/LAyer/LType/Material/multileadeRstyle/textStyle/
Tablestyle/MLinestyle/Ucs/VIew/VPort/layOut]:
```

3. Enter the old name and then the new one (they can contain spaces). Enter in either of them ends without changing anything.

```
Enter old name:
```

Notes

- The rules are those of RENAME: what cannot be renamed and invalid or repeated names are reported.
- With an asterisk several are renamed at once: AB* to CD* changes AB1 to CD1, AB2 to CD2…, what the asterisk covers is kept. AutoCAD only accepts it in the RENAME dialog; LinceCAD accepts it here too.

```
With an asterisk in the old name, put another in the new one (AB* to CD*).
```

- Besides the AutoCAD types, LinceCAD renames multiline styles and layouts.

See also

RENAME · -LAYER · SCRIPT

AREA —Area

Calculates area and perimeter.

- **Command:** AREA · **Global name:** _AREA · **Alias:** AA
- Ribbon: Home › Utilities
- Menu: Tools

Calculates the area and perimeter of a boundary defined by points or of a closed object. With the Add area and Subtract area modes it keeps a running total, for example to subtract openings from a surface. The drawing does not change.

Usage

1. Enter AREA (or AA) or click its button.

```
Specify first corner point or [Object/Add area/Subtract area] <Object>:
```

2. To measure by points, specify the corners of the boundary one after another. The figure closes by itself: there is no need to go back to the first point.

```
Specify next point or [Undo]:
Specify next point or [Undo/Total] <Total>:
```

3. Press Enter (or type T) to calculate. The command line shows the area and the perimeter.
4. To measure an object, press Enter at the first prompt (or type O) and select the object.

```
Select objects:
```

Options

- **Object** (O): measures a circle, arc, ellipse, polyline, spline or hatch. For an open object the area is closed by the chord between its endpoints and its length is given instead of the perimeter.
- **Add area** (A): switches to add mode. Every boundary or object you measure is added to the total, which is shown after each measurement.

```
Specify first corner point or [Object/Subtract area]:
(ADD mode) Select objects:
```

- **Subtract area** (S): switches to subtract mode. Every measurement is subtracted from the total.

```
Specify first corner point or [Object/Add area]:
(SUBTRACT mode) Select objects:
```

- **Undo** (U): removes the last point of the boundary.
- **Total** (T): closes the boundary and calculates.

In add and subtract modes, Enter at the object prompt goes back to the mode prompt; Enter at the mode prompt ends the command and shows the total.

Notes

- In a hatch, the inner boundaries (islands) subtract their area.
- Regions and 2D solids cannot be measured yet.
- Results are written with the units and precision of the drawing (UNITS).

See also

MEASUREGEOM · DIST · LIST · HATCH · BOUNDARY

CAL —Calculator

Evaluates a mathematical expression.

- **Command:** `CAL` · **Global name:** `_CAL`
- Ribbon: Home › Utilities

Evaluates a mathematical expression typed on the command line and shows the result. It does not change the drawing.

Use

1. Type CAL or click its button.
2. Type the expression and press Enter.

```
Expression to evaluate:
```

3. The result appears on the command line with the drawing format (UNITS).

Expressions

- Operators: `+`, `-`, `*`, `/`, `^` (power) and the sign; use parentheses to group.
- Constants: `pi` and `e`.
- Functions: `sin`, `cos`, `tang`, `asin`, `acos`, `atan`, `sqrt`, `sqr` (square), `abs`, `ln`, `log` (base 10), `exp`, `round`, `trunc`, `r2d` and `d2r`.

Points and vectors

- Points and vectors are written `[x,y]` or `[x,y,z]`. They add and subtract; `number*point` scales them; `a*b` is the dot product and `a&b` the cross product.
- Geometric functions: `dist(p1,p2)`, `ang(p1,p2)` (degrees in the XY plane; also `ang(vertex,p1,p2)`), `vec(p1,p2)`, `vec1(p1,p2)` (unit), `mee(p1,p2)` (midpoint), `ill(p1,p2,p3,p4)` (intersection of two lines), `pld(p1,p2,d)` (point at distance d from p1 towards p2), `plt(p1,p2,t)` (parametric point), `rot(p,origin,angle)`, `xof`, `yof`, `zof` and `mag`; `cvunit(value,from,to)` converts units (mm, cm, m, km, in, ft, yd, mi, deg, rad, grad, g, kg, lb, oz, c, f, k).
- Drawing points: `cur` (any point) and the snaps `end`, `mid`, `int`, `ins`, `nea`, `nod`, `qua`, `cen`, `per` and `tan` ask for a point with the mouse; `rad` asks for an arc or circle and gives its radius. Example: `dist(cur,cur)` or `(end+end)/2`.
- The snap functions do not force the mode: the point comes out with the active snaps of the status bar (OSNAP). Coordinates are flat (Z = 0).

Notes

- Trigonometric angles are in **degrees**, like the AutoCAD calculator; `atan` returns degrees.
- Examples: `2+3*4`, `sqrt(2)*10`, `sin(30)`, `(1+2)^3`.

See also

DIST · AREA · ID · UNITS

COMPARE —Compare drawings

Compares the drawing with another DWG or DXF and highlights the differences.

- **Command:** `COMPARE` · **Global name:** `_COMPARE` · **Alias:** `DWGCOMPARE`
- Ribbon: Manage › Drawing

Compares the open drawing with another DWG or DXF (for example, two revisions of the same sheet) and highlights the differences in model space.

Usage

1. Type COMPARA or click **Compare drawings**.
2. Enter the file to compare with (or press Enter to pick it in the browser).

```
Enter the DWG or DXF to compare with (or Enter to pick it):
```

3. The current drawing is marked and a change summary is shown on the command line.

```
Markup: green = added, red = deleted, yellow = modified (UNDO to remove it).
```

How it matches

- First by each entity's **handle**: the same handle in both drawings is the same entity; if its geometry or properties changed, it counts as *modified*.
- The rest is matched by **geometry**: an entity with the same outline but a different handle (after renumbering) is treated as unchanged, not a change.

Notes

- The markup is overlaid on the drawing itself and is undoable with a single UNDO: modified in yellow, added in green, and entities deleted from the other drawing brought in red (on the current layer).
- Only model-space entities are compared.

Differences from AutoCAD

- In AutoCAD the command is COMPARE; here `_DWGCOMPARE` is still valid as an alias. There is no separate comparison drawing, comparison toolbar or revision clouds: the mark is overlaid on the drawing itself and undone with one UNDO. The colors (yellow, green, red) are fixed.

See also

AUDIT · UNDO

DBLIST — List All

Displays the data of every object in the drawing.

- **Command:** `DBLIST` · **Global name:** `_DBLIST`
- Ribbon: Home › Utilities

Displays the data of every object in the drawing, one after another, as if all were selected with LIST: type, layer, space, handle and geometry.

Usage

1. Enter DBLIST. The list is shown on the command line and in the text window (F2).

Notes

- The objects of model space and of the layouts are listed, in drawing order; those inside blocks are not.
- In a large drawing the list is very long: for a few objects, use LIST.

See also

LIST · STATUS

DIST — Distance

Measures the distance and angle between two points.

- **Command:** `DIST` · **Global name:** `_DIST` · **Alias:** `DI`
- Ribbon: Home › Utilities
- Menu: Tools

Measures the distance between two points and writes it on the command line, together with the angle of the line that joins them in the XY plane and the X, Y and Z deltas. The drawing does not change.

Usage

1. Enter DIST (or DI) or click its button.
2. Specify the first point with a click, an object snap or by typing its coordinates.

```
Specify first point:
```

3. Specify the second point. While you move the cursor, the line being measured is shown.

```
Specify second point:
```

4. The result appears on the command line and the command ends.

Notes

- Numbers are written with the format and precision of the drawing (UNITS): decimal, feet and inches, degrees, degrees/minutes/seconds…The angle follows the base and direction of ANGBASE and ANGDIR.
- LinceCAD is 2D: the angle from the XY plane and the Z delta are always 0.
- Use object snaps (Endpoint, Midpoint, Center…)to measure between exact points of the geometry. See Object snaps and precision.
- To measure several things in a row, press Enter or Space: it repeats the last command.

See also

MEASUREGEOM · AREA · ID · LIST · UNITS

DWGPROPS — Drawing Properties

Title, subject, author, keywords and other properties of the drawing.

- **Command:** `DWGPROPS` · **Global name:** `_DWGPROPS`
- Ribbon: Home › Utilities

Opens the Drawing Properties dialog box, with the data that describe the drawing, as in AutoCAD.

Usage

1. Enter DWGPROPS.
2. Choose the tab: - **General**: file name, location and size. - **Summary**: Title:, Subject:, Author:, Keywords:, Comments: and Hyperlink base:. - **Statistics**: creation and modification dates, who saved it and the editing time. - **Custom**: custom properties, with their name and value (Add, Delete).
3. Click OK to keep the changes in the drawing or Cancel to discard them.

Notes

- Summary and Custom are saved in the DWG (SummaryInfo section) and AutoCAD reads them the same; the fields of the Document category (Author, Title…)use them.

- Changes are undone with UNDO.

See also

TIME · STATUS · FIELD

FILTER — Filter

Selects with a filter of types and properties.

- **Command:** `FILTER` · **Global name:** `_FILTER` · **Alias:** `FI`
- Ribbon: Home › Utilities
- Menu: Tools

Selects the objects that match a filter: a list of conditions on the object type and its properties (layer, color, linetype, lineweight, length, radius, text height…), which can be grouped with AND, OR, XOR and NOT. It is AutoCAD's Object Selection Filters dialog.

Usage

1. Enter FILTER (or FI). If objects are selected, the filter applies to them; otherwise, to the whole current space.
2. Choose an object type or, with "(property)", a property, an operator and a value, and click **Add to List**. Each condition is added below the selected one.
3. To combine conditions, enclose them in **Begin OR** … **End OR** (or AND, XOR, NOT).
4. Click **Apply**: what matches the filter is selected. If a command is asking for objects, it receives them.

Conditions

- Without groups, all conditions must be met (AND).
- **OR**: one is enough. **XOR**: exactly one. **NOT**: the condition (or the group) is not met.
- The operators are = (equal), != (not equal), > and < (for numbers: lengths, radii, heights…) and * (wildcards, as in QSELECT: `WALL*`). Texts are compared ignoring case.

Named filters

Enter a name and click **Save As**: the filter is stored in the `filtros.nfl` file of the LinceCAD configuration folder and works in any drawing. Choose it in the list to load it; **Delete Current Filter List** removes it.

Notes

- AutoCAD, after Apply, asks "Select objects" and filters what you select; here what is already selected, or the whole space, is filtered. To use the filter inside a command, open FILTER from the menu while the command asks for objects.

See also

QSELECT · SELECTSIMILAR · Selecting objects

HYPERLINK — Hyperlink

Attaches a file or a URL to objects of the drawing.

- **Command:** `HYPERLINK` · **Global name:** `_HYPERLINK`
- Ribbon: Home › Utilities

Attaches a file or a URL to objects of the drawing, as in AutoCAD. The link is stored in each object's `PE_URL` extended data, so other programs can read it. There is no dialog box: everything is done on the command line

(HYPERLINK and -HYPERLINK are the same).

Usage

1. Enter HYPERLINK.

```
Enter an option [Remove/Insert] <Insert>:
```

2. Select the objects and press Enter.
3. With **Insert**: enter the file or URL and, if you wish, a description.

```
Enter the file or URL of the hyperlink:
Enter a description <none>:
```

4. With **Remove**: the link is removed from the selected objects.

Notes

- An object has one link only: inserting another replaces the previous one.
- The change can be undone with UNDO.

See also

DWGPROPS · PROPERTIES

ID — ID Point

Displays the coordinates of a point.

- **Command:** `ID` · **Global name:** `_ID`
- Ribbon: Home › Utilities
- Menu: Tools

Shows the coordinates of a point on the command line. Use it to check where a vertex, a center or any point of the drawing is. The drawing does not change.

Usage

1. Enter ID or click its button.
2. Specify the point, usually with an object snap (Endpoint, Center, Intersection…).

```
Specify point:
```

3. The X, Y and Z coordinates appear on the command line and the command ends.

Notes

- Coordinates are written with the format and precision of the drawing (UNITS). Z is always 0.
- The coordinates are those of the drawing's world coordinate system.
- The status bar also shows the coordinates of the cursor while it moves.

See also

DIST · LIST · Coordinates and input · Object snaps and precision

LIMITS —Limits

Sets the drawing limits and their checking.

- **Command:** `LIMITS` · **Global name:** `_LIMITS`
- Ribbon: Home › Utilities

Sets the model space limits, a rectangle that marks the working area (for instance, the sheet size), and whether drawing outside it is checked.

Usage

1. Enter LIMITS.

```
Reset Model space limits:
```

2. Specify the lower left corner (Enter keeps the current one).

```
Specify lower left corner or [ON/OFF] <…>:
```

3. Specify the upper right corner.

```
Specify upper right corner <…>:
```

Options

- **ON**: turns limits checking on (LIMCHECK = 1).
- **OFF**: turns it off (LIMCHECK = 0).

Notes

- The limits are saved in the drawing (variables LIMMIN and LIMMAX). A new drawing has those of acadiso.dwt: from 0,0 to 420,297.
- The upper right corner must be above and to the right of the lower left one.

```
The upper right corner must be above and to the right of the lower left one.
```

See also

STATUS · GRID · LIMCHECK

LIST —List

Displays property data for selected objects.

- **Command:** `LIST` · **Global name:** `_LIST` · **Alias:** `LI`, `LS`, `SHOWMAT`
- Ribbon: Home › Utilities
- Menu: Tools

Shows the data of the selected objects: type, layer, space, handle and geometry. The listing is written on the command line and the text window (F2) opens so you can read all of it. The drawing does not change.

Usage

1. Enter LIST (or LI) or click its button. If objects are already selected, they are listed at once.
2. Select the objects and press Enter.

```
Select objects:
```

3. Read the listing in the text window. Press F2 to close it or open it again.

What is listed

- For every object: type, layer, model or paper space and handle; and the color and linetype when they are not ByLayer.
- **Line**: start and end points, length, angle and deltas.
- **Circle** and **arc**: center, radius, circumference and area, or start and end angles and length.
- **Polyline**: open or closed, constant width, area, perimeter or length, and every vertex (with the center, radius and angles of the curved segments).
- **Ellipse**, **point**, **text**, **multiline text**, **spline**, **hatch** (pattern and area) and **dimension** (measurement and text).
- **Block reference**: name, insertion point, scales, rotation and attributes.

Notes

- Other types (tables, images, solids…) only show the common data.
- Measurements are written with the units of the drawing (UNITS).

See also

PROPERTIES · DIST · AREA · ID

MASSPROP — Mass Properties

Calculates the mass properties of regions and solids.

- **Command:** `MASSPROP` · **Global name:** `_MASSPROP`
- Ribbon: Home › Utilities

Calculates the mass properties of the selected regions and 3D solids, with AutoCAD's density of 1, and writes them in the text window (F2).

Usage

1. Enter MASSPROP.

```
Select objects:
```

2. Select regions or solids and press Enter. The report is shown: - regions: Area:, Perimeter:, Bounding box:, Centroid:, Moments of inertia:, Product of inertia:, Radii of gyration: and the principal moments with their directions; - solids: Mass:, Volume:, Bounding box:, Centroid:, Moments of inertia:, Products of inertia:, Radii of gyration: and the principal moments.
3. Say whether to save the analysis to a text file (.mpr).

```
Write analysis to a file? [Yes/No] <N>:
```

Notes

- Several regions are measured together, as one; holes subtract. Arcs are measured with very fine polygons.
- Solids are measured by their mesh: exact with planar faces, very close with curved ones.
- Moments and products of inertia are about the origin; the principal ones, about the centroid.

See also

AREA · REGION · LIST · MEASUREGEOM

MEASUREGEOM —Measure

Measures distances, radii and areas.

- **Command:** `MEASUREGEOM` · **Global name:** `_MEASUREGEOM` · **Alias:** `MEA`
- Ribbon: Home › Utilities
- Menu: Tools

Brings the most common measurements together in one command: distance between two points, radius of an arc or circle, angle, area, volume and quick measurement of objects. Each option works like the equivalent command and writes the result on the command line.

Usage

1. Enter MEASUREGEOM (or MEA) or click its button.

```
Enter an option [Distance/Radius/ANgle/ARea/Volume/Quick] <Distance>:
```

2. Choose the option. Enter, or specifying a point straight away, starts measuring a distance.
3. Follow the prompts of the chosen option. The command ends after the measurement.

Options

- **Distance** (D): distance, angle and deltas between two points, like DIST.
- **Radius** (R): select an arc or a circle; shows its radius and diameter.

```
Select arc or circle:
```

- **ARea** (AR): area and perimeter by points or of an object, with the add and subtract modes, like AREA.
- **ANgle** (AN): select an arc (its angle), a circle (pick two points on it), a line and then another (the angle between them, depending on where you pick them) or **Vertex** (V, Enter): vertex and two endpoints.

```
Select arc, circle, line, or [Vertex] <Vertex>:
```

- **Volume** (V): base by points (Enter calculates the area) or **Object** (a solid, or a closed curve that is given a height) and, for a flat base, the height.

```
Specify first corner point of the base or [Object] <Object>:
```

- **Quick** (Q): select objects and their measurements are shown (length and angle of a line, radius and diameter of an arc or circle, area and perimeter of a closed object) until you press Enter; it also switches to the other options.

```
Select an object or [Distance/Radius/ANgle/ARea/Volume/eXit] <eXit>:
```

Notes

- Quick does not follow the cursor as in AutoCAD: you click the object. There is no separate QUICKMEASURE command, and AutoCAD 2027 has none either.
- Results are written with the units and precision of the drawing (UNITS).

See also

DIST · AREA · ID · LIST

PTYPE — Point Style

Sets how points are displayed and their size.

- **Command:** `PTYPE` · **Global name:** `_PTYPE` · **Alias:** `DDPTYPE`
- Ribbon: Home › Utilities
- Menu: Format

Chooses how point objects are drawn (the symbol) and their size. The change affects all the points in the drawing, including those created by DIVIDE and MEASURE.

Usage

1. Enter PTYPE (or DDPTYPE) or click **Point Style**. The **Point Style** dialog box opens.
2. Click one of the 20 symbols.
3. Enter the **Point Size** and whether it is measured as a percentage of the screen size or in drawing units.
4. Click **OK**. **Cancel** leaves the style as it was.

On the command line (for example, in a script) you can type the values of the variables:

```
Enter new value for PDMODE (or use the Point Style dialog box):
Enter new value for PDSIZE:
```

Notes

- The symbol is the PDMODE variable: 0 to 4 (dot, nothing, cross, X, tick), plus 32 to add a circle, 64 for a square or 96 for both.
- The size is PDSIZE: a positive value is in drawing units; a negative one, a percentage of the screen.
- Both variables are saved in the drawing.

See also

POINT · DIVIDE · MEASURE · System variables

PURGE — Purge

Removes unused items from the drawing: blocks, layers, linetypes, styles…

- **Command:** `PURGE` · **Global name:** `_PURGE` · **Alias:** `PU`
- Ribbon: Manage › Drawing
- Menu: File

Removes from the drawing what is not used: block definitions, layers, linetypes and styles that nothing refers to, plus zero-length geometry and empty text objects. It makes the file lighter and keeps the layer and style lists tidy.

Usage

1. Enter PURGE (or PU) or click its button. The **Purge** dialog box opens with the **Purgeable items:** list, grouped by type and with the number of items in each group.
2. Expand the groups and check what you want to remove.
3. Click **Purge Checked Items**, or **Purge All** to remove everything in the list. The command line says how many items were purged.

Options

- **Show items you cannot purge**: Items that other objects still use; they are not purged.

- **Purge nested items** (checked by default): Also removes what becomes unused when the selected items are removed (for example, layers only used by a purged block).

What can be purged

Blocks, Layers, Linetypes, Text styles, Shapes, Dimension styles, Multileader styles, Table styles, Multiline styles, Plot styles, Registered applications, Zero-length geometry and Empty text objects.

Notes

- An item is only removed if nothing uses it: no object (including those inside blocks), no other table and not the drawing header. Layer 0, the Standard styles and other default items, the current ones and those of external references are never purged.
- Purging is undone with UNDO.
- To purge from the command line or in a script, use -PURGE.

See also

-PURGE · AUDIT · OVERKILL · LAYER

QSELECT —Quick Select

Selects objects by type and by the value of a property.

- **Command:** `QSELECT` · **Global name:** `_QSELECT`
- Ribbon: Home › Utilities
- Menu: Tools

Selects the objects that meet a condition on their type and the value of a property: all lines on a layer, circles with a radius greater than 5, texts of a certain height…The selection can then be edited with any command or with the Properties palette.

Usage

1. Enter QSELECT or click its button. The **Quick Select** dialog box opens.
2. Fill in the condition:

- **Apply to:** Entire drawing or, if objects were selected, Current selection.
- **Object type:** Multiple (all types) or one of the types present, with its number of objects.
- **Properties:** the general ones (layer, color, linetype…) and, once a type is chosen, its own (length, radius, height…), the same as in the Properties palette.
- **Operator:** = Equals, <> Not Equal, > Greater than, < Less than, * Wildcard Match or Select All.
- **Value:** the value to compare with. For the layer it is chosen from a list.

3. Under **How to apply:** choose Include in new selection set or Exclude from new selection set, and check **Append to current selection set** to add the result to what was already selected.
4. Click **OK**. The objects are selected.

Notes

- Numbers are compared as numbers, in the drawing units; texts, ignoring case. With the wildcard operator, * (any text) and ? (one character) can be used.
- The objects of the current space (model or the open layout) are searched.

See also

SELECTSIMILAR · AI_SELALL · PROPERTIES · Selecting objects

QUICKCALC —QuickCalc

Opens the calculator palette (QuickCalc).

- **Command:** `QUICKCALC` · **Global name:** `_QUICKCALC` · **Alias:** `QC`
- Ribbon: Home › Utilities

Opens the calculator palette (QuickCalc): a floating window to evaluate expressions without leaving the drawing.

Use

1. Type QUICKCALC (or QC) or click its button.
2. Type an expression and press Enter or the = button.
3. The result appears below the expression. The "To command line" button sends it: - if a command is running and asks for a value (a distance, a number), that command receives it; - otherwise it is left on the command line.
4. The history remembers the last expressions; click one to reuse it.

Expressions

The same as CAL: `+ - * / ^`, parentheses, constants (`pi`, `e`) and functions (`sin`, `cos`, `tang`, `sqrt`, `ln`, `log`, `abs` …), with angles in degrees.

Variables, units and scientific functions

- `name=expression` stores a variable (for example `width=2*3`); use it by name in any expression. It appears in the **Variables (name=expression)** list, where a click adds it to the expression.
- **Scientific functions**: buttons that type `sin(`, `cos(`, `tang(`, `sinh(`, `cosh(`, `tanh(`, `log(`, `ln(`, `exp(`, `exp10(`, `sqrt(`, `sqr(`, `abs(`, `r2d(` and `d2r(`.
- **Unit conversion**: value, source unit and target unit. Length (mm, cm, m, km, in, ft, yd, mi), angle (deg, rad, grad), mass (g, kg, lb, oz) and temperature (c, f, k). `cvunit(value,from,to)` also works inside an expression.
- It also takes points `[x,y,z]` and the geometric functions of CAL (`dist`, `ang`, `mee`…), except those that ask for points in the drawing.

See also

CAL · DIST · UNITS

RENAME —Rename

Changes the name of layers, blocks, styles and other named objects.

- **Command:** `RENAME` · **Global name:** `_RENAME` · **Alias:** `REN`
- Ribbon: Home › Utilities
- Menu: Format

Changes the name of blocks, dimension styles, layers, linetypes, multileader, text, table and multiline styles, UCSs, views, viewport configurations and layouts. Everything that refers to them is updated: the inserts of a block, the entities of a layer or a style, the layers of a linetype and the current values of the drawing.

Usage

1. Enter RENAME (or REN).
2. In the Rename dialog, choose the type on the left and the item on the right (or type its name in **Old name**).
3. Type the new name next to **Rename to** and click that button. You can rename several in a row; **OK** closes the

dialog.

Notes

- Layer 0 and Defpoints, the ByLayer, ByBlock and Continuous linetypes, the Standard styles, the space blocks and the anonymous ones (`*U…`), and what comes from an external reference cannot be renamed.
- The new name cannot be in use already nor contain `<>/\":;?*|,=` or a backquote. Changing only the case is allowed.
- All the renames of one session are undone at once with U. To do it on the command line, use -RENAME.

See also

-RENAME · LAYER · STYLE · DIMSTYLE · LAYOUT

SELECT —Select

Selects objects and keeps them selected for the next command.

- **Command:** `SELECT` · **Global name:** `_SELECT`

Selects objects and keeps them selected: the next command uses them without asking again, as if they had been selected before starting it. It is the AutoCAD SELECT used by many scripts and AutoLISP routines.

Usage

1. Enter SELECT (or DESIGNA).

```
Select objects:
```

2. Select the objects (click, window, crossing or ALL for all) and press Enter.
3. Start the command that will use them: ERASE, MOVE, COPY…

Notes

- AutoCAD's Previous (P) option to reuse this selection inside another command is not available yet: the objects stay selected and the next command takes them.

See also

QSELECT · SELECTSIMILAR · AI_SELALL · Selecting objects

SELECTSIMILAR —Select Similar

Selects objects similar to the selected ones.

- **Command:** `SELECTSIMILAR` · **Global name:** `_SELECTSIMILAR`
- Ribbon: Home › Utilities
- Menu: Tools

Selects all objects similar to the ones already selected: of the same type, on the same layer and, for block references, texts and dimensions, with the same block or style name.

Usage

1. Select one or more sample objects, with no command active.
2. Enter SELECTSIMILAR or click its button.
3. All objects similar to any of the samples are selected; the command line says how many.

Notes

- With no objects selected, the command only warns: Select the sample objects first.
- The objects of the current space (model or the open layout) are searched.
- The criteria are fixed (type, layer and name, as AutoCAD's default setting); the Settings option to choose them is not available yet.

See also

QSELECT · AI_SELALL · MATCHPROP · Selecting objects

STATUS —Status

Displays the drawing status: limits, extents and current settings.

- **Command:** `STATUS` · **Global name:** `_STATUS`
- Ribbon: Home › Utilities

Displays the drawing status in the text window (F2): how many objects it has, its limits, the area used by the model, the insertion base, the snap and grid spacing, the current layer, color and linetype, the elevation, the status bar modes and the running object snaps.

Usage

1. Enter STATUS. The report is shown on the command line and in the text window.

Notes

- "Model space uses" is the extent of the objects in model space; with none, it says *Nothing*.
- Unlike AutoCAD, free disk space and memory are not shown.

See also

LIMITS · TIME · DBLIST · SETVAR

TIME —Time

Displays the dates and editing time of the drawing.

- **Command:** `TIME` · **Global name:** `_TIME`
- Ribbon: Home › Utilities

Displays the dates and times of the drawing: the current time, when it was created and last saved, the total editing time and the user elapsed timer.

Usage

1. Enter TIME. The report is written on the command line:

```
Times for this drawing:
```

2. Choose an option or press Enter to finish.

