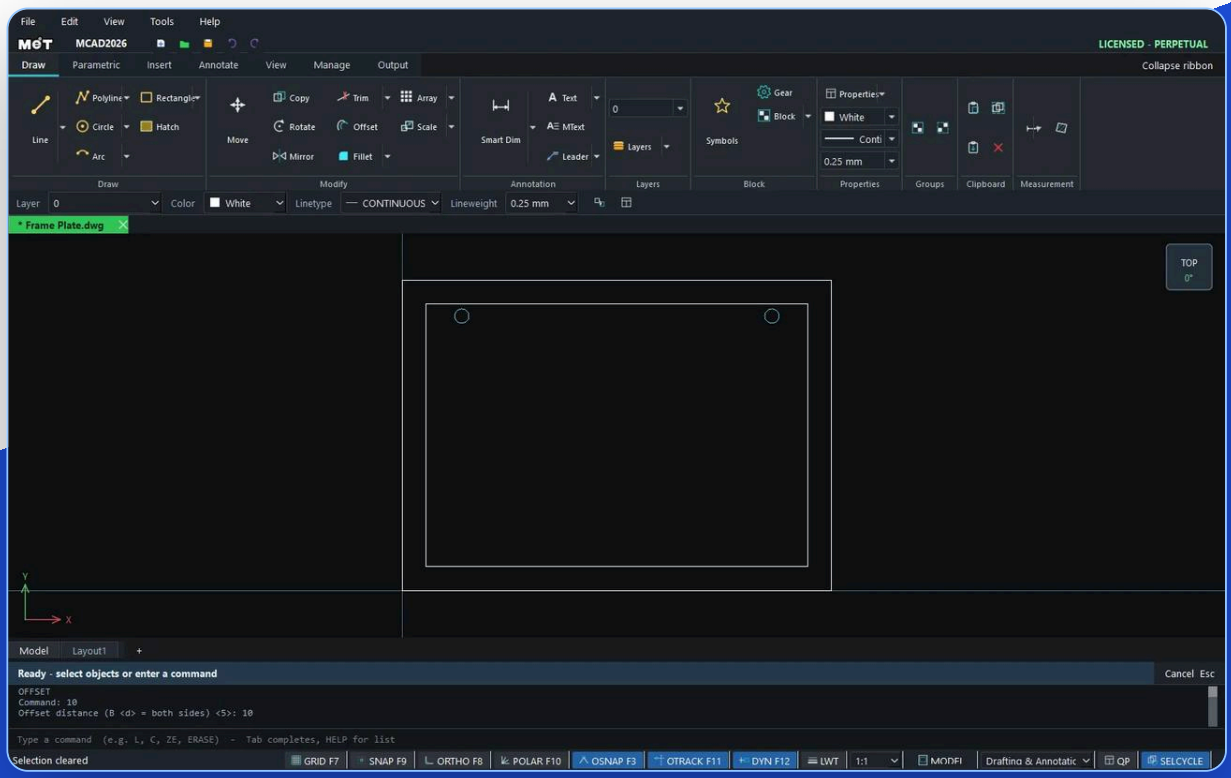
MECHATRONICS DESIGNED CO., LTD.

USER MANUAL

MCAD2026

2D drafting with native DWG/DXF



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MCAD2026 — User Manual
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26/230 Moo 18, Soi Mitsubishi, Paholyothin Rd., Klong Nueng, Klong Luang, Pathumthani 12120, Thailand
sales@mechdesigned.com · +66 2 520 4575 · LINE @mechdesigned

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Text and screenshots come from the real application; menus and defaults may change in later versions. The latest online manual is at www.mechdesigned.com/en/help/mcad

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How to use this manual

The manual has 11 chapters in the order you will use the program. Each topic tells you **what it is for**, **where to find it**, the **steps** and a table of options. Every chapter ends with a 6-question **quiz**; the answers are in Appendix A.

- **Bold** = a button, menu or command exactly as it appears; › separates menu levels.
- The Standard Expert Premium badge shows the lowest edition that includes the command.
- **Try it** boxes are short exercises with the finished result, ready to follow in the program.
- Blue links in the PDF jump to the referenced topic.

CHAPTER 1

Getting started

1. Install MCAD 2026
   Install MCAD 2026 with the MET installer or from the Microsoft Store, and check the version.
2. Activate Your Licence
   Activate your MCAD 2026 licence with a License key, move it to another computer, and import an offline licence.
3. Open DWG/DXF Files and the Start Center
   Open DWG/DXF files with the built-in reader, and use the Start Center and multiple tabs.
4. Draw your first part (Bracket Plate)
   Practise by drawing a 240 x 140 mm plate with holes and dimensions, then save it, from start to finish.
5. Command Index
   A list of all MCAD 2026 commands with aliases, descriptions and links to the help pages.

Install MCAD 2026

Install MCAD 2026 with the MET installer or from the Microsoft Store, and check the version.

What it does

MCAD 2026 is a 2D CAD program for Windows that reads and saves DWG/DXF files directly. This page explains both ways to install it, how to update it, and how to check the version you are running.

System requirements

Item	Minimum
Operating system	Windows 10 / Windows 11, 64-bit
Memory	8 GB (16 GB recommended for large DWG files)
Disk space	About 1 GB
Internet	Needed to activate the licence and to verify entitlement periodically

Option 1: Installer from MET (recommended)

1. Download the installer from license.mechdesigned.com/download/MCAD. The file is named `MCAD2026-Setup-<version>.exe` and is digitally signed by MET Mechatronics Designed.
2. Double-click the file and follow the installer steps (the default location is `C:\Program Files\MCAD2026`).
3. The installer includes the DWG reader and MET's symbol library (547 symbols in the electrical, pneumatic, mechanical and architecture groups). You do not need to install a separate converter.
4. Open MCAD 2026 from the Desktop icon or the Start menu.
5. The first time you open it, a window asks you to activate your licence. See Activate your licence.

Updating the program (MET installer)

- The program checks for a new version by itself shortly after you start it.
- Or run it yourself with **Help › Check for Updates...**.

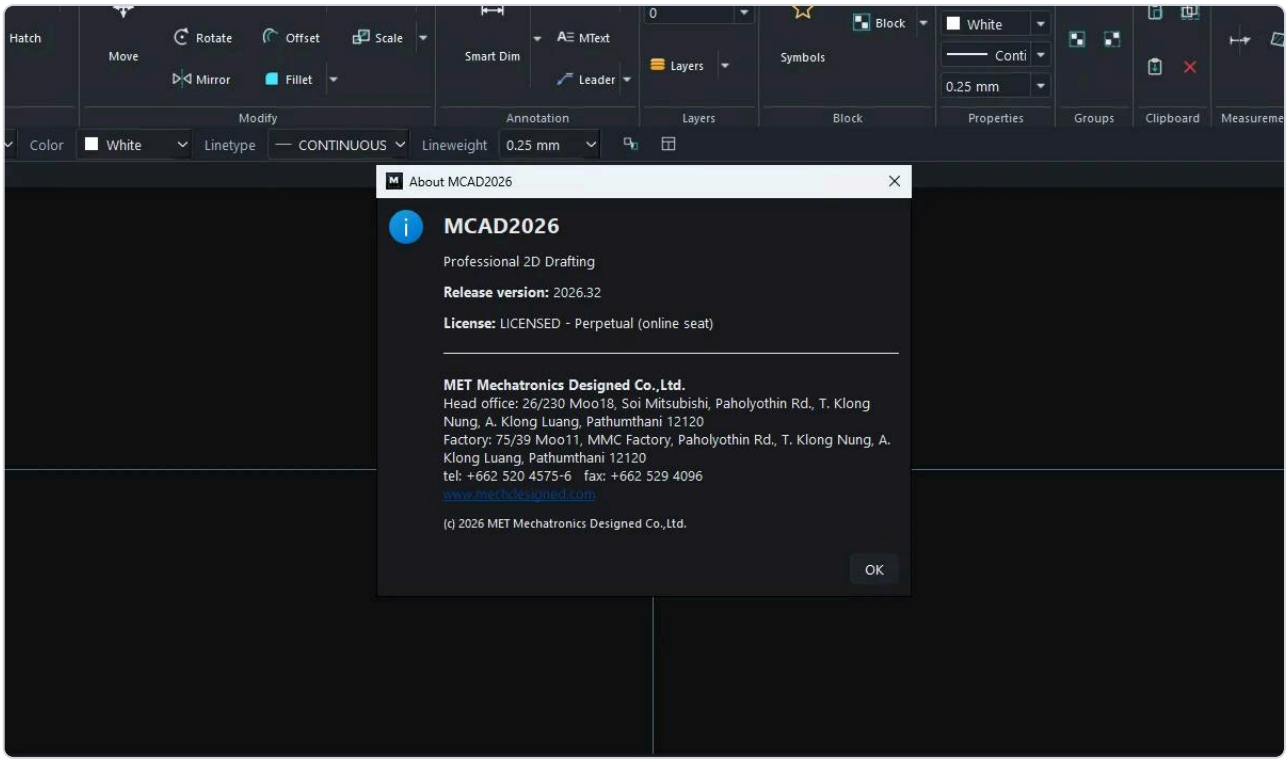
Option 2: Microsoft Store

You can also install MCAD 2026 from the Microsoft Store. It is the same program with the same licence; the Store copy updates through the Store and may trail the MET installer slightly.

> **Note:** If you already installed MCAD from the Store, it keeps working. The Store version updates only through the Microsoft Store. **Help › Check for Updates...** does not download an installer for it.

Check the version

Choose **Help › About MCAD2026**. It shows the version number (Release version) and the licence status.



About MCAD2026 window showing the version number and licence status

> **Note:** When you report a problem to support, always include the Release version from this window.

Try it: Set up MCAD on an interior designer's new laptop

Scenario: an interior designer has a new laptop and must open a client's condo DWG plan this afternoon.

1. Open a browser at license.mechdesigned.com/download/MCAD and download the installer.
2. Double-click `MCAD2026-Setup-<version>.exe` and click Next until it finishes (it takes a few minutes).
3. Start MCAD 2026 from the Desktop and fill in the Activate License window as described in Activate your licence.
4. Choose **Help › About MCAD2026** and note the Release version.
5. Press `Ctrl+O` and open the condo DWG straight away. No extra file converter is needed.

See also

- Activate your licence
- Open DWG/DXF files and the Start Center
- Common problems and fixes

Activate Your Licence

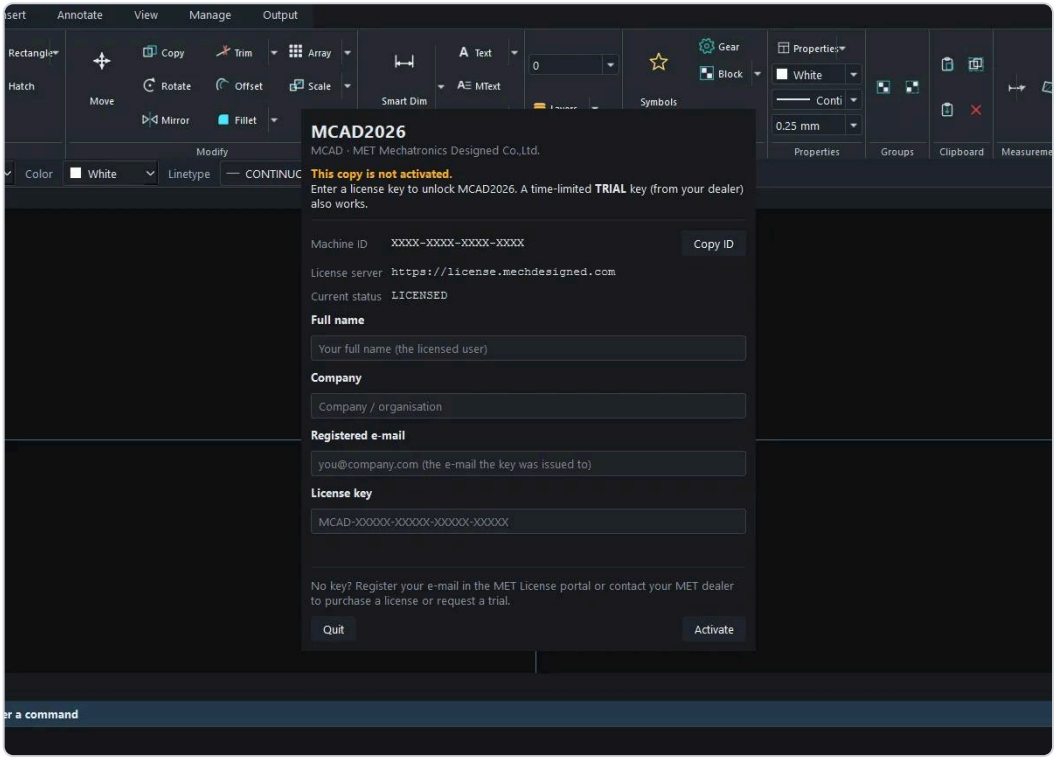
Activate your MCAD 2026 licence with a License key, move it to another computer, and import an offline licence.

What it does

MCAD 2026 must be activated with a License key before you can use it. A TRIAL key (a trial with an expiry date) from a reseller also works.

Where to find it

- The first time you start the program (with no licence yet), the **Activate License** window opens by itself.
- Menu **Help › Activate License (online)...**
- The licence status is shown at the top right of the Ribbon (for example `LICENSED - PERPETUAL`).