```
Enter option [Display/ON/OFF/Reset]:
```

Options

- **Display** (D): writes the report again with the current time.

- **ON** / **OFF**: starts or stops the user elapsed timer (the USRTIMER variable of the drawing).
- **Reset** (R): resets the timer to zero.

Notes

- The dates are saved in the drawing (TDCREATE and TDUPDATE). The creation date is that of the first save in AutoCAD or LinceCAD; the last update changes when you save.
- The editing time adds the previous sessions and this one.

See also

STATUS · DBLIST

UNITS —Units

Sets the drawing units and precision.

- **Command:** `UNITS` · **Global name:** `_UNITS` · **Alias:** `UN`, `DDUNITS`
- Ribbon: Manage › Drawing
- Menu: Format

Sets how the lengths and angles of the drawing are written and the unit used to scale inserted content. The choice is saved in the drawing and used by the command line, the status bar, the Properties palette and the inquiry commands (DIST, AREA, LIST…).

Usage

1. Enter UNITS (or UN) or click its button. The **Drawing Units** dialog box opens.
2. Choose the formats. The **Sample Output** box shows how a coordinate and an angle will look.
3. Click **OK**. The change can be undone with UNDO.

The dialog box

- **Length**: **Type:** (Architectural, Decimal, Engineering, Fractional or Scientific) and **Precision:**, from 0 to 8 decimal places (or the smallest fraction).
- **Angle**: **Type:** (Decimal Degrees, Deg/Min/Sec, Grads, Radians or Surveyor's Units), **Precision:** and the **Clockwise** check box.
- **Insertion scale**: Units to scale inserted content: millimeters, meters, inches, feet and the rest of AutoCAD's units, or Unitless.
- **Base angle direction (degrees):** 0 = East, 90 = North, 180 = West, 270 = South.

Notes

- They correspond to the system variables LUNITS, LUPREC, AUNITS, AUPREC, ANGDIR, ANGBASE and INSUNITS, which can also be changed with SETVAR.
- The decimal separator (comma or point) belongs to the application, not to the drawing: choose it in OPTIONS.
- Changing the units does not change the size of anything: only how the numbers are written.

See also

SETVAR · OPTIONS · Coordinates and input

Tools

AIACTIONS — AI Actions

Shows the changes requested by AI assistants (MCP) so you can accept or reject them, and the log of API calls.

- **Command:** `AIACTIONS` · **Global name:** `_AIACTIONS` · **Alias:** `AIA`
- Ribbon: Manage › Applications

Opens the **AI Actions** palette: the changes requested by AI assistants connected through MCP, so you can accept or reject them, and the log of API calls.

Usage

1. Enable automation in OPTIONS, **Automation** tab, with **Confirm the changes of AI assistants (AI Actions palette)**.
2. When an assistant wants to change the drawing (create or delete entities, run a command, save…), the palette opens by itself and shows the client, the method and its data.
3. Click **Accept** to run it or **Reject** to deny it; the assistant gets the answer. **Accept all in this session** accepts what is pending and whatever arrives until LinceCAD is closed; **Ask again** undoes that.

Enter AIACTIONS (or AIA) to show or hide the palette at any time.

Notes

- Reads (querying entities, capturing the view…) are not asked: only changes.
- Each accepted action is one UNDO step.
- The log keeps the last 500 calls of every client (API and MCP), with the time (UTC) and the error, if any.
- Without a window (`lince --servidor`, `lince --mcp`) nothing is asked: the read-only mode and the allowed folders apply.

See also

Automation (API and MCP) · OPTIONS · UNDO

ALIASEDIT — Edit Aliases

Edits the command aliases (lince.pgp file, with the acad.pgp syntax).

- **Command:** `ALIASEDIT` · **Global name:** `_ALIASEDIT`
- Ribbon: Manage › Customization

Edits the user command aliases: your own abbreviations, typed on the command line instead of the full name. They are saved in `lince.pgp`, with the same syntax as AutoCAD's `acad.pgp`, and an AutoCAD one can be imported as it is.

Usage

1. Enter ALIASEDIT or click the button. **Customize** opens on the **Aliases** tab, with the file path at the top.
2. To add one, type the alias and the command (its English or Spanish name; the suggestions complete it) and click **Add**. If the alias already existed, it now points to the new command.
3. To remove one, click ? on its row.
4. **Import acad.pgp…** adds the aliases of a `.pgp` file; existing ones with the same name are replaced.

Changes apply at once, without restarting the program.

The lince.pgp file

One line per alias, `ALIAS, *COMMAND`; whatever follows `;` is a comment:

```
; my aliases
LL,      *LINE
CC,      *CIRCLE
```

The external command lines of acad.pgp (those without `*`) are ignored. If an alias is repeated, the last one wins.

Notes

- A user alias wins over a built-in alias with the same letters, but never over a command name: `LINE` is still LINE even if there is a `LINE` alias.
- With `_` or `.` in front (`_LL`) the user aliases are not looked up: the command is searched by its global name.
- **Built-in aliases**, at the bottom of the tab, lists the aliases that come with the program.
- The file is in the configuration folder (`%APPDATA%\lince`; on Linux, `~/.config/lince`) and is read when the program starts.

See also

CUI · Command aliases · Keyboard shortcuts

APPLOAD — Load Application

Loads AutoLISP routines (.lsp) and chooses the ones loaded at startup.

- **Command:** `APPLOAD` · **Global name:** `_APPLOAD` · **Alias:** `AP`
- Ribbon: Manage › Applications

Loads AutoLISP routines (`.lsp`) and manages the ones loaded at startup and the load security. Once loaded, a routine that defines `C:NAME` is used by entering NAME on the command line.

Usage

1. Enter APPLOAD (or AP) or click its button. The **Load Applications (APPLOAD)** dialog box opens.
2. Click **Load a .lsp file…** and choose the file. It is loaded at once.
3. Close the dialog box. Changes to the lists and to the security are saved immediately.

The dialog box

- **Startup suite (loaded at startup):**: files loaded every time LinceCAD starts. Add them with **Add to the startup suite…** and remove them with ?.
- **Load security (SECURELOAD):**: Load from any folder without asking, Ask outside the trusted folders or Load only from the trusted folders. It is equivalent to AutoCAD's SECURELOAD variable.
- **Trusted folders (the first one is the user folder):**: the folders from which files are loaded without asking. Add them with **Add trusted folder…**.

Notes

- Outside the trusted folders, and with the intermediate security, the program asks for permission before loading: a `.lsp` file can run commands on the drawing.
- On the command line, anything that starts with `(` is evaluated as AutoLISP: `(load "routine.lsp")` also loads.
- Only plain-text `.lsp` files are loaded; `.fas`, `.vlx`, `.arx` and `.dll` files are not.

See also

SCRIPT · OPTIONS · AutoLISP

CUI —Customize

Customizes the ribbon, toolbars, keyboard shortcuts and aliases.

- **Command:** `CUI` · **Global name:** `_CUI`
- Ribbon: View › Interface
- Ribbon: Manage › Customization

Opens the **Customize** window, to adapt the interface: which commands each ribbon panel and each toolbar of the Classic workspace has, your own keyboard shortcuts and command aliases. It is a reduced version of AutoCAD's CUI editor.

Usage

1. Enter CUI or click the button (it is also in the menu of the ? gear on the status bar).
2. Choose the tab and make the changes: they apply at once and are saved automatically.

Ribbon and Toolbars tab

- **Panel or toolbar:** chooses the ribbon panel (tab › panel) or the toolbar. Those with changes are marked `*`.
- The list shows its commands in order: ? (Remove) removes one, ? and ? (Move up, Move down) move it.
- **Add command:**: type part of a command name and click one of the suggestions to add it at the end.
- **Reset** returns the panel or toolbar to its built-in commands.

Keyboard Shortcuts tab

- Type the keys (`Ctrl+Shift+L`, `Alt+G`, `F6`...) and the command, and click **Add**. The shortcut must include Ctrl or Alt, or be a function key, so that it does not take keys away from the command line.
- A user shortcut wins over a built-in one with the same keys. ? removes it.
- **Built-in shortcuts** lists the shortcuts that come with the program.

Aliases tab

The same list that ALIASEDIT opens: the user aliases, saved in `lince.pgp`.

Notes

- The ribbon, toolbars and shortcuts are saved in `personal.txt` and the aliases in `lince.pgp`, both in the configuration folder (`%APPDATA%\lince`; on Linux, `~/.config/lince`).
- Only existing panels and toolbars can be changed: no new tabs, panels, toolbars or menus.
- Shortcut keys are typed as text; they are not captured by pressing them.

See also

ALIASEDIT · WORKSPACE · The user interface · Keyboard shortcuts

DSETTINGS —Drafting Settings…

Snap, grid, polar tracking, object snap and dynamic input.

- **Command:** `DSETTINGS` · **Global name:** `_DSETTINGS` · **Alias:** `DS`, `SE`, `DDRMODES`
- Ribbon: Manage › Drawing

Opens the **Drafting Settings** dialog box with the drafting aids, as in AutoCAD. Changes apply at once and are the same as those of the status bar and SETVAR.

Usage

1. Enter DSETTINGS (or DS or SE) or click the button.

```
Drafting Settings: use the dialog box; Enter closes it.
```

2. Choose the tab:

- **Snap and Grid**: Snap On (F9), Grid On (F7), snap and grid spacing in X and Y, and how many lines apart the major one is.
- **Polar Tracking**: Polar Tracking On (F10), Object Snap Tracking On (F11) and the Increment angle.
- **Object Snap**: Object Snap On (F3) and the 13 object snap modes.
- **Dynamic Input**: Enable Dynamic Input (F12).

3. Click **Close** or press Enter.

Notes

- Variables: SNAPMODE, GRIDMODE, AUTOSNAP, POLARANG, OSMODE and DYNMODE.
- It does not include the isometric snap style or the object snap tracking options: for isometric use ISODRAFT.

See also

GRID · SNAP · OSNAP · ORTHO · OPTIONS

ELEV —Elevación

Elevación y altura de objeto por defecto.

- **Comando:** ELEV · **Nombre global:** _ELEV
- Cinta: Vista › Coordenadas

Fija la elevación (Z) y la altura de objeto por defecto de las entidades nuevas, como ELEV de AutoCAD.

Uso

1. Escriba ELEV.

```
Precise la nueva elevación por defecto:
```

2. Indique la nueva elevación (Intro deja la actual).

```
Precise la nueva altura de objeto por defecto:
```

3. Indique la nueva altura de objeto (THICKNESS).

Notas

- La elevación levanta en Z las entidades planas nuevas (líneas, círculos, arcos, polilíneas, textos, puntos…); la altura de objeto les da un «muro» vertical de esa altura, que se ve en las vistas 3D.
- Se guardan en las variables ELEVATION y THICKNESS; con los dos a 0 (lo normal) el dibujo 2D queda igual.
- No afecta a las entidades que ya existen: para cambiarlas, use CHANGE o la paleta de propiedades.

Véase también

CHANGE · UCS · -VIEW

ISODRAFT —Isometric Drafting

Turns isometric drafting on in an isoplane, or back to orthographic.

- **Command:** `ISODRAFT` · **Global name:** `_ISODRAFT` · **Alias:** `ISOD`
- Ribbon: View › Coordinates
- Menu: Tools

Turns on isometric drafting in one of its three planes, or goes back to orthographic drafting. In isometric mode you draw an axonometric view of an object in a flat drawing: edges run at 30°, 90° and 150°, depending on the plane (left, top or right) you draw in.

Usage

1. Enter ISODRAFT (or ISOD) or click the button.
2. Choose the plane; Enter, the top one.

```
Enter an option [Orthographic/Left/Top/Right] <Top>:
```

Options

- **Left** (L), **Top** (T) and **Right** (R): turn on isometric drafting in that plane.
- **Orthographic** (O): goes back to normal drafting, with horizontal and vertical axes.

With isometric drafting

- Ortho (F8), the grid, snap (F9) and the crosshairs follow the two axes of the current plane.
- F5 or Ctrl+E switch to the next plane (left, top, right); so does ISOPLANE.
- The **ISO** button on the status bar shows the plane and, with a click, turns the mode on or off.
- ELLIPSE offers the **Isocircle** option: the ellipse that represents a circle in the current plane, given by its center and its radius or diameter.

```
Specify radius of isocircle or [Diameter]:
```

Notes

- The mode is saved in the drawing as in AutoCAD, in the SNAPSTYL and SNAPISOPAIR variables.
- It is an aid for drawing in 2D: it does not create a 3D model.

See also

ISOPLANE · ELLIPSE · ORTHO · SNAP · Object snaps and precision

ISOPLANE —Isoplane

Specifies the current isometric plane.

- **Command:** `ISOPLANE` · **Global name:** `_ISOPLANE`
- Ribbon: View › Coordinates

Chooses the current isometric plane: left, top or right. Ortho, the grid, snap and the crosshairs follow the axes of that plane. It is the same as pressing F5 or Ctrl+E.

Usage

1. Enter ISOPLANE.
2. Choose the plane; Enter switches to the next one.

```
Enter isometric plane setting [Left/Top/Right] <next>:
```

3. The program shows the chosen plane.

```
<…>
```

Options

- **Left** (L): axes at 90° and 150°.
- **Top** (T): axes at 30° and 150°.
- **Right** (R): axes at 30° and 90°.

Notes

- The plane is saved in the SNAPISOPAIR variable. If isometric drafting is not on, the plane is kept for when it is turned on with the **ISO** button on the status bar.

See also

ISODRAFT · ELLIPSE · Object snaps and precision

OPTIONS —Options…

Application preferences.

- **Command:** `OPTIONS` · **Global name:** `_OPTIONS` · **Alias:** `OP`, `PREFERENCES`
- Ribbon: Manage › Drawing
- Menu: LinceCAD
- Menu: Tools

Opens the application preferences: the interface language, how numbers are written, the QNEW template and the start page. They belong to LinceCAD, not to the drawing: they apply to every drawing and are kept when the program closes.

Usage

1. Enter OPTIONS (or OP) or click its button. The **Options** dialog box opens.
2. Change what you need. The changes show at once in the whole window.
3. Click **OK** (or **Apply**, which saves without closing). **Cancel** goes back to the previous preferences.

Interface tab

- **Language**: the language of the menus, ribbon, dialog boxes, command line and help (Spanish or English). Global command names (`_LINE`) work in any language.
- **Country**: the country you work in. Online maps are framed to its extent (or, if it does not fit, to its capital) when the view lies outside the layer. With **Automatic (from the system)** it is taken from the system regional settings or the drawing coordinate system; if it cannot be told, LinceCAD asks once (with the **Remember my choice** box) and stores the answer here.
- **Theme**: CAD Optimus (default: blue-grey dark, AutoCAD style), Follow system, Light or Dark.
- **Number format**: Automatic (by language), Decimal comma (1234,56) or Decimal point (1234.56).
- **Decimal places**: from 0 to 8, the decimal places the interface uses to show numbers.
- **Example**: a coordinate written with the chosen format.

Open and Save tab

- **QNEW template**: the template QNEW starts drawings with, without asking (also the startup one). With **None (ask)**, QNEW asks like NEW. The **…**button chooses another one.

- **Start page**: shows the start page (templates, open and recent files) while nothing has been drawn.

Display tab

Display lineweight (LWDISPLAY), Fill hatches and solids (FILLMODE), Quick text (QTEXTMODE), Show the control polygon of splines (SPLFRAME) and Display transparency (TRANSPARENCYDISPLAY).

Drafting tab

AutoSnap (AUTOSNAP): Marker, Magnet and Display AutoSnap tooltip. Selection: Select a group with one of its objects (PICKSTYLE) and Delete source objects when creating boundaries and regions (DELOBJ). For snap, grid and object snap modes use DSETTINGS.

Notes

- Preferences are saved in `%APPDATA%\lince\preferencias.toml` (on Linux, `~/.config/lince`). The first time, the system language is used if there is a catalog for it.
- Drawing measurements (DIST, AREA, LIST, dimensions) follow the format and precision of UNITS.
- There are five tabs: Interface, Open and Save, Automation (the local API and the MCP server), Display and Drafting; these two change drawing variables (like SETVAR) and are not application preferences. AutoCAD's other tabs (Plot and Publish, System…) are not available. The AutoLISP preferences are in APPLOAD.

See also

UNITS · SETVAR · APPLOAD · Options

SCRIPT — Run Script

Runs a command script (.scr).

- **Command:** `SCRIPT` · **Global name:** `_SCRIPT` · **Alias:** `SCR`
- Ribbon: Manage › Applications
- Menu: Tools

Runs a script: a `.scr` text file whose lines are typed on the command line one after another, as if the user entered them. Use it to repeat tasks (setting up layers, purging, changing variables…) in one or more drawings.

Usage

1. Enter SCRIPT (or SCR) or click its button.
2. Choose the `.scr` file in the dialog box. (In a script that calls another one, its path is typed.)

```
Enter script file name:
```

3. The lines run in order. Meanwhile the keyboard does not reach the command line.

How a script is written

- Every space and every end of line counts as pressing Enter, as in AutoCAD: `CIRCLE 0,0 5` draws a circle.
- When a command expects free text (the text of TEXT, a layer or style name…), the rest of the line is taken whole, with its spaces.
- In other prompts, a value with spaces is written between quotes: `"Exterior wall"`.
- Lines that start with `;` are comments.
- A blank line is an Enter (it repeats the last command or ends the active one).

```
; Current layer, linetype scale and purge
CLAYER Exterior wall
LTSCALE 50
```

```
-PURGE A * N
```

Notes

- Commands with a dialog box are used in their command-line version (-PURGE, -LAYER, -INSERT…).
- AutoCAD's RSCRIPT, DELAY and RESUME are not available yet.

See also

SETVAR · APPLOAD · -PURGE · AutoLISP

SETVAR —System Variables

Lists or changes a system variable.

- **Command:** SETVAR · **Global name:** _SETVAR · **Alias:** SET
- Ribbon: Manage › Drawing
- Menu: Tools

Shows or changes the value of a system variable of the drawing: units, current layer, linetype scale, fillet radius… The variable name can also be entered directly, as if it were a command.

Usage

1. Enter SETVAR (or SET).

```
Enter variable name or [?]:
```

2. Enter the variable name, or ? to see the list of all of them with their values (in the text window, F2).
3. Enter the new value. Enter keeps the current value, shown between < >.

```
Enter new value for … <…>:
```

Shortcut: enter the variable name on the command line (for example LTSCALE) and go straight to step 3.

Variables

ANGBASE · ANGDIR · ATTMODE · AUNITS · AUPREC · CECOLOR · CELTSCALE · CELTYPE · CHAMFERA · CHAMFERB · CLAYER · CTABLESTYLE · DIMSCALE · DIMSTYLE · ELEVATION · FILLETRAD · FILLMODE · IMAGEFRAME · INSBASE · INSUNITS · LTSCALE · LUNITS · LUPREC · LWDISPLAY · MIRRTEXT · PDMODE · PDSIZE · PICKSTYLE · PLINEWID · SPLINESEGS · SPLINETYPE · TEXTSIZE · TEXTSTYLE · THICKNESS

Notes

- A value out of range or of the wrong type is rejected with a warning and the variable does not change.
- Variables that name something (CLAYER, CELTYPE, TEXTSTYLE, DIMSTYLE, CTABLESTYLE) require it to exist: they do not create the layer or the style. Names with spaces are accepted.
- The change is saved in the drawing and undone with UNDO.
- Only these variables are available; AutoCAD has many more. See System variables.

See also

UNITS · OPTIONS · System variables

SHEETSET — Sheet Set

Groups layouts from several drawings as numbered sheets of a set.

- **Command:** `SHEETSET` · **Global name:** `_SHEETSET` · **Alias:** `SSM`
- Ribbon: Output › Plot
- Menu: File

Opens the **Sheet Set** palette, to group layouts from several drawings as numbered "sheets" of a set, open them and batch-plot them to PDF. It is the equivalent of the AutoCAD Sheet Set Manager.

Usage

1. Type SHEETSET (or SSM) or click the button: the palette opens and can be docked to a side.
2. Click **Add drawings…** to add other drawings (their Model and layouts are added) or **Add current** for the open drawing (it must be saved).
3. Edit each sheet's number and title, reorder them with ? and ?, or remove them with ?.
4. **Number** numbers the sheets from 1 to n in list order.
5. Double-click a sheet to open its drawing and switch to its layout.
6. **Publish set to PDF…** loads the sheets into PUBLISH to batch-plot them to PDF.

Saving the set

- **Save** and **Save as…** save the set to a `.dst` file.
- The LinceCAD `.dst` is a plain-text file of its own (drawing paths are saved relative to its folder when they are inside it). AutoCAD `.dst` files, which use a proprietary format, are neither read nor written.

Notes

- Sheets from other drawings are plotted with their page setups; as in PUBLISH, their external references are not loaded when publishing.

See also

PUBLISH · PAGESETUP · Layouts

UCS — UCS

Manages user coordinate systems.

- **Command:** `UCS` · **Global name:** `_UCS`
- Ribbon: View › Coordinates
- Menu: Tools

Changes the user coordinate system (UCS): an origin and X and Y axes of your own, moved or rotated with respect to the world coordinate system (WCS). It is useful to draw aligned with a rotated building or to measure from a corner of the plot.

Usage

1. Enter UCS or click the button.
2. Specify the new origin, or choose an option. Enter goes back to the world system.

```
Specify origin of UCS or [NAmed/OBject/Previous/World/Z] <World>:
```

3. Specify a point on the positive X axis; Enter keeps the axes as they were and only changes the origin.

```
Specify point on X-axis or <accept>:
```

4. Press Enter (or specify any point) to accept.

```
Specify point on the XY plane or <accept>:
```

Options

- **NAmed** (NA): saves the current UCS with a name, restores a saved one or deletes it; `?` lists them.

```
Enter an option [Restore/Save/Delete/?]:
Enter name to save current UCS or [?]:
```

- **OBject** (OB): aligns the UCS with an object: a line (origin at the endpoint nearest to the selected point), a circle or an arc (origin at the center), a polyline (its first segment), a text or a block (its point and rotation), an ellipse or a point.

```
Select object to align UCS:
```

- **Previous** (P): goes back to the previous UCS. The last ten of the session are remembered.
- **World** (W): goes back to the world system.
- **Z**: rotates the UCS about its origin; Enter rotates 90°.

```
Specify rotation angle about Z axis <90>:
```

- **X** and **Y**: rotate the UCS about its X or Y axis (Enter: 90°).

```
Specify rotation angle about X axis <90>:
```

- **ZAxis**: new origin and a point on the positive Z axis; X and Y come from AutoCAD's arbitrary axis algorithm. Type `x,y,z` to give the Z.

```
Specify new origin point <0,0,0>:
```

- **View**: UCS parallel to the screen, with the current origin.
- **Move**: changes the origin without rotating the axes; **Zdepth** moves it along the Z axis.

```
Specify new origin point or [Zdepth] <0,0,0>:
```

- **Orthographic**: the six standard UCSs (Top, Bottom, Front, Back, Left and Right) with the current origin.

```
Enter an option [Top/Bottom/Front/BAck/Left/Right] <Top>:
```

- **Apply**: applies the current UCS to the viewports (here, the only active one).
- **Face**: not available: it would need the face of a 3D solid to be selected.

Notes

- 2D drawing works in the XY plane: a UCS rotated about X or Y (X, Y, ZAxis, View, Orthographic) is saved and shown in the icon, the 3D view and AutoCAD, but 2D commands treat it as the World UCS.
- With a UCS, the coordinates you type and those shown in the status bar are in the UCS. To type a point in the world system, put an asterisk in front: `*100,50` or `@*20,0`.
- Ortho, the grid, snap and the crosshairs are aligned with the UCS, and so are RECTANG and the rotation of TEXT. Polar tracking, dynamic input and the angles of other commands stay on the world axes.
- Only UCSs in the plane: the 3-point option does not tilt the XY plane.
- It is saved as in AutoCAD (UCS table and the UCSORG, UCSXDIR and UCSNAME variables): AutoCAD opens the drawing with the same UCS.
- PLAN rotates the view so that the UCS X axis is horizontal. The icon is controlled with UCSICON.

See also

UCSMAN · PLAN · UCSICON · Coordinates and data entry

UCSICON —UCS Icon

Controls the visibility and placement of the UCS icon.

- **Command:** `UCSICON` · **Global name:** `_UCSICON`
- Ribbon: View › Coordinates
- Menu: Tools

Shows or hides the UCS icon (the X and Y axis arrows in the corner of model space) and decides whether it is drawn at the UCS origin.

Usage

1. Enter UCSICON or click the button.
2. Choose an option; Enter shows it.