Activate License window

Activation steps

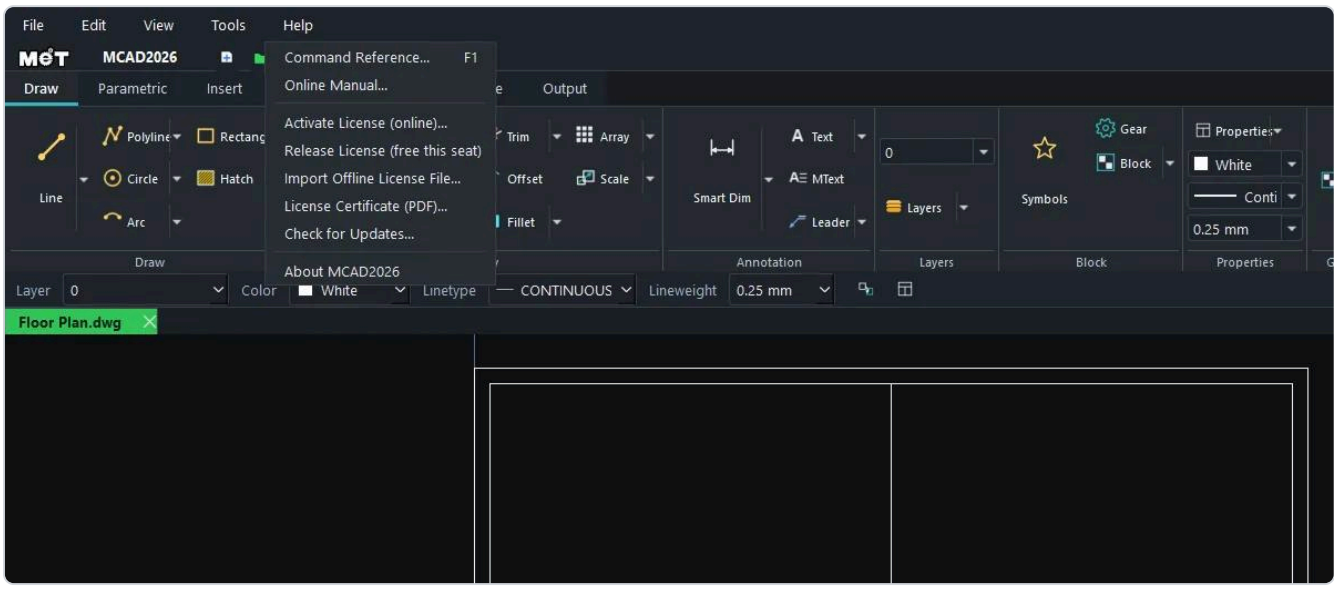
1. Connect to the internet.
2. Enter your **Full name** (the name of the authorized user) and **Company**.
3. Enter the **Registered e-mail**. It must match the e-mail you used to request the key.
4. Enter the **License key** in the format `MCAD-XXXXX-XXXXX-XXXXX-XXXXX`.
5. Click **Activate**. The program contacts the License server and shows the message License activated, with the Edition and the expiry date (if any).

Options in the window

Option	Meaning
Machine ID	The ID of this computer. Give it to support when needed. Click **Copy ID** to copy it
License server	The MET licence server (`https://license.mechdesigned.com`)
Current status	The current licence status of this computer
Full name / Company	The licence holder's name, printed on the licence certificate
Registered e-mail	The e-mail registered with the key
License key	The key you received from MET or a reseller
Quit / Activate	Exit the program / activate

Other licence commands in the Help menu

Menu	Meaning
Release License (free this seat)	Returns this computer's seat to the server. Use it before you move to a new computer or reinstall Windows
Import Offline License File...	Imports a `.json` licence file signed by MET, for computers without internet access
License Certificate (PDF)...	Prints the licence-holder certificate as a PDF



Help menu

> **Note:** The program checks your licence with the server periodically (about every 25 minutes while online). If you are briefly offline, you can keep working.

Try it: Move your licence from the office PC to a laptop before a site visit

Scenario: tomorrow you are measuring a shop fit-out on site and want MCAD on your laptop instead of the office PC. Your licence has 1 seat.

1. On the office PC: save all your work, then choose **Help › Release License (free this seat)**. The seat goes back to the server.
2. On the laptop: if the program is not installed yet, download the installer from license.mechdesigned.com/download/MCAD or get it from the Microsoft Store.
3. Start the program, enter the same Full name, Company, Registered e-mail and License key, then click **Activate**.
4. Check that the top right of the Ribbon shows `LICENSED`.
5. If you are told all seats are in use because you forgot to release the old PC, click **Copy ID** and send the Machine ID to MET so they can free the seat.

See also

- Install MCAD 2026
- Common Problems and Solutions

Open DWG/DXF Files and the Start Center

Open DWG/DXF files with the built-in reader, and use the Start Center and multiple tabs.

What it does

MCAD 2026 opens AutoCAD **DWG** and **DXF** files directly with the DWG reader built into the program. You do not need to install an extra file converter. It also opens `.mprj` project files (several sheets at once).

Where to find it

- Ribbon: **Output › File › Open**
- Quick Access Toolbar (the folder icon at the top)
- **File › Open...** menu
- Command line: `OPEN`
- Shortcut: `Ctrl+O`
- Start Center: **File › Start Center...**, then click **Open Drawing...**

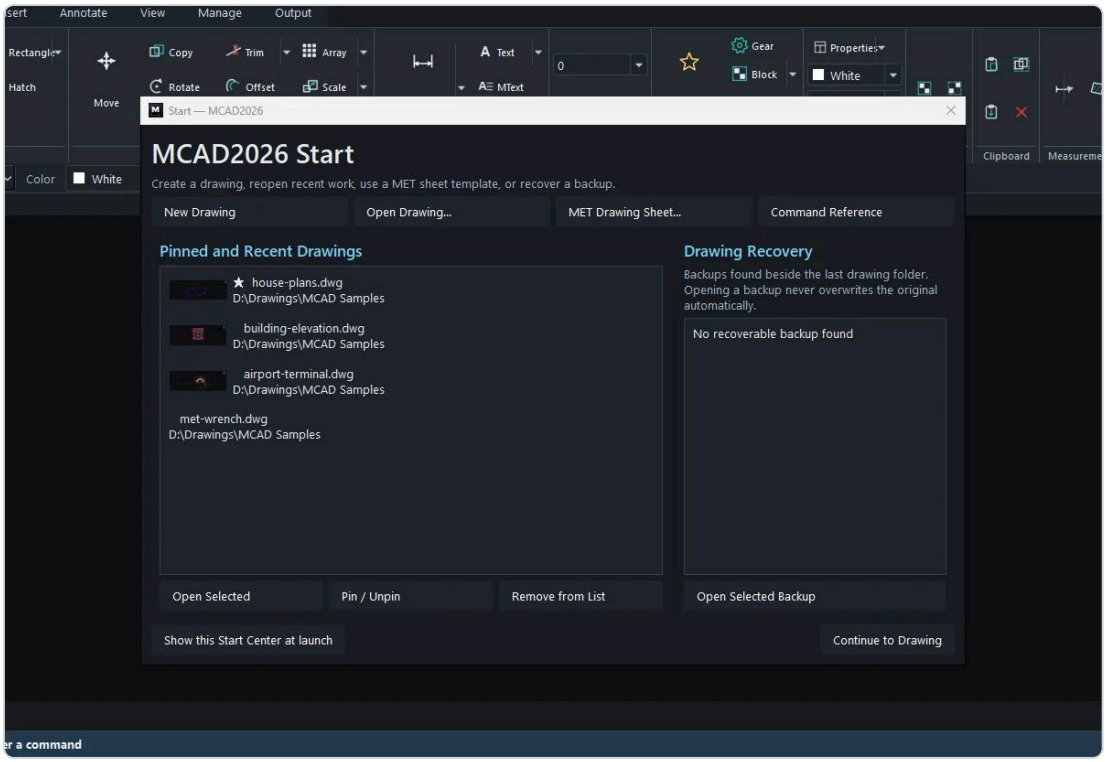
Steps

1. Press `Ctrl+O`, or type `OPEN` and press Enter.
2. Choose the file type `2D CAD (*.dwg *.dxf *.mprj)` and select a file.
3. Click **Open**. The file opens in a new tab (the existing tabs stay open). Large files load in the background, and you can keep working in other tabs meanwhile.
4. Switch tabs with `Ctrl+Tab` / `Ctrl+Shift+Tab`.

> **Note:** You can also double-click a `.dwg` or `.dxf` file in Windows Explorer to open it in MCAD (if MCAD is set as the default program). Explorer then shows thumbnails of the drawings.

Start Center

The Start Center is the start-up window. It shows recent files, pinned files, and backup files you can recover.



Start Center showing recent files and the start buttons

Option	Meaning
New Drawing	Create a new drawing (same as `NEW`)
Open Drawing...	Open a file
MET Drawing Sheet...	Create a new drawing from a standard MET A4–A0 sheet with a title block. See TITLEBLOCK and MET Drawing Sheet
Command Reference	The table of all commands
Pinned and Recent Drawings	Recent files (★ = pinned). Double-click to open
Open Selected / Pin / Unpin / Remove from List	Open, pin, or remove from the list
Drawing Recovery / Open Selected Backup	Opens backup files found in recent folders, without overwriting the original
Show this Start Center at launch	Turn the Start Center on or off at start-up
Continue to Drawing	Close the Start Center and go to the drawing area

Damaged files

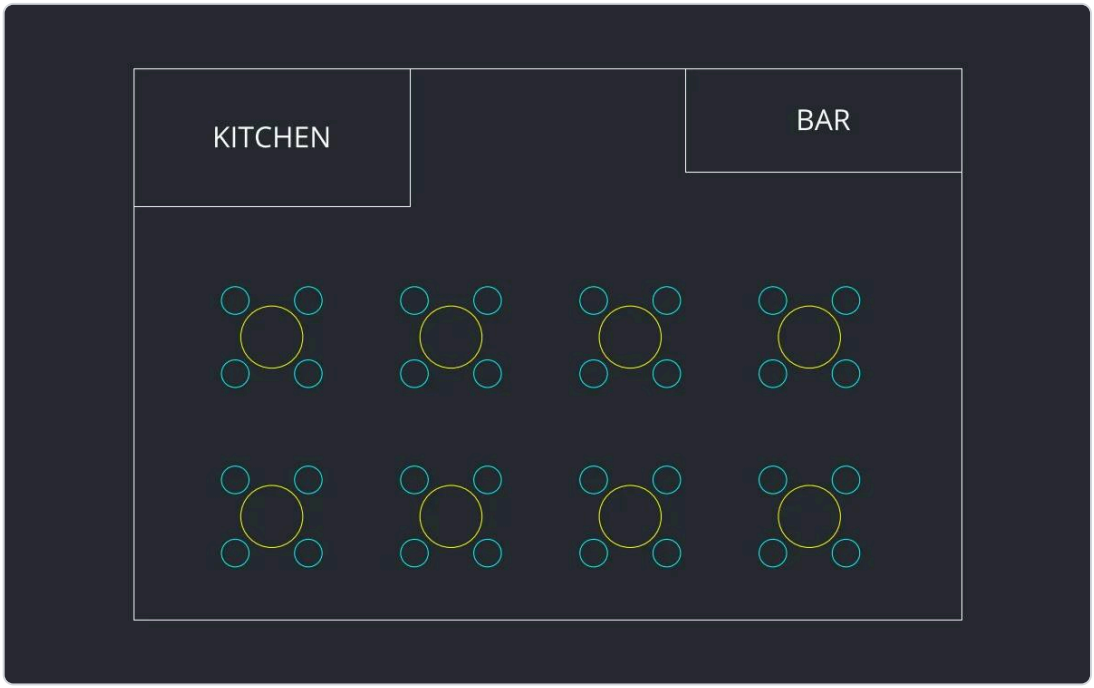
If a DXF file will not open, try `RECOVER`. See AUDIT, RECOVER and PURGE.

Try it: Open an architect's restaurant plan next to your own drawing

Scenario: an architect sends you `restaurant-plan.dwg` so you can place the signs and furniture, while your sign drawing is already open in another tab.

1. Press `Ctrl+O`, choose `2D CAD (*.dwg *.dxf *.mprj)` and open `restaurant-plan.dwg`. It opens in a new tab and your existing tab stays open.
2. If the file is large, you can keep working in the sign tab while it loads.

3. Press `Ctrl+Tab` to switch between the plan and the sign drawing. Use `DIST` on the plan to measure the shopfront wall.
4. Open **File › Start Center...**, click `restaurant-plan.dwg` in the recent list and click **Pin**, so you find it straight away next time.



Restaurant plan opened from a DWG: kitchen, bar and 8 table sets

Challenge: Select one table set in the plan, press `Ctrl+C`, then `Ctrl+V` in the other tab to reuse the furniture in your drawing. See Clipboard.

See also

- NEW, SAVE and SAVEAS
- SAVEPROJECT - Multiple Tabs as a Project
- Draw Your First Part

Draw your first part (Bracket Plate)

Practise by drawing a 240 x 140 mm plate with holes and dimensions, then save it, from start to finish.

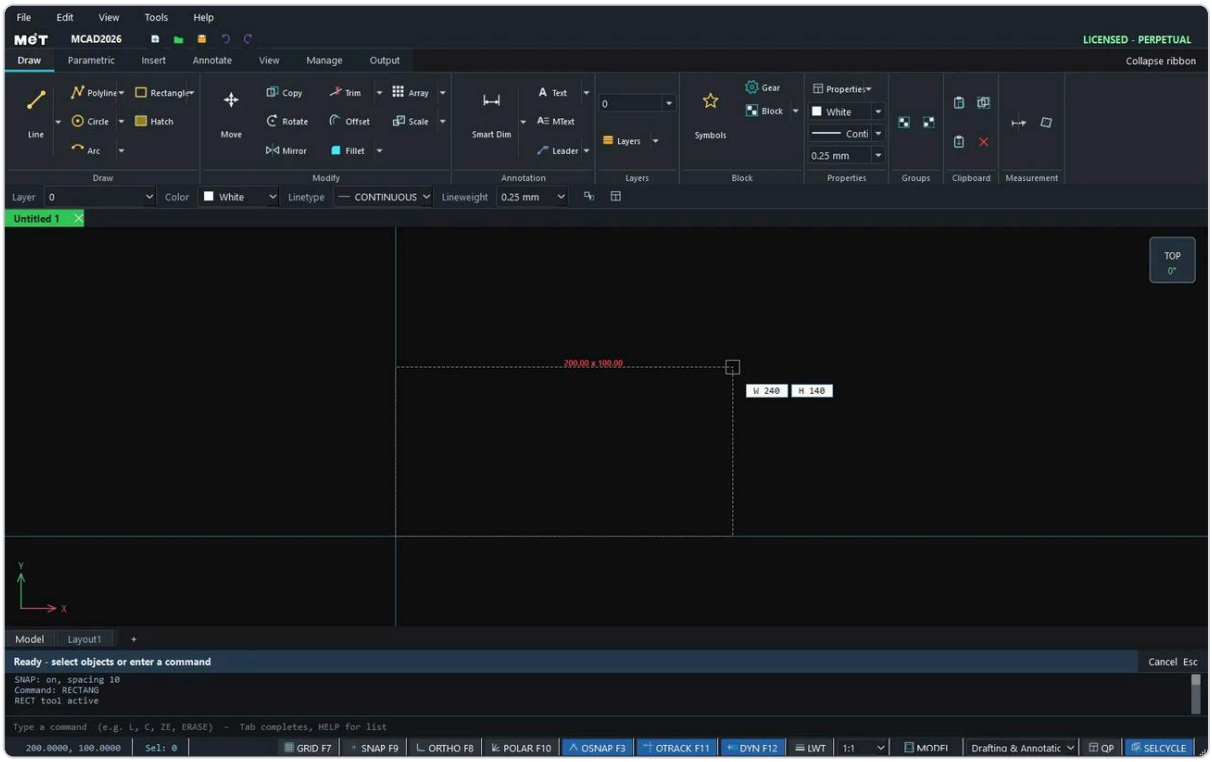
What it does

This tutorial walks you through drawing a bracket plate that is 240 x 140 mm, with a Ø50 centre hole and four Ø12 mounting holes in the corners. You add dimensions and save the file. It takes about 5 minutes.

Step 1: Turn on grid snap and draw the rectangle

1. Type `SNAP 10` and press Enter. This turns on grid snap at 10 mm, so your clicks land exactly on grid points.
2. Type `REC` and press Enter (or use the Ribbon **Draw › Draw › Rectangle**).
3. Click the first corner at point 0,0.
4. Move the mouse to the upper right. The Smart Input box next to the cursor shows W and H.

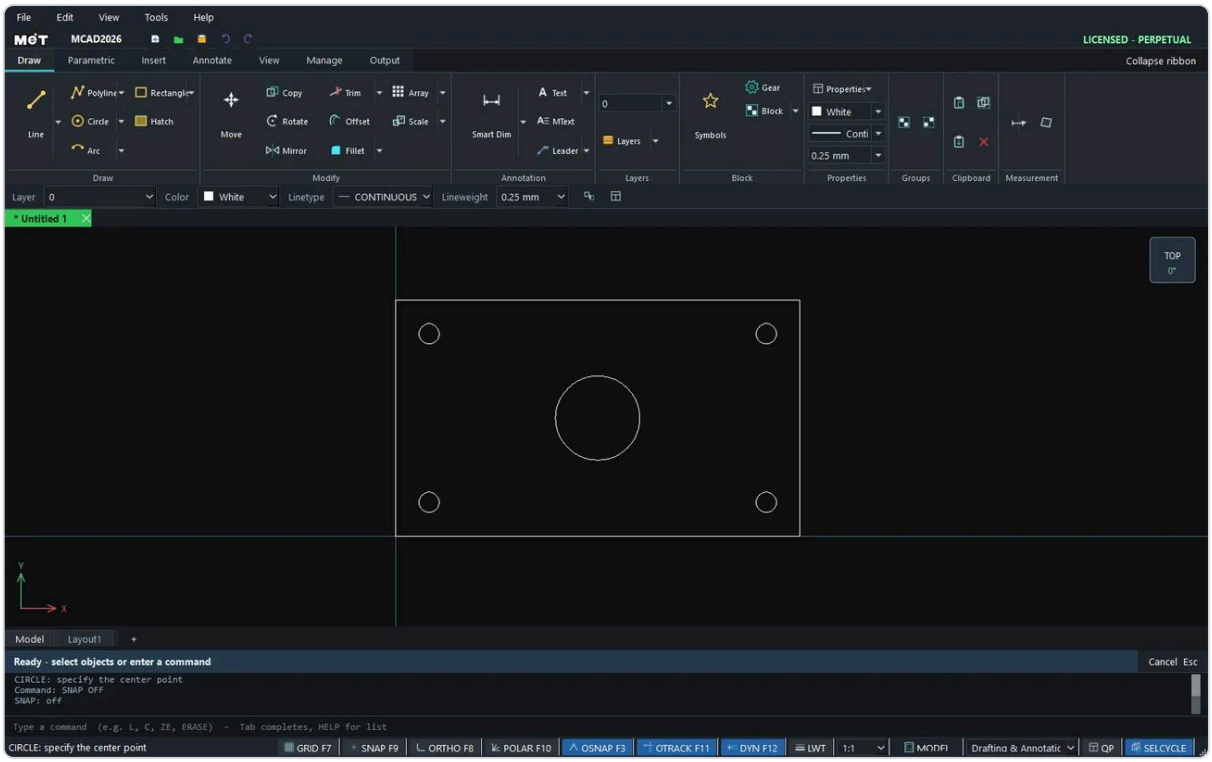
5. Type `240`, press `Tab`, type `140`, then press Enter.



Drawing the rectangle by typing W 240 and H 140 in the Smart Input box

Step 2: Draw the holes

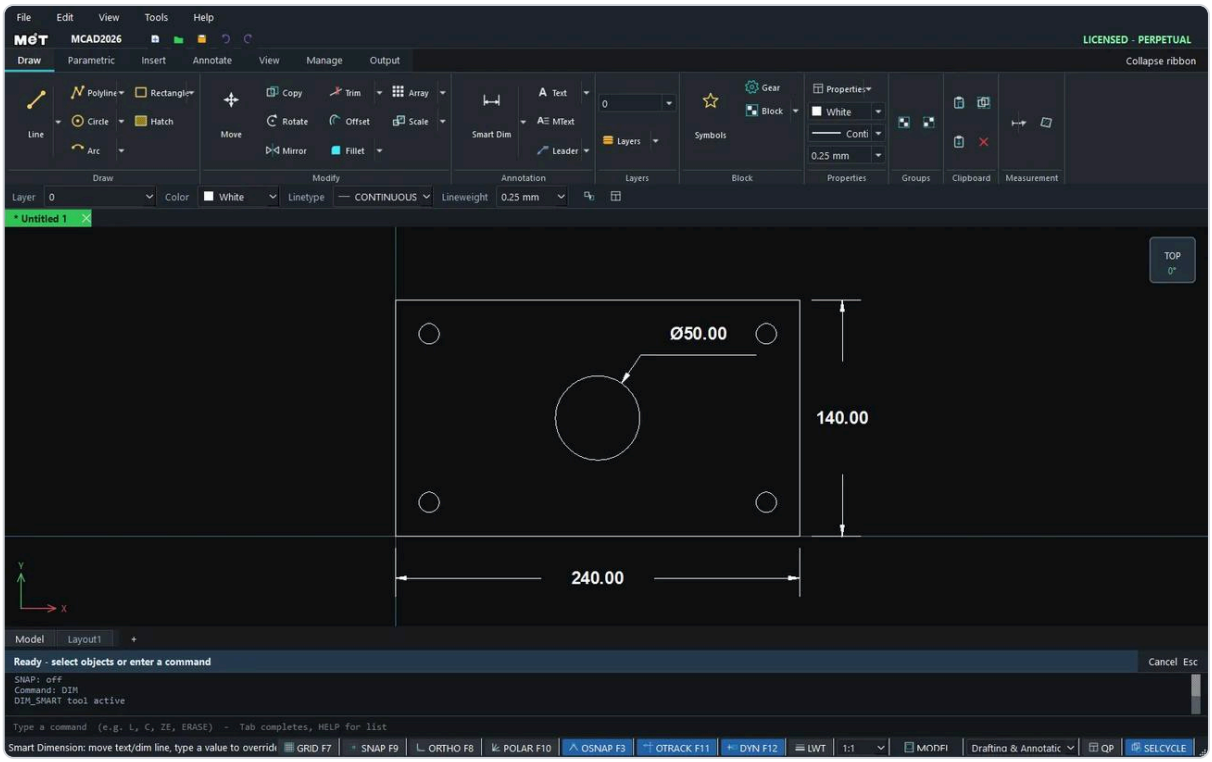
1. Type `C` and press Enter. Click the centre point at 120,70.
2. Type the radius `25` and press Enter.
3. Repeat for the 4 mounting holes at 20,20 / 220,20 / 20,120 / 220,120 with radius `6`.
4. Type `SNAP OFF` to turn off grid snap. Object Snap keeps working as usual.



The plate with the centre hole and 4 mounting holes

Step 3: Add dimensions with Smart Dimension

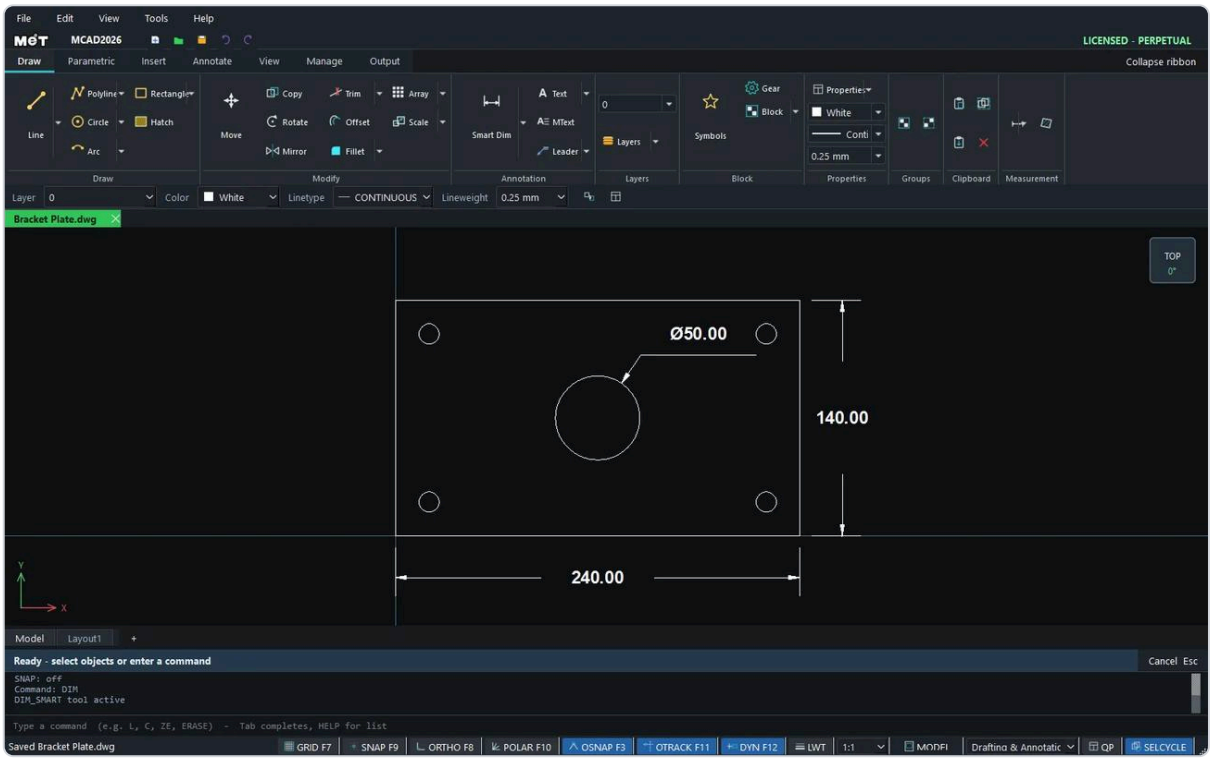
1. Type `DIM` and press Enter (or use the Ribbon **Draw › Annotation › Smart Dim**).
2. Click the bottom edge, then click an empty area below the part. A 240 dimension is attached to the mouse. Click again to place it.
3. Click the right edge, click an empty area, then click to place it. This gives 140.
4. Click the circumference of the centre hole. This gives Ø50. Move it into position and click.
5. Press `Esc` to end the command.



The plate with the 240, 140 and Ø50 dimensions added

Step 4: Save

1. Press `Ctrl+S`. The new drawing has no name yet, so the Save As dialog opens.
2. Choose the type `AutoCAD Drawing (*.dwg)` or `DXF (*.dxf)`, enter a name and click **Save**.
3. The tab at the top changes to the file name, and the Status bar shows `Saved ...`.



Saved, with the tab showing the file name

> **Note:** `FILLET` works on LINE objects (and a LINE with an ARC or CIRCLE) only, not on an outline drawn with `REC` or `PLINE`. For R12 rounded corners, draw the rectangle as four `LINE`s, then `FILLET`. See FILLET and CHAMFER.

Try it: Your second drawing, a house-number plaque

You have drawn the plate. Now make something for your front gate: a 300 x 150 mm house-number plaque with screw holes, ready to laser-cut from a DXF file.

> Don't have the program yet? Download the installer from license.mechdesigned.com/download/MCAD or get it from the Microsoft Store.

1. `REC` `0,0`, then `300` `Tab` `150` Enter.
2. Screw holes: `C` `20,75` radius `4` and `C` `280,75` radius `4`.
3. Type `TEXT`, set the height to `90`, click in the middle of the plaque and type your number, for example `27`.
4. Press `Ctrl+S`, choose `DXF (*.dxf)` and name it `house-number`. It is ready for the laser shop.



300 x 150 mm house-number plaque showing 27

Challenge: Redraw the plaque outline as four `LINE`s (FILLET does not work on a REC outline), then round all four corners with `FILLET` radius `15`. See FILLET and CHAMFER.