```
Enter an option [ON/OFF/All/Noorigin/ORigin/Selectable/Properties] <ON>:
```

Options

- **ON** (ON) and **OFF** (OFF): shows or hides the icon.
- **ORigin** (OR): draws it at the UCS origin, if the origin is visible on screen; otherwise, at the lower left.
- **Noorigin** (N): always draws it at the lower left.
- **All** (A): in AutoCAD it applies the option to all active viewports; LinceCAD has a single model viewport, so it already applies to all.
- **Selectable** (S): toggles the setting. Here the icon cannot be selected to move it with grips.
- **Properties** (P): the **UCS Icon Properties** dialog with the **Icon size** (5 to 95 %), the **Line width** (1 to 3) and an **Own icon color**. There is no 2D/3D style or cone: the icon is 2D in plan views and 3D in 3D views.

Notes

- The setting is the UCSICON variable and it is saved with the drawing, as in AutoCAD. It can also be changed in UCSMAN.
- In the world system the icon has a small square where the axes meet; with a UCS, it does not.

See also

UCS · UCSMAN · Coordinates and data entry

UCSMAN —Named UCS

Manages defined user coordinate systems.

- **Command:** `UCSMAN` · **Global name:** `_UCSMAN` · **Alias:** `UC`, `DDUCS`, `DDUCSP`
- Ribbon: View › Coordinates
- Menu: Tools

Opens the **Named UCSs** dialog box, which lists the world system and the drawing's named UCSs to make them current, rename them or delete them, and sets up the UCS icon.

Usage

1. Enter UCSMAN (or UC) or click the button.
2. At the top you see **Current UCS: …** In the list, the current one is marked with ?.
3. Click a UCS in the list and use the buttons:

- **Set Current**: makes it current (the world system too).
- **Delete**: deletes it from the drawing.
- **Rename**: gives it the name typed in the box next to it.

4. At the bottom, **Display the UCS icon** and **Display at UCS origin point** change UCSICON.
5. Click **Close**.

Orthographic UCSs and settings

- **Orthographic UCSs (relative to World):** the six standard UCSs (Top, Bottom, Front, Back, Left, Right): a button makes the chosen one current.
- **Update view to Plan when UCS is changed (UCSFOLLOW)** puts the view in plan each time the UCS changes; **Save UCS with viewport (UCSVP)** is stored as a setting (LinceCAD has a single model viewport).

Notes

- Named UCSs are created with the Named > Save option of UCS.
- Names follow the rules for block names. The world system cannot be renamed or deleted.
- Each change can be undone with UNDO.

See also

UCS · UCSICON · PLAN · Coordinates and data entry

WORKSPACE — Workspace

Sets, saves, renames or deletes workspaces.

- **Command:** `WORKSPACE` · **Global name:** `_WORKSPACE`
- Ribbon: View › Interface

Opens the **Workspaces** window, which switches workspaces and saves, renames or deletes user workspaces. A workspace remembers how the interface is laid out:

- the ribbon or the menus and toolbars of the Classic workspace, and the ribbon state (full, tabs only or panel titles);
- the command line, docked or floating, and its transparency;
- for each palette (Properties, Layers, Blocks, Tool Palettes, DesignCenter and external references): whether it is open, whether it floats or is docked left or right, whether it auto-hides, its width and its position.

Usage

1. Enter WORKSPACE or click the button. The window lists the workspaces; the current one is marked **(current)**.
2. Choose one and click **Set Current** (or double-click it): the interface changes at once.
3. To save the current layout, type a name in **Name:** and click **Save Current As…** or press Enter.

Buttons

- **Set Current**: applies the selected workspace and makes it current.
- **Delete**: deletes the selected workspace. The built-in ones and the current one cannot be deleted.
- **Rename**: gives the selected workspace the name typed in **Name:**. Built-in ones cannot be renamed.
- **Save Current As…**: saves the current layout with the typed name; a workspace with that name is replaced.

Notes

- Two are built in: **Drafting & Annotation** and **Classic**. You can overwrite them by saving with their name.

- The ? gear on the status bar shows the current workspace: its menu lists them all to switch with one click, and also has this window, save current and the interface customization.
- Workspaces and which one is current are saved in `espacios.txt`, in the configuration folder (`%APPDATA%\lince`; on Linux, `~/.config/lince`), and restored when the program starts.
- Until they are overwritten, the built-in workspaces have every palette closed and floating.

See also

WSSAVE · CUI · The user interface

WSSAVE —Save Current As…

Saves the current interface layout as a workspace.

- **Command:** `WSSAVE` · **Global name:** `_WSSAVE`
- Ribbon: View › Interface

Saves the current interface layout as a named workspace, so you can return to it whenever you need: the ribbon or the Classic workspace, the ribbon state, the command line and the open palettes with their place and size.

Usage

1. Arrange the interface the way you want it: open palettes, dock them, minimize the ribbon, float the command line...
2. Enter WSSAVE or choose **Save Current As…** in the menu of the ? gear on the status bar. The **Workspaces** window opens with the cursor in **Name:**.
3. Type the name and press Enter (or click **Save Current As…**).

The saved workspace becomes current, and the status bar says so:

```
Workspace saved: …
```

Notes

- A workspace with the same name is replaced without asking. With the name of a built-in one (**Drafting & Annotation**, **Classic**), that one is overwritten.
- The name cannot be empty, start with `@` or contain brackets.
- It is saved in `espacios.txt`, in the user configuration folder: it applies to every drawing.

See also

WORKSPACE · The user interface

File

-PLOT — Plot (command line)

Plots the drawing to PDF, PNG or SVG asking at the command line.

- **Command:** `-PLOT` · **Global name:** `_-PLOT` · **Alias:** `-PRINT`

Command-line version of PLOT, with the same questions and in the same order as AutoCAD. It is the one scripts use.

Usage

1. Enter -PLOT.

```
Detailed plot configuration? [Yes/No] <No>:
```

2. With **No**, it asks for the layout, a saved page setup, the output device if the layout has none, the file name, whether to save the changes to the page setup and whether to proceed.
3. With **Yes**, it asks in addition for the paper, its units, the orientation, the area (with **Window**, the two corners; with **View**, the name of a view of the drawing), the scale, the offset, the plot style table, the lineweights and, in a layout, the paper space options.

```
Enter paper size or [?] <…>:
Enter plot scale (Plot …=Drawing Units) or [Fit] <…>:
Enter file name <…>:
```

Notes

- The file extension (`.pdf`, `.png` or `.svg`) decides the format.
- Only the active layout is plotted: change to its tab first.
- The shade plot settings other than as displayed and hidden are not available.
- If the plot stamp is on (PLOTSTAMP), it appears on the sheet.
- Plotting is synchronous: there is no background plotting.

See also

PLOT · PUBLISH · PAGESETUP · Layouts

-PLOTSTAMP — Plot Stamp (command line)

Sets the plot stamp on the command line.

- **Command:** `-PLOTSTAMP` · **Global name:** `_-PLOTSTAMP`

Sets the plot stamp on the command line. It is the dialog-less version of PLOTSTAMP.

Usage

```
Enter an option [ON/OFF/Fields/User fields/loG file/Location/Text properties/Units] :
```

Enter ends and saves the stamp.

See also

PLOTSTAMP

-PUBLISH —Publish (command line)

Publishes a sheet list to a multi-page PDF.

- **Command:** `-PUBLISH` · **Global name:** `_-PUBLISH`

Command-line version of PUBLISH. It only asks for the name of the sheet list, as AutoCAD does.

Usage

1. Enter -PUBLISH.

```
Enter name of sheet list <>:
```

2. Type the path of an AutoCAD sheet list (`.dsd`) or of a list saved by PUBLISH (`.dsl`). The sheets are plotted, with their page setups, to a single multi-page PDF.

Notes

- In a `.dsd`, the sheets are the `[DWF6Sheet:...]` sections (keys `DWG` and `Layout`) and the PDF is the `DWF` key of `[Target]` (or its `OUT` folder with the name of the first drawing). Relative paths start in the folder of the list. A `.dsl` is published next to the list, with its name and the `.pdf` extension.
- Only the single multi-sheet PDF is available: one PDF per sheet and DWF files are not.

See also

PUBLISH · -PLOT · Layouts

AUDIT —Audit

Evaluates the integrity of the drawing and fixes the safe errors.

- **Command:** `AUDIT` · **Global name:** `_AUDIT`
- Ribbon: Manage › Drawing
- Menu: File

Checks the integrity of the open drawing and, if asked, fixes the errors that can be repaired safely. Useful with drawings that come from other programs or that cause problems when saving.

Usage

1. Enter AUDIT or click its button.
2. Say whether you want the errors fixed.

```
Fix any errors detected? [Yes/No] <N>:
```

3. The command line shows how many records were checked, the errors and warnings found and the first problems; with Yes, also how many repairs were made and how many errors remain.

Options

- **Yes** (Y): repairs what is safe: indexes, null handles, owners of block entities and the handle seed.
- **No** (N, or Enter): only reports; the drawing does not change.

Notes

- Doubtful issues (duplicate handles, references to objects that do not exist) are never fixed: they are only reported.

- Repairs are undone with UNDO. Save the drawing to keep them.
- If the file does not even open, use RECOVER.

See also

RECOVER · PURGE · QSAVE

CLOSE —Close

Closes the current drawing.

- **Command:** `CLOSE` · **Global name:** `_CLOSE`
- Menu: LinceCAD
- Menu: File

Closes the current drawing and moves to the drawing in the next tab. If it is the only one open, an empty drawing takes its place.

Usage

1. Enter CLOSE, click the **×** on its tab or choose **Close** in the tab's shortcut menu.
2. If the drawing has unsaved changes (its tab shows `*`), the program asks:

```
Save changes to …?
```

3. Answer **Yes** to save and close, **No** to close without saving or **Cancel** to keep working.

Notes

- With Yes, an unnamed drawing opens the SAVEAS dialog box; if that is cancelled, the drawing does not close.
- Closing deletes its automatic save.
- The shortcut menu of a tab also has **Close All Other Drawings**, which closes all but that one.

See also

CLOSEALL · QSAVE · QUIT

CLOSEALL —Close All

Closes all open drawings.

- **Command:** `CLOSEALL` · **Global name:** `_CLOSEALL`
- Menu: LinceCAD

Closes all open drawings, one after another. At the end an empty drawing remains and LinceCAD stays open.

Usage

1. Enter CLOSEALL or choose it in the Window menu.
2. For each drawing with unsaved changes, the program moves to its tab and asks:

```
Save changes to …?
```

3. Answer **Yes** (save), **No** (discard the changes) or **Cancel**.

Notes

- If you cancel on a drawing, that one and the remaining ones stay open.
- To close the program as well, use QUIT.

See also

CLOSE · QUIT · QSAVE

CONVERTCTB —Convert CTB

Converts a color-dependent plot style table (.ctb) into a named one (.stb).

- **Command:** `CONVERTCTB` · **Global name:** `_CONVERTCTB`

Converts a color-dependent plot style table (.ctb) into a named plot style table (.stb).

Usage

1. Type CONVERTCTB.

```
Enter the color-dependent plot style table (.ctb) to convert:
```

2. Choose the .ctb file (or type its path, or its name if it is in the LinceCAD plot styles folder).

Notes

- The .stb is created with the same name in the folder of the .ctb. If it already exists it is not overwritten.
- It has the "Normal" style and one style per pen (Color_1, Color_2...) keeping its color, grayscale, screening and lineweight.
- To make a drawing use the new table, see CONVERTPSTYLES.

See also

CONVERTPSTYLES · PAGESETUP · PLOT

CONVERTPSTYLES —Convert Plot Styles

Makes the drawing use named or color-dependent plot styles.

- **Command:** `CONVERTPSTYLES` · **Global name:** `_CONVERTPSTYLES`

Makes the drawing use named or color-dependent plot styles, depending on the table you choose.

Usage

1. Type CONVERTPSTYLES.

```
Enter the plot style table (.stb: named; .ctb: color-dependent):
```

2. Choose an .stb (named styles) or a .ctb (color-dependent styles).

Notes

- All the layouts and the model use that table, objects go back to ByLayer and layers to "Normal" (with color-dependent styles they carry none).
- PSTYLEMODE becomes 0 (named) or 1 (color-dependent).

See also

CONVERTCTB · PAGESETUP · PLOT

DIGITALSIGN —Digital signature

Signs a drawing with an integrity stamp and validates it.

- **Command:** `DIGITALSIGN` · **Global name:** `_DIGITALSIGN`
- Ribbon: Output › Export

Signs a drawing with a digital integrity signature and later checks whether the file has changed since it was signed.

Usage

1. Type FIRMADIG or click **Digital signature**.
2. Choose Sign or Validate.

```
Sign or validate the signature? [Sign/Validate] <Validate>:
```

3. Pick the DWG or DXF file in the file dialog box (with FILEDIA set to 0, type its path).

```
Drawing to sign or validate (DWG or DXF):
```

4. When signing, type the signer name (or press Enter for the system user).

```
Signer name (or Enter for the system user):
```

How it works

- Computes the file's **SHA-256** and signs it with **Ed25519**, using the user's own key pair, created the first time and stored in `<config>/lince/firma/` (the private key with restricted permissions, the public one in `ed25519.pub`).
- The stamp is saved next to the drawing, in `<name>.lincefirma`, with the signer, the date (UTC), the algorithm, the public key and the signature.
- Validating recomputes the SHA-256 and verifies the signature with the stamp's public key: valid, file changed, signed with a different key, or no signature.

Limitations

- It is a **self-signed integrity signature**: it proves the file has not changed and which key it came from, but not third-party trust. That needs a **certificate** (code signing), still pending by the user's decision.
- The signature is not embedded inside the DWG like AutoCAD's; it travels in the accompanying `.lincefirma` file.
- Sign the file already saved on disk: the signature is of the file bytes, not of the in-memory drawing.

See also

ETRANSMIT · QSAVE

DRAWINGRECOVERY —Drawing Recovery

Opens the drawing recovery manager.

- **Command:** `DRAWINGRECOVERY` · **Global name:** `_DRAWINGRECOVERY` · **Alias:** `DRM`

Opens the drawing recovery manager, like AutoCAD DRAWINGRECOVERY.

Usage

Enter DRAWINGRECOVERY. The list of drawings that LinceCAD saved when it closed after repeated errors appears.

Notes

- Each drawing can be opened; the list can also be left for later or the drawings discarded. It is the same dialog that appears at start-up after a failure.
- If there are none, the command line says so.

See also

RECOVER · AUDIT

ETRANSMIT —eTransmit

Packages the drawing and its dependent files in a ZIP.

- **Command:** `ETRANSMIT` · **Global name:** `_ETRANSMIT`
- Ribbon: Output › Export

Packages the drawing with every file it needs to look the same on another computer: external references, images, PDF underlays, SHX and TTF fonts and plot style tables. The result is a ZIP file with a transmittal report.

Usage

1. Save the drawing and enter ETRANSMIT. The **eTransmit** dialog lists the files in the package with the path they will have inside the ZIP; files that cannot be found show as *(not found)* and are left out.
2. Check **Keep the drawing paths as they are** if the packaged drawing should keep its original paths. Unchecked, the packaged drawing points to its files with paths relative to the package, which is usually what you need to send it.
3. Click **Create Transmittal…** and specify the ZIP name. The dialog shows the **Transmittal Report**.

What the package contains

- The drawing and, with its relative path, each dependent file. Files outside the drawing folder go into subfolders: `xrefs/`, `imagenes/`, `pdf/`, `fuentes/` and `estilos_trazado/`.
- **Transmittal report.txt**: each file with its path in the drawing and in the package, and the missing ones.

Notes

- The original drawing is not changed: rewriting the paths only affects the packaged copy.
- The drawing must be saved before packaging it.

See also

PUBLISH · XREF · SAVEAS · External references

EXPORT —Export

Exports the drawing to another format.

- **Command:** `EXPORT` · **Global name:** `_EXPORT` · **Alias:** `EXP`
- Ribbon: Output › Export
- Menu: LinceCAD
- Menu: File

Exports model space or the current layout to vector PDF or SVG, or to a PNG image. It uses the same **Plot** dialog box as PLOT: paper, area, scale and plot styles are chosen the same way.

Usage

1. Enter EXPORT (or EXP) or click its button. The **Plot** dialog box opens.
2. Set the paper, area, scale and plot style table (see PLOT). For a PNG, set the resolution in **PNG:**.
3. Click **OK**, enter the file name and choose its type. The extension decides the format:

- `.pdf`: vector PDF. Texts are drawn from their font (SHX strokes or TrueType outlines), so they look the same on any computer.
- `.svg`: vector drawing for web browsers and design programs.
- `.png`: image at the chosen resolution.

Notes

- To convert the drawing to DXF or another DWG version, use SAVEAS.
- Tables are exported to CSV with TABLEEXPORT and geographic data with MAPEXPORT.
- AutoCAD's WMF, BMP, STL and similar formats are not available.

See also

PLOT · SAVEAS · PREVIEW · File formats

EXPORTLAYOUT —Export Layout

Saves what the layout shows as a new drawing, in its model space.

- **Command:** `EXPORTLAYOUT` · **Global name:** `_EXPORTLAYOUT`

Saves what the active layout shows as a new drawing, in its model space.

Usage

1. In a layout, type EXPORTLAYOUT.

```
Enter the name of the new drawing:
```

2. Choose or type the .dwg file.

Notes

- What is drawn on the sheet and, for each viewport that is on, the model it shows go to the new drawing at paper size. Layers that are off, frozen or frozen in the viewport are not exported.
- What goes beyond the edge of a viewport is not clipped.
- On the Model tab the command does nothing.

See also

LAYOUT · CHSPACE · EXPORT

IMPORT —Import 3D mesh

Brings an STL, OBJ, glTF, DAE or FBX mesh into the drawing.

- **Command:** `IMPORT` · **Global name:** `_IMPORT` · **Alias:** `IMPORTARMALLA`, `IMP`
- Ribbon: Insert › Import

Brings a mesh from another program (STL, OBJ, glTF/GLB, COLLADA or FBX) into the drawing as mesh entities.

Usage

1. Type IMPORTAR3D (or IMPORTARMALLA).
2. Give the file; the format is detected from the extension.

```
Mesh file (STL, OBJ, glTF, DAE, FBX):
```

Formats

- **STL** (ASCII and binary), **OBJ**, **glTF/GLB**, **COLLADA (.dae)** and **FBX**.

Notes

- Faces are added to model space, on the layers the file carries.
- They are meshes (MESH/PFACE): they show in the 3D viewer and export again with STLOUT.

See also

STLOUT

NEW —New

Starts an empty drawing.

- **Command:** `NEW` · **Global name:** `_NEW` · **Shortcut:** Ctrl+N
- Menu: LinceCAD
- Menu: File

Starts a new drawing from a **template** (`.dwt`, or a `.dwg` used as a template), in a new tab. The drawings already open stay in their tabs.

Usage

1. Enter NEW, press Ctrl+N or click the **+** button to the right of the file tabs.
2. The **Select template** dialog box opens in the LinceCAD templates folder. Choose one and click Open.
3. A tab opens with an unnamed drawing (Drawing1, Drawing2…) that has the layers, styles, units and layouts of the template. Save it with QSAVE; the first time it asks for a name and a folder.

On the command line (scripts) the prompt is:

```
Enter template file name or [. (for none)] <…>:
```

Enter the name of a template in the folder (with or without `.dwt`) or a full path; Enter takes the one shown between `<` `>` and `.` starts an empty metric drawing without a template.

Standard templates

Template	Contents
`lince_iso.dwt`	LinceCAD metric template: millimetres, ISO-25 dimension style and a landscape A3 layout.
`lince_imperial.dwt`	LinceCAD inch template: the acad.dwt dimension style and an ANSI B (17 × 11 in) layout.

They are installed with the program in the `plantillas` folder, next to the executable. Any drawing can be saved as a template with SAVEAS › DWT.

Notes

- If the current drawing is untouched (no name and nothing drawn), the new one takes its tab.
- To always start with the same template without being asked, choose it in OPTIONS › Open and Save and use QNEW (the Quick Access toolbar button).
- To switch drawings, click their tab or use Ctrl+Tab and Ctrl+Shift+Tab.

See also

QNEW · OPEN · SAVEAS · CLOSE · UNITS

OPEN —Open

Opens a DWG or DXF drawing.

- **Command:** `OPEN` · **Global name:** `_OPEN` · **Alias:** `OPENURL`, `DXFIN` · **Shortcut:** Ctrl+O
- Menu: LinceCAD
- Menu: File
- Toolbar: Standard

Opens a DWG or DXF drawing in a new tab. The file is read in the background, with a progress bar, so the window keeps responding even with large files.

Usage

1. Enter OPEN or press Ctrl+O.
2. Choose the file in the dialog box. You can also drag the file onto the LinceCAD window.
3. When it finishes, the status bar says how many entities it has and how long it took, and the view shows the drawing extents.

Notes

- If the file is already open, its tab becomes current instead of opening it again.
- If the current drawing is untouched (Drawing1 with no changes), the opened one takes its tab, as in AutoCAD.
- Its external references and the text fonts in the drawing's folder are loaded too.
- If there is an automatic save of the same drawing newer than the file (from a session that did not close properly), the command line warns about it and says where it is.
- If the file is damaged and does not open, try RECOVER.

See also

NEW · RECOVER · QSAVE · XREF · File formats

PAGESETUP —Page Setup

Manages the page setups of the drawing.

- **Command:** `PAGESETUP` · **Global name:** `_PAGESETUP`
- Ribbon: Output › Plot
- Menu: File

Manages the page setups of the drawing: the one of each layout (and of model space) and the named page setups, which can be applied to several layouts or brought in from other drawings. They are saved in the drawing, as in AutoCAD, and are the ones PLOT uses.

Usage

1. Enter PAGESETUP or click its button. The **Page Setup Manager** opens, showing Current layout: …
2. In **Page setups:** choose the one of the current layout (between asterisks) or a named one.
3. Use the buttons:

- **Set Current**: Applies the selected page setup to the current layout.
- **New…** creates a named page setup. It asks for the **New page setup name:** and what to **Start with:** (the layout's own or an existing one).
- **Modify…** opens the edit dialog box, with the same controls as the Plot dialog box (plotter, paper, area, offset, scale, plot style table, options and orientation).
- **Import…** Copies the named page setups of another drawing (DWG or DXF).

4. At the bottom, **Selected page setup details** sums up the device, paper, plot size, scale and plot style table.

Notes

- Two page setups cannot have the same name.
- The controls are explained in PLOT, including **Properties…** (custom paper sizes and printable area of the plotter, for this page setup only or saved to the plotter file).

See also

PLOT · PREVIEW · Layouts

PDF3D — Export to 3D PDF

Plots the drawing to a PDF with the navigable 3D model embedded (U3D).

- **Command:** `PDF3D` · **Global name:** `_3DPDF`
- Ribbon: Output › Export

Plots the drawing to a PDF with the 3D model embedded as a navigable U3D object: opening it in Adobe lets you orbit, rotate and zoom the model inside the page itself.

Use

1. Type 3DPDF (or click **Export to 3D PDF**).
2. Give the output PDF file.

```
3D PDF file:
```

Notes

- The same 3D tessellation the 3D viewer shows is embedded (solids, surfaces, regions and meshes); its fineness comes from FACETRES.
- Each triangle carries the colour of its object or layer.
- The model is shown flat (unlit), with no textures or animation. PRC is not supported.
- A reader with 3D support (Adobe Acrobat/Reader) is needed to navigate the model.

See also

STLOUT · PLOT

PLOT — Plot

Plots the drawing to PDF, to an image or to a printer.

- **Command:** `PLOT` · **Global name:** `_PLOT` · **Alias:** `PRINT`, `DWFOUT` · **Shortcut:** Ctrl+P
- Ribbon: Output › Plot
- Menu: LinceCAD
- Menu: File
- Toolbar: Standard

Plots model space or the current layout to a vector PDF, PNG or SVG file, with the paper, area, scale and plot style table chosen in the **Plot** dialog box.

Usage

1. Enter PLOT (or PRINT), press Ctrl+P or click its button. The dialog box says what is plotted: Plot: …
2. Choose a saved **Page setup:**, or <None>, or <Previous plot> (the one of the last plot), and adjust what is needed.
3. Click **Preview…** to check the sheet (PREVIEW).
4. Click **OK**, enter the file name and choose its type: the extension (`.pdf`, `.png` or `.svg`) decides the format.

The dialog box

- **Printer/plotter:** `DWG To PDF.pc3` (lince PDF), None or an installed printer (with its paper sizes and **Copies:**): OK prints to it instead of saving a file. **Properties…** opens the plotter's paper size editor (see below).
- **Paper size:** ISO (A4, A3…), ANSI and architectural sizes, portrait or landscape (or those of the printer driver), preceded by the custom sizes of that plotter.
- **What to plot:** Layout (only in a layout), Extents, Display, Limits or Window. With Window, click **Window <** and specify the two corners in the drawing.

```
Specify first corner:
Specify opposite corner:
```

- **Plot offset (origin set to printable area):** X and Y in mm, or **Center the plot**.
- **Plot scale: Fit to paper** or a scale from the list (1:1, 1:50, 1:100…) or a custom one: so many mm (or inches) of paper = so many drawing units. **Scale lineweights** thickens lineweights with the scale.
- **Plot style table (pen assignments):** None, `monochrome.ctb`, `grayscale.ctb` or the `.ctb` tables (`.stb` if the drawing uses named plot styles) in `%APPDATA%\lince\estilos_trazado` and in the drawing's folder.
- **Plot options:** Plot object lineweights, Plot with plot styles, Plot paperspace last and Hide paperspace objects (the last two, in layouts).
- **Drawing orientation:** Portrait or Landscape, and Plot upside-down.
- **Apply to Layout**: Saves this page setup to the layout (or to the model) when plotting.
- **PNG:** the resolution, in dots per inch (30 to 2400), when saving as PNG.

Custom paper sizes and printable area

Like AutoCAD's Plotter Configuration Editor (its "Custom Paper Sizes" and "Modify Standard Paper Sizes"), **Properties…** opens the **Plotter Configuration Editor** of the chosen plotter or printer:

1. **Paper sizes saved for this plotter**: the custom sizes and the standard ones with a modified printable area (modified printable area). Click one to edit it.

- **New from scratch**: Custom paper size: name, width, height and margins.
- **New from this one**: Custom size with the dimensions and margins of the paper being edited.
- **Delete**: Removes the custom size from the list (a standard size gets back the plotter's printable area).
- **Import .pc3/.pmp…**: Reads the custom paper sizes and modified printable areas of an AutoCAD plotter

configuration (.pc3) or plot model parameter file (.pmp). With a `.pc3`, its linked `.pmp` is read too (at the path it stores or in the `PMP Files` folder next to it).

2. **Paper size (media bounds)**: Name:, Units: (mm or inches), Width: and Height:. If the paper is a standard one of the plotter (Standard size of the plotter: only its printable area is changed.), only the margins change.
3. **Printable area (non-printable margins)**: Top:, Bottom:, Left: and Right:, in the paper units. The remaining printable area is shown below.
4. **Apply the changes**, like AutoCAD's "Changes to a Printer Configuration File" prompt:

- **To the current plot only**: Not saved in the plotter file: it stays in the page setup (and in the drawing if applied to the layout), like AutoCAD's "Apply changes for the current plot only".
- **Save to the plotter file:** `%APPDATA%\lince\trazadores\<plotter>.pc3` and `PMP Files\<plotter>.pmp`: The sizes are kept for every drawing that uses this plotter, in a .pc3 and its .pmp in AutoCAD format (in the LinceCAD folder), like AutoCAD's "Save changes to the following file". **Open plotters folder** opens it in the file manager.

5. Click **OK**: the edited paper goes to the settings being plotted.

Custom sizes appear in that plotter's paper list in PLOT and PAGESETUP, and PUBLISH plots them with each layout's page setup. The PDF uses the size and margins as they are; a Windows printer gets the custom size (if its driver supports it). When the drawing is saved, the layout keeps the name, size and margins, and AutoCAD opens it with that paper.

In a script, `!traz medida width height left bottom right top name` (in mm) sets a custom paper for that plot only.

Notes

- The settings are saved in the drawing as an AutoCAD page setup; named page setups are managed with PAGESETUP.
- Plotting is always in wireframe.
- The `.pc3`/`.pmp` files saved by LinceCAD use the AutoCAD format (the header of its PDF or Windows printer driver, and the sizes as its wizard writes them) and stay in the LinceCAD folder, never in AutoCAD's. To use them in AutoCAD, copy the `.pc3` to its plotters folder and the `.pmp` to its `PMP Files` (AutoCAD Plotter Manager); back up its `DWG To PDF.pc3` first if you are going to replace it. Earlier versions saved a `.toml`: it is converted to `.pc3`/`.pmp` automatically (and kept as `.toml.migrado`).

See also

PREVIEW · PAGESETUP · EXPORT · Layouts · Tutorial: from drawing to PDF

PLOTSTAMP —Plot Stamp

Sets the plot stamp: a line of drawing data plotted in a corner of the sheet.

- **Command:** `PLOTSTAMP` · **Global name:** `_PLOTSTAMP` · **Alias:** `DDPLOTSTAMP`

Sets the plot stamp: a line of drawing data plotted in a corner of the sheet.

Usage

1. Type PLOTSTAMP: the Plot Stamp dialog opens.
2. Turn the stamp on, tick the fields, type up to two own fields and choose the corner, the font and the units.

The command-line version is -PLOTSTAMP.

```
Enter an option [ON/OFF/Fields/User fields/loG file/Location/Text properties/Units] :
```

Notes

- The fields are the file name, the layout, the date and time, the user, the device, the paper and the scale; on the sheet they are separated by " - ".
- The stamp is applied when plotting to PDF, PNG or SVG and is saved with the LinceCAD settings (not with the drawing).
- The log file option is saved but does not write any file yet.

See also

PLOT · PAGESETUP

PLOTTERMANAGER — Plotter Manager

Lists the plotters and edits their paper sizes and margins.

- **Command:** `PLOTTERMANAGER` · **Global name:** `_PLOTTERMANAGER`

Lists the plotters (the LinceCAD PDF and the system printers) and edits their paper sizes and margins.

Usage

1. Type PLOTTERMANAGER: the manager opens.
2. Choose a plotter and click Edit paper and margins.
3. In the editor create a new size, copy one, adjust the dimensions and the non-printable margins (top, bottom, left and right) and choose whether the change is temporary or permanent.

Notes

- It is the same editor opened from the plotter properties in PLOT.
- Without the dialog (scripts), the command asks for a plotter and writes its own sizes with the margins.

See also

PLOT · PAGESETUP · STYLESMANAGER

PREVIEW — Preview

Shows the page as it will be plotted.

- **Command:** `PREVIEW` · **Global name:** `_PREVIEW` · **Alias:** `PRE`
- Ribbon: Output › Plot
- Menu: File

Shows the sheet full screen exactly as it will be plotted: the paper, the printable area and what is plotted, with the settings of the Plot dialog box (scale, area, orientation and plot style table included).

Usage

1. Enter PREVIEW or click its button. It also opens from the PLOT dialog box with **Preview…**
2. Inspect the sheet: Wheel: zoom · Drag: pan.
3. Click **Plot** to plot to a file, **Fit** to see the whole sheet again or **Close** to leave.

Notes

- The preview is an image of the page: with a plot style table such as `monochrome.ctb` it already shows in black.

- To change the paper, scale or area, close the preview and change them in PLOT or PAGESETUP.

See also

PLOT · PAGESETUP · EXPORT

PUBLISH —Publish

Batch-plots a sheet list to PDF.

- **Command:** `PUBLISH` · **Global name:** `_PUBLISH`
- Ribbon: Output › Plot

Plots several sheets to PDF in one go: model space and the layouts of the open drawing and of other drawings, each sheet with its own page setup. The result is a single multi-sheet PDF or one PDF per sheet.

Usage

1. Save the drawing and enter PUBLISH. The **Publish** dialog opens with the **Sheet List**: model space and the layouts of the current drawing.
2. Check the sheets you want to plot; **?** removes a sheet from the list.
3. Use **Add Drawings…** to add model space and the layouts of other DWG or DXF files.
4. Choose **Single multi-sheet PDF** or **One PDF per sheet** and, if you wish, **Include layer information** and **PDF/A**.
5. Click **Publish** and specify the PDF file (or the folder, with one PDF per sheet). Plotting runs in the background with a progress bar; you can keep drawing meanwhile.

Sheet list

- **Save Sheet List…** saves the list in a LinceCAD `.dsl` file, with drawing paths relative to the list; **Load Sheet List…** loads it again to repeat the same publication.
- Each sheet is plotted with the page setup stored in its layout (PAGESETUP): paper, plot area, scale and plot style table.

Notes

- The drawing must be saved before publishing.
- Sheets of other drawings are plotted without their external references.
- AutoCAD `.dsl` files are not read: the list is LinceCAD's own.

See also

PLOT · PAGESETUP · ETRANSMIT · Layouts and plotting

QNEW —Quick new

Starts a drawing with the template set in OPTIONS.

- **Command:** `QNEW` · **Global name:** `_QNEW`
- Toolbar: Standard

Starts a new drawing with the template chosen in OPTIONS › Open and Save › **QNEW template**, without asking. It is what the **New** button of the Quick Access toolbar and of the Standard toolbar does.

Usage

1. Enter QNEW or click the **New** button of the Quick Access toolbar.

2. A tab opens with an unnamed drawing made from that template.

Notes

- With **None (ask)** (the LinceCAD default, as in AutoCAD), QNEW asks for the template like NEW.
- With a template chosen, the startup drawing (Drawing1) also comes from it.

See also

NEW · OPTIONS · SAVEAS

QSAVE —Save

Saves the current drawing.

- **Command:** `QSAVE` · **Global name:** `_QSAVE` · **Alias:** `SAVEURL` · **Shortcut:** Ctrl+S
- Menu: LinceCAD
- Menu: File
- Toolbar: Standard

Saves the current drawing to its file, in the same format and version it was opened with. A new drawing, which has no name yet, asks where to save it.

Usage

1. Enter QSAVE, press Ctrl+S or click its button.
2. If the drawing has no name, the SAVEAS dialog box opens with the DWG 2018 format.
3. The status bar confirms the path, the version and the time taken. The `*` on the tab disappears.

Notes

- Saving is safe: the file is first written next to it under another name; the previous one is kept as `.bak` and only then replaced. If anything fails, the original is left untouched.
- DWG files older than version 2000 are saved as DWG 2018.
- A read-only file cannot be overwritten.
- Every 10 minutes with unsaved changes, LinceCAD makes an automatic save in `%TEMP%\lince` (`.sv$` files), without touching your file. It is deleted when the drawing is closed.
- In the block editor, the whole drawing is saved.

See also

SAVEAS · OPEN · CLOSE · File formats

QUIT —Exit

Closes LinceCAD.

- **Command:** `QUIT` · **Global name:** `_QUIT` · **Alias:** `EXIT` · **Shortcut:** Ctrl+Q
- Menu: LinceCAD
- Menu: File

Closes LinceCAD. Before that it asks about every drawing with unsaved changes.

Usage

1. Enter QUIT (or EXIT), press Ctrl+Q or close the window (X button or Alt+F4).

2. For each drawing with unsaved changes, the program moves to its tab and asks:

```
Save changes to …?
```

3. Answer **Yes** (save), **No** (discard the changes) or **Cancel** (do not quit).

Notes

- If you cancel on any drawing, LinceCAD stays open with all its drawings.
- The preferences (OPTIONS) and the AutoLISP startup suite (APPLOAD) are already saved: nothing needs to be done with them when quitting.

See also

CLOSEALL · CLOSE · QSAVE

RECOVER —Recover

Opens a damaged drawing, recovering what can be read.

- **Command:** `RECOVER` · **Global name:** `_RECOVER`
- Ribbon: Manage › Drawing
- Menu: LinceCAD

Opens a damaged DWG or DXF drawing, recovering everything that can be read. Records that cannot be understood are skipped instead of stopping the reading, and then the same safe repairs as AUDIT are applied.

Usage

1. Enter RECOVER or click its button.
2. Choose the file in the dialog box. It opens in a new tab.
3. The command line shows the report: how many records were damaged or could not be read, the audit result and the repairs made.
4. Check the drawing and save it with QSAVE or SAVEAS to keep what was recovered.

Notes

- The recovered drawing is marked as modified: until you save it, the original file does not change.
- It is a good idea to save it under another name (SAVEAS) to keep the original.
- If a drawing opens fine but has errors, AUDIT is enough.

See also

AUDIT · OPEN · SAVEAS

SAVEAS —Save As

Saves with another name, format or version.

- **Command:** `SAVEAS` · **Global name:** `_SAVEAS` · **Alias:** `DXFOUT` · **Shortcut:** Ctrl+Shift+S
- Menu: LinceCAD
- Menu: File

Saves the drawing under another name, in another folder, in another format (DWG or DXF) or in another version, so that older programs can open it. Afterwards, the open drawing becomes the new file.

Usage

1. Enter SAVEAS or press Ctrl+Shift+S. The save dialog box opens with the DWG 2018 format.
2. In the menus (application menu or File), **Save As** opens a list of target versions: choose one and then the file name.
3. Enter the name and click Save. The status bar confirms the path and the version.

Target versions

Target	Opened by
DWG 2018 (AutoCAD 2018–2027)	AutoCAD 2018 and later, up to 2027
DWG 2013	AutoCAD 2013 to 2017
DWG 2010	AutoCAD 2010 to 2012
DWG 2007	AutoCAD 2007 to 2009
DWG 2004	AutoCAD 2004 to 2006
DWG 2000	AutoCAD 2000 to 2002
DWG R14	AutoCAD R14 and older programs (proxies are left out)
DXF 2018	Any program that reads AutoCAD 2018 DXF
DXF 2013, 2010, 2007 and 2004	Programs that read the DXF of those AutoCAD versions
DXF 2000	Programs that only read old DXF files
DXF R12	Programs that only read R12 DXF (what R12 lacks is exploded or left out)
DWT (template)	A drawing template for NEW (DWG 2018)

Saving as a template

With **DWT (template)** you choose the `.dwt` file and the **Template Options** dialog box opens: enter the **Description** (shown by the start page when the mouse is over the template) and choose the **Measurement** (Metric or English, the MEASUREMENT variable). They are saved in the file, where AutoCAD saves them, and AutoCAD shows them too.

Notes

- When saving to an older version, proxy objects of external applications are kept or left out the way AutoCAD does it (depending on each object version), and the status bar says how many are left out.
- AutoCAD Architecture objects of a DWG 2007 or later drawing cannot be saved to DWG 2000, 2004 or 2007: the program warns and does not save.
- It is saved with the same protections as QSAVE: temporary file and `.bak` copy.
- For PDF, PNG or SVG use EXPORT or PLOT.

See also

QSAVE · EXPORT · OPEN · File formats

STEPOUT —Export to STEP

Saves the 3D solids to a STEP file (AP214, millimetres) for other CAD programs.

- **Command:** `STEPOUT` · **Global name:** `_STEPOUT` · **Alias:** `STEP`
- Ribbon: Output › Export

Saves the drawing's 3D solids to a STEP file (ISO 10303, AP214 protocol, in millimetres), the exchange format that mechanical CAD programs read.

Usage

1. Type STEPOUT (or STEP) or click **Export to STEP**.
2. Select the 3D solids, or press Enter to export all of them.

```
Select 3D solids <all>:
```

3. Give the output file (`.step` is added when it has no extension).

```
STEP file:
```

What is exact and what is faceted

- **Exact** (analytic surfaces, not approximated): box, wedge, pyramid, cylinder, cone, sphere and torus, and any planar-faced solid without holes (for example, the extrusion of a polygon).
- **Faceted**: everything else (extrusions with arcs, revolutions of general profiles, fillets, results of curved Booleans…) is written as flat triangulated faces at the FACETRES fineness. When it finishes, the command line tells you how many solids were faceted.

Notes

- Only 3DSOLIDs are exported (not surfaces, regions or meshes; use STLOUT for those).
- If a faceted solid does not close, it is written as an open surface instead of a solid, and it is also counted in the warning.
- Solids that cannot be tessellated are skipped, and you are told how many.

See also

STLOUT · EXPORT

STLOUT — Export to STL/OBJ

Saves the drawing's 3D tessellation as STL or OBJ for 3D printing.

- **Command:** `STLOUT` · **Global name:** `_STLOUT` · **Alias:** `3DPRINT`, `3DP`, `3DPLOT`, `RAPIDPROTOTYPE`
- Ribbon: Output › Export

Saves the drawing's 3D tessellation (solids, surfaces, regions and meshes) to an STL or OBJ file, for 3D printing or to move it to another program.

Usage

1. Type STLOUT (or 3DPRINT) or click **Export to STL/OBJ**.
2. Give the output file; its extension picks the format.

```
STL or OBJ file:
```

3. For STL, choose binary (more compact) or ASCII (text).

```
Binary STL? [Yes/No] <Yes>:
```

Formats

- **.stl** — triangles with their normal, binary (default) or ASCII.
- **.obj** — Wavefront OBJ with vertices and faces.

Notes

- It exports the same tessellation the 3D viewer shows; facet fineness comes from FACETRES.

- ACIS solids, surfaces and regions and meshes (FACE3D, MESH, PFACE, 3DMESH) in model space are included.
- Geometry inside blocks (INSERT) and THICKNESS extrusions are not exported yet.

See also

IMPORT · EXPORT

STYLESMANAGER —Plot Style Manager

Edits the plot style tables (.ctb and .stb).

- **Command:** `STYLESMANAGER` · **Global name:** `_STYLESMANAGER`

Edits the plot style tables (color-dependent .ctb and named .stb).

Usage

1. Type STYLESMANAGER: the manager opens.
2. Choose a table from the list and a pen (or style); change its fixed color, grayscale, screening or lineweight.
3. Click Save.

Notes

- Tables are saved to the LinceCAD plot styles folder. Built-in tables (monochrome.ctb, grayscale.ctb) are saved as a copy with the same name, which from then on takes precedence over the built-in one.
- Without the dialog (scripts), the command asks for the table name and lists its pens.
- Linetypes, end styles and fill styles are not edited.

See also

CONVERTCTB · CONVERTPSTYLES · PAGESETUP

Edit and Clipboard

AI_SELALL — Select All

Selects all visible objects.

- **Command:** `AI_SELALL` · **Global name:** `_AI_SELALL` · **Shortcut:** Ctrl+A
- Ribbon: Home › Utilities
- Menu: Edit

Selects all visible objects in the current space (model or the open layout). The selection can then be edited with any command or with the Properties palette.

Usage

1. Press Ctrl+A or enter AI_SELALL with no command active.
2. All objects are selected.

Inside a command that asks to select objects, Ctrl+A (or typing ALL) adds all objects to the command's selection.

Notes

- Objects on locked, off or frozen layers are left out.
- To remove objects from the selection, click them with Shift held down. Esc clears it.
- Ctrl+A does not act if another text box (for example, in the Properties palette) has the keyboard.

See also

QSELECT · SELECTSIMILAR · Selecting objects

COPYBASE — Copy with Base Point

Copies the selection to the clipboard with a specified base point.

- **Command:** `COPYBASE` · **Global name:** `_COPYBASE` · **Shortcut:** Ctrl+Shift+C
- Ribbon: Home › Clipboard
- Menu: Edit

Copies the selected objects to the clipboard with the base point you specify. When pasting with PASTECLIP, that point goes exactly to the insertion point, so the copy can be placed precisely.

Usage

1. Press Ctrl+Shift+C or enter COPYBASE.
2. Select the objects and press Enter.

```
Select objects:
```

3. Specify the base point, usually with an object snap (a corner, a center…).

```
Specify base point:
```

4. The command line says how many objects were copied.

Notes

- As with COPYCLIP, the layers, styles and blocks the objects use are copied too, and the copy can be pasted

into any open drawing.
- The drawing does not change.

See also

COPYCLIP · PASTECLIP · PASTEBLOCK · Object snaps and precision

COPYCLIP —Copy Clip

Copies the selection to the clipboard.

- **Command:** `COPYCLIP` · **Global name:** `_COPYCLIP` · **Shortcut:** Ctrl+C
- Ribbon: Home › Clipboard
- Menu: Edit

Copies the selected objects to the LinceCAD clipboard, without changing the drawing. They can then be pasted into this drawing or another open one with PASTECLIP, PASTEORIG or PASTEBLOCK.

Usage

1. Select the objects and press Ctrl+C (or enter COPYCLIP). With objects already selected, they are copied at once.
2. If nothing was selected, select the objects and press Enter.

```
Select objects:
```

3. The command line says how many objects were copied.

Notes

- The base point is the lower-left corner of the selection: when pasting, that corner goes to the point you specify. To choose another base point, use COPYBASE.
- The layers, linetypes, styles and blocks the objects use are copied too, so the paste is the same in another drawing.
- The clipboard belongs to LinceCAD: it works between the open drawings, but not for pasting into AutoCAD or other programs. The Windows clipboard only receives a text summary (how many objects and of which types).
- Ctrl+C only acts with an empty command line and no active command; in a text box it copies the text.

See also

CUTCLIP · COPYBASE · PASTECLIP · COPY

CUTCLIP —Cut

Moves the selection to the clipboard and erases it.

- **Command:** `CUTCLIP` · **Global name:** `_CUTCLIP` · **Shortcut:** Ctrl+X
- Ribbon: Home › Clipboard
- Menu: Edit

Moves the selected objects to the LinceCAD clipboard and erases them from the drawing. They can then be pasted into this drawing or another one with PASTECLIP, PASTEORIG or PASTEBLOCK.

Usage

1. Select the objects and press Ctrl+X (or enter CUTCLIP). With objects already selected, they are cut at once.
2. If nothing was selected, select the objects and press Enter.

```
Select objects:
```

3. The command line says how many objects were copied. The objects disappear from the drawing.

Notes

- The base point is the lower-left corner of the selection. To choose it, use COPYBASE and then ERASE.
- The layers, linetypes, styles and blocks the objects use are copied too.
- The cut is undone with UNDO; the copy stays on the clipboard.
- Ctrl+X only acts with an empty command line and no active command.

See also

COPYCLIP · PASTECLIP · COPYBASE · Editing objects

MREDO —Multiple redo

Redoes several undone operations.

- **Command:** `MREDO` · **Global name:** `_MREDO`

Redoes several undone operations at once.

Usage

1. Enter MREDO with no command active.
2. Enter how many operations to redo, or choose an option.

```
Enter number of actions or [All/Last] <1>:
```

Options

- **All** (A): redoes everything that was undone.
- **Last** (L): redoes only one (the same as Enter or REDO).

Notes

- An UNDO BEgin…Endgroup counts as one operation.
- As soon as a new change is made, what was undone can no longer be redone.

See also

REDO · UNDO · U

OOPS —Oops

Restores the objects erased last.

- **Command:** OOPS · **Global name:** _OOPS

Restores the objects erased by the last ERASE (or by the last BLOCK or WBLOCK that deleted the originals), without undoing what was done afterwards.

Usage

1. Enter OOPS with no command active.
2. The objects come back with the same properties and the same identifier (handle). The command line says how many were restored.

Notes

- Only the last erasure is restored, and only once; if there is nothing to restore, it says so.
- OOPS is one more operation: it can be undone with U.

See also

ERASE · UNDO · U

PASTEBLOCK — Paste as Block

Pastes the clipboard contents as a block.

- **Command:** `PASTEBLOCK` · **Global name:** `_PASTEBLOCK` · **Shortcut:** Ctrl+Shift+V
- Ribbon: Home › Clipboard
- Menu: Edit

Pastes the copy as a single object: it creates a new block definition with it and inserts it at the point you specify. The pasted content can then be moved, copied or erased in one go.

Usage

1. Copy the objects with COPYCLIP, CUTCLIP or COPYBASE.
2. Press Ctrl+Shift+V or enter PASTEBLOCK.
3. Specify the insertion point. While you move the cursor, the outline of the copy is shown.

```
Specify insertion point:
```

4. The block is inserted at scale 1 with no rotation.

Notes

- The block gets an automatic name, `A$C` followed by 8 characters, as in AutoCAD.
- The base point of the block is the base point of the copy.
- To get the loose objects back, use EXPLODE.

See also

PASTECLIP · BLOCK · EXPLODE · Blocks

PASTECLIP — Paste

Inserts the contents of the clipboard.

- **Command:** `PASTECLIP` · **Global name:** `_PASTECLIP` · **Shortcut:** Ctrl+V
- Ribbon: Home › Clipboard
- Menu: Edit

Pastes into the current drawing what was copied with COPYCLIP, CUTCLIP or COPYBASE. The pasted objects are ordinary, independent objects, with their layers and styles.

Usage

1. Press Ctrl+V or enter PASTECLIP.
2. Specify the insertion point. While you move the cursor, the outline of the copy is shown.

```
Specify insertion point:
```

3. The objects are placed with their base point at the specified point.

Notes

- You can paste several times in a row: pasting does not empty the clipboard.
- The layers, linetypes, styles and blocks the objects use that are missing from the drawing are added with them.
- If the clipboard is empty, the command says so and ends.
- To paste at the same place they had, use PASTEORIG; to paste as a single block, PASTEBLOCK.
- Ctrl+V only acts with an empty command line and no active command; in a text box it pastes the text.

See also

PASTEORIG · PASTEBLOCK · COPYCLIP · COPYBASE

PASTEORIG — Paste to Original Coordinates

Pastes the copied objects at their original coordinates.

- **Command:** `PASTEORIG` · **Global name:** `_PASTEORIG`
- Ribbon: Home › Clipboard
- Menu: Edit

Pastes the copy at the same coordinates it had in the source drawing, without asking for a point. Use it to move objects from one drawing to another without shifting them.

Usage

1. Copy the objects in one drawing with COPYCLIP or CUTCLIP.
2. Switch to the target drawing (its tab or Ctrl+Tab).
3. Enter PASTEORIG or choose it in the Edit menu. The objects appear where they were.

Notes

- As with PASTECLIP, the layers, styles and blocks the objects use that are missing from the drawing are added.
- Pasting into the same drawing leaves an exact copy on top of the original.
- If the clipboard is empty, the command says so and ends.

See also

PASTECLIP · COPYCLIP · PASTEBLOCK

PASTESPEC — Paste Special

Pastes the clipboard choosing the format.

- **Command:** `PASTESPEC` · **Global name:** `_PASTESPEC` · **Alias:** `PA`
- Ribbon: Home › Clipboard

Pastes the clipboard choosing the format, like AutoCAD PASTESPEC: the **Paste Special** dialog lists the formats in the clipboard (LinceCAD objects, as a block, or system clipboard text as multiline text).

Usage

1. Enter PASTESPEC (or PA), choose the format and press OK.

See also

PASTECLIP · PASTEBLOCK

REDO — Redo

Redoes the last undone action.

- **Command:** `REDO` · **Global name:** `_REDO` · **Shortcut:** Ctrl+Y
- Menu: Edit
- Toolbar: Standard

Redoes what was just undone with UNDO. It can be repeated to redo several steps, in the same order they were undone.

Usage

1. Press Ctrl+Y, click **Redo** on the Quick Access toolbar or enter REDO with no command active.
2. The command line says what was done. If there is nothing left to redo, it says so.

Notes

- Undone steps can be redone as long as you make no other change to the drawing: the first new command empties the REDO list.
- Each tab has its own list.

See also

UNDO · Editing objects

U — Undo one

Undoes the last operation.

- **Command:** `U` · **Global name:** `_U`

Undoes the last operation: the last command that changed the drawing or, if UNDO BEgin and End were used, the whole group. It is the same as UNDO with 1, without asking.

Usage

1. Enter U with no command active, press Ctrl+Z or click the **Undo** button of the Quick Access toolbar.
2. The command line says what was done; if there is nothing left to undo, it says so.

Notes

- To undo several operations at once, return to a mark or group commands, use UNDO.
- What was undone comes back with REDO or MREDO.

See also

UNDO · REDO · MREDO · OOPS

UNDO —Undo

Undoes the last command.

- **Command:** `UNDO` · **Global name:** `_UNDO` · **Shortcut:** Ctrl+Z
- Menu: Edit
- Toolbar: Standard

Undoes one or more commands that changed the drawing, as in AutoCAD. Each command is undone as a whole (all the lines of a LINE, all the copies of a COPY…). Its options return to a mark, group several commands to undo them at once and limit or disable the history.

Usage

1. Enter UNDO with no command active.
2. Enter how many operations to undo and press Enter (Enter alone undoes one), or choose an option.

```
Enter the number of operations to undo or [Auto/Control/BEgin/End/Mark/Back] <1>:
```

3. The command line says how many were undone. To get them back, use REDO (Ctrl+Y) or MREDO.

Ctrl+Z, the **Undo** button of the Quick Access toolbar and U undo one operation without asking.

Options

- **Mark** (M): places a mark in the history.
- **Back** (B): undoes back to the last mark and removes it. If there is none, everything is undone.
- **BEgin** (BE) and **End** (E): everything you do between UNDO BEgin and UNDO End counts as a single operation (it is undone and redone at once). A BEgin with a group open closes it and opens another one.
- **Auto** (A): ON or OFF. Kept for compatibility; here each command is always one operation already.

```
Enter UNDO Auto mode [ON/OFF] <ON>:
```

- **Control** (C): **All** keeps the whole history; **None** clears it and disables UNDO and U (only the Control option is left, to turn it on again); **One** keeps only the last operation. **Combine** and **Layer** are kept for compatibility (zoom, pan and the layer changes of the palette do not take separate steps).

```
Enter an UNDO control option [All/None/One/Combine/Layer] <All>:
```

What is undone

- Drawing and editing commands.
- Changes made in the Properties palette or on the ribbon (layer, color, linetype…).
- Changes to layers, styles and linetypes, and those of UNITS, SETVAR, PURGE and AUDIT.

Notes

- The last 50 operations of each drawing are kept; each tab has its own list.
- Inquiries (DIST, LIST…), zoom and pan do not change the drawing and do not take up a step.
- Inside a command such as LINE or PLINE, the Undo option of its prompt removes only the last segment.
- To restore only what was erased, without undoing what was done afterwards, use OOPS.

See also

U · REDO · MREDO · OOPS · Editing objects

Status Bar

-OSNAP —Object Snap (command line)

Sets the running object snap modes from the command line.

- **Command:** `-OSNAP` · **Global name:** `_-OSNAP` · **Alias:** `-OS`

Sets the running object snap modes from the command line, like the object snap settings but typing their short names. Useful in scripts and in LISP.

Usage

1. Enter -OSNAP. If some modes are on, the program says which.

```
Current osnap modes: …
```

2. Type the modes separated by commas and press Enter.

```
Enter list of object snap modes:
```

Modes

END (endpoint), MID (midpoint), CEN (center), NOD (node), QUA (quadrant), INT (intersection), INS (insertion), PER (perpendicular), TAN (tangent), NEA (nearest), GCE (geometric center), APP (apparent intersection) and EXT (extension). NON clears them all.

Notes

- What you type replaces the previous modes (it is the OSMODE variable).

```
Invalid object snap modes.
```

See also

OSNAP · SNAP · Object snaps and precision

GRID —GRID

Displays the grid.

- **Command:** `GRID` · **Global name:** `_GRID` · **Shortcut:** F7

Shows or hides the grid of the drawing area, a reference mesh that helps to estimate distances at a glance, and sets its spacing. Also with F7 and the **GRID** button on the status bar.

Usage

1. Enter GRID.

```
Specify grid spacing(X) or [ON/OFF/Snap/Major/aDaptive/Limits/Follow/Aspect] <…>:
```

2. Type the spacing (the grid turns on with it), choose an option or press Enter to leave it as it is.

Options

- **ON** / **OFF**: turns the grid on or off.
- **Snap** (S): the grid follows the snap spacing (SNAP).
- **Major** (M): how many lines between major lines.

```
Specify major line every (1-100):
```

- **aDaptive** (D): whether the grid adapts to the zoom.

```
Turn adaptive behavior on [Yes/No] <Yes>:
```

- **Limits** (L): whether it is drawn beyond the drawing limits (LIMITS).

```
Display grid beyond Limits [Yes/No] <Yes>:
```

- **Follow** (F): whether it follows the dynamic UCS.

```
Follow Dynamic UCS [Yes/No] <No>:
```

- **Aspect** (A): different spacings in X and Y.