See also

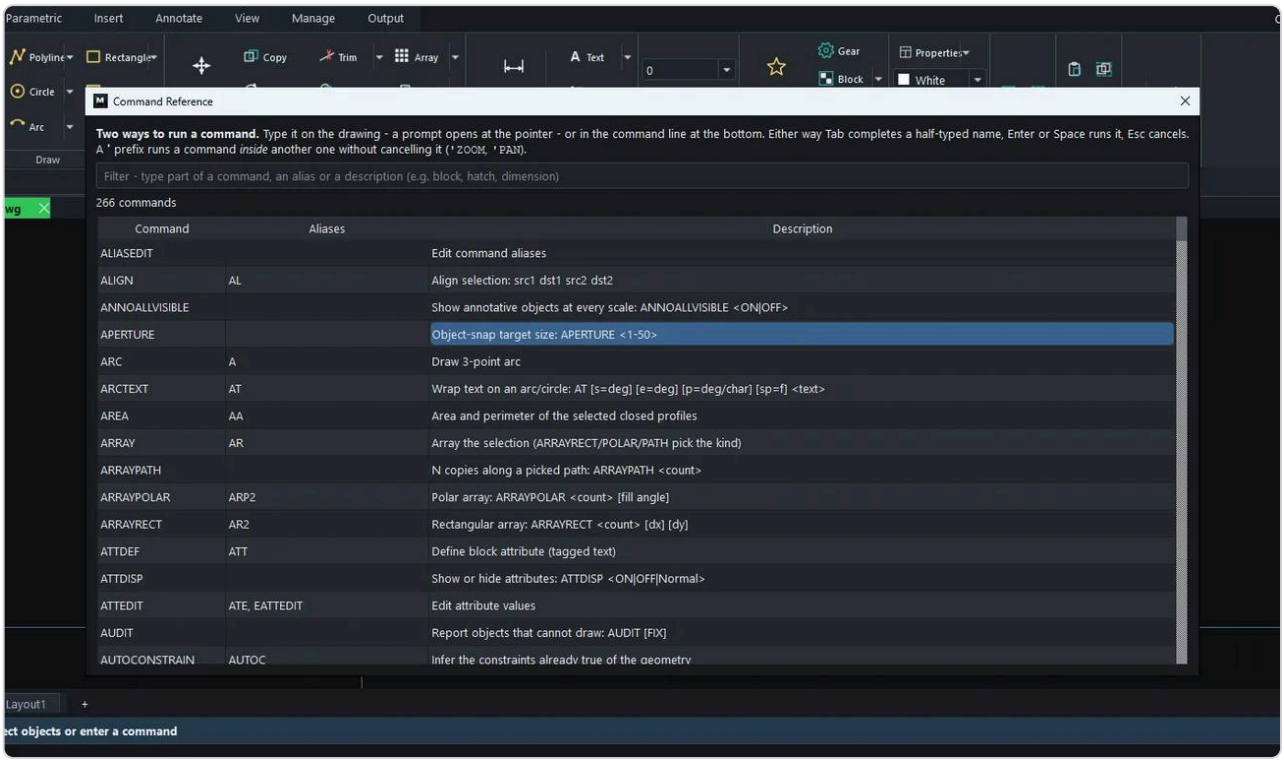
- Command Line, Aliases and Smart Input
- Smart Dimension (DIM)
- NEW, SAVE and SAVEAS

Command Index

A list of all MCAD 2026 commands with aliases, descriptions and links to the help pages.

What it does

This table lists all 266 typed commands in MCAD 2026, with their aliases and links to the help pages. In the program, you can open the same table from **Help › Command Reference...** (`F1`), or type `HELP` (`?`) to list the commands on the Command line.



Command Reference window in the program

> **Note:** The descriptions in the table are the same text that the program shows in its command suggestions (in English). In the syntax, `<...>` = a required value and `[...]` = an optional value.

Command list

Command	Alias	Description (from the program)	Help page
`ALIASEDIT`	–	Edit command aliases	Options, Shortcuts and Aliases (OPTIONS / CUI)
`ALIGN`	`AL`	Align selection: src1 dst1 src2 dst2	ALIGN
`ANNOALLVISIBLE`	–	Show annotative objects at every scale: ANNOALLVISIBLE <ON|OFF>	Annotation Scale (OBJECTSCALE, ANNOALLVISIBLE)
`APERTURE`	–	Object-snap target size: APERTURE <1-50>	Status Bar: Snap, Ortho, Polar, OSnap
`ARC`	`A`	Draw 3-point arc	ARC (3-point arc)

Command	Alias	Description (from the program)	Help page
ARCTEXT	AT	Wrap text on an arc/circle: AT [s=deg] [e=deg] [p=deg/char] [sp=f] <text>	Text Tools (ARCTEXT, TALIGN, TCOUNT, FIND ...)
AREA	AA	Area and perimeter of the selected closed profiles	Measuring and inquiry (DIST, AREA, LIST)
ARRAY	AR	Array the selection (ARRAYRECT/POLAR/PATH pick the kind)	ARRAY - Rectangular / Polar / Path
ARRAYPATH	–	N copies along a picked path: ARRAYPATH <count>	ARRAY - Rectangular / Polar / Path
ARRAYPOLAR	ARP2	Polar array: ARRAYPOLAR <count> [fill angle]	ARRAY - Rectangular / Polar / Path
ARRAYRECT	AR2	Rectangular array: ARRAYRECT <count> [dx] [dy]	ARRAY - Rectangular / Polar / Path
ATTDEF	ATT	Define block attribute (tagged text)	Attributes (ATTDEF, ATTEDIT, ATTDISP)
ATTDISP	–	Show or hide attributes: ATTDISP <ON|OFF|Normal>	Attributes (ATTDEF, ATTEDIT, ATTDISP)
ATTEDIT	ATE , EATTEDIT	Edit attribute values	Attributes (ATTDEF, ATTEDIT, ATTDISP)
AUDIT	–	Report objects that cannot draw: AUDIT [FIX]	AUDIT, RECOVER and PURGE
AUTOCONSTRAIN	AUTOC	Infer the constraints already true of the geometry	Geometric Constraints (Parametric)
BASE	–	Insertion base point for this drawing: BASE <x> <y>	Units (UNITS) and System Variables
BCLOSE	–	Close Block Editor: BCLOSE SAVE or BCLOSE DISCARD	Block Editor and Dynamic Blocks (BEDIT, BPARAM)
BEDIT	BE	Open a selected/named definition in the isolated Block Editor	Block Editor and Dynamic Blocks (BEDIT, BPARAM)
BLOCK	B , BMAKE	Define a block from the selection: BLOCK <name> [baseX baseY]	BLOCK, INSERT and WBLOCK
BLOCKLIST	BL	List the drawing's blocks and their dynamic parameters	BLOCK, INSERT and WBLOCK
BOUNDARY	BO	Trace a closed polyline around the selected profiles	BOUNDARY, REGION and Booleans (UNION/SUBTRACT/INTERSECT)
BPARAM	BPAR	Add a dynamic parameter to the selected block ('BPARAM' shows the forms)	Block Editor and Dynamic Blocks (BEDIT, BPARAM)
BREAK	BR	Break: 2 points on an entity remove the middle	BREAK, JOIN and PEDIT
BSAVE	–	Save the open block definition and update every reference	Block Editor and Dynamic Blocks (BEDIT, BPARAM)

Command	Alias	Description (from the program)	Help page
BSET	BPROP	Set a dynamic parameter on the selected block: BSET <param> <value>	Block Editor and Dynamic Blocks (BEDIT, BPARAM)
CAL	–	Calculator: CAL 25.4*3, CAL sqrt(2), CAL sin(30)	Command Line, Aliases and Smart Input
CENTERLINE	CL	Centreline between two selected lines	CENTERMARK and CENTERLINE
CENTERMARK	DIMCENTER, DCE	Centre cross on the selected circles/arcs: CENTERMARK [size]	CENTERMARK and CENTERLINE
CHAMFER	CHA	Chamfer corner (CHAMFER [D] <distance>)	FILLET and CHAMFER - Corner rounding and bevelling
CHPROP	CHANGE	Retag the selection: CHPROP layer=NAME color=#RRGGBB	PROPERTIES, MATCHPROP and CHPROP
CIRCLE	C	Draw circle	CIRCLE and DONUT
COLOR	COL, DDCOLOR	Colour for new objects	COLOR, LINETYPE and LWEIGHT
CONSTRAINTBAR	CBAR	List the drawing's constraints	Geometric Constraints (Parametric)
CONSTRAINTSETTINGS	CSETTINGS	Constraint options: CONSTRAINTSETTINGS [INFER ON|OFF]	Geometric Constraints (Parametric)
COPY	CO, CP	Copy selection	MOVE and COPY
COPYBASE	–	Copy with base point (paste in any window)	Clipboard: COPYCLIP, CUTCLIP, PASTECLIP, COPYBASE
COPYCLIP	CC	Copy the selection to the clipboard	Clipboard: COPYCLIP, CUTCLIP, PASTECLIP, COPYBASE
CUI	–	Shortcut and UI customization	Options, Shortcuts and Aliases (OPTIONS / CUI)
CUTCLIP	CX	Cut the selection to the clipboard	Clipboard: COPYCLIP, CUTCLIP, PASTECLIP, COPYBASE
DATALINK	DL	Table from a spreadsheet: DATALINK <file> [sheet] | UPDATE | DETACH	TABLE and DATALINK - Tables
DBLIST	–	Tally every object in the drawing	Measuring and inquiry (DIST, AREA, LIST)
DCANGULAR	DCA	Dimensional constraint: angle between two lines	Dimensional Constraints (DCLINEAR, DCRADIUS ...)
DCCONVERT	–	Turn dimensions into dimensional constraints	Dimensional Constraints (DCLINEAR, DCRADIUS ...)
DCDIAMETER	DCD	Dimensional constraint: circle diameter	Dimensional Constraints (DCLINEAR, DCRADIUS ...)
DCLINEAR	DCL	Dimensional constraint: distance between two lines' start points	Dimensional Constraints (DCLINEAR, DCRADIUS ...)
DCRADIUS	DCR	Dimensional constraint: circle radius	Dimensional Constraints (DCLINEAR, DCRADIUS ...)

Command	Alias	Description (from the program)	Help page
DDPTYPE	PTYPE	Point display size: DDPTYPE <size>	Units (UNITS) and System Variables
DELCONSTRAINT	DELC	Remove a constraint: DELCONSTRAINT [n|all]	Geometric Constraints (Parametric)
DIM	DIMENSION	Smart dimension	Smart Dimension (DIM)
DIMALIGNED	DAL	Aligned dimension	DIMLINEAR, DIMALIGNED and DIMVERTICAL
DIMANGULAR	DAN	Angular dimension	DIMANGULAR, DIMRADIUS, DIMDIAMETER, DIMARC, DIMJOGGED
DIMARC	DAR	Arc length dimension	DIMANGULAR, DIMRADIUS, DIMDIAMETER, DIMARC, DIMJOGGED
DIMARROW	ARCHTICK	Arrowheads: DIMARROW [arrow|tick] (tick = construction slash)	Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)
DIMBASELINE	DBA	Baseline dimension chain (QDIM B)	QDIM, DIMCONTINUE and DIMBASELINE
DIMBREAK	–	Gap dimension lines where geometry crosses: DIMBREAK [size|REMOVE]	Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)
DIMCONSTRAINT	DIMCON	Apply a dimensional constraint: DIMCONSTRAINT <type> [value]	Dimensional Constraints (DCLINEAR, DCRADIUS ...)
DIMCONTINUE	DCO	Continuous dimension chain (QDIM C)	QDIM, DIMCONTINUE and DIMBASELINE
DIMDIAMETER	DDI	Diameter dimension	DIMANGULAR, DIMRADIUS, DIMDIAMETER, DIMARC, DIMJOGGED
DIMEDIT	DED	Edit dimension text: DIMEDIT <Home|New <text>|Rotate <deg>>	Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)
DIMJOGGED	DJO , JOG	Jogged radius dimension: DIMJOGGED [Remove]	DIMANGULAR, DIMRADIUS, DIMDIAMETER, DIMARC, DIMJOGGED
DIMJOGLINE	DJL	Not-to-scale jog on a dimension line: DIMJOGLINE [0-1|REMOVE]	Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)
DIMLINEAR	DLI	Linear dimension	DIMLINEAR, DIMALIGNED and DIMVERTICAL
DIMORDINATE	DOR	Ordinate dimension	DIMORDINATE - Ordinate Dimension
DIMOVERRIDE	DOV	Override one dimension setting: DIMOVERRIDE <precision|text> <value>	Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)
DIMPOST	DP	Dimension prefix/suffix: DIMPOST %%c<> or DIMPOST <> mm	Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)

Command	Alias	Description (from the program)	Help page
DIMRADIUS	DRA	Radius dimension	DIMANGULAR, DIMRADIUS, DIMDIAMETER, DIMARC, DIMJOGGED
DIMREGEN	–	Re-measure every dimension	Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)
DIMSCALE	–	Overall dimension feature scale: DIMSCALE <value>	Units (UNITS) and System Variables
DIMSNAP	–	Smart Dimension object snap (End/Mid/Center): DIMSNAP [ON|OFF]	Status Bar: Snap, Ortho, Polar, OSnap
DIMSPACE	–	Even out parallel dimension offsets: DIMSPACE [spacing] (0 = align)	Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)
DIMSTYLE	D, DST, DDIM	Dimension styles	DIMSTYLE and STYLE - Style Manager
DIMTEDIT	DIMTED	Place dimension text: DIMTEDIT <Left|Right|Center|Home|Angle deg>	Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)
DIMVERTICAL	DVE	Vertical dimension	DIMLINEAR, DIMALIGNED and DIMVERTICAL
DIST	DI	Distance and angle between two points	Measuring and inquiry (DIST, AREA, LIST)
DIVIDE	DIV	Copy the selection at N equal divisions of a path: DIV <count>	ARRAY - Rectangular / Polar / Path
DLINE	DLN	Dual line: two parallels along one picked path (DLINE [width] [C|E])	DLINE and MLINE - Parallel lines
DONUT	DO	Two concentric circles: DONUT [inner] [outer]	CIRCLE and DONUT
DRAWORDER	DR	Move the selection in the draw order: DRAWORDER <Front|Back>	DRAWORDER, OVERKILL, REVERSE and FLATTEN
DSETTINGS	SE2, DDRMODES	Drafting settings: snap, grid and object snap	Status Bar: Snap, Ortho, Polar, OSnap
ELLIPSE	EL	Draw ellipse	ELLIPSE
ERASE	E, DELETE, DEL	Erase selection	ERASE, OOPS, UNDO and REDO
ETRANSMIT	ZIP	Package the drawing and its references: ETRANSMIT [file.zip]	ETRANSMIT - Package a drawing for sending
EXPLODE	X	Break blocks, tables, multileaders and splines into primitives	EXPLODE and XPLODE
EXPORT	EXP, DXFOUT	Export the drawing to another format	EXPORT - Exporting files
EXTEND	EX	Extend	TRIM, EXTEND and EXTRIM
EXTRIM	–	Trim everything at a selected boundary: EXTRIM [Outside]	TRIM, EXTEND and EXTRIM