```
Specify horizontal spacing(X):
```

Notes

- With no spacing set, the grid is drawn every power of 10 (1, 10, 100...) depending on the zoom, so that lines are at least about 25 pixels apart.
- The spacing, major lines and settings are saved in the drawing (GRIDUNIT, GRIDMAJOR and GRIDDISPLAY); LinceCAD still draws every line alike, in plan, without telling the major ones apart.

See also

SNAP · ORTHO · LIMITS · Object snaps and precision

LWDISPLAY —LWT

Show lineweights.

- **Command:** `LWDISPLAY` · **Global name:** `_LWDISPLAY`

Will show or hide the lineweight of objects on screen. It is not available yet as a toggle: it cannot be typed and it has no button on the status bar.

Notes

- For now, objects are always drawn on screen with their lineweight.
- The LWDISPLAY variable is read and saved with the drawing, but it does not change the display yet.
- The lineweight of new objects is set with LWEIGHT; that of a layer, in the LAYER.

See also

LWEIGHT · PROPERTIES · LAYER

ORTHO —ORTHO

Ortho mode.

- **Command:** `ORTHO` · **Global name:** `_ORTHO` · **Shortcut:** F8

Turns ortho mode on and off: while a command asks for a point from another, the cursor is restricted to the horizontal or vertical of the last point. Use it to draw straight lines by hand without typing coordinates. Also with F8 or the **ORTHO** button on the status bar.

Usage

1. Enter ORTHO, or press F8 or the **ORTHO** button.

```
Enter mode [ON/OFF] <…>:
```

2. Choose ON or OFF; Enter leaves the state as it is.

Notes

- Ortho and polar tracking (POLAR) exclude each other: turning one on turns the other off.
- The state is the ORTHOMODE variable.

See also

SNAP · POLAR · GRID · Object snaps and precision

OSNAP —OSNAP

Object snap.

- **Command:** OSNAP · **Global name:** _OSNAP · **Alias:** OS, DDOSNAP · **Shortcut:** F3

Turns running object snaps on and off: while a command asks for a point, the cursor jumps to nearby key points (endpoints, midpoints, centers, intersections...) and a marker shows which one was found. It is not typed as a command: use the F3 key or the **OSNAP** button on the status bar.

Usage

1. Press F3 or click the **OSNAP** button to turn them on or off.
2. Right-click the button to choose the modes: **Endpoint**, **Midpoint**, **Center**, **Intersection**, **Perpendicular**, **Tangent**, **Quadrant**, **Insertion**, **Node**, **Extension** and **Nearest**. **Select All** and **Clear All** check or clear them all.

Temporary snaps

When a command asks for a point, type the first three letters of a mode to use it for that point only, even with running snaps off: END, MID, CEN, INT, PER, TAN, QUA, INS, NOD, EXT or NEA (the Spanish names FIN, MED, CUA, CER... also work).

Notes

- Object snaps take priority over tracking, SNAP and Ortho.
- Object snap tracking (**OTRACK**, F11) needs object snaps on to acquire points.
- They are on at startup.

See also

POLAR · ORTHO · SNAP · Object snaps and precision

POLAR —POLAR

Polar tracking.

- **Command:** POLAR · **Global name:** _POLAR · **Shortcut:** F10

Turns polar tracking on and off: while a command asks for a point from another one, when the cursor passes close to a direction that is a multiple of the increment angle, it snaps to it, a dashed alignment path is drawn and a tooltip

shows the distance and the angle. It is not typed as a command: use the F10 key or the **POLAR** button on the status bar.

Usage

1. Press F10 or click the **POLAR** button to turn it on or off.
2. Right-click the button to choose the increment angle: 90, 45, 30, 22.5, 18, 15, 10 or 5 degrees.
3. With the alignment path in view, click: the point lies on it.

Notes

- Polar tracking and Ortho (ORTHO) exclude each other: turning one on turns the other off.
- The **OTRACK** button (F11), next to it, turns on object snap tracking: rest the cursor half a second on an object snap to acquire it (green cross) and horizontal and vertical paths come out of it.
- POLAR, at 90°, and object snap tracking are on at startup.

See also

ORTHO · OSNAP · Object snaps and precision

QUICKPROPERTIES —Quick Properties

Turns the quick properties panel on or off (QPMODE).

- **Command:** `QUICKPROPERTIES` · **Global name:** `_QUICKPROPERTIES` · **Alias:** `QP`

Turns the quick properties panel on or off: a small floating panel that appears next to the cursor when you select objects with no command running, with the key editable properties of the selection.

Usage

1. Enter QUICKPROPERTIES (or click its button) to turn it on or off. The state is kept in the variable QPMODE (1 on, 0 off), which can also be changed with SETVAR.
2. With the panel on, select one or more objects with no command running: the panel appears next to the cursor.
3. Edit the properties directly in the panel, as in the PROPERTIES palette.
4. The panel closes when you deselect or start a command.

Properties shown

The general ones (layer and color) and the key properties of the selected type:

- Line: length and angle.
- Circle: radius and diameter.
- Arc: radius, total angle and arc length.
- Polyline: length and whether it is closed.
- Text: contents, height and rotation.
- Block: name and rotation.
- Dimension: style and measurement.

With objects of several types only the common (general) properties are shown.

Notes

It is a reduced, floating version of the PROPERTIES palette; it shares with it the logic for reading and editing properties by type. It does not appear in test scripts.

See also

PROPERTIES · QPMODE · SETVAR

SNAP —SNAP

Snap to grid.

- **Command:** SNAP · **Global name:** _SNAP · **Alias:** SN · **Shortcut:** F9

Turns the snap on and off and sets its spacing: while a command asks for a point, the cursor jumps to the nodes of an invisible grid with that spacing. Also with F9 or the **SNAP** button on the status bar.

Usage

1. Enter SNAP.

```
Specify snap spacing or [ON/OFF/Aspect/Style/Type] <…>:
```

2. Type the spacing (snap turns on with it), choose an option or press Enter to leave it as it is.

Options

- **ON** / **OFF**: turns snap on or off.
- **Aspect** (A): different spacings in X and Y.

```
Specify horizontal spacing(X):
```

- **Style** (S): standard or isometric snap (SNAPSTYL).

```
Enter snap grid style [Standard/Isometric] <S>:
```

- **Type** (T): grid or polar snap.

```
Enter snap type [Polar/Grid] <Grid>:
```

Notes

- With no spacing set, the spacing is that of the grid on screen, a power of 10 that changes with the zoom.
- The spacing is saved in the drawing (SNAPUNIT). Polar type is saved, but the cursor still snaps to the grid.
- Object snaps and tracking take precedence over SNAP; ortho is applied afterwards.

See also

GRID · ORTHO · OSNAP · Object snaps and precision

Help

ABOUT —About

Version, credits and third-party licenses.

- **Command:** `ABOUT` · **Global name:** `_ABOUT`
- Ribbon: Manage › Help
- Menu: LinceCAD
- Menu: Help

Opens the **About LinceCAD** window, with the installed version of LinceCAD, the copyright notice and the license notices of the third-party libraries included in the program.

Usage

1. Enter ABOUT or choose it on the application menu or the Help menu.
2. Expand **Third-party notices** to read the full licenses.
3. Close the window with its close button.

Notes

- Quote the version shown in this window when you report a problem.

See also

HELP · What's new

CHECKFORUPDATES —Check for updates

Checks whether a new version of LinceCAD is available and lets you download and install it.

- **Command:** `CHECKFORUPDATES` · **Global name:** `_CHECKFORUPDATES`
- Ribbon: Manage › Help
- Menu: LinceCAD
- Menu: Help

Checks whether a new version of LinceCAD is available, shows what is new and, if you want, downloads and installs it.

Usage

1. Type CHECKFORUPDATES or choose **Check for updates** on the application menu or on the Manage tab, Help group.
2. The **LinceCAD update** dialog shows the **Installed version** and, if there is one, the **New version**, its date, size and release notes. If there is nothing new, it says **LinceCAD is up to date**.
3. Choose an option: - **Download and install**: downloads the installer in the background, with a progress bar and **Cancel**. You can keep working meanwhile. - **Later**: closes the dialog; the notice stays on the status bar. - **Skip this version**: no more notices for that version (later ones are still announced). Mandatory versions cannot be skipped.
4. When the download finishes, click **Install and restart**. If any drawing has unsaved changes, the button is **Save all and install**: it saves each one (with «Save as» if it has no name) and, if you cancel any, nothing is installed.
5. LinceCAD closes, the installer updates the program without questions and LinceCAD opens again by itself.

Automatic check

- With **Check for updates automatically** on (OPTIONS, Open and Save tab), LinceCAD checks at startup and every 12 hours, in the background: it does not slow down startup or interrupt your work.
- If there is a new version, a discreet notice appears on the status bar, **Update … available**, which opens this dialog.
- A **mandatory** version (for example, an important fix) is announced more visibly and opens the dialog once, but it **never blocks** your work: you can install it whenever you want.
- The **channel** is chosen in OPTIONS: **Stable** (default) or **Beta**, which also gets preview versions.

License and maintenance

- The perpetual license includes **one year of updates**. Only versions released up to the end of your maintenance are offered; if there are several, the newest of them.
- If there is an even newer version, the dialog says so (**«Version X requires renewing maintenance»**) with the **Renew maintenance…** button, without downloading it.
- If you manually install a version released after your maintenance, LinceCAD switches to read-only with a courtesy save and this dialog offers to **reinstall** the latest entitled version. See License and maintenance.

What is sent and how it is protected

- Only the installed version, the channel, the language, a single-use random value and, if there is a license, the same activation token used by the 12-hour validation are sent to `https://lincecad.com/api`. For gradual rollouts, a SHA-256 digest of the machine fingerprint is also sent, never the fingerprint itself. Nothing from your drawings is sent.
- The answer is **signed** with the license server key; a copied or forged answer is discarded.
- The installer is saved in `%LOCALAPPDATA%\lince\actualizaciones\` and its **SHA-256** is checked before running it: if it does not match the published one, it is deleted and not installed.

Notes

- Not available in scripts, in automation (API, MCP) or if LinceCAD was installed from the Microsoft Store, which updates the program itself.
- Offline, the automatic check announces nothing; when run by hand, it says there is no connection.

See also

LICENSE · OPTIONS · License and maintenance

HELP — Help

Displays the LinceCAD help.

- **Command:** `HELP` · **Global name:** `_HELP` · **Shortcut:** F1
- Ribbon: Manage › Help
- Menu: LinceCAD
- Menu: Help

Opens the LinceCAD help viewer. If a command is running, it opens the page of that command directly.

Usage

1. Press F1, enter HELP or choose **Help** on the Help menu.
2. While a command is running, F1 opens its page; with the mouse over a button of the ribbon or of a toolbar, F1 opens the page of that button's command. Otherwise the general help opens.

The viewer

- **Contents**: the table of contents as a tree (general help, user manual, commands, system variables and AutoLISP functions). Click an entry to show it.
- **Index**: alphabetical list of commands and terms; click one to go to its page.
- **Search**: looks for words in the text of all pages and lists the pages that contain them.
- **Back** and **Forward**: step through the pages you have seen, as in a web browser.
- **Manual (PDF)**: opens the complete manual as a PDF, in the language of the interface, to read or print it.

Command pages start with the command name, its English global name, its alias, its shortcut and where it is on the ribbon and in the menus. Prompt texts are the same ones the command line shows.

Notes

- Links lead to other help pages; those starting with `https://` open in the web browser.
- The help works without an Internet connection: the pages are installed with the program.

See also

ABOUT · User interface · Keyboard · Welcome

LICENSE — License

Shows the license status and lets you activate it, start a trial or release it on this machine.

- **Command:** `LICENSE` · **Global name:** `_LICENSE`
- Ribbon: Manage › Help
- Menu: LinceCAD
- Menu: Help

Shows the LinceCAD license status and lets you buy the license, activate a seat, redeem an invitation, start a 30-day trial or release the license on this machine to move it to another one.

Usage

1. Type LICENSE or click **License**.
2. At the top, a badge sums up the status: **Full license**, **Temporary license · until D**, **Trial · days left**, **Offline · grace days left** or **Read-only**. Below it, the type (perpetual, temporary or trial), email, seats, expiry and the date of the last online validation.
3. To **buy** it, click **Buy license…** (see below).
4. To **activate**, paste the key you received by email when you bought it (or the one support gave you) and click **Activate**.
5. If you have an **invitation code**, click **I have an invitation…**, enter it and click **Redeem** (see below).
6. To **try it**, enter your email and click **Start trial**.
7. To **move** the license to another machine, click **Release on this machine…** and confirm. Then activate it on the new machine with the same key.

Buying the license

- The **Buy a license** section appears while there is no perpetual license (no license, on trial or with the trial expired). It shows the price of the perpetual license for one machine; taxes are calculated at checkout for your country.
- **Buy license…** asks the license server for a **Stripe** payment page for **this machine** and opens it in your browser. The server only receives the machine fingerprint (stored as a SHA-256 digest); Stripe only gets a purchase reference and that digest.

- While you pay, the dialog shows **Waiting for the payment…** and checks every few seconds, also with the dialog closed. **Open the payment page again** reopens it; **Stop waiting** stops checking (if the payment completes anyway, **Check the pending payment** picks it up).
- Once the payment is confirmed, the server creates the perpetual license and **activates it on this machine** (it uses its seat): LinceCAD switches to **Full license** by itself, without typing any key.
- You receive the **license key** by email (keep it: you need it to move the license to another machine) and, separately, the payment **receipt and invoice** from the payment platform.
- If you close LinceCAD before the payment arrives, reopening LICENSE resumes the check. An unused payment page expires after one hour and the purchase is cancelled.
- Without a connection to the server you cannot buy from here; the **Pricing and terms on lincecad.com** link opens the website.

Invitation

- LinceCAD can give away full licenses, **perpetual** or **temporary**, with an **invitation code** of the form `LINCE-XXXX-XXXX-XXXX-XXXX` (for example for the press, collaborators, schools or giveaways). It usually arrives by email, with what it includes and the deadline to redeem it.
- Click **I have an invitation…**, enter the code and click **Redeem**. Lower case, spaces or missing hyphens do not matter.
- The server creates the license and **activates it on this computer**: LinceCAD shows **Perpetual license activated** or **Temporary license activated until D** and switches to the full license, without typing any key.
- If the code does not exist, has already been used or is past its deadline, the dialog says so; for security it does not tell those cases apart, and after several failed attempts you have to wait a while.
- Redeeming the same code again on the same computer does not use it up again: it returns the same license.
- A **temporary license** works like a perpetual one until its end date. **15 days before**, the start screen and this dialog warn you: «Your temporary license ends on D». When the date arrives (also with the program open) it switches to **read-only** with the courtesy save; to keep editing, buy the license or redeem another invitation.

Start screen

With no license, a running trial or an expired trial, the start screen says so and waits for a button:

- **No license:** "To activate the 30-day trial version, press the button" and **Activate the trial version**, which opens this dialog.
- **Trial running:** "Trial version: N days left" and **Buy license…**
- **Trial expired:** "The trial version has expired" and **Buy license…**
- **Temporary license 15 days or less from its end:** "Your temporary license ends on D" and **Buy license…**; once ended, "Your temporary license has ended".
- **Continue** (or Esc) closes the screen. With a perpetual license the start screen goes away by itself, as always.

Moving to another machine

- When you release it, the server deletes this machine's activation and LinceCAD deletes the local token: this machine goes back to **no license** (read-only) and the seat is free for another one.
- Each license allows **3 transfers in a rolling 12 months** (every release in the last 365 days counts). Beyond that, LinceCAD tells you so and only **LinceCAD support** can release it from the panel (for example, for a broken or stolen machine; those releases do not use up transfers).
- A trial can also be moved, but it does not entitle you to another trial on the same machine.

Online validation and grace period

- At startup and every **12 hours** while the program is open, LinceCAD checks the license with the server **in the background**: it does not delay startup or block your work.
- If the server answers that the license is **revoked**, **does not exist** or is **no longer activated on this machine** (it was released elsewhere), LinceCAD switches to **read-only** immediately.

- If there is **no answer** (no internet or the server is down), the license stays full for **7 days** from the last successful validation; the badge shows the grace days left. After 7 days, it switches to **read-only**.
- As soon as it validates successfully again, the **full license** comes back with no further action.
- This applies to perpetual licenses and trials; the trial expiry (30 days) is still checked on the machine itself.

Maintenance (updates)

- The perpetual license includes **one year of updates**. The **Updates until** row and the **Maintenance** section show **Updates included until: …**or, if it has passed, **Maintenance expired on …**
- **Renew maintenance…**opens the Stripe payment, like the purchase; once paid, maintenance is extended one year from its end date (or from today, if it had expired) and a confirmation email is sent.
- If this version was released **after** the end of maintenance (for example, installed by hand), LinceCAD switches to read-only with a courtesy save, also offline, and explains it here and on the start screen. See License and maintenance.

Courtesy save

If the license is cut off or expires (trial or temporary) while you work, LinceCAD warns you and opens this dialog. Each drawing that had **unsaved changes** at that moment can be saved **once** (SAVE or SAVEAS) so you do not lose your work; after that it is read-only like the rest.

Evaluation mode (read-only)

Without a valid license, LinceCAD runs in **evaluation mode**: you can open, view, pan and zoom, query, plot and export to PDF, PNG or SVG and print, but you **cannot edit or save** the drawing. Drawing and editing commands and SAVE/SAVEAS are blocked, with a discreet notice in the status bar.

How it works

- The license is signed by the server with **Ed25519**. The client embeds the **public key** and verifies the signature **offline**.
- Activation, trial sign-up and release send a **machine fingerprint** (a stable machine identifier); the server only stores its SHA-256 digest. The returned token is stored in `<config>/lince/licencia.json`.
- The online validation answer is **signed** too and includes a random value the program picks for each check: a copied or forged answer does not work.
- A **trial** lasts 30 days; when it expires, LinceCAD returns to evaluation mode.
- The pending purchase is stored in `<configuration>/lince/compra.json` (reference and query code, no payment data) and deleted once paid or cancelled.

Limitations

- The grace period is measured with the machine's clock.
- The license is bound to the machine by its fingerprint: copying `licencia.json` to another machine does not activate it there.

See also

ABOUT · CHECKFORUPDATES · License and maintenance

System Variables

System variables

System variables hold drawing settings: the units, the current layer and color, the fillet radius, how points look… LinceCAD uses the same names and the same values as AutoCAD, and stores them in the drawing header: when the DWG or DXF file is opened in AutoCAD they are kept, and the other way round.

Reading and changing a variable

1. Enter SETVAR (or SET).

```
Enter variable name or [?]:
```

2. Enter the variable name, or `?` to list all of them with their values in the text window.

```
Enter new value for … <…>:
```

3. Enter the new value, or press Enter to keep the current one (shown between `< >`).

The variable name can also be typed directly on the command line, as if it were a command: `LTSCALE` asks for the new value of LTSCALE. The `_` and `.` prefixes are accepted, as in AutoCAD.

Notes

- Real numbers can be typed with a decimal comma or point (`2,5` or `2.5`). Distances also accept feet and inches (`1'-3 1/2"`).
- If the value is not valid, the program says so and keeps the previous one.

```
Invalid value for …: …
```

- Changes can be undone with UNDO and are saved with the drawing, except for the variables marked as session ones, which last until the program is closed.
- From AutoLISP they are read and changed with getvar and setvar, which also know other variables (DIMZIN, UNITMODE, OSMODE, ORTHOMODE, DWGNAME…).See AutoLISP.

List of variables

Variable	What it controls
AFLAGS	Modes of the next attribute definition
ANGBASE	Direction of angle 0
ANGDIR	Direction of positive angles
ANNOAUTOSCALE	Scale added when the annotation scale changes
ATTDIA	Whether INSERT asks for attributes in a dialog box
ATTMODE	Attribute visibility
ATTREQ	Whether INSERT asks for attribute values
AUNITS	Angle format
AUPREC	Angle precision
BINDTYPE	Names when binding external references
CANNOSCALE	Current annotation scale

CECOLOR	Color of new objects
CELTSCALE	Linetype scale of new objects
CELTYPE	Linetype of new objects
CENTERCROSSGAP	Gap between the cross and the extensions of center marks
CENTERCROSSSIZE	Size of the cross of center marks
CENTEREXE	Overshoot of center marks and centerlines
CENTERLAYER	Layer of center marks and centerlines
CENTERLTSCALE	Linetype scale of center marks and centerlines
CENTERLTYPE	Linetype of center marks and centerlines
CENTERMARKEXE	Extension lines of center marks
CETRANSPARENCY	Transparency of new objects
CHAMFERA	First chamfer distance
CHAMFERB	Second chamfer distance
CLAYER	Current layer
CMLJUST	Justification of new multilines
CMLSCALE	Scale of new multilines
COORDS	How cursor coordinates are shown on the status bar
CTABLESTYLE	Current table style
DELOBJ	Whether commands that build objects from others
DIMASSOC	Associativity of new dimensions
DIMCEN	Center mark of DIMCENTER, DIMRADIUS and DIMDIAMETER
DIMSCALE	Overall dimension scale
DIMSTYLE	Current dimension style
DISPSILH	Silhouette of solids in wireframe
DRAGMODE	Dragging of objects while points are picked
DYNMODE	Dynamic input on or off
ELEVATION	Elevation of new objects
EXTMAX	Upper right corner of the drawing extents
EXTMIN	Lower left corner of the drawing extents
FACETRES	Smoothness of the tessellation of solids and surfaces
FIELDDISPLAY	Gray background of fields
FILEDIA	File dialog box
FILLETRAD	Fillet radius
FILLMODE	Fill of solids and wide polylines
HALOGAP	Gap left by objects hidden behind others, as a % of a unit
HIDETEXT	Whether HIDE takes texts into account
HPGAPTOL	Hatch gap tolerance
IMAGEFRAME	Image frames
INDEXCTL	Layer and spatial indexes saved with the drawing
INSBASE	Insertion base point of the drawing
INSUNITS	Insertion units
ISOLINES	Isolines on the curved faces of solids

LAYOUTCREATEVIEWPORT	Viewport in new layouts
LENSLENGTH	Lens length of the perspective
LAYLOCKFADECTL	Fading of locked layers
LIMCHECK	Drawing limits checking
LIMMAX	Upper right corner of the limits
LIMMIN	Lower left corner of the limits
LTSCALE	Global linetype scale
LUNITS	Length format
LUPREC	Length precision
LWDISPLAY	Lineweight display
MAXACTVP	Maximum number of viewports active at once in a layout
MEASUREMENT	Drawing units for hatch patterns and linetypes
MIRRTEXT	Mirroring of text
PDFFRAME	Frame of PDF underlays
PDMODE	Point symbol
PDSIZE	Point size
PELEVATION	Paper space elevation
PELLIPSE	Whether ELLIPSE creates true ellipses
PERSPECTIVE	Perspective or parallel view
PICKSTYLE	Group selection
PLINEGEN	Whether the linetype of new polylines runs along the whole polyline
PLINEWID	Default polyline width
POLYSIDES	Default sides of POLYGON
PROXYGRAPHICS	Whether proxy object images are saved in the drawing
PSLTSCALE	Whether linetype scaling in viewports follows each viewport scale
PSOLHEIGHT	Height of new polysolids
PSOLWIDTH	Width of new polysolids
PSTYLEMODE	Whether the drawing uses color-dependent
QTEXTMODE	Quick text
REFPATHTYPE	Path type of new references
REGENMODE	Automatic regeneration of the drawing
SETBYLAYERMODE	Properties SETBYLAYER sets to ByLayer
SHADEDGE	How faces and edges are shaded
SHADEDIF	Diffuse versus ambient light when shading, in %
SKETCHINC	SKETCH increment
SKPOLY	SKETCH type
SKTOLERANCE	SKETCH fit tolerance
SNAPISOPAIR	Current isometric plane
SNAPSTYL	Normal or isometric snap
SORTENTS	Which operations follow the draw order of objects
SPLFRAME	Whether the control frames of spline-fit polylines and invisible faces are shown
SPLINESEGQS	Segments of the surfaces of smoothed mesh polylines

SPLINESEGS	Segments per span when spline-fitting polylines
SPLINETYPE	Curve type when spline-fitting polylines
SURFTYPE	Surface type when smoothing meshes
SURFU	Density of smoothed surfaces in the M direction
SURFV	Density of smoothed surfaces in the N direction
TARGET	Target of the view
TEXTQLTY	Resolution of TrueType texts when plotting, from 0 to 100
TEXTSIZE	Default text height
TEXTSTYLE	Current text style
THICKNESS	Thickness of new objects
TILEMODE	Whether the Model tab
TRANSPARENCYDISPLAY	Display transparency
UCSICON	UCS icon
UNITMODE	Whether fractional and feet-and-inches units are shown as typed
USRTIMER	Whether the TIME user timer is running
VIEWCTR	Center of the view
VIEWDIR	Viewing direction
VIEWSIZE	Height of the view
VIEWTWIST	View twist
VISRETAIN	Changes to reference layers
VSCURRENT	Visual style of the current viewport
XCLIPFRAME	Boundary of clipped references
XFADECTL	Fading of external references

AFLAGS

Type: integer · Saved in: session · Initial value: 16

Modes the next attribute definition (ATTDEF) starts with; each definition leaves them as it used them. It is the sum of the values of the active modes:

Value	Mode
1	Invisible
2	Constant
4	Verify
8	Preset
16	Lock position

ANGBASE

Type: real · Stored in: drawing · Initial value: 0

Direction of angle 0, from which the angles written by the program are measured (in LIST, DIST, the PROPERTIES palette…)It is entered in degrees or in any angle format (`90`, `90d0'0"`, `100g`, `1.5708r`); getvar returns it in radians, as in AutoCAD.

- `0`: east (positive X axis). `90`: north. `180`: west. `270`: south.

Notes

Typed polar coordinates (`@50<30`) are always measured in degrees from the X axis, counterclockwise.

See also

UNITS · ANGDIR · AUNITS

ANGDIR

Type: integer · Stored in: drawing · Initial value: 0

Direction in which the angles written by the program increase, starting from ANGBASE.

Value	Direction
0	Counterclockwise
1	Clockwise

See also

UNITS · ANGBASE · AUNITS

ANNOAUTOSCALE

Type: integer · Stored in: session · Initial value: -4

When the annotation scale (CANNOSCALE) changes, adds the new scale to the annotative objects that supported the previous one, so that they keep their paper size.

Value	Objects the scale is added to
1	All, except those on layers that are off, frozen or locked
2	All, except those on layers that are off or frozen
3	All, except those on locked layers
4	All
0 or negative	None: the scale is not added

Notes

It applies to the session: it is not saved in the drawing (in AutoCAD it is a program setting). A negative value turns the feature off while remembering which value would be used, as in AutoCAD.

See also

CANNOSCALE · OBJECTSCALE · Annotation scales

ATTDIA

Type: integer · Stored in: drawing · Initial value: 0

Whether INSERT asks for attributes in a dialog box (1) or on the command line (0).

ATTMODE

Type: integer · Stored in: drawing · Initial value: 1

Visibility of block attributes.

Value	Attributes
0	All hidden
1	Normal: visible ones shown, invisible ones hidden
2	All shown, including invisible ones

Notes

When it changes, blocks are redrawn with the new value.

See also

ATTDEF · ATTEDIT · BATTMAN

ATTREQ

Type: integer · Stored in: drawing · Initial value: 1

Whether INSERT asks for attribute values (1) or uses the defaults (0).

AUNITS

Type: integer · Stored in: drawing · Initial value: 0

Format in which the program writes angles.

Value	Format	Example
0	Decimal degrees	`45.5`
1	Degrees/minutes/seconds	`45d30'0"`
2	Grads	`50.5556g`
3	Radians	`0.7941r`
4	Surveyor's units (bearings)	`N 44d30' E`

Angles can be typed in any of these formats whatever AUNITS is.

See also

UNITS · AUPREC · ANGBASE · angtos

AUPREC

Type: integer · Stored in: drawing · Initial value: 0

Number of decimal places of the angles written by the program, from 0 to 8. In degrees/minutes/seconds it sets how far they go: 0 degrees only, 1-2 minutes, 3-4 seconds and higher, decimals of the seconds.

See also

UNITS · AUNITS

AUTOSNAP

Type: integer · Saved in: session · Initial value: 63

Cursor markers and tracking, as a sum of bits. LinceCAD uses 8 (polar tracking, the POLAR button and F10) and 16 (object snap tracking, F11); the other bits are accepted and change nothing.

See also

POLAR · POLARANG · Object snaps and precision

BINDTYPE

Type: integer · Stored in: session · Initial value: 0

How the layers, linetypes, styles and blocks of an external reference are named when it is bound to the drawing with the Bind option of -XREF (or the button in XREF).

Value	Names
0	Bind: `reference$0$name` (the WALLS layer of the `base` reference becomes `base$0$WALLS`)
1	Insert: the name they had in the reference (`WALLS`); if it already exists in the drawing, the drawing's own is used

Notes

It applies to the session: it is not saved in the drawing (in AutoCAD it is a program setting).

See also

-XREF · XREF · External references

CANNOSCALE

Type: text · Stored in: drawing · Initial value: 1:1

Current annotation scale of model space: the name of a scale in the drawing's scale list (`1:50`). Annotative objects are shown with the representation of this scale and new ones are created with it: a 2.5 text at 1:50 measures 125 drawing units.

Notes

- It can also be changed with the scale selector on the status bar. The scale must be in the drawing's list (SCALELISTEDIT); otherwise you are warned:

```
Scale … is not in the drawing scale list.
```

- Layout viewports show annotative objects at their own scale, not this one.
- It is saved in the drawing as AutoCAD does (variable dictionary), so it is kept both ways.

See also

OBJECTSCALE · ANNOAUTOSCALE · DIMSCALE · Annotation scales

CECOLOR

Type: text · Stored in: drawing · Initial value: BYLAYER

Color given to new objects.

- `BYLAYER`: the color of the object's layer.
- `BYBLOCK`: the color of the block it is inserted in.
- A number from 1 to 255: an AutoCAD Color Index color (1 red, 2 yellow, 3 green, 4 cyan, 5 blue, 6 magenta, 7 white or black).

See also

COLOR · PROPERTIES · CLAYER

CELTSCALE

Type: real · Stored in: drawing · Initial value: 1

Linetype scale given to new objects. It multiplies LTSCALE: with 2, the dashes and gaps of new objects are twice as long as those of the rest. It must be greater than 0.

See also

LINETYPE · LTSCALE · CELTYPE

CELTYPE

Type: text · Stored in: drawing · Initial value: ByLayer

Linetype given to new objects: `BYLAYER` (the layer's), `BYBLOCK` (the block's) or the name of a linetype loaded in the drawing (`CONTINUOUS`, `DASHED`…)A linetype that is not loaded is rejected: load it first with LINETYPE.

See also

LINETYPE · CELTSCALE · LTSCALE

CENTERCROSSGAP

Type: string · Stored in: session · Initial value: 0.05x

Gap between the cross of the center mark and its extension lines. With a trailing `x` it is relative to the diameter of the circle or arc (0.05x, 5 %); without it, in drawing units.

See also

CENTERMARK · CENTERLINE

CENTERCROSSSIZE

Type: string · Stored in: session · Initial value: 0.1x

Length of the cross of the center mark. With a trailing `x` it is relative to the diameter (0.1x, 10 %); without it, in drawing units.

See also

CENTERMARK · CENTERLINE

CENTEREXE

Type: real · Stored in: session · Initial value: 3.5

How far the extension lines of a center mark go beyond the circle or arc, and how far a centerline goes beyond the ends of its lines.

See also

CENTERMARK · CENTERLINE

CENTERLAYER

Type: string · Stored in: session · Initial value: .

Layer on which center marks and centerlines are created; `.` is the current layer. It is also changed with the Layer option of both commands.

See also

CENTERMARK · CENTERLINE

CENTERLTSCALE

Type: real · Stored in: session · Initial value: 1

Linetype scale of new center marks and centerlines.

See also

CENTERMARK · CENTERLINE

CENTERLTYPE

Type: string · Stored in: session · Initial value: (empty)

Linetype of the extensions of center marks and of centerlines. Empty, CENTER2, which is loaded if it is not in the drawing.

See also

CENTERMARK · CENTERLINE

CENTERMARKEXE

Type: integer · Stored in: session · Initial value: 1

It is stored and read as in AutoCAD (1 on, 0 off). As in AutoCAD 2027, new center marks get their extension lines with either value.

See also

CENTERMARK · CENTERLINE

CETRANSPARENCY

Type: text · Stored in: drawing · Initial value: BYLAYER

Transparency given to new objects.

- `BYLAYER`: the transparency of the object's layer.
- `BYBLOCK`: the transparency of the block it is inserted in.
- A number from 0 (opaque) to 90: the transparency percentage.

Notes

- It is the one the Properties palette shows with nothing selected; it can also be changed there.
- Transparency is saved in DWG files from version 2004 (group code 440). When plotting, it goes into the PDF, SVG and PNG output.

See also

PROPERTIES · TRANSPARENCYDISPLAY · CECOLOR

CHAMFERA

Type: real · Stored in: drawing · Initial value: 0

First chamfer distance: the one trimmed from the first selected object. CHAMFER proposes it and stores it when it is changed. With both distances at 0, the objects are joined in a sharp corner.

See also

CHAMFER · CHAMFERB · FILLETRAD

CHAMFERB

Type: real · Stored in: drawing · Initial value: 0

Second chamfer distance: the one trimmed from the second selected object.

See also

CHAMFER · CHAMFERA

CLAYER

Type: text · Stored in: drawing · Initial value: 0

Current layer: new objects are created on it. Enter the name of an existing layer (case does not matter). A frozen layer cannot be made current.

See also

LAYER · LAYMCUR · LAYERP · CECOLOR

CMLJUST

Type: integer · Stored in: drawing · Initial value: 0

Justification of new multilines: 0 top, 1 middle, 2 bottom.

CMLSCALE

Type: real · Stored in: drawing · Initial value: 1

Scale of new multilines.

COORDS

Type: integer · Stored in: drawing · Initial value: 1

How cursor coordinates are shown on the status bar (0 to 3).

CTABLESTYLE

Type: text · Stored in: drawing · Initial value: Standard

Current table style: the one TABLE uses for new tables. Enter the name of an existing style.

See also

TABLE · TABLESTYLE

DELOBJ

Type: integer · Stored in: drawing · Initial value: 1

Whether commands that build objects from others (REGION, EXTRUDE…) delete the source ones.

DIMASSOC

Type: integer · Stored in: drawing · Initial value: 2

Associativity of new dimensions.

Value	New dimensions
0	Exploded: separate lines, arrowheads and text, as in AutoCAD
1	Not associated with the geometry
2	Associative: they follow the objects they measure when these are moved, stretched or scaled

Notes

- With 2, a dimension is associated when its points coincide with key points of the objects (endpoint, midpoint, center, quadrant, node). Changing the variable does not affect existing dimensions: use DIMREASSOCIATE and DIMDISASSOCIATE.

See also

DIMREASSOCIATE · DIMDISASSOCIATE · Dimensions

DIMCEN

Type: real · Stored in: drawing · Initial value: 2.5 (ISO-25)

Size of the center mark of DIMCENTER and of radius and diameter dimensions: positive, a cross of that half size; negative, also the center lines; 0, no mark.

Notes

When a dimension style is made current, this variable takes its value.

See also

DIMCENTER · DIMSTYLE

DIMSCALE

Type: real · Stored in: drawing · Initial value: 1

Overall dimension scale: it multiplies the text heights, arrow sizes and distances of the dimension style. A value of 0 or greater.

Notes

New dimensions use the DIMSCALE of their dimension style. When a style is made current, this variable takes its value; to rescale the dimensions, modify the style with DIMSTYLE.

See also

DIMSTYLE · DIMSTYLE

DIMSTYLE

Type: text · Stored in: drawing · Initial value: Standard

Current dimension style: the one used by new dimensions. When it changes, the dimension variables of the header (DIMSCALE, arrow size, text height…) take the values of the style, as in AutoCAD.

See also

DIMSTYLE · DIMSCALE · DIMLINEAR

DISPSILH

Type: integer · Stored in: drawing · Initial value: 0

With `1`, solids and surfaces (3DSOLID, BODY, SURFACE) show in wireframe the silhouette of their curved faces as seen from the viewing direction, besides their edges and isolines. With `0`, only edges and isolines.

Notes

- The silhouette is computed on the tessellation (see FACETRES): with a low FACETRES it may look jagged.

See also

ISOLINES · FACETRES

DRAGMODE

Type: integer · Stored in: drawing · Initial value: 2

Dragging of objects while points are picked: 0 off, 1 on request, 2 always.

DYNMODE

Type: integer · Saved in: session · Initial value: 3

Whether dynamic input (F12) is on: a positive value turns it on and a negative one off, as in AutoCAD (from -3 to 3; 0 also turns it off).

Notes

LinceCAD does not tell pointer input from dimensional input: any positive value turns both on and, when read, the variable is 3 or -3.

See also

OSNAP · ORTHO

ELEVATION

Type: real · Stored in: drawing · Initial value: 0

Elevation (Z coordinate) of new objects, in AutoCAD.

Notes

LinceCAD is a 2D program: it stores the value in the drawing for AutoCAD, but draws new objects at Z = 0.

See also

THICKNESS

EXTMAX

Type: 2D point · Stored in: drawing

Upper right corner of the drawing extents (read-only; updated on save).

EXTMIN

Type: 2D point · Stored in: drawing

Lower left corner of the drawing extents (read-only; updated on save).

FACETRES

Type: real · Stored in: drawing · Initial value: 0.5

How finely the curved faces of solids, regions and surfaces are divided into triangles, from 0.01 (very coarse) to 10 (very fine). With 0.5 a circle or a cylinder is divided into 6° steps; doubling the value halves the steps. It affects how they look and what is computed on the tessellation (silhouettes).

Notes

- B-spline surfaces are divided by their knot spans, more finely the higher FACETRES is.
- Faces the program cannot evaluate (rolling-ball fillets and other procedural surfaces without an approximation saved in the drawing) are shown only with their edges.

See also

ISOLINES · DISPSILH

FIELDDISPLAY

Type: integer · Stored in: session · Initial value: 1

Background of fields (FIELD) on screen.

Value	Fields
0	No background: they look like any other text
1	On a gray background, which is not plotted

Notes

It applies to the session: it is not saved in the drawing (in AutoCAD it is a program setting).

See also

FIELD · UPDATEFIELD · Text and styles

FILEDIA

Type: integer · Stored in: session · Initial value: 1

How commands ask for a file (GEOIMAGE, XREF, PDFIMPORT, MAPIMPORT, SCRIPT…).

Value	File
0	The path is typed on the command line; Enter, ~ or Browse open the dialog box
1	The Windows file dialog box opens directly, with the formats of the command

Notes

- In scripts (SCRIPT and `--guion`), AutoLISP routines and the API or MCP the path is still typed, even when FILEDIA is 1, as in AutoCAD.
- Cancelling the dialog box cancels the command.
- It applies to the session: it is not saved in the drawing (in AutoCAD it is a program setting).

See also

SCRIPT · FIELDDISPLAY

FILLETRAD

Type: real · Stored in: drawing · Initial value: 0

Fillet radius. FILLET proposes it and stores it when it is changed with its Radius option. With 0, the objects are trimmed or extended until they meet in a sharp corner.

See also

FILLET · CHAMFERA

FILLMODE

Type: integer · Stored in: drawing · Initial value: 1

In AutoCAD, whether solids, donuts and wide polylines are filled.

Value	Fill
0	Outline only
1	Filled

Notes

With 0 no fill is drawn (solid and gradient hatches, solids and wide polylines): solids keep their outline and wide polylines their centreline. It also applies when plotting.

See also

SOLID · DONUT · PLINE

GRIDMODE

Type: integer · Saved in: session · Initial value: 1

1 shows the grid and 0 hides it, like F7, GRID and the status bar button.

See also

GRID · SNAPMODE

HALOGAP

Type: integer · Stored in: drawing · Initial value: 0

Gap left by objects hidden behind others, as a % of a unit.

HIDETEXT

Type: integer · Stored in: drawing · Initial value: 1

Whether HIDE takes texts into account.

HPGAPTOL

Type: real · Stored in: session · Initial value: 0

Hatch gap tolerance: when looking for the boundary with an internal point, HATCH closes gaps of up to this length between loose endpoints, as if they were joined. With 0, the boundary must be fully closed. It accepts 0 to 5000, in drawing units.

Notes

As in AutoCAD, it is not saved in the drawing. It only counts for boundaries found from an internal point, not for selected objects.

See also

HATCH · BOUNDARY · Hatches

IMAGEFRAME

Type: integer · Stored in: drawing · Initial value: 1

Frame of the raster images in the drawing.

Value	Frame
0	Hidden (images cannot be selected by their edge)
1	Displayed and plotted
2	Displayed but not plotted

Notes

With 2 the frame shows on screen but is not plotted or exported. The setting is stored in the image dictionary of the drawing, as in AutoCAD.

See also

IMAGEATTACH · IMAGECLIP · IMAGEADJUST

INDEXCTL

Type: integer · Stored in: drawing · Initial value: 0

Layer and spatial indexes saved with the drawing (0 to 3).

INSBASE

Type: point · Stored in: drawing · Initial value: 0,0

Base point of the drawing: the point placed at the insertion point when this file is inserted as a block into another drawing. It is entered as `x,y` coordinates.

Notes

WBLOCK stores the base point of the block as the INSBASE of the new file.

See also

INSERT · WBLOCK

INSUNITS

Type: integer · Stored in: drawing · Initial value: 0

Unit in which the drawing is drawn. When an image is attached with IMAGEATTACH, LinceCAD uses it together with the resolution of the file to give the image its real size; AutoCAD also uses it to scale blocks and references inserted from drawings in another unit.

Value	Unit	Value	Unit
0	Unitless	11	Angstroms
1	Inches	12	Nanometers
2	Feet	13	Microns
3	Miles	14	Decimeters
4	Millimeters	15	Decameters
5	Centimeters	16	Hectometers
6	Meters	17	Gigameters
7	Kilometers	18	Astronomical units
8	Microinches	19	Light years
9	Mils	20	Parsecs
10	Yards		

See also

UNITS · INSERT · IMAGEATTACH

ISOLINES

Type: integer · Stored in: drawing · Initial value: 4

Number of isolines per direction used to draw the curved faces of solids and surfaces in wireframe (0 to 2047): lines of constant parameter on the surface, such as the rulings of a cylinder or the meridians and parallels of a sphere. With `0` only the edges are shown.

See also

FACETRES · DISPSILH

LAYOUTCREATEVIEWPORT

Type: integer · Saved in: session · Initial value: 1

Whether new layouts get a viewport showing the model extents: `0` no, `1` yes.

See also

LAYOUT · MVIEW

LAYLOCKFADECTL

Type: integer · Saved in: session · Initial value: 50

Fading (0 to 90) of the objects on locked layers; LAYISO with Lock and fade sets it too.

Notes

LinceCAD keeps the value, but the view does not fade locked layers yet.

See also

LAYISO · LAYLCK

LIMCHECK

Type: integer · Stored in: drawing · Initial value: 0

Whether drawing outside the drawing limits is checked.

Value	Checking
0	Off
1	On

See also

LIMITS · LIMMIN · LIMMAX

LIMMAX

Type: 2D point · Stored in: drawing · Initial value: 420,297

Upper right corner of the model space limits.

See also

LIMITS · LIMMIN

LIMMIN

Type: 2D point · Stored in: drawing · Initial value: 0,0

Lower left corner of the model space limits.

See also

LIMITS · LIMMAX

LTSCALE

Type: real · Stored in: drawing · Initial value: 1

Global linetype scale: it multiplies the length of the dashes and gaps of all objects. For drawings in millimeters at 1:100, a value around 100 makes dashed lines look dashed. It must be greater than 0.

See also

LINETYPE · CELTSCALE

LUNITS

Type: integer · Stored in: drawing · Initial value: 2

Format in which the program writes lengths and coordinates.

Value	Format	Example
1	Scientific	`1.5500E+01`
2	Decimal	`15.5000`
3	Engineering	`1'-3.5000"`
4	Architectural	`1'-3 1/2"`
5	Fractional	`15 1/2`

In the engineering and architectural formats the drawing unit is the inch, as in AutoCAD.

See also

UNITS · LUPREC · rtos

LUPREC

Type: integer · Stored in: drawing · Initial value: 4

Number of decimal places of the lengths written by the program, from 0 to 8. In the fractional formats it sets the smallest denominator: 1 halves, 2 quarters, 3 eighths…up to 8 (1/256).

See also

UNITS · LUNITS

LWDISPLAY

Type: integer · Stored in: drawing · Initial value: 0

Whether the lineweights of objects are displayed on screen.

Value	Lineweights
0	Not displayed: everything with the same width
1	Displayed

Notes

LinceCAD stores the value in the drawing and the view follows it: the status bar toggle (LWDISPLAY) changes it too.

See also

LWEIGHT · LWDISPLAY

MAXACTVP

Type: integer · Stored in: drawing · Initial value: 64

Maximum number of viewports active at once in a layout.

MEASUREMENT

Type: integer · Stored in: drawing · Initial value: 1

Drawing units for hatch patterns and linetypes: 0 imperial, 1 metric.

MIRRTEXT

Type: integer · Stored in: drawing · Initial value: 0

How MIRROR handles text, multiline text and attributes.

Value	Text
0	Moved, but still reads normally
1	Mirrored like everything else (reads backwards)

See also

MIRROR · ALIGN

ORTHOMODE

Type: integer · Saved in: session · Initial value: 0

1 turns on Ortho mode (F8, ORTHO): the cursor moves horizontally or vertically from the last point. Turning it on turns polar tracking off, as in AutoCAD.

See also

ORTHO · AUTOSNAP

OSMODE

Type: integer · Saved in: session · Initial value: 4133

Running object snaps, as a sum: 1 endpoint, 2 midpoint, 4 center, 8 node, 16 quadrant, 32 intersection, 64 insertion, 128 perpendicular, 256 tangent, 512 nearest and 4096 extension. Adding 16384 turns them off without losing the modes (F3, OSNAP).

See also

OSNAP · Object snaps and precision

PDFFRAME

Type: integer · Stored in: session · Initial value: 1

Frame of the PDF underlays in the drawing.

Value	Frame
0	Hidden (the frame of an underlay whose file is not found is shown anyway)
1	Displayed and plotted
2	Displayed but not plotted

Notes

With 2 the frame shows on screen but is not plotted or exported. The setting applies to the session (AutoCAD saves it in the drawing).

See also

PDFATTACH · PDFCLIP · IMAGEFRAME · XCLIPFRAME

PDMODE

Type: integer · Stored in: drawing · Initial value: 0

Symbol used to draw point objects. It is the sum of a figure and, optionally, an outline.

Value	Figure
0	A dot
1	Nothing
2	Cross (+)
3	X
4	Short vertical tick

Add	Outline
32	Circle
64	Square

For example, 35 = an X inside a circle. Valid values: 0-4, 32-36, 64-68 and 96-100.

See also

PTYPE · POINT · DIVIDE · MEASURE · PDSIZE

PDSIZE

Type: real · Stored in: drawing · Initial value: 0

Size of the point symbol (see PDMODE).

- `0`: 5 % of the window height.
- Greater than 0: size in drawing units (it grows and shrinks with the zoom).
- Less than 0: percentage of the window height (`-10` = 10 %), the same at any zoom.

See also

PTYPE · POINT · PDMODE

PELEVATION

Type: real · Stored in: drawing · Initial value: 0

Paper space elevation.

PELLIPSE

Type: integer · Stored in: drawing · Initial value: 0

Whether ELLIPSE creates true ellipses (0) or polylines (1).

PICKSTYLE

Type: integer · Stored in: drawing · Initial value: 1

Whether selecting a member of a group selects the whole group.

Value	Selection
0	Single objects
1	Whole groups (the selectable ones)
2	Single objects (in AutoCAD, associative hatches with their boundary)
3	Whole groups (in AutoCAD, associative hatches too)

Ctrl+Shift+A turns group selection on and off.

Notes

LinceCAD only uses the group part; values 2 and 3 are kept for AutoCAD.

See also

GROUP · GROUPEDIT · UNGROUP

PLINEGEN

Type: integer · Stored in: drawing · Initial value: 0

Whether the linetype of new polylines runs along the whole polyline (1) or restarts at each vertex (0).

PLINEWID

Type: real · Stored in: drawing · Initial value: 0

Default width of new polylines, in AutoCAD. A value of 0 or greater.

Notes

PLINE and POLYGON create polylines with this width, and the PLINE Width option proposes it.

See also

PLINE · PEDIT

POLYSIDES

Type: integer · Stored in: session · Initial value: 4

Number of sides that POLYGON proposes: the last one used (3 to 1024), as in AutoCAD.

See also

POLYGON

POLARANG

Type: real · Saved in: session · Initial value: 90

Polar tracking increment angle, in degrees (the one in the POLAR button menu).

See also

POLAR · AUTOSNAP

PROXYGRAPHICS

Type: integer · Stored in: drawing · Initial value: 1

Whether proxy object images are saved in the drawing.

PSLTSCALE

Type: integer · Stored in: drawing · Initial value: 1

Whether linetype scaling in viewports follows each viewport scale.

PSOLHEIGHT

Type: real · Stored in: drawing · Initial value: 80

Height of new polysolids.

PSOLWIDTH

Type: real · Stored in: drawing · Initial value: 5

Width of new polysolids.

PSTYLEMODE

Type: integer · Stored in: drawing · Initial value: 1

Whether the drawing uses color-dependent (1) or named (0) plot styles (read-only).

QPMODE

Type: integer · Stored in: session · Initial value: 1

Whether the quick properties panel appears when objects are selected with no command running.

Value	Quick properties panel
0	Off
1	On

Notes

The panel is a reduced, floating version of the PROPERTIES palette: next to the cursor it shows the layer, the color and the key properties of the selected type (radius, length, contents…), and it closes when the selection is cleared or a command starts. With objects of several types it only shows the common properties. It does not appear in test scripts.

See also

QUICKPROPERTIES · PROPERTIES

QTEXTMODE

Type: integer · Stored in: drawing · Initial value: 0

Quick text: whether texts are shown as rectangles.

REFPATHTYPE

Type: integer · Stored in: session · Initial value: 1

How the path of the external references and PDF underlays you attach is saved.

Value	Path
0	No path: just the file name, which is looked for in the drawing's folder
1	Relative to the drawing's folder
2	Full

Notes

- XATTACH offers it in its pathType option. PDFATTACH saves a relative path with 1 and a full one with 0 or 2.
- If the drawing has not been saved yet or the file is on another drive, the relative path is saved as a full one.
- It applies to the session: it is not saved in the drawing (in AutoCAD it is a program setting).

See also

XATTACH · -XREF · External references

REGENMODE

Type: integer · Stored in: drawing · Initial value: 1

Automatic regeneration of the drawing.

SETBYLAYERMODE

Type: integer · Saved in: session · Initial value: 255

Which properties SETBYLAYER sets to ByLayer: the sum of 1 (color), 2 (linetype), 4 (lineweight), 8 (material), 16 (plot style) and 128 (transparency). Bits 32 (change ByBlock too) and 64 (include blocks) are kept, but the command asks for them every time.

See also

SETBYLAYER

SHADEDGE

Type: integer · Stored in: drawing · Initial value: 3

How faces and edges are shaded (0 to 3).

SHADEDIF

Type: integer · Stored in: drawing · Initial value: 70

Diffuse versus ambient light when shading, in %.

SKETCHINC

Type: real · Stored in: session · Initial value: 1

Distance between the points that SKETCH records when the mouse moves with the pen down (Increment option).

See also

SKETCH

SKPOLY

Type: integer · Stored in: session · Initial value: 0

What SKETCH creates: 0 lines, 1 polylines and 2 splines (Type option).

See also

SKETCH

SKTOLERANCE

Type: real · Stored in: session · Initial value: 0.5

How far the spline of SKETCH may deviate from the points of the stroke (fitTolerance option; only with SKPOLY = 2).

See also

SKETCH

SNAPISOPAIR

Type: integer · Stored in: drawing · Initial value: 0

Current isometric plane, the one followed by ortho, the grid, snap and the crosshairs when isometric drafting is on.

Value	Plane
0	Left (axes at 90° and 150°)
1	Top (axes at 30° and 150°)
2	Right (axes at 30° and 90°)

Notes

F5 and Ctrl+E switch to the next plane. It is saved in the drawing's current viewport, as in AutoCAD.

See also

ISOPLANE · ISODRAFT · SNAPSTYL

SNAPMODE

Type: integer · Saved in: session · Initial value: 0

1 snaps the cursor to the grid nodes (F9, SNAP).

See also

SNAP · GRIDMODE

SNAPSTYL

Type: integer · Stored in: drawing · Initial value: 0

Snap and grid style: normal or isometric.

Value	Style
0	Rectangular: horizontal and vertical axes (or those of the UCS)
1	Isometric, in the plane of SNAPISOPAIR

Notes

It is what ISODRAFT and the ISO button on the status bar change. It is saved in the drawing's current viewport, as in AutoCAD.

See also

ISODRAFT · ISOPLANE · SNAP · Object snaps and precision

SORTENTS

Type: integer · Stored in: drawing · Initial value: 127

Which operations follow the draw order of objects (sum of bits).

SPLFRAME

Type: integer · Stored in: drawing · Initial value: 0

Whether the control frames of spline-fit polylines and invisible faces are shown.

SPLINESEGQS

Type: integer · Stored in: drawing · Initial value: 0

Segments of the surfaces of smoothed mesh polylines.

SPLINESEGS

Type: integer · Stored in: drawing · Initial value: 8

Number of straight segments with which the Spline option of PEDIT approximates each span of the polyline. More segments give a smoother curve. From 1 to 32767.

See also

PEDIT · SPLINETYPE

SPLINETYPE

Type: integer · Stored in: drawing · Initial value: 6

Curve type used by the Spline option of PEDIT.

Value	Curve
5	Quadratic B-spline
6	Cubic B-spline

See also

PEDIT · SPLINESEGS · SPLINE

SURFTYPE

Type: integer · Stored in: drawing · Initial value: 6

Surface type when smoothing meshes: 5 quadratic B-spline, 6 cubic, 8 Bezier.

SURFU

Type: integer · Stored in: drawing · Initial value: 6

Density of smoothed surfaces in the M direction.

SURFV

Type: integer · Stored in: drawing · Initial value: 6

Density of smoothed surfaces in the N direction.

TEXTQLTY

Type: integer · Stored in: drawing · Initial value: 50

Resolution of TrueType texts when plotting, from 0 to 100.

TEXTSIZE

Type: real · Stored in: drawing · Initial value: 2.5

Default text height, when the current text style has no fixed height. TEXT, MTEXT and ATTDEF propose it and store here the last height chosen; it is also used by the attributes created by BATTMAN and by texts created with entmake without a height. It must be greater than 0.

See also

ATTDEF · TEXT · STYLE · TEXTSTYLE

TEXTSTYLE

Type: text · Stored in: drawing · Initial value: Standard

Current text style: the one used by new text and attributes. Enter the name of an existing style.

See also

STYLE · TEXT · MTEXT · TEXTSIZE

THICKNESS

Type: real · Stored in: drawing · Initial value: 0

Extrusion thickness (in Z) of new objects, in AutoCAD.

Notes

LinceCAD is a 2D program: it stores the value in the drawing for AutoCAD, but creates objects without thickness.

See also

ELEVATION

THUMBSAVE

Type: integer · Stored in: session · Initial value: 1

Whether saving a DWG includes the drawing's preview (thumbnail).

Value	Preview
0	Not updated: the DWG keeps the one it had
1	Redone on save from the model view

Notes

The thumbnail is shown by the Open dialog and by the file explorer. It is only written in DWG version 2013 or later (in earlier ones LinceCAD cannot write it). It is a picture of the whole model on white paper, 256 × 256 px.

TILEMODE

Type: integer · Stored in: drawing · Initial value: 1

Whether the Model tab (1) or a layout (0) is active (read-only: change it with the tabs).

TRANSPARENCYDISPLAY

Type: integer · Stored in: session · Initial value: 1

Whether the transparency of objects and layers is displayed on screen.

Value	Transparency
0	Not displayed: everything is drawn opaque
1	Displayed

Notes

- It is the same as the TRANSP button on the status bar.
- It only changes how the drawing looks: it does not change the objects or the plotted output.
- It lasts for the session: it is not saved in the drawing and goes back to 1 when the program starts.

See also

CETRANSPARENCY · PROPERTIES · LAYER

UCSICON

Type: integer · Stored in: drawing · Initial value: 3

UCS icon in model space. It is the sum of these values:

Value	Icon
0	Hidden
1	Displayed
2	At the UCS origin, if it is visible on screen (otherwise, at the lower left)

With 3, displayed and at the origin.

See also

UCSICON · UCSMAN · UCS

UNITMODE

Type: integer · Stored in: drawing · Initial value: 0

Whether fractional and feet-and-inches units are shown as typed (1) or with spaces (0).

USRTIMER

Type: integer · Stored in: drawing · Initial value: 1

Whether the TIME user timer is running.

VISRETAIN

Type: integer · Stored in: drawing · Initial value: 1

What happens to the layers of external references (`reference|layer`) when they are reloaded.

Value	Layers
0	They take again the color, linetype and state they have in the reference file
1	They keep what has been changed in this drawing (color, on, frozen...)

See also

XREF · LAYER · External references

XCLIPFRAME

Type: integer · Stored in: drawing · Initial value: 2

Boundary of the external references and blocks clipped with XCLIP.

Value	Boundary
0	Hidden
1	Displayed and plotted
2	Displayed but not plotted

Notes

With 2 the boundary shows on screen but is not plotted or exported. It is drawn on the layer and in the color of the insert.

See also

XCLIP · PDFFRAME · IMAGEFRAME

XFADECTL

Type: integer · Stored in: session · Initial value: 70

How much the content of external references (what is on `reference|layer` layers) is faded on screen, from 0 (not at all) to 90 %, to tell it apart from what is drawn in this drawing.

Notes

- It only affects the screen: when plotting, references come out in their colors.
- In AutoCAD this fading is XDWGFADECTL; its XFADECTL is the one for in-place reference editing, which LinceCAD does not have.
- It applies to the session: it is not saved in the drawing.

See also

XREF · XATTACH · External references

LENSLENGTH

Type: real · Stored in: drawing · Read-only · Initial value: 50

Lens length in millimeters of the perspective view of the current viewport (with a 35 mm film).

See also

PERSPECTIVE · 3DORBIT

PERSPECTIVE

Type: integer · Stored in: drawing · Initial value: 0

Value	View
0	Parallel
1	Perspective (zooming moves the viewer towards the target)

See also

LENSLENGTH · 3DORBIT · -VIEW

TARGET

Type: 3D point · Stored in: drawing · Read-only · Initial value: 0,0,0

Target of the view of the current viewport, in the WCS: the point 3DORBIT rotates around.

VIEWCTR

Type: 3D point · Stored in: drawing · Read-only

Center of the view of the current viewport, on the XY plane along the viewing direction, as AutoCAD gives it.

VIEWDIR

Type: 3D vector · Stored in: drawing · Read-only · Initial value: 0,0,1

Viewing direction of the current viewport, from the target to the viewer. It is changed with VPOINT, -VIEW, 3DORBIT or PLAN.

VIEWSIZE

Type: real · Stored in: drawing · Read-only

Height of the view of the current viewport, in drawing units.

VIEWTWIST

Type: angle · Stored in: drawing · Read-only · Initial value: 0

Twist of the view of the current viewport (set by PLAN with a rotated UCS and by 3DFORBIT).

VSCURRENT

Type: text · Stored in: drawing (in the viewport) · Initial value: 2dWireframe

Name of the visual style of the current viewport; changing it does the same as VSCURRENT.

OFFSETDIST

Type: real · Stored in: session · Initial value: -1

Distance for OFFSET. A negative value means "Through" (the default option): the offset passes through the point you give. OFFSET remembers it and offers it between < >.

See also

OFFSET · OFFSETGAPTYPE

OFFSETGAPTYPE

Type: integer · Stored in: session · Initial value: 0

How OFFSET closes the gaps between polyline segments: 0 extends, 1 fillets, 2 chamfers. It is stored, but offset always applies "extend".

See also

OFFSET · OFFSETDIST

SELECTSIMILARMODE

Type: integer · Stored in: session · Initial value: 130

Properties that must match in SELECTSIMILAR, besides the object type: the sum of 1 color, 2 layer, 4 linetype, 8 linetype scale, 16 lineweight, 32 plot style, 64 object style (text, dimension or table style) and 128 name (of a

block). 130 is layer and name, as in AutoCAD.

See also

SELECTSIMILAR

ARRAYASSOCIATIVITY

Type: integer · Stored in: session · Initial value: 1

Whether new arrays of ARRAY, ARRAYRECT and ARRAYPOLAR are associative (1) or separate objects (0). The Associative option of those prompts changes it, as in AutoCAD.

See also

ARRAY · ARRAYEDIT

TRIMEXTENDMODE

Type: integer · Stored in: session · Initial value: 1

Mode of TRIM and EXTEND: 1 quick (every object is an edge, none to select) and 0 standard (edges are selected).

See also

TRIM · EXTEND · EDGEMODE · PROJMODE

EDGEMODE

Type: integer · Stored in: session · Initial value: 0

Edges of TRIM and EXTEND in standard mode: 0 does not extend them and 1 extends them (Edge > Extend option).

See also

TRIM · TRIMEXTENDMODE

PROJMODE

Type: integer · Stored in: session · Initial value: 1

Projection of TRIM and EXTEND: 0 none, 1 UCS and 2 view. In 2D it does not change the result.

See also

TRIM · TRIMEXTENDMODE

TRIMMODE

Type: integer · Stored in: session · Initial value: 1

Whether FILLET and CHAMFER trim the objects (1) or only add the arc or the chamfer (0).

See also

FILLET · CHAMFER · CHAMMODE

CHAMMODE

Type: integer · Stored in: session · Initial value: 0

Method of CHAMFER: 0 two distances and 1 length and angle.

See also

CHAMFER · CHAMFERC · CHAMFERD

CHAMFERC

Type: real · Stored in: session · Initial value: 0

Chamfer length on the first line when the method is angle.

See also

CHAMFER · CHAMMODE · CHAMFERD

CHAMFERD

Type: real · Stored in: session · Initial value: 0

Chamfer angle with the first line (in radians) when the method is angle. CHAMFER asks for it in degrees.

See also

CHAMFER · CHAMMODE · CHAMFERC

COPYMODE

Type: integer · Stored in: session · Initial value: 0

Mode of COPY: 0 multiple (keeps asking for points) and 1 single (one copy and it ends).

See also

COPY · MOVE

AutoLISP

AutoLISP

LinceCAD has its own AutoLISP interpreter to run routines written for AutoCAD or progeCAD: the same functions, the same data types and the same error messages ("; error: bad argument type…" as in AutoCAD, because many routines compare them in their `*error*`). Routines ask for points and select objects like any command, with object snaps, rubber band and typed coordinates.

Typing expressions on the command line

When no command is active, anything that starts with `(` is evaluated as AutoLISP and the result is written below. If parentheses or quotes are left open, the program shows `(_>` and waits for more lines.