Command	Alias	Description (from the program)	Help page
FIELD	–	Self-updating text: FIELD <expression> ('FIELD ?' lists them)	FIELD and UPDATEFIELD
FILLET	F	Fillet corner (FILLET [R] <radius>)	FILLET and CHAMFER - Corner rounding and bevelling
FILTER	FI	Select by property: FILTER <prop>= <value> ... ('FILTER ?' lists them)	Selecting Objects (SELECT, QSELECT, FILTER)
FIND	–	Search drawing text: FIND <text> [replacement]	Text Tools (ARCTEXT, TALIGN, TCOUNT, FIND ...)
FLATTEN	–	Drop every Z to zero	DRAWORDER, OVERKILL, REVERSE and FLATTEN
GCCOINCIDENT	GCC	Make two objects' start points coincide	Geometric Constraints (Parametric)
GCCOLLINEAR	GCCOL	Constrain two lines onto the same infinite line	Geometric Constraints (Parametric)
GCCONCENTRIC	GCCO	Constrain two circles concentric	Geometric Constraints (Parametric)
GCEQUAL	GCE	Equal length (lines) or equal radius (circles)	Geometric Constraints (Parametric)
GCFIX	GCF, FIXCONSTRAINT	Fix the selected line or circle in place	Geometric Constraints (Parametric)
GCHORIZONTAL	GCH	Constrain the selected line horizontal	Geometric Constraints (Parametric)
GCODE	CAM, NCOUT	Generate CNC, laser or plotter G-code from 2D drawing geometry	GCODE - Generate G-code for CNC and laser
GCPARALLEL	GCPA	Constrain two lines parallel	Geometric Constraints (Parametric)
GCPERPENDICULAR	GCPE	Constrain two lines perpendicular	Geometric Constraints (Parametric)
GCSMOOTH	GCSM	Smooth continuity for a line-circle or circle-circle pair	Geometric Constraints (Parametric)
GCSYMMETRIC	GCSYM	Constrain two matching objects about a selected line axis	Geometric Constraints (Parametric)
GCTANGENT	GCT	Constrain two objects tangent	Geometric Constraints (Parametric)
GCVERTICAL	GCV	Constrain the selected line vertical	Geometric Constraints (Parametric)
GEOMCONSTRAINT	GEOMCON	Apply a geometric constraint: GEOMCONSTRAINT <type>	Geometric Constraints (Parametric)
GETVAR	GET	Read a system variable: GETVAR <name>	Units (UNITS) and System Variables

Command	Alias	Description (from the program)	Help page
GRADIENT	GD	Gradient fill: GRADIENT [#RRGGBB] [#RRGGBB] [angle]	HATCH and GRADIENT
GROUP	G	Bind the selection into a group	GROUP, UNGROUP and GROUPEDIT
GROUPEDIT	GROUPED	Change a group's members: GROUPEDIT <Add|Remove>	GROUP, UNGROUP and GROUPEDIT
HATCH	H, BH, H2, BHATCH	Fill closed profiles: HATCH [pattern] [scale] [angle] ('HATCH ?' lists)	HATCH and GRADIENT
HATCHEDIT	HE	Restyle selected hatches: HATCHEDIT <pattern> [scale] [angle]	HATCH and GRADIENT
HATCHTOBACK	HTB	Send every hatch behind the linework	HATCH and GRADIENT
HELP	?	List commands	Command Index
HIDEOBJECTS	HIDEOBJ	Hide the selection (UNISOLATEOBJECTS restores it)	Selecting Objects (SELECT, QSELECT, FILTER)
ID	–	Report the coordinates of a point	Measuring and inquiry (DIST, AREA, LIST)
IMAGE	IM, IMAGEATTACH	Raster underlay: IMAGE <Attach file|List|Detach>	Images and PDF Underlays (IMAGE, PDFATTACH)
INSERT	I	Place a block: INSERT <name> [x y] [scale] [rot] (no name: Symbols panel)	BLOCK, INSERT and WBLOCK
INSERTOBJ	IMAGEATT	Insert a picture from a file	Images and PDF Underlays (IMAGE, PDFATTACH)
INTERSECT	IN	Keep only the overlapping area	BOUNDARY, REGION and Booleans (UNION/SUBTRACT/INTERSECT)
ISOLATEOBJECTS	ISOLATE	Hide everything except the selection	Selecting Objects (SELECT, QSELECT, FILTER)
ISOPLANE	ISO	Isometric drafting: ISOPLANE <Left|Top|Right|OFF>	Status Bar: Snap, Ortho, Polar, OSnap
JOIN	J	Join collinear/touching lines & polylines	BREAK, JOIN and PEDIT
JUSTIFYTEXT	JT	Re-justify selected text in place: JUSTIFYTEXT <Left|Center|Right>	Text Tools (ARCTEXT, TALIGN, TCOUNT, FIND ...)
LANDXMLIMPORT	LANDXML	Import LandXML points, surface faces and alignments	Import DXF, PDF, Point Cloud and LandXML
LAYER	LA	Layer manager	LAYER - Layer Manager
LAYERSTATE	LAS, LMAN	Named layer setups: LAYERSTATE <Save|Restore|List|Delete> [name]	LAYER - Layer Manager
LAYFRZ	–	Freeze a layer (LAYON will not bring it back)	Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)

Command	Alias	Description (from the program)	Help page
LAYISO	LI2	Isolate the selected object's layer	Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)
LAYLCK	–	Lock a layer: visible but not selectable	Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)
LAYMCUR	LAYCUR	Make the selected object's layer current	Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)
LAYOFF	–	Hide the selected object's layer	Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)
LAYON	–	Show every layer	Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)
LAYOUT	LO	Sheets: LAYOUT <New|Rename|Delete|List> [name]	Layouts and viewports (LAYOUT, MVIEW, PAGESETUP)
LAYTHW	–	Thaw a frozen layer, or all of them	Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)
LAYULK	–	Unlock a locked layer, or all of them	Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)
LAYUNISO	–	Undo LAYISO	Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)
LEADER	LEAD	Arrow with a note	QLEADER, LEADER, MLEADER and Callout
LENGTHEN	LEN	Extend a line end by delta: LEN [delta]	STRETCH and LENGTHEN
LIMITS	–	Nominal sheet area: LIMITS <x1> <y1> <x2> <y2> | OFF	Units (UNITS) and System Variables
LINE	L	Draw line	LINE
LINETYPE	LT, DDLTYPE	Linetype for new objects	COLOR, LINETYPE and LWEIGHT
LIST	LI	List selection info	Measuring and inquiry (DIST, AREA, LIST)
LTSCALE	LTS	Global linetype scale: LTSCALE <value>	Units (UNITS) and System Variables
LWEIGHT	LINEWEIGHT	Lineweight for new objects	COLOR, LINETYPE and LWEIGHT
M2P	MTP	Snap to the midpoint between two points (can be used transparently)	Command Line, Aliases and Smart Input
MATCHPROP	MA	Match properties (source, then targets)	PROPERTIES, MATCHPROP and CHPROP
MEASURE	ME	Copy selection along a path: ME <spacing>	ARRAY - Rectangular / Polar / Path
MEASUREGEOM	MEA	Measure: MEASUREGEOM <Distance|Angle|Area|Radius>	Measuring and inquiry (DIST, AREA, LIST)
MIRROR	MI	Mirror	MIRROR

Command	Alias	Description (from the program)	Help page
MLEADER	MLD	Multileader: arrow, elbow and note in three picks	QLEADER, LEADER, MLEADER and Callout
MLINE	ML	Parallel runs in one pass: MLINE <lines> <offset> [Zero|Top|Bottom]	DLINE and MLINE - Parallel lines
MOVE	M	Move (snap)	MOVE and COPY
MSPACE	MS	Switch to model space	Layouts and viewports (LAYOUT, MVIEW, PAGESETUP)
MTEXT	T, MT	Multi-line text	TEXT, MTEXT and TEXTEDIT
MVIEW	MV	Add a viewport: MVIEW [width] [height]	Layouts and viewports (LAYOUT, MVIEW, PAGESETUP)
NEST	NESTING, SHEETNEST	Nest closed parts on sheets for laser / plasma / wire-cut output	NEST - Arrange Parts on a Sheet
NEW	QNEW	New drawing	NEW, SAVE and SAVEAS - Save DWG/DXF
OBJECTSCALE	ANNOOBJECTSCALE	Annotative scales: OBJECTSCALE <Add|Delete|List> [scale]	Annotation Scale (OBJECTSCALE, ANNOALLVISIBLE)
OFFSET	O	Offset selection: O [distance]	OFFSET - Parallel Copies
OOPS	–	Bring back the last erased objects (not UNDO)	ERASE, OOPS, UNDO and REDO
OPEN	–	Open...	Open DWG/DXF Files and the Start Center
OPTIONS	OP	Application options and customization	Options, Shortcuts and Aliases (OPTIONS / CUI)
ORTHO	–	Right-angle drawing: ORTHO <ON|OFF> (F8)	Status Bar: Snap, Ortho, Polar, OSnap
OSMODE	–	Running object-snap bitmask: OSMODE <value>	Status Bar: Snap, Ortho, Polar, OSnap
OSNAP	OS, DDOSNAP	Running object snap: OSNAP <ON|OFF>	Status Bar: Snap, Ortho, Polar, OSnap
OVERKILL	OV	Delete duplicate/overlapping geometry	DRAWORDER, OVERKILL, REVERSE and FLATTEN
PAGESETUP	PAGE	Sheet size for a layout: PAGESETUP <A3|420x297> [Portrait|Landscape] [margin]	Layouts and viewports (LAYOUT, MVIEW, PAGESETUP)
PAN	P	Pan view (can be used transparently)	Zoom, Pan and View Controls
PASTECLIP	PASTE, PA	Paste from clipboard	Clipboard: COPYCLIP, CUTCLIP, PASTECLIP, COPYBASE
PDFADJUST	–	Fade a selected PDF underlay: PDFADJUST <0..100>	Images and PDF Underlays (IMAGE, PDFATTACH)

Command	Alias	Description (from the program)	Help page
PDFATTACH	PDFUNDERLAY	Attach a PDF underlay: PDFATTACH <file> [page]	Images and PDF Underlays (IMAGE, PDFATTACH)
PDFCLIP	–	Clip a selected PDF underlay to a closed profile	Images and PDF Underlays (IMAGE, PDFATTACH)
PDFIMPORT	PDFIN	Import a vector PDF's geometry: PDFIMPORT <file> [page]	Import DXF, PDF, Point Cloud and LandXML
PEDIT	PE	Join selected chain into polyline	BREAK, JOIN and PEDIT
PICKBOX	PICKBOXSIZE	Selection pickbox half-size in pixels: PICKBOX <1-50>	Status Bar: Snap, Ortho, Polar, OSnap
PLINE	PL, POLYLINE	Draw polyline	PLINE - Polyline
PLOT	PRINT	Print the drawing	PLOT and PREVIEW - Print and PDF
POINT	PO	Place a point object	POINT
POINTCLOUDATTACH	POINTCLOUD, PCATTACH	Import XYZ/PTS/CSV point cloud: POINTCLOUDATTACH <file> [max_points]	Import DXF, PDF, Point Cloud and LandXML
POLAR	–	Polar tracking: POLAR <ON|OFF> (F10)	Status Bar: Snap, Ortho, Polar, OSnap
POLYGON	POL	Draw polygon	POLYGON - Regular Polygon
PREVIEW	PRE	Print preview	PLOT and PREVIEW - Print and PDF
PROPERTIES	CH, MO, PR, DDMODIFY	What the selection is, and where to change it	PROPERTIES, MATCHPROP and CHPROP
PSPACE	PS	Switch to paper space	Layouts and viewports (LAYOUT, MVIEW, PAGESETUP)
PUBLISH	PUB	Batch all layouts to one PDF	PUBLISH - Combine All Layouts into One PDF
PURGE	PU	Remove unused layers/styles	AUDIT, RECOVER and PURGE
QDIM	–	Quick dimension chain across selected geometry	QDIM, DIMCONTINUE and DIMBASELINE
QLEADER	LE	Leader	QLEADER, LEADER, MLEADER and Callout
QPMODE	QUICKPROPERTIES, QP	Quick Properties box at the cursor: QPMODE <ON|OFF> (Ctrl+Shift+P)	Workspaces, Palettes and Clean Screen
QSELECT	QSE	Select by kind: QSELECT <kind> [layer] ('QSELECT ?' lists kinds)	Selecting Objects (SELECT, QSELECT, FILTER)
QTEXT	–	Draw text as boxes for speed: QTEXT <ON|OFF>	Units (UNITS) and System Variables
RAY	–	Construction line from a point, one direction only	XLINE and RAY - Construction Lines
RECOVER	–	Open a damaged drawing: RECOVER <file>	AUDIT, RECOVER and PURGE