```
(setq r (* 2 pi))
6.28319
```

Loading .lsp files

- APPLOAD opens the Load Applications (APPLOAD) dialog box: **Load a .lsp file…** loads a file now; the **startup suite** is loaded every time the program starts.
- load loads a file from a routine or from the command line: `(load "myroutine")`.
- When a drawing is opened, `acaddoc.lsp` and `lince.lsp` are loaded automatically if they are in its folder. At startup, those in the user folder are loaded too.
- Security (like SECURELOAD): files in the **trusted folders** (the user folder, the `lisp` folder next to the program and the ones added in APPLOAD) are loaded without asking; files in other folders are confirmed first. APPLOAD can also allow loading from any folder or only from trusted ones.
- Files can be in UTF-8 or in Windows-1252, like those of older AutoCAD versions.

New commands with c:

A function whose name starts with `c:` becomes a command: type its name on the command line like any other.

```
(defun c:area2 (/ e)
  (setq e (car (entsel "\nSelect a circle: ")))
  (princ (strcat "\nArea: " (rtos (* pi (expt (cdr (assoc 40 (entget e))) 2)) 2 2)))
  (princ))
```

Data types

32-bit integers (which overflow as in AutoCAD), reals, strings, symbols, lists and dotted pairs, entity names, selection sets, files and functions. Angles are in radians; points are lists of two or three numbers. Comments go from `;` to the end of the line or between `;|` and `|;`. Strings accept `\n`, `\t`, `\"`, `\\` and `\e`.

Limits

- There are no ActiveX functions (`vla-…`, `vlax-…`), DCL dialog boxes, reactors or separate namespaces per drawing.
- entget and entmod work with LINE, CIRCLE, ARC, LWPOLYLINE, TEXT, MTEXT, POINT, INSERT and ATTRIB; entmake creates the same ones except ATTRIB; DIMENSION can be read but not changed; for other entities only the common codes (layer, color…) are read. There is no extended data (XDATA).
- trans converts between the WCS and the UCS; with an entity name or with paper space it returns the point

unchanged.

- While a command is active, `(` is not evaluated (AutoCAD evaluates it and uses the result as the answer). There is no `!variable` either to see the value of a variable: type `(princ variable)`.
- Expressions can be nested up to about 8000 levels (recursion included). Esc stops a running routine.

Functions by family

Special forms and control: quote · function · lambda · setq · defun · defun-q · if · cond · while · repeat · foreach · progn · and · or · t · pi · pause · error

Arithmetic: + · - · * · / · 1+ · 1- · abs · fix · float · rem · gcd · min · max · sqrt · expt · exp · log · sin · cos · atan · ~ · logand · logior · lsh

Comparison and predicates: = · /= · < · > · <= · >= · eq · equal · not · null · atom · listp · vl-consp · numberp · zerop · minusp · boundp · type · vl-symbolp

Lists: car · cdr · cadr and the other c…r · cons · list · append · reverse · length · vl-list-length · nth · member · assoc · subst · last · vl-remove · vl-position · vl-sort · vl-remove-if · vl-remove-if-not · vl-some · vl-every · vl-member-if · acad_strlsort · mapcar · apply

Evaluation and symbols: eval · set · read · vl-symbol-name · vl-symbol-value

Strings: strcat · strlen · substr · strcase · itoa · atoi · atof · chr · ascii · wcmatch · vl-string-trim · vl-string-left-trim · vl-string-right-trim · vl-string-search · vl-string-subst · vl-string-translate · vl-string-position · vl-string->list · vl-list->string · vl-string-elt · vl-princ-to-string · vl-prin1-to-string

Units: rtos · distof · angtos · angtof

Geometry: distance · angle · polar · inters · trans

User input: initget · getpoint · getcorner · getdist · getangle · getorient · getreal · getint · getstring · getkword · entsel

Messages: princ · prin1 · print · terpri · prompt · alert · textscr · graphscr

Commands: command · command-s · vl-cmdf

Entities: entlast · entnext · entget · entmod · entmake · entdel · entupd · handent

Selection sets: ssget · ssname · sslength · ssadd · ssdel · ssmemb

Tables, variables and display: tblsearch · tblnext · getvar · setvar · redraw · regen

Files and loading: open · close · read-line · write-line · findfile · getenv · load · vl-filename-base · vl-filename-directory · vl-filename-extension

Errors and system: exit · quit · vl-catch-all-apply · vl-catch-all-error-p · vl-catch-all-error-message · vl-load-com · ver · gc

quote

`(quote expression)` or `'expression`

Returns the expression without evaluating it.

```
'(1 2 3)          ; (1 2 3)
(quote a)         ; A
```

function

`(function symbol-or-lambda)`

Like quote, but tells that the argument is a function. Used with mapcar and the like.

```
(mapcar (function (lambda (x) (* x x))) '(1 2 3))   ; (1 4 9)
```

lambda

`(lambda (arguments [/ locals]) expression...)`

Creates an unnamed function. Returns the function.

```
((lambda (a b) (+ a b)) 2 3)   ; 5
```

setq

`(setq symbol expression [symbol expression]...)`

Evaluates each expression and assigns it to the symbol before it, which is not evaluated. Returns the last value assigned.

```
(setq a 10 b (* a 2))   ; 20
```

defun

`(defun name (arguments [/ locals]) expression...)`

Defines a function. The arguments and the local variables (after `/`) get their previous value back on exit; other variables are global. Returns the name. If the name starts with `C:`, the function becomes a command.

```
(defun double (x) (* 2 x))
(double 4)   ; 8
```

defun-q

`(defun-q name (arguments [/ locals]) expression...)`

Same as defun.

Differences with AutoCAD

In AutoCAD the function is kept as a list and can be modified with `defun-q-list-set`; in LinceCAD it is a normal function.

if

`(if test then [else])`

Evaluates the test; if it is not nil, evaluates and returns `then`; if it is nil, `else` (or nil if missing).

```
(if (> a 0) "positive" "zero or negative")
```

cond

`(cond (test expression...) ...)`

Evaluates the tests in order and, for the first one that is not nil, its expressions. Returns the last one evaluated (or the test, if the clause has no expressions). `T` as the last test gives the default case.

```
(cond ((< x 0) "negative") ((= x 0) "zero") (T "positive"))
```

while

`(while test expression...)`

Repeats the expressions while the test is not nil. Returns the value of the last expression evaluated.

```
(setq i 0)
(while (< i 3) (setq i (1+ i)))   ; 3
```

repeat

`(repeat integer expression...)`

Evaluates the expressions the given number of times. Returns the value of the last one.

```
(setq n 1)
(repeat 4 (setq n (* n 2)))   ; 16
```

foreach

`(foreach symbol list expression...)`

Gives the symbol each element of the list and evaluates the expressions. Returns the value of the last one; the symbol gets its value back at the end.

```
(foreach c '("A" "B") (princ c))   ; prints AB
```

progn

`(progn expression...)`

Evaluates the expressions in order and returns the last one. Used to put several where one is expected, as in if.

```
(if ok (progn (princ "Done") (setq n (1+ n))))
```

and

`(and expression...)`

Evaluates the expressions until one is nil. Returns T if none is nil (also with no arguments) and nil otherwise.

```
(and (> 3 1) "text")   ; T
```

or

`(or expression...)`

Evaluates the expressions until one is not nil. Returns T if it finds one and nil if all are nil.

```
(or nil 5)   ; T
```

t

`T`

Symbol whose value is T: the "true" value of tests. Predicates that hold return it.

```
(= 1 1)   ; T
```

pi

`pi`

Variable holding the number ? (3.14159…)It can be changed with setq, as in AutoCAD, though it is not a good idea.

```
(* 2 pi)   ; 6.28319
```

pause

`pause`

Variable whose value is `"\\"`. In command it makes the command wait for the user to give that input.

```
(command "_circle" pause 10)
```

error

`(defun *error* (message) ...)`

Function the program calls when a routine fails, with the error message, before unwinding the stack: the local variables of the running functions still have their values. Used to restore variables and close files. Without `*error*`, the program writes `; error: message`.

```
(defun *error* (msg)
  (if (/= msg "Function cancelled") (princ (strcat "\nError: " msg)))
  (princ))
```

+

`(+ [number number]...)`

Adds the numbers. Integer if all are integers (32-bit, overflowing as in AutoCAD); real if any is real. With no arguments, 0.

```
(+ 1 2 3)     ; 6
(+ 1 2.5)     ; 3.5
```

-

`(- [number number]...)`

Subtracts the others from the first. With a single number, changes its sign. With no arguments, 0.

```
(- 10 3 2)   ; 5
```

```
(- 4)        ; -4
```

*

`(* [number number]...)`

Multiplies the numbers. Integer if all are integers. With no arguments, 0.

```
(* 2 3 4)   ; 24
```

/

`(/ [number number]...)`

Divides the first by the others. Between integers, the result is an integer and is truncated; with any real, real. Dividing by zero gives the error `divide by zero`.

```
(/ 7 2)     ; 3
(/ 7 2.0)   ; 3.5
```

1+

`(1+ number)`

Adds 1 to the number.

```
(1+ 5)   ; 6
```

1-

`(1- number)`

Subtracts 1 from the number.

```
(1- 5)   ; 4
```

abs

`(abs number)`

Absolute value, of the same type as the number.

```
(abs -2.5)   ; 2.5
```

fix

`(fix number)`

Integer part of the number, truncated toward zero, as an integer. If it does not fit in 32 bits, returns a truncated real.

```
(fix 3.7)    ; 3
(fix -3.7)   ; -3
```

float

`(float number)`

The number as a real.

```
(float 3)   ; 3.0
```

rem

`(rem number number...)`

Remainder of dividing the first by the others, from left to right. Integer if all are integers.

```
(rem 17 5)     ; 2
(rem 7.5 2)    ; 1.5
```

gcd

`(gcd integer integer)`

Greatest common divisor of two integers.

```
(gcd 12 18)   ; 6
```

min

`(min number...)`

The smallest of the numbers. If any is real, the result is real.

```
(min 3 1 2)     ; 1
(min 3 1.0)     ; 1.0
```

max

`(max number...)`

The largest of the numbers. If any is real, the result is real.

```
(max 3 1 2)   ; 3
```

sqrt

`(sqrt number)`

Square root, as a real. A negative number is an error.

```
(sqrt 16)   ; 4.0
```

expt

`(expt base exponent)`

Power. Integer if both are integers and the exponent is not negative; otherwise real.

```
(expt 2 10)    ; 1024
(expt 2 0.5)   ; 1.41421
```

exp

`(exp number)`

e raised to the number, as a real.

```
(exp 1)   ; 2.71828
```

log

`(log number)`

Natural logarithm of a positive number, as a real.

```
(log 10)   ; 2.30259
```

sin

`(sin angle)`

Sine of an angle in radians.

```
(sin (/ pi 2))   ; 1.0
```

cos

`(cos angle)`

Cosine of an angle in radians.

```
(cos 0)   ; 1.0
```

atan

`(atan y [x])`

Arctangent in radians. With one number, between -?/2 and ?/2; with two, the angle of `y/x` in the right quadrant, between -? and ?.

```
(atan 1)      ; 0.785398
(atan 1 -1)   ; 2.35619
```

~

`(~ integer)`

Bitwise NOT of the integer.

```
(~ 5)   ; -6
```

logand

`(logand [integer]...)`

Bitwise AND of the integers. Useful to test a bit of a value such as OSMODE.

```
(logand 7 12)   ; 4
```

logior

`(logior [integer]...)`

Bitwise inclusive OR of the integers.

```
(logior 1 4)   ; 5
```

lsh

`(lsh integer bits)`

Shifts the bits of the integer to the left (or to the right, if `bits` is negative).

```
(lsh 1 4)    ; 16
(lsh 16 -2)  ; 4
```

=

`(= value value...)`

T if all the values are equal. Compares numbers (an integer and a real are equal if their values are) or strings.

```
(= 2 2.0)       ; T
(= "a" "A")     ; nil
```

/=

`(/= value value...)`

T if each value is different from the next one. Numbers or strings.

```
(/= 1 2)   ; T
```

<

`(< value value...)`

T if each value is less than the next one. Numbers or strings (by character order).

```
(< 1 2 3)       ; T
(< "abc" "abd") ; T
```

>

`(> value value...)`

T if each value is greater than the next one.

```
(> 3 2 1)   ; T
```

<=

`(<= value value...)`

T if each value is less than or equal to the next one.

```
(<= 1 1 2)   ; T
```

>=

`(>= value value...)`

T if each value is greater than or equal to the next one.

```
(>= 2 2 1)   ; T
```

eq

`(eq expression expression)`

T if both are the same object: the same symbol, the same number or the same list (not an equal copy). To compare contents, use equal.

```
(setq a '(1 2) b a)
(eq a b)          ; T
(eq a '(1 2))     ; nil
```

equal

`(equal expression expression [fuzz])`

T if both have the same contents. With a fuzz, numbers may differ by up to that amount (useful to compare points).

```
(equal '(1 2) '(1 2))                  ; T
(equal '(1.0 2.0) '(1.0 2.0001) 0.001) ; T
```

not

`(not expression)`

T if the expression is nil; nil otherwise. Same as null.

```
(not nil)   ; T
```

null

`(null expression)`

T if the expression is nil (or the empty list).

```
(null '())   ; T
```

atom

`(atom expression)`

T if the expression is not a list with elements (nil is an atom too).

```
(atom 'a)      ; T
(atom '(1))    ; nil
```

listp

`(listp expression)`

T if the expression is a list or nil.

```
(listp nil)   ; T
```

vl-consp

`(vl-consp expression)`

T if the expression is a list with elements (not nil).

```
(vl-consp nil)    ; nil
(vl-consp '(1))   ; T
```

numberp

`(numberp expression)`

T if the expression is an integer or a real.

```
(numberp 3.5)   ; T
```

zerop

```
(zerop number)
```

T if the number is 0.

```
(zerop 0.0)   ; T
```

minusp

```
(minusp number)
```

T if the number is negative.

```
(minusp -1)   ; T
```

boundp

```
(boundp symbol)
```

T if the symbol has a value other than nil.

```
(boundp 'pi)   ; T
```

type

```
(type expression)
```

Type of the expression as a symbol: INT, REAL, STR, SYM, LIST, SUBR (built-in function), USUBR (user function), ENAME, PICKSET, FILE or VL-CATCH-ALL-APPLY-ERROR. For nil, nil.

```
(type 1.5)     ; REAL
(type "a")     ; STR
```

vl-symbolp

```
(vl-symbolp expression)
```

T if the expression is a symbol (nil is not).

```
(vl-symbolp 'a)   ; T
```

car

```
(car list)
```

First element of the list. For nil, nil. In a dotted pair, the first part.

```
(car '(1 2 3))   ; 1
(car '(8 . "0")) ; 8
```

cdr

`(cdr list)`

The list without its first element. In a dotted pair, the second part.

```
(cdr '(1 2 3))    ; (2 3)
(cdr '(8 . "0"))  ; "0"
```

caar

`(caar list)`

First element of the first element: the first number of a point at the start of the list. Same as `(car (car list))`. If any step is not a list, it is an error.

```
(caar list)   ; = (car (car list))
```

caaar

`(caaar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(car (car (car list)))`. If any step is not a list, it is an error.

```
(caaar list)   ; = (car (car (car list)))
```

cdaar

`(cdaar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (car (car list)))`. If any step is not a list, it is an error.

```
(cdaar list)   ; = (cdr (car (car list)))
```

caaaar

`(caaaar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(car (car (car (car list))))`. If any step is not a list, it is an error.

```
(caaaar list)   ; = (car (car (car (car list))))
```

cadaar

`(cadaar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(car (cdr (car (car list))))`. If any step is not a list, it is an error.

```
(cadaar list)   ; = (car (cdr (car (car list))))
```

cdaaar

`(cdaaar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (car (car (car list))))`. If any step is not a list, it is an error.

```
(cdaaar list)   ; = (cdr (car (car (car list))))
```

cddaar

`(cddaar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (cdr (car (car list))))`. If any step is not a list, it is an error.

```
(cddaar list)   ; = (cdr (cdr (car (car list))))
```

cadr

`(cadr list)`

Second element of the list. Of a point, the Y coordinate. Same as `(car (cdr list))`. If any step is not a list, it is an error.

```
(cadr list)   ; = (car (cdr list))
```

caadr

`(caadr list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(car (car (cdr list)))`. If any step is not a list, it is an error.

```
(caadr list)   ; = (car (car (cdr list)))
```

cdadr

`(cdadr list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (car (cdr list)))`. If any step is not a list, it is an error.

```
(cdadr list)   ; = (cdr (car (cdr list)))
```

caaadr

`(caaadr list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(car (car (car (cdr list))))`. If any step is not a list, it is an error.

```
(caaadr list)   ; = (car (car (car (cdr list))))
```

cadadr

`(cadadr list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(car (cdr (car (cdr list))))`. If any step is not a list, it is an error.

```
(cadadr list)   ; = (car (cdr (car (cdr list))))
```

cdaadr

`(cdaadr list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (car (car (cdr list))))`. If any step is not a list, it is an error.

```
(cdaadr list)   ; = (cdr (car (car (cdr list))))
```

cddadr

`(cddadr list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (cdr (car (cdr list))))`. If any step is not a list, it is an error.

```
(cddadr list)   ; = (cdr (cdr (car (cdr list))))
```

cdar

`(cdar list)`

The first element without its first element. Same as `(cdr (car list))`. If any step is not a list, it is an error.

```
(cdar list)   ; = (cdr (car list))
```

cadar

`(cadar list)`

Second element of the first element. Same as `(car (cdr (car list)))`. If any step is not a list, it is an error.

```
(cadar list)   ; = (car (cdr (car list)))
```

cddar

`(cddar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (cdr (car list)))`. If any step is not a list, it is an error.

```
(cddar list)   ; = (cdr (cdr (car list)))
```

caadar

`(caadar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(car (car (cdr (car list))))`. If any step is not a list, it is an error.

```
(caadar list)   ; = (car (car (cdr (car list))))
```

caddar

`(caddar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(car (cdr (cdr (car list))))`. If any step is not a list, it is an error.