Command	Alias	Description (from the program)	Help page
RECTANG	REC, RECTANGLE	Draw rectangle	RECTANG - Rectangle
REDO	–	Redo	ERASE, OOPS, UNDO and REDO
REGEN	RE, REGENALL, REDRAW, REDRAWALL	Rebuild the display from the model	Zoom, Pan and View Controls
REGION	REG	Turn closed profiles into a region (inner rings become holes)	BOUNDARY, REGION and Booleans (UNION/SUBTRACT/INTERSECT)
RENAME	REN	Rename a layer: RENAME <old> <new>	LAYER - Layer Manager
REVCLOUD	RC	Revision cloud around the selection: REVCLOUD [arc length]	SOLID, WIPEOUT and REVCLOUD
REVERSE	REV	Flip the direction of selected lines/polylines	DRAWORDER, OVERKILL, REVERSE and FLATTEN
ROTATE	RO	Rotate selection	ROTATE
SAVE	QSAVE	Save	NEW, SAVE and SAVEAS - Save DWG/DXF
SAVEAS	–	Save the drawing under a new name	NEW, SAVE and SAVEAS - Save DWG/DXF
SAVEPROJECT	SP, PROJ	Save all open tabs as an .mprj project	SAVEPROJECT - Save Multiple Tabs as a Project (.mprj)
SCALE	SC	Scale selection: SC [factor]	SCALE and SCALEREF - Resize Objects
SCALEREF	SR	Scale by reference: SCALEREF [measured] [real]	SCALE and SCALEREF - Resize Objects
SCRIPT	SCR, MACRO	Run Python macro (mcad API): SCRIPT [path.py]	SCRIPT - Python Macro API
SELECT	SE	Select tool	Selecting Objects (SELECT, QSELECT, FILTER)
SELECTLAST	SELL	Select the most recently created visible object	Selecting Objects (SELECT, QSELECT, FILTER)
SELECTPREVIOUS	SELP	Restore the previous selection set (AutoCAD P selection keyword)	Selecting Objects (SELECT, QSELECT, FILTER)
SELECTSIMILAR	SELSIM	Select every object of the same kind and layer	Selecting Objects (SELECT, QSELECT, FILTER)
SETVAR	SET	Set a system variable: SETVAR <name> <value> ('SETVAR ?' lists them)	Units (UNITS) and System Variables
SHEETSET	SSM, SHEETSETMANAGER	Multi-drawing sheet sets: SHEETSET <New|Add|List|Remove|Save|Open|Publish>	SHEETSET - Sheet Set Manager
SKETCH	–	Freehand: SKETCH [increment], drag to trace	SPLINE, SPLINEDIT and SKETCH

Command	Alias	Description (from the program)	Help page
`SLOT`	–	Straight slot	SLOT (Straight Slot)
`SNAP`	`SN`	Grid snap: SNAP <spacing|ON|OFF>	Status Bar: Snap, Ortho, Polar, OSnap
`SOLID`	`SO`	Filled triangle or quadrilateral	SOLID, WIPEOUT and REVCLOUD
`SPELL`	`SP2`	Check text for common drafting misspellings, repeats and brackets	Text Tools (ARCTEXT, TALIGN, TCOUNT, FIND ...)
`SPLINE`	`SPL`	Smooth curve through the points you click	SPLINE, SPLINEDIT and SKETCH
`SPLINEDIT`	`SPE`	Edit a spline: SPLINEDIT <Close|Open|Delete n|Add n x y>	SPLINE, SPLINEDIT and SKETCH
`STATUS`	–	Summarise what is in the drawing	Measuring and inquiry (DIST, AREA, LIST)
`STRETCH`	`S`	Stretch (crossing box + base + dest)	STRETCH and LENGTHEN
`STYLE`	`ST`	Text styles	DIMSTYLE and STYLE - Style Manager
`SUBTRACT`	`SU`	First profile minus the rest	BOUNDARY, REGION and Booleans (UNION/SUBTRACT/INTERSECT)
`TABLE`	`TB2`	Place a table: TABLE <rows> <cols> [colw] [rowh] | TABLE CSV <path>	TABLE and DATALINK - Tables
`TABLECELL`	`TCELL`	Write into the selected table: TABLECELL <row> <col> <text>	TABLE and DATALINK - Tables
`TABLEDIT`	`TED`	Edit a cell of the selected table: TABLEDIT <row> <col> <text>	TABLE and DATALINK - Tables
`TABLEEXPORT`	`TBEXP`	Write the selected table out as CSV	TABLE and DATALINK - Tables
`TALIGN`	`TA`, `ALIGNTEXT`	Align selected text onto a line (rotate + drop on)	Text Tools (ARCTEXT, TALIGN, TCOUNT, FIND ...)
`TCOUNT`	`TC`	Auto-number selected text: TCOUNT [start] [step] [p=pre] [s=suf]	Text Tools (ARCTEXT, TALIGN, TCOUNT, FIND ...)
`TEXT`	`DT`, `DTEXT`	Single-line text	TEXT, MTEXT and TEXTEDIT
`TEXTALIGN`	–	AutoCAD's name for TALIGN - align text onto a line	Text Tools (ARCTEXT, TALIGN, TCOUNT, FIND ...)
`TEXTEDIT`	`ED`, `DDEDIT`, `MTEDIT`	Edit the selected text: TEXTEDIT [new text]	TEXT, MTEXT and TEXTEDIT
`TEXTSIZE`	–	Default height for new text: TEXTSIZE <value>	Units (UNITS) and System Variables
`TEXTTOFRONT`	`TTF`	Lift annotation above the geometry: TEXTTOFRONT [Text|Dim|Both]	Text Tools (ARCTEXT, TALIGN, TCOUNT, FIND ...)
`TIME`	–	Drawing session time and edit count	Measuring and inquiry (DIST, AREA, LIST)

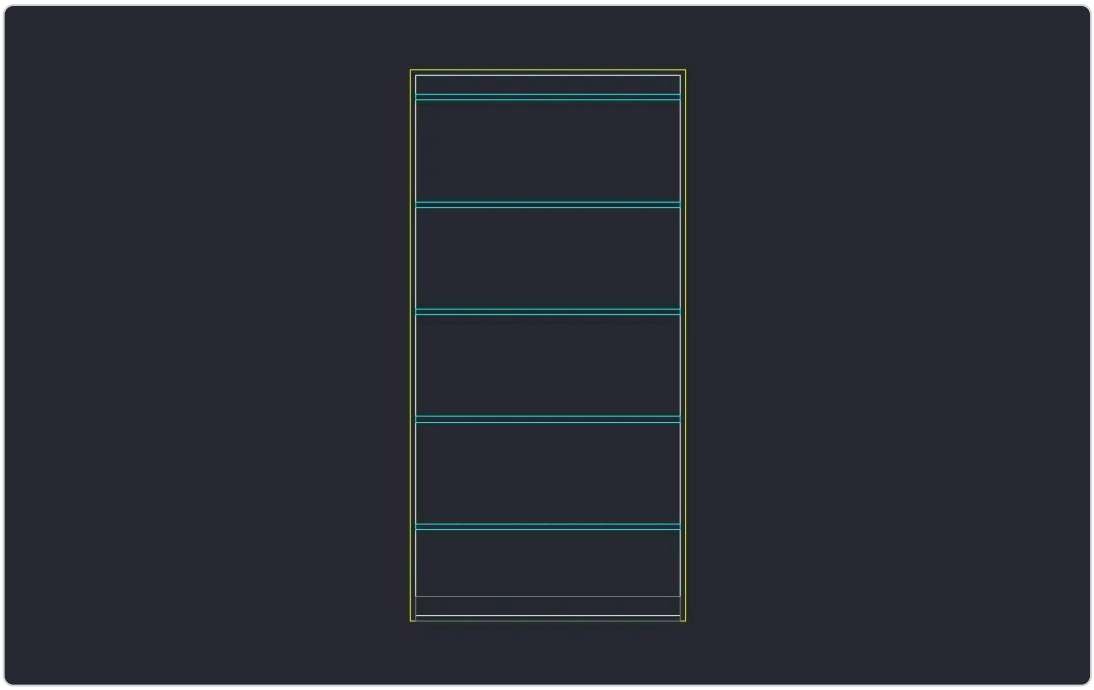
Command	Alias	Description (from the program)	Help page
TITLEBLOCK	TB	Insert title block on the layout sheet	TITLEBLOCK and MET Drawing Sheet
TOLERANCE	TOL	GD&T frame: TOLERANCE <symbol> <value> [datum] ('TOLERANCE ?' lists)	TOLERANCE - GD&T Feature Control Frame
TRIM	TR	Trim	TRIM, EXTEND and EXTRIM
UCS	–	User coordinate system: UCS <World|Origin x,y|Z angle|Previous|?>	Zoom, Pan and View Controls
UCSICON	–	The UCS axis marker: UCSICON <ON|OFF|ORIGIN|NOORIGIN>	Zoom, Pan and View Controls
UNDO	U	Undo	ERASE, OOPS, UNDO and REDO
UNGROUP	UNG	Break the selected group apart	GROUP, UNGROUP and GROUPEDIT
UNION	UNI	Merge selected profiles into one outline	BOUNDARY, REGION and Booleans (UNION/SUBTRACT/INTERSECT)
UNISOLATEOBJECTS	UNISOLATE	Bring back what ISOLATEOBJECTS hid	Selecting Objects (SELECT, QSELECT, FILTER)
UNITS	UN , DDUNITS	Report the drawing's unit settings	Units (UNITS) and System Variables
UPDATEFIELD	UPDATEF	Re-evaluate every field in the drawing	FIELD and UPDATEFIELD
VIEW	V	Named views: VIEW <Save|Restore|List|Delete> [name]	Zoom, Pan and View Controls
VIEWCUBE	NAVVCUBE	2D orientation cube: VIEWCUBE <ON|OFF|RESET>	Zoom, Pan and View Controls
VIEWLOCK	VL	Lock the view centre - only PAN/ZOOM may move it (can be used transparently)	Zoom, Pan and View Controls
VPCLIP	–	Clip a viewport to the selected profile: VPCLIP [Delete] [viewport]	Layouts and viewports (LAYOUT, MVIEW, PAGESETUP)
VPLAYER	–	Per-viewport layers: VPLAYER <Freeze|Thaw|List> <layer> [viewport]	Layouts and viewports (LAYOUT, MVIEW, PAGESETUP)
WBLOCK	W	Write selection to DXF	BLOCK, INSERT and WBLOCK
WIPEOUT	WO	Mask that hides whatever is under it	SOLID, WIPEOUT and REVCLOUD
WIRING	WW	Wiring Wizard: TIA IO tags -> A4 wiring pages	Wiring Wizard (WIRING) - PLC IO as A4 Pages
WSCURRENT	WORKSPACE	Current workspace: WSCURRENT [name]	Workspaces, Palettes and Clean Screen
WSDELETE	–	Forget a workspace: WSDELETE <name>	Workspaces, Palettes and Clean Screen

Command	Alias	Description (from the program)	Help page
`WSSAVE`	–	Save the arrangement: WSSAVE <name>	Workspaces, Palettes and Clean Screen
`XATTACH`	`XA`	Attach external DWG/DXF underlay	External Reference (XREF, XATTACH, XCLIP, XBIND)
`XBIND`	`XB`	Copy an xref's geometry into this drawing: XBIND [index]	External Reference (XREF, XATTACH, XCLIP, XBIND)
`XCLIP`	`XC`	Show only part of an xref: XCLIP [Delete] [index]	External Reference (XREF, XATTACH, XCLIP, XBIND)
`XLINE`	`XL`	Infinite construction line	XLINE and RAY - Construction Lines
`XPLODE`	`XP`	Explode with control: XPLODE [Layer <name>|Color <#RRGGBB>]	EXPLODE and XPLODE
`XREF`	`XR`	External references manager	External Reference (XREF, XATTACH, XCLIP, XBIND)
`ZOOM`	`Z`, `ZE`, `ZA`, `ZW`	Zoom (E/A/W/I/O) (can be used transparently)	Zoom, Pan and View Controls

Try it: Find the commands for a bookshelf elevation

You want a 1.8 m bookshelf with 5 evenly spaced shelves, but you cannot remember the name of the repeat-in-a-row command. Look it up in the Command Reference.

1. Press `F1` to open the Command Reference and type `array` in the Filter box. Only the ARRAY commands remain, with their syntax, for example `ARRAYRECT <count> [dx] [dy]`.
2. Carcass: `REC` `0,0`, then `900` `Tab` `1800`, and the inside `REC` `18,18`, then `864` `Tab` `1764` (18 mm boards).
3. First shelf: `REC` `18,300`, then `864` `Tab` `18`.
4. Select the shelf and type `ARRAYRECT 5 0 350`. You get 5 shelves 350 mm apart.
5. Kick plate: `REC` `18,0`, then `864` `Tab` `80`.



900 x 1800 bookshelf with 5 shelves

Challenge: Type `hatch` in the Filter box and work out how to give the kick plate a `LINE` pattern so it reads as timber.

See also

- Command Line, Aliases and Smart Input
- Options, Shortcuts and Aliases

Chapter 1 quiz: Getting started

Choose the one best answer. Answers are in Appendix A

1. **With the MET installer, how do you check for a new version yourself?**

   A. Help › About MCAD2026
   B. Help › Check for Updates...
   C. File › Start Center...
   D. Download it again from the Microsoft Store

2. **Which window shows the Release version that you should include when you report a problem to support?**

   A. Help › About MCAD2026
   B. Help › Activate License (online)...
   C. The Start Center
   D. Help › License Certificate (PDF)...

3. **You want to move your licence to a new computer. What should you do on the old computer first?**

   A. Uninstall MCAD only
   B. Delete the licence .json file
   C. Run Help › Import Offline License File...
   D. Run Help › Release License (free this seat)

4. **How do you activate MCAD on a computer that has no internet access?**

   A. Enter the License key without an e-mail
   B. Use a TRIAL key
   C. Import a .json licence file signed by MET with Help › Import Offline License File...
   D. Copy the program folder from another licensed computer

5. **Several drawings are open in tabs. Which shortcut switches to the next file tab?**

   A. Alt+Tab
   B. Ctrl+Tab
   C. Ctrl+PgDown
   D. F6

6. **In the first-drawing tutorial, why is a rounded-corner outline drawn as four LINEs instead of with REC?**

   A. FILLET works on LINE objects, not on an outline drawn with REC or PLINE
   B. REC cannot draw a 240 x 140 mm rectangle
   C. LINE draws faster than REC
   D. REC outlines cannot be saved as DXF

CHAPTER 2

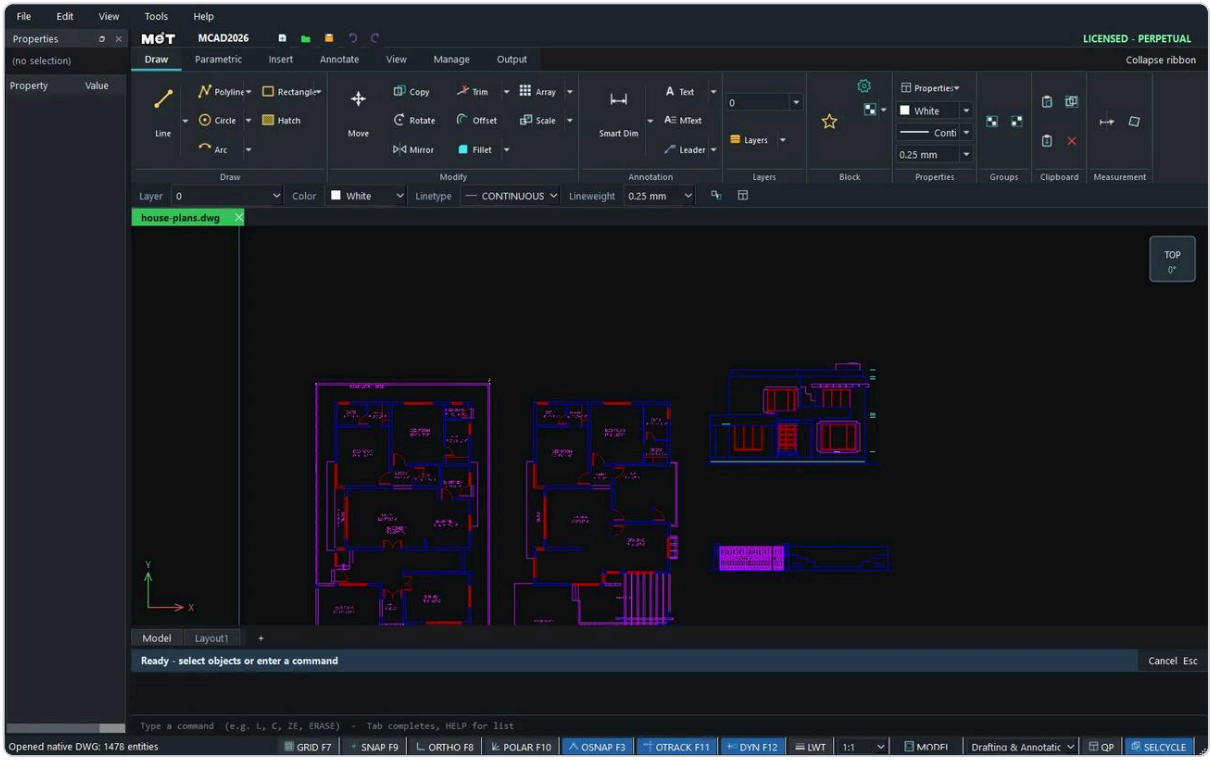
Interface & settings

1. The MCAD 2026 interface
   Learn the parts of the MCAD 2026 screen - the Ribbon, file tabs, drawing area, command line, Status bar and palettes.
2. Ribbon and Menus
   The buttons on all 7 Ribbon tabs, the contextual tabs, and the menu bar of MCAD 2026.
3. Command Line, Aliases and Smart Input
   Type commands and aliases, use command autocomplete and Smart Input, enter coordinates, and use CAL and M2P.
4. Status Bar: Snap, Ortho, Polar, OSnap
   Drafting aid buttons on the Status bar: GRID, SNAP, ORTHO, POLAR, OSNAP, OTRACK, DYN, and related commands.
5. Zoom, Pan and View Controls
   Zoom, pan, save views, lock the view, rotate the view, and the UCS coordinate system.
6. Options, Shortcuts and Aliases (OPTIONS / CUI)
   The Options window: general settings, keyboard shortcuts, command aliases, and exporting/importing profiles.
7. Workspaces, Palettes and Clean Screen
   Save and switch Workspaces, show/hide palettes, Clean Screen, and Quick Properties.
8. Units (UNITS) and System Variables
   Set drawing units (UNITS), LIMITS, the BASE point, and System Variables with SETVAR.
9. Measuring and inquiry (DIST, AREA, LIST)
   Measure distance, angle, area, radius and point coordinates, and summarise a drawing with DIST, AREA, ID, LIST and STATUS.

The MCAD 2026 interface

Learn the parts of the MCAD 2026 screen - the Ribbon, file tabs, drawing area, command line, Status bar and palettes.

Screen overview



MCAD 2026 main window

Part	Location	Purpose
Menu bar	Top	File, Edit, View, Tools, Help
Quick Access Toolbar	Next to the MET logo	New, Open, Save, Undo, Redo
Ribbon	Below the menu	The Draw, Parametric, Insert, Annotate, View, Manage and Output tabs. See Ribbon and menus.
Licence status	Top right	For example `LICENSED - PERPETUAL`
Quick Properties bar	Below the Ribbon	Layer, Color, Linetype and Lineweight for new objects or the selected objects
File tabs	Above the drawing area	One tab per file (`*` = not saved yet)
Drawing area (canvas)	Centre of the screen	Draw and edit here. It has the X/Y axis icon and the ViewCube (TOP).
Model / Layout tabs	Below the drawing area	Switch between model space and the paper layout
Prompt bar	Below the Layout tabs	Shows the next step of the command. It has a **Cancel Esc** button.

Part	Location	Purpose
Command line	Below the prompt bar	The last 3 lines of command history and the box where you type commands
Status bar	Bottom	Cursor coordinates, selection count, the GRID/SNAP/ORTHO/POLAR/OSNAP/OTRACK/DYN/LWT buttons, Annotation scale, Workspace
Properties palette	Left	The properties of the selected objects
Content Browser	Right	The block and symbol libraries. See Symbol Library.

Using the mouse

Action	Result
Left-click	Select an object / specify a point.
Drag left to right	Window selection (selects only objects that lie entirely inside the box).
Drag right to left	Crossing selection (selects objects that cross the box).
Roll the mouse wheel	Zoom in or out around the cursor position.
Hold the wheel and drag	Pan.
Right-click	The context menu. See Clipboard.
Double-click text	Edit the text in place.

Try it: Draw a doormat and watch the screen respond

The quickest way to learn the screen is to draw something small and watch what each part tells you. Try an 800 x 500 mm doormat.

1. Move the mouse in the drawing area. The X/Y coordinates on the left of the Status bar follow it.
2. Type `REC`. The Prompt bar tells you the next step. Type `0,0` Enter, then `800` `Tab` `500` Enter. The file tab at the top now shows `*`, meaning not saved yet.
3. Inner border: `REC` `50,50`, then `700` `Tab` `400`.
4. Click the outer rectangle. The Properties palette on the left shows Area `400000` and Perimeter `2600` under **Measured**.
5. While it is selected, change its colour on the Quick Properties bar below the Ribbon, then roll the mouse wheel to zoom in.



800 x 500 doormat with an inner border and WELCOME

Challenge: Drag a crossing window from right to left over the inner border and read the selection count on the Status bar.

See also

- Command Line, Aliases and Smart Input
- Status Bar: Snap, Ortho, Polar, OSnap
- Workspaces, palettes and Clean Screen

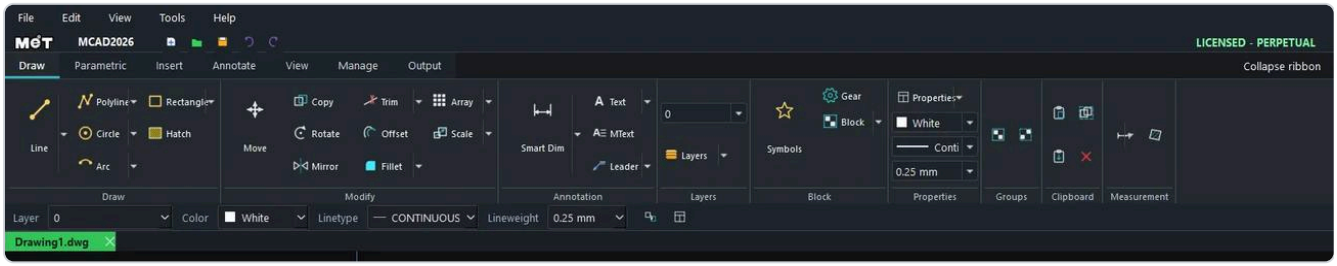
Ribbon and Menus

The buttons on all 7 Ribbon tabs, the contextual tabs, and the menu bar of MCAD 2026.

What it does

The Ribbon groups the most-used commands into tabs and groups. A button with an arrow ▾ has a flyout submenu that holds other variations of the command. Double-click a tab name, or click **Collapse ribbon**, to collapse the Ribbon.

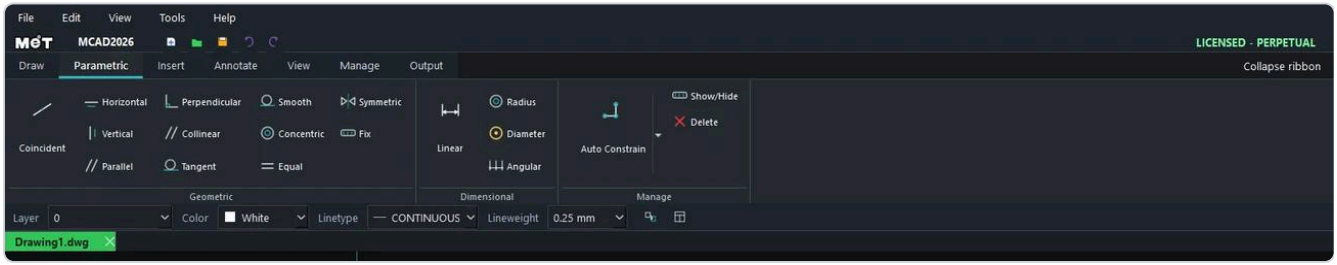
Draw tab



Draw tab

Group	Buttons (submenu)
Draw	Line (2-Point, From Midpoint, Dual Line, Construction Line, Ray, Multiline, Dashed Line), Polyline (Open/Closed Loop, Spline, Freehand Sketch, Boundary), Circle (Center-Radius, Donut, Point), Arc (3-Point), Rectangle (Corner, Center, Revision Cloud, 2D Solid, Wipeout), Hatch
Modify	Move, Copy, Rotate, Mirror, Trim (Extend, Stretch, EXTRIM, Break, Join, Lengthen, Align, Explode, Bring to Front, Send to Back), Offset, Fillet (Chamfer), Array (Rectangular, Polar, Array along Curve), Scale (Scale by Reference)
Annotation	Smart Dim (Angle between 2 lines, Leader, Callout box, Ordinate), Text (Single Line / Multiline), MText, Leader (Multileader, Callout, Ordinate)
Layers	Current Layer box, Layers (Layer tools)
Block	Symbols, Gear, Block (Insert Block, Picture / Logo, Block Editor)
Properties	Properties (Properties Palette, Match Properties), Color, Linetype, Lineweight
Groups	Group, Ungroup
Clipboard	Copy, Copy Window, Paste, Delete
Measurement	Measure (Distance, Angle, Radius, Area, Coordinates), Area

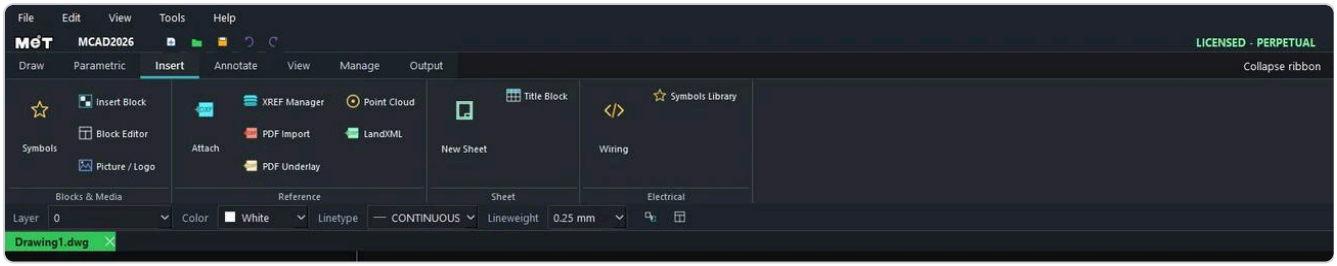
Parametric tab



Parametric tab

Geometric (Coincident, Horizontal, Vertical, Parallel, Perpendicular, Collinear, Tangent, Smooth, Concentric, Equal, Symmetric, Fix) · Dimensional (Linear, Radius, Diameter, Angular) · Manage (Auto Constrain, Show/Hide, Delete). See Geometric Constraints.

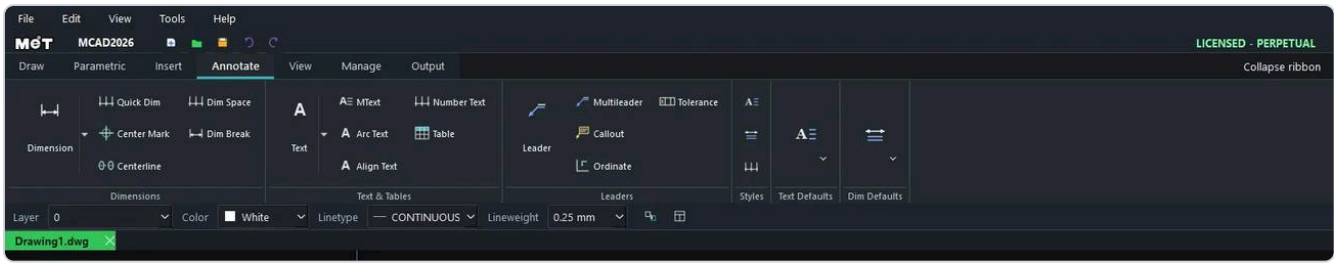
Insert tab



Insert tab

Blocks & Media (Symbols, Insert Block, Block Editor, Picture / Logo) · Reference (Attach, XREF Manager, PDF Import, PDF Underlay, Point Cloud, LandXML) · Sheet (New Sheet, Title Block) · Electrical (Wiring, Symbols Library)