```
(caddar list)   ; = (car (cdr (cdr (car list))))
```

cdadar

`(cdadar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (car (cdr (car list))))`. If any step is not a list, it is an error.

```
(cdadar list)   ; = (cdr (car (cdr (car list))))
```

cdddar

`(cdddar list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (cdr (cdr (car list))))`. If any step is not a list, it is an error.

```
(cdddar list)   ; = (cdr (cdr (cdr (car list))))
```

cddr

`(cddr list)`

The list without its first two elements. Same as `(cdr (cdr list))`. If any step is not a list, it is an error.

```
(cddr list)   ; = (cdr (cdr list))
```

caddr

`(caddr list)`

Third element of the list. Of a 3D point, the Z coordinate. Same as `(car (cdr (cdr list)))`. If any step is not a list, it is an error.

```
(caddr list)   ; = (car (cdr (cdr list)))
```

cdddr

`(cdddr list)`

The list without its first three elements. Same as `(cdr (cdr (cdr list)))`. If any step is not a list, it is an error.

```
(cdddr list)   ; = (cdr (cdr (cdr list)))
```

caaddr

`(caaddr list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(car (car (cdr (cdr list))))`. If any step is not a list, it is an error.

```
(caaddr list)   ; = (car (car (cdr (cdr list))))
```

cadddr

`(cadddr list)`

Fourth element of the list. Same as `(car (cdr (cdr (cdr list))))`. If any step is not a list, it is an error.

```
(cadddr list)   ; = (car (cdr (cdr (cdr list))))
```

cdaddr

`(cdaddr list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (car (cdr (cdr list))))`. If any step is not a list, it is an error.

```
(cdaddr list)   ; = (cdr (car (cdr (cdr list))))
```

cddddr

`(cddddr list)`

A combination of car and cdr: the letters between c and r are applied from right to left (a = car, d = cdr). Same as `(cdr (cdr (cdr (cdr list))))`. If any step is not a list, it is an error.

```
(cddddr list)   ; = (cdr (cdr (cdr (cdr list))))
```

cons

`(cons element list-or-atom)`

With a list, returns it with the element in front. With an atom, creates a dotted pair, like those of entget.

```
(cons 0 '(1 2))   ; (0 1 2)
(cons 8 "Walls")  ; (8 . "Walls")
```

list

`(list [expression]...)`

List with the values of the expressions. With no arguments, nil.

```
(list 1 "a" 2.5)   ; (1 "a" 2.5)
```

append

`(append [list]...)`

Joins the lists into one.

```
(append '(1 2) '(3) '(4 5))   ; (1 2 3 4 5)
```

reverse

`(reverse list)`

The list with its elements in reverse order.

```
(reverse '(1 2 3))   ; (3 2 1)
```

length

`(length list)`

Number of elements of the list.

```
(length '(1 2 3))   ; 3
```

vl-list-length

`(vl-list-length list)`

Number of elements of the list, same as length.

```
(vl-list-length '(a b))   ; 2
```

nth

`(nth index list)`

Element at the given position, starting at 0. Outside the list, nil.

```
(nth 1 '(a b c))   ; B
```

member

`(member expression list)`

Looks for the expression in the list (with equal) and returns the rest of the list from it, or nil if it is not there.

```
(member 2 '(1 2 3))   ; (2 3)
```

assoc

`(assoc key list)`

In a list of pairs, the first one whose first part is the key (with equal), or nil. It is the way to read a code from an entget list.

```
(cdr (assoc 8 (entget (entlast))))   ; layer name
```

subst

`(subst new old list)`

Copy of the list with every element equal to `old` replaced by `new`. Used to change a pair of an entget list before entmod.

```
(subst '(8 . "New") (assoc 8 e) e)
```

Differences with AutoCAD

Only the elements of the top level of the list are replaced, not those inside sublists.

last

`(last list)`

Last element of the list.

```
(last '(1 2 3))   ; 3
```

vl-remove

`(vl-remove element list)`

The list without the elements equal to the given one.

```
(vl-remove 2 '(1 2 3 2))   ; (1 3)
```

vl-position

`(vl-position element list)`

Position (from 0) of the first occurrence of the element, or nil.

```
(vl-position "b" '("a" "b"))   ; 1
```

vl-sort

`(vl-sort list function)`

The list sorted with the comparison function, which receives two elements and returns non-nil if the first one goes first.

```
(vl-sort '(3 1 2) '<)   ; (1 2 3)
```

Differences with AutoCAD

The sort is stable and keeps repeated elements (AutoCAD may remove them).

vl-remove-if

```
(vl-remove-if function list)
```

The list without the elements for which the function returns non-nil.

```
(vl-remove-if 'minusp '(1 -2 3))   ; (1 3)
```

vl-remove-if-not

```
(vl-remove-if-not function list)
```

Only the elements for which the function returns non-nil.

```
(vl-remove-if-not 'numberp '(1 "a" 2))   ; (1 2)
```

vl-some

```
(vl-some function list [list]...)
```

Applies the function to the elements of the lists, in parallel, and returns the first result that is not nil (or nil).

```
(vl-some '> '(1 5) '(2 3))   ; T
```

vl-every

```
(vl-every function list [list]...)
```

T if the function returns non-nil for all the elements of the lists.

```
(vl-every 'numberp '(1 2.5))   ; T
```

vl-member-if

```
(vl-member-if function list)
```

The rest of the list from the first element for which the function returns non-nil, or nil.

```
(vl-member-if 'minusp '(1 -2 3))   ; (-2 3)
```

acad_strlsort

```
(acad_strlsort list-of-strings)
```

The list of strings sorted alphabetically, ignoring case.

```
(acad_strlsort '("b" "A" "c"))   ; ("A" "b" "c")
```

mapcar

```
(mapcar function list [list]...)
```

List with the result of applying the function to each element (with several lists, to the elements at the same position, up to the shortest one).

```
(mapcar '+ '(1 2) '(10 20))   ; (11 22)
```

apply

```
(apply function list)
```

Calls the function with the elements of the list as arguments.

```
(apply 'max '(3 7 2))   ; 7
```

eval

```
(eval expression)
```

Evaluates the value of the expression: used to run a list built by the routine.

```
(eval '(+ 1 2))   ; 3
```

set

```
(set symbol expression)
```

Like setq, but the symbol is evaluated too: the variable can be chosen at run time.

```
(set (read "radius") 5)   ; radius is 5
```

read

```
(read [string])
```

Reads the first expression of the string and returns it without evaluating it. With no argument, nil.

```
(read "(1 2)")   ; (1 2)
(read "abc")     ; ABC
```

vl-symbol-name

```
(vl-symbol-name symbol)
```

Name of the symbol as a string, in uppercase.

```
(vl-symbol-name 'radius)   ; "RADIUS"
```

vl-symbol-value

`(vl-symbol-value symbol)`

Value of the symbol.

```
(vl-symbol-value 'pi)   ; 3.14159
```

strcat

`(strcat [string]...)`

Joins the strings. With no arguments, "".

```
(strcat "Layer " "0")   ; "Layer 0"
```

strlen

`(strlen [string]...)`

Number of characters of the string (with several, the sum).

```
(strlen "Lince")   ; 5
```

substr

`(substr string start [length])`

Part of the string from position `start` (the first one is 1), with the given length or up to the end.

```
(substr "LinceCAD" 6)     ; "2D"
(substr "LinceCAD" 1 5)   ; "Lince"
```

strcase

`(strcase string [lowercase])`

The string in uppercase; if the second argument is not nil, in lowercase.

```
(strcase "Wall")     ; "WALL"
(strcase "Wall" T)   ; "wall"
```

itoa

`(itoa integer)`

The integer as a string.

```
(itoa 25)   ; "25"
```

atoi

`(atoi string)`

Integer written at the start of the string (0 if there is none).

```
(atoi "12abc")   ; 12
```

atof

`(atof string)`

Real written at the start of the string (0.0 if there is none). The decimal separator is the point.

```
(atof "2.5 m")   ; 2.5
```

chr

`(chr code)`

Character with that (Unicode) code, as a string.

```
(chr 65)   ; "A"
```

ascii

`(ascii string)`

Code of the first character of the string (0 if it is empty).

```
(ascii "A")   ; 65
```

wcmatch

`(wcmatch string pattern)`

T if the string matches the pattern. Wildcards: `*` (any text), `?` (one character), `#` (one digit), `@` (one letter), `.` (one non-alphanumeric character), `[abc]` and `[a-z]` (one of them), `[~abc]` (none of them), `~` at the start negates the pattern, `,` separates several patterns and the reverse quote makes the next character literal. It is case-sensitive.

```
(wcmatch "DIM-12" "DIM-##")   ; T
(wcmatch "Wall" "W*,P*")      ; T
```

vl-string-trim

`(vl-string-trim characters string)`

The string without the given characters at the start and at the end.

```
(vl-string-trim " " "  hello  ")   ; "hello"
```

vl-string-left-trim

`(vl-string-left-trim characters string)`

The string without the given characters at the start.

```
(vl-string-left-trim "0" "0042")   ; "42"
```

vl-string-right-trim

`(vl-string-right-trim characters string)`

The string without the given characters at the end.

```
(vl-string-right-trim ".0" "2.500")   ; "2.5"
```

vl-string-search

`(vl-string-search pattern string [start])`

Position (from 0) where the pattern appears in the string, searching from `start`, or nil.

```
(vl-string-search "2D" "LinceCAD")   ; 5
```

vl-string-subst

`(vl-string-subst new pattern string [start])`

The string with the first occurrence of the pattern (from `start`) replaced by `new`.

```
(vl-string-subst "B" "A" "AXA")   ; "BXA"
```

vl-string-translate

`(vl-string-translate from to string)`

Replaces each character of `from` found in the string by the one at the same position in `to`.

```
(vl-string-translate "," "." "2,5")   ; "2.5"
```

vl-string-position

`(vl-string-position code string [start [from-end]])`

Position (from 0) of the character with that code, or nil. With a non-nil fourth argument, finds the last occurrence.

```
(vl-string-position (ascii "\\") "C:\\dwg\\a.dwg" nil T)   ; 6
```

vl-string->list

`(vl-string->list string)`

List with the codes of the characters of the string.

```
(vl-string->list "AB")   ; (65 66)
```

vl-list->string

```
(vl-list->string list-of-codes)
```

String with the characters of those codes.

```
(vl-list->string '(76 50))   ; "L2"
```

vl-string-elt

```
(vl-string-elt string position)
```

Code of the character at that position (from 0).

```
(vl-string-elt "AB" 1)   ; 66
```

vl-princ-to-string

```
(vl-princ-to-string expression)
```

The expression as princ writes it, in a string.

```
(vl-princ-to-string '(1 "a"))   ; "(1 a)"
```

vl-prin1-to-string

```
(vl-prin1-to-string expression)
```

The expression as prin1 writes it (strings in quotes), in a string.

```
(vl-prin1-to-string "a")   ; "\"a\""
```

rtos

```
(rtos number [mode [precision]])
```

The number as text. Modes: 1 scientific, 2 decimal, 3 engineering, 4 architectural, 5 fractional. Without mode or precision, the drawing's LUNITS and LUPREC are used. It follows DIMZIN (extra zeros) and uses the decimal point.

```
(rtos 15.5 2 2)   ; "15.50"
(rtos 15.5 4 2)   ; "1'-3 1/2\""
```

distof

```
(distof string [mode])
```

Reads a distance written in any of the rtos formats. Returns a real, or nil if it cannot be read.

```
(distof "1'-3 1/2\"")   ; 15.5
```

angtos

`(angtos angle [mode [precision]])`

An angle in radians as text. Modes: 0 degrees, 1 degrees/minutes/seconds, 2 grads, 3 radians, 4 surveyor's units. Without mode or precision, AUNITS and AUPREC are used.

```
(angtos (/ pi 4) 0 2)   ; "45.00"
(angtos (/ pi 4) 1 4)   ; "45d0'0\""
```

angtof

`(angtof string [mode])`

Reads an angle written in any angtos format (the `d`, `g` and `r` marks and bearings are recognized on their own; without a mark, according to the mode, degrees by default). Returns radians, or nil.

```
(angtof "90")   ; 1.5708
```

distance

`(distance point point)`

Distance between two points (2D or 3D).

```
(distance '(0 0) '(3 4))   ; 5.0
```

angle

`(angle point point)`

Angle in radians of the line from the first point to the second, measured from the X axis counterclockwise (from 0 to 2?).

```
(angle '(0 0) '(0 5))   ; 1.5708
```

polar

`(polar point angle distance)`

Point at that distance from the given one, in the direction of the angle (radians).

```
(polar '(0 0) 0 10)   ; (10.0 0.0 0.0)
```

inters

`(inters p1 p2 p3 p4 [on-segments])`

Intersection point of line p1-p2 with line p3-p4, or nil if they are parallel. If the fifth argument is nil, the lines are extended; if it is missing or not nil, the point must lie on both segments.

```
(inters '(0 0) '(10 10) '(0 10) '(10 0))   ; (5.0 5.0 0.0)
```

trans

```
(trans point from to [displacement])
```

Converts a point from one coordinate system to another. Systems are given as a number: 0 the world system (WCS), 1 the current UCS and 2 the display system (in 2D, the same as the UCS). With a non-nil fourth argument, the point is treated as a displacement: it is only rotated, without taking the UCS origin into account.

```
(trans '(0 0) 1 0)      ; the UCS origin, in world coordinates
(trans '(10 0) 1 0 T)   ; 10 units along the UCS X direction, in the WCS
```

Differences with AutoCAD

An entity name (object coordinate system) or 3 (paper space) return the point unchanged.

initget

```
(initget [bits] [keywords])
```

Sets up the next `get…` function, entsel or ssget. Bits that can be added: 1 disallows Enter, 2 disallows zero, 4 disallows negatives, 128 accepts any text (returned as a string). Keywords are separated by spaces; their capitals are the abbreviation. After _ the global names can be given.

```
(initget 1 "Yes No")
(getkword "\nContinue? [Yes/No]: ")
```

Differences with AutoCAD

Bits 8, 16, 32 and 64 do nothing.

getpoint

```
(getpoint [base-point] [prompt])
```

Asks for a point: a click (with object snaps) or typed coordinates. With a base point, shows a rubber band from it. Returns the point, nil on Enter, or the initget keyword.

```
(setq p (getpoint "\nSpecify center: "))
```

getcorner

```
(getcorner base-point [prompt])
```

Asks for the opposite corner of a rectangle starting at the base point.

```
(getcorner p "\nOpposite corner: ")
```

getdist

```
(getdist [base-point] [prompt])
```

Asks for a distance: typed (in any length format) or with two points. Returns a real.

```
(setq d (getdist "\nDistance: "))
```

getangle

`(getangle [base-point] [prompt])`

Asks for an angle: typed (in any angle format) or with two points. Returns radians.

```
(setq a (getangle "\nAngle: "))
```

Differences with AutoCAD

It does not take ANGBASE or ANGDIR into account (with the initial values the result is the same).

getorient

`(getorient [base-point] [prompt])`

Same as getangle.

Differences with AutoCAD

In AutoCAD it always measures from east, even when ANGBASE is not 0; in LinceCAD getangle does not use ANGBASE either, so both give the same result.

getreal

`(getreal [prompt])`

Asks for a real number (with a decimal point; the comma separates coordinates).

```
(setq k (getreal "\nFactor: "))
```

getint

`(getint [prompt])`

Asks for an integer between -32768 and 32767.

```
(initget 7)
(setq n (getint "\nNumber of copies: "))
```

getstring

`(getstring [spaces] [prompt])`

Asks for a text. If the first argument is not nil, Space is part of the text and only Enter ends it. Returns "" on Enter.

```
(setq name (getstring T "\nLayer name: "))
```

getkword

`(getkword [prompt])`

Asks for one of the initget keywords. Returns the whole keyword (or its global name) and nil on Enter.

```
(initget "Line Arc")
```

```
(getkword "\n[Line/Arc]: ")
```

entsel

`(entsel [prompt])`

Asks to select one object with a click. Returns a list with the entity name and the picked point, or nil.

```
(setq s (entsel "\nSelect a line: "))
(entget (car s))
```

princ

`(princ [expression [file]])`

Writes the expression on the command line (or to the file), strings without quotes. Returns the expression. `(princ)` with no arguments writes nothing: at the end of a routine it hides the last value.

```
(princ "\nDone.")
(princ)
```

prin1

`(prin1 [expression [file]])`

Like princ, but strings are written in quotes and with their escape characters.

```
(prin1 "a")   ; writes "a"
```

print

`(print [expression [file]])`

Like prin1, with a line break before and a space after.

```
(print 42)
```

terpri

`(terpri)`

Writes a line break. Returns nil.

prompt

`(prompt string)`

Writes the string on the command line. Returns nil.

```
(prompt "\nSelect the walls: ")
```

alert

`(alert string)`

Shows the string in a message box with an OK button (and writes it on the command line). In scripts, it only writes it.

```
(alert "There is no WALLS layer.")
```

textscr

`(textscr)`

In AutoCAD, switches to the text window. In LinceCAD it does nothing. Returns nil.

graphscr

`(graphscr)`

In AutoCAD, goes back to the graphics window. In LinceCAD it does nothing. Returns nil.

command

`(command [argument]...)`

Types the arguments on the command line, like a script: strings as they are (`""` is Enter), numbers and points as text, entity names and selection sets as a selection, and pause for the user to give that input. nil is skipped. `(command)` with no arguments cancels the active command. Returns nil.

```
(command "_line" '(0 0) '(100 0) "")
(command "_circle" pause 25)
```

Notes

Use global names (`_line`) so that the routine works with the interface in any language. With CMDECHO set to 0 the command line does not repeat the prompts or what `command` types (results are still shown).

command-s

`(command-s [argument]...)`

Same as command.

Differences with AutoCAD

In AutoCAD, command-s does not accept `pause` and can be used inside `*error*`; in LinceCAD it behaves like command.

vl-cmdf

`(vl-cmdf [argument]...)`

Like command, but returns T.

Differences with AutoCAD

AutoCAD checks the arguments before typing them; LinceCAD types them directly.

entlast

`(entlast)`

Name of the last main entity created in the drawing, or nil if it is empty.

```
(command "_circle" '(0 0) 5)
(setq c (entlast))
```

entnext

`(entnext [entity])`

With no argument, the first entity of the drawing; with an entity, the next one (or nil at the end). It goes through model space and then the layouts; after an INSERT with attributes come its ATTRIBs and the SEQEND, as in AutoCAD.

```
(setq e (entnext))
(while e (setq e (entnext e)))
```

entget

`(entget entity [applications])`

List of DXF pairs of the entity: `(-1 . name)`, `(0 . "LINE")`, `(8 . "layer")`, `(10 x y z)`... Angles are in radians. Returns nil if the entity does not exist.

```
(cdr (assoc 10 (entget (car (entsel)))))   ; start point or center
```

Differences with AutoCAD

Applications (extended data) are accepted but not returned. For types other than LINE, CIRCLE, ARC, LWPOLYLINE, TEXT, MTEXT, POINT, INSERT, ATTRIB and DIMENSION only the common codes are returned.

entmod

`(entmod list)`

Changes the entity of code -1 with the values of the list (the others stay the same). If the layer does not exist, it is created. Returns the list, or nil if it cannot.

```
(setq e (entget (entlast)))
(entmod (subst '(62 . 1) (assoc 62 e) e))
```

Differences with AutoCAD

Only for LINE, CIRCLE, ARC, LWPOLYLINE, TEXT, MTEXT, POINT, INSERT and ATTRIB.

entmake

`(entmake list)`

Creates an entity from the list of DXF pairs: LINE, CIRCLE, ARC, POINT, LWPOLYLINE, TEXT, MTEXT or INSERT (of an existing block), or a layer with `(0 . "LAYER")`. Without code 8 it goes on the current layer; texts without a height take TEXTSIZE. Returns the list, or nil if it cannot.

```
(entmake '((0 . "CIRCLE") (10 0.0 0.0 0.0) (40 . 5.0)))
```

Differences with AutoCAD

INSERTs are created without attributes (there are no ATTRIB/SEQEND sequences) and blocks are not created.

entdel

`(entdel entity)`

Erases the entity. If it had already been erased with entdel in this session, it is restored. Returns the name, or nil.

```
(entdel (entlast))
```

entupd

`(entupd entity)`

In AutoCAD, redraws the entity after changing it. In LinceCAD the drawing updates on its own: it returns the name and does nothing.

handent

`(handent handle)`

Name of the entity with that handle (code 5, in hexadecimal), or nil if it does not exist. The handle is kept when the drawing is saved.

```
(handent "2F")
```

ssget

`(ssget [mode] [point [point]] [filter])`

Selection set, or nil if it is empty. Without a mode, the user selects (with `":S"`, only once). Modes: `"X"` or `"A"` (the whole drawing), `"I"` (what was selected before the routine), `"P"` (the previous one of ssget), `"L"` (the last entity), `"W"` and `"C"` with two points (window and crossing) or a single point. The filter is a list of DXF pairs, with wildcards in strings and the `-4` operators (`"<AND"`, `"<OR"`, `"<XOR"`, `"<NOT"`, `"<"`, `">="`, `"*"`…).

```
(ssget "X" '((0 . "CIRCLE") (8 . "AXES")))
(ssget '((0 . "LINE,LWPOLYLINE")))
```

ssname

`(ssname set index)`

Name of the entity at that position (from 0) of the set, or nil.

```
(ssname ss 0)
```

sslength

`(sslength set)`

Number of entities in the set.

```
(sslength (ssget "X"))
```

ssadd

`(ssadd [entity [set]])`

With no arguments, an empty set; with an entity, a set holding it; with both, adds the entity to the set (if it was not there) and returns the set.

```
(setq ss (ssadd))
(ssadd (entlast) ss)
```

ssdel

`(ssdel entity set)`

Removes the entity from the set. Returns the set, or nil if the entity was not in it.

ssmemb

`(ssmemb entity set)`

Returns the entity if it is in the set, or nil.

tblsearch

`(tblsearch table name [set-tblnext])`

DXF list of the `name` record of the table, or nil. Tables: `"LAYER"`, `"LTYPE"`, `"STYLE"`, `"BLOCK"` and `"DIMSTYLE"`. With a non-nil third argument, the next tblnext continues after this record.

```
(if (not (tblsearch "LAYER" "WALLS")) (princ "\nThe WALLS layer is missing."))
```

Differences with AutoCAD

The VIEW, UCS, VPORT and APPID tables are not available.

tblnext

`(tblnext table [rewind])`

Next record of the table, or nil at the end. With a non-nil second argument, goes back to the first one.

```
(setq r (tblnext "LAYER" T))
(while r (princ (strcat "\n" (cdr (assoc 2 r)))) (setq r (tblnext "LAYER")))
```

getvar

`(getvar name)`

Value of a system variable, or nil if it is unknown. It reads those of the drawing (CLAYER, LUNITS, TEXTSIZE, DIMZIN…; ANGBASE in radians) and those of the program: OSMODE, ORTHOMODE, SNAPMODE, CMDACTIVE, CMDECHO, DWGNAME, DWGPREFIX, SECURELOAD, ACADVER, PRODUCT and PLATFORM.

```
(getvar "CLAYER")   ; "0"
(getvar "OSMODE")
```

Notes

Variables such as PICKBOX, FILEDIA, CMDDIA, ATTREQ or EXPERT exist with their usual value so that routines do not fail, but they change nothing. See System variables.

setvar

`(setvar name value)`

Changes a system variable and returns the value. Those of the drawing can be undone; OSMODE, ORTHOMODE and SNAPMODE change the program's object snaps, ortho and snap. An invalid value gives the error `AutoCAD variable setting rejected`.

```
(setq old (getvar "OSMODE"))
(setvar "OSMODE" 0)
(setvar "OSMODE" old)
```

redraw

`(redraw [entity [mode]])`

In AutoCAD, redraws the screen or an entity. In LinceCAD it is not needed: it returns nil and does nothing.

regen

`(regen)`

Regenerates the whole drawing, like REGEN. Returns nil.

open

`(open file mode)`

Opens a text file: `"r"` to read, `"w"` to write (emptying it) or `"a"` to append. Returns the file, or nil if it cannot be opened.

```
(setq f (open "C:/data/points.txt" "r"))
```

close

`(close file)`

Closes the file (and saves what was written). Returns nil.

```
(close f)
```

read-line

`(read-line file)`

Next line of the file, without the line break, or nil at the end.

```
(while (setq l (read-line f)) (princ l))
```

Differences with AutoCAD

Without a file it does not read from the keyboard: use getstring.

write-line

`(write-line string [file])`

Writes the string and a line break to the file (or on the command line). Returns the string.

```
(write-line "X;Y" f)
```

findfile

`(findfile name)`

Path of the file if it exists as given, in the drawing folder or in the APPLOAD trusted folders; otherwise nil.

```
(findfile "blocks.lsp")
```

getenv

`(getenv name)`

Value of an environment variable of the operating system, or nil.

```
(getenv "TEMP")
```

Differences with AutoCAD

It does not read the AutoCAD variables stored in the Windows registry.

load

`(load file [on-failure])`

Loads a .lsp file (the extension can be omitted), looking for it as given, in the drawing folder and in the trusted folders, and evaluates its expressions. Returns the value of the last one. If it is not found, returns `on-failure` or gives the error `LOAD failed`. Outside the trusted folders it may ask first, depending on the APPLOAD security setting.

```
(load "tools" nil)
```

vl-filename-base

`(vl-filename-base path)`

File name without folder or extension.

```
(vl-filename-base "C:/job/plan.dwg")   ; "plan"
```

vl-filename-directory

`(vl-filename-directory path)`

Folder of the path, without the file name.

```
(vl-filename-directory "C:/job/plan.dwg")   ; "C:/job"
```

vl-filename-extension

`(vl-filename-extension path)`

Extension of the file, with the dot, or nil if it has none.

```
(vl-filename-extension "plan.dwg")   ; ".dwg"
```

exit

`(exit)`

Ends the routine with the error `quit / exit abort` (which reaches error).

quit

`(quit)`

Same as exit.

vl-catch-all-apply

`(vl-catch-all-apply function list)`

Like apply, but an error neither stops the routine nor calls `*error*`: it returns an error object, which is checked with vl-catch-all-error-p.

```
(setq r (vl-catch-all-apply '/ '(1 0)))
(if (vl-catch-all-error-p r) (princ (vl-catch-all-error-message r)))
```

vl-catch-all-error-p

`(vl-catch-all-error-p value)`

T if the value is an error object of vl-catch-all-apply.

vl-catch-all-error-message

`(vl-catch-all-error-message error)`

Message of the error object.

```
(vl-catch-all-error-message (vl-catch-all-apply '/ '(1 0)))   ; "divide by zero"
```

vl-load-com

`(vl-load-com)`

In AutoCAD, loads the ActiveX functions. LinceCAD does not have them: it returns nil so that routines that call it at the start do not fail.

ver

`(ver)`

Version of the interpreter, such as `"LinceCAD LISP 0.x.y"`.

gc

`(gc)`

In AutoCAD, frees memory. In LinceCAD it is not needed: it returns nil.