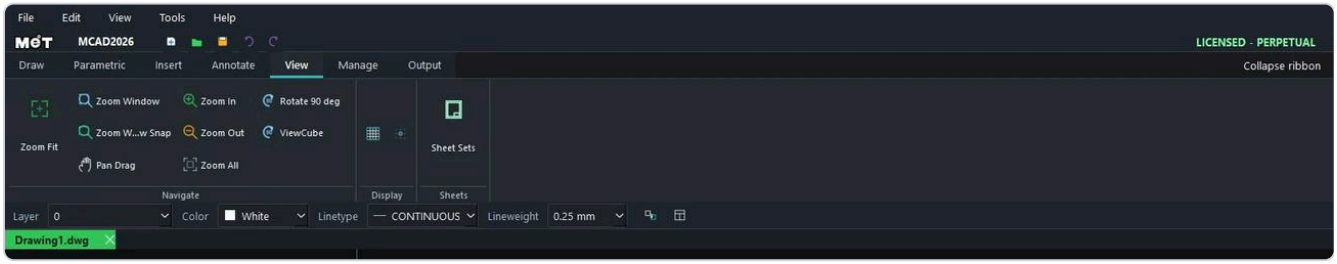
Annotate tab



Annotate tab

Dimensions (Dimension, Quick Dim, Center Mark, Centerline, Dim Space, Dim Break) · Text & Tables (Text, MText, Arc Text, Align Text, Number Text, Table) · Leaders (Leader, Multileader, Callout, Ordinate, Tolerance) · Styles (Text Style, Dim Style, Add Object Scale) · Text Defaults · Dim Defaults

View tab



View tab

Navigate (Zoom Fit, Zoom Window, Zoom Window Snap, Pan Drag, Zoom In, Zoom Out, Zoom All, Rotate 90 deg, ViewCube) · Display (Grid Display, Grid Snap) · Sheets (Sheet Sets)

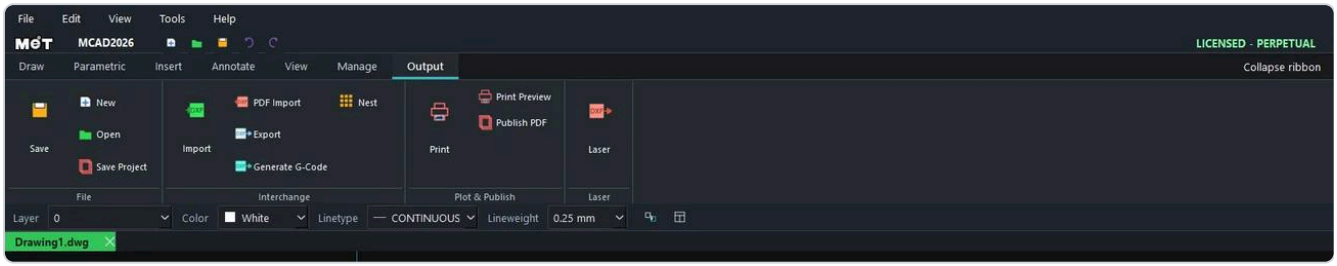
Manage tab



Manage tab

Customization (Options, Drafting Settings, Keyboard/Aliases) · CAD Standards (Layers, Text Style, Dim Style, Drawing Units) · Drawing Cleanup (Audit, Purge, Delete Duplicates, Drawing Status)

Output tab



Output tab

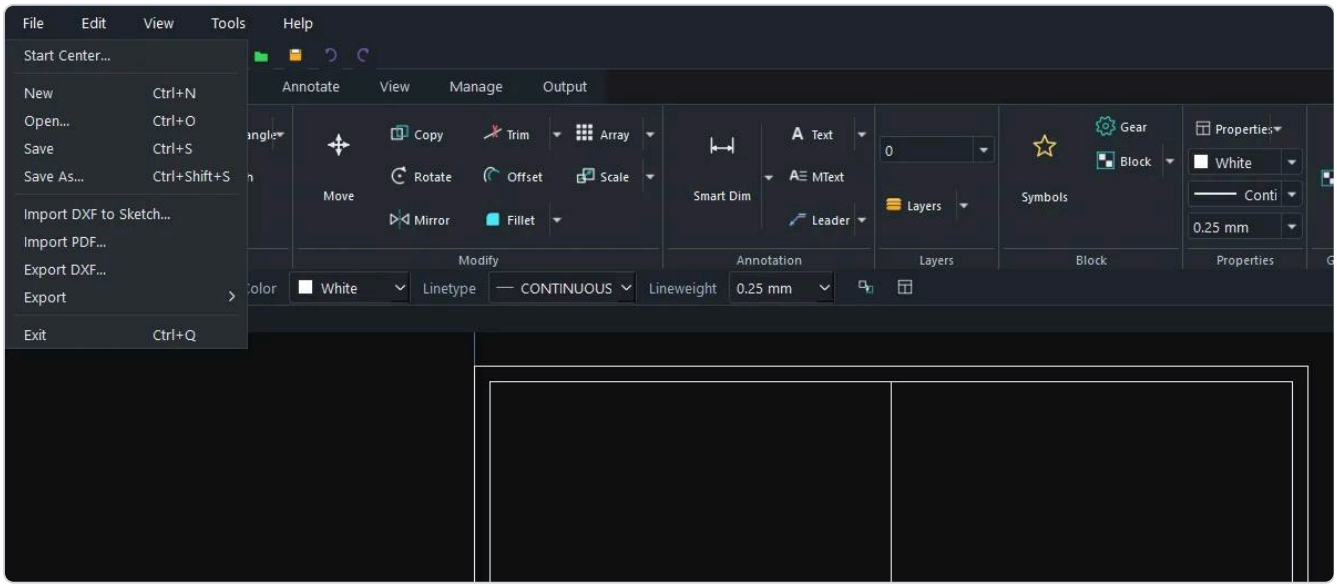
File (Save, New, Open, Save Project) · Interchange (Import, PDF Import, Export, Generate G-Code, Nest) · Plot & Publish (Print, Print Preview, Publish PDF)

Contextual tabs

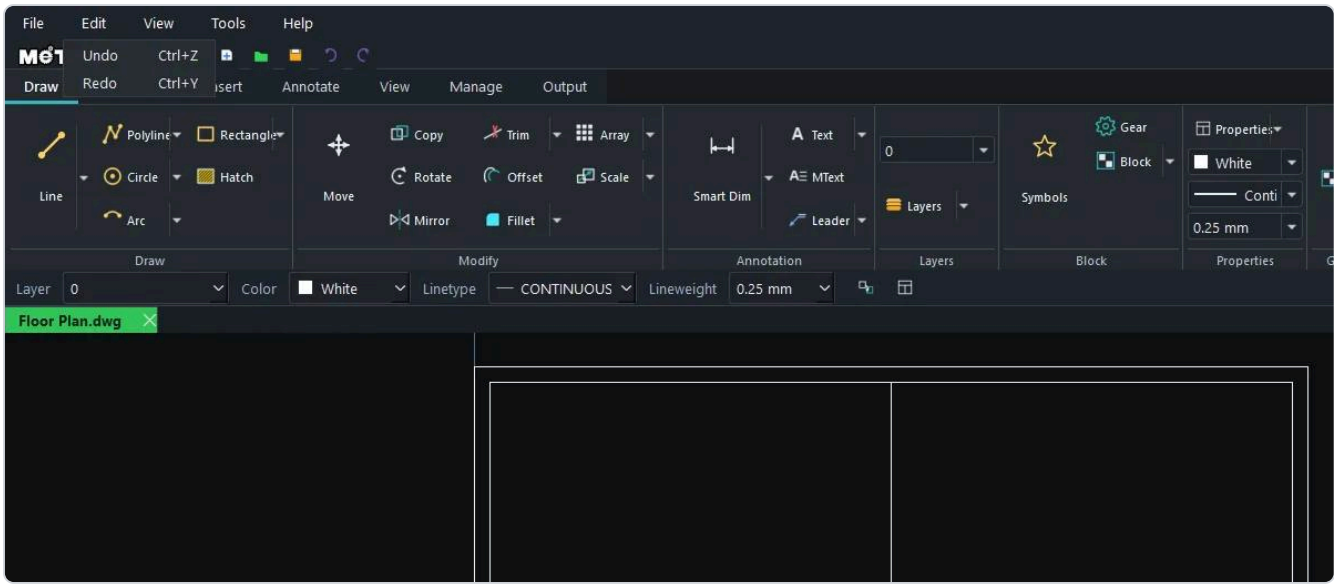
- **Hatch** – appears when you select a Hatch, so you can change the Pattern, Scale, and Angle right away. See HATCH.
- **Text Editor** – appears while you type text in the drawing area (Font, size, Bold/Italic/Underline, Justify, Bullets). See TEXT, MTEXT.

Menu bar

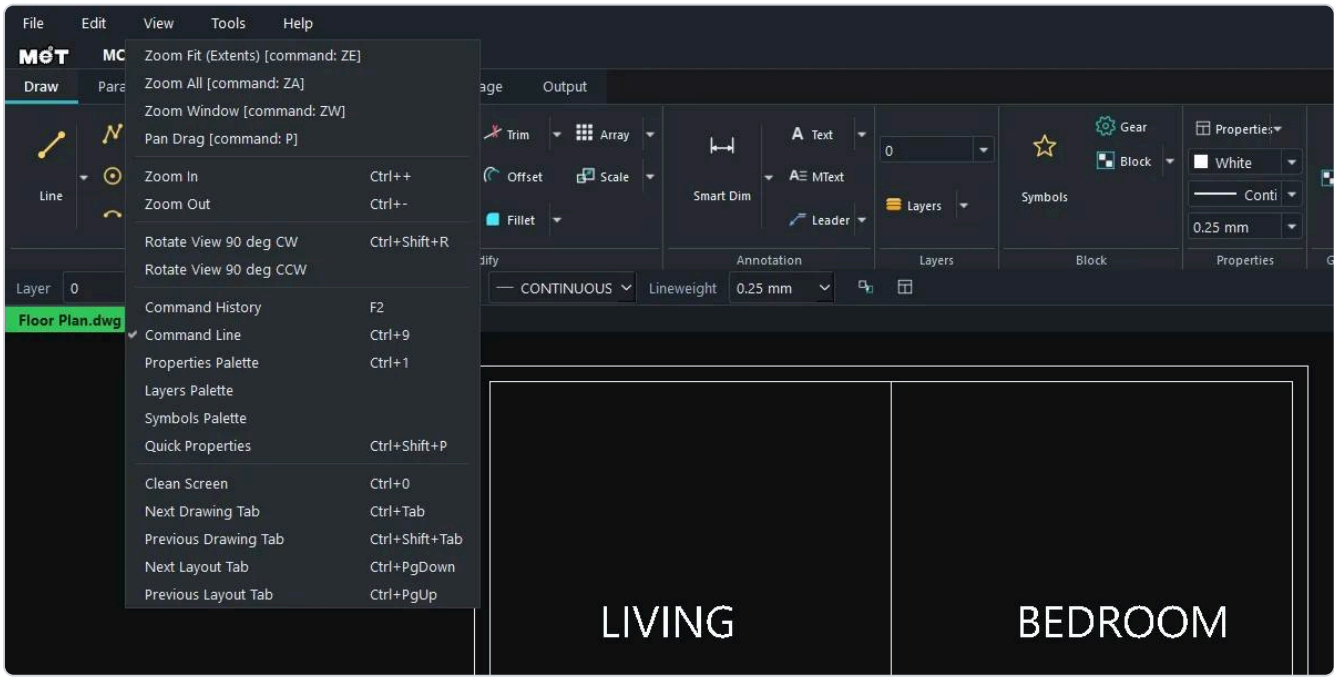
Menu	Items
File	Start Center..., New, Open..., Save, Save As..., Import DXF to Sketch..., Import PDF..., Export DXF..., Export › Generate G-Code..., Exit
Edit	Undo, Redo
View	Zoom Fit/All/Window, Pan Drag, Zoom In/Out, Rotate View 90°, Command History, Command Line, Properties Palette, Layers Palette, Symbols Palette, Quick Properties, Clean Screen, Next/Previous Drawing Tab, Next/Previous Layout Tab
Tools	Options..., Customize Shortcuts and Aliases...
Help	Command Reference... (F1), Activate License, Release License, Import Offline License File..., License Certificate (PDF)..., Check for Updates..., About



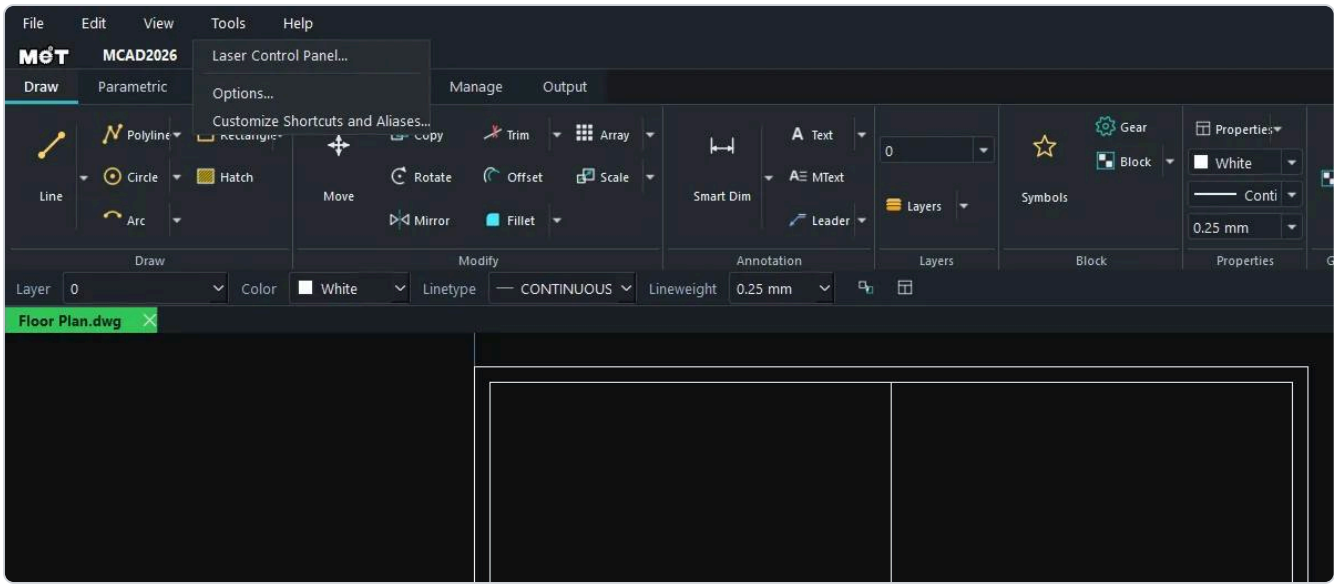
File menu



Edit menu



View menu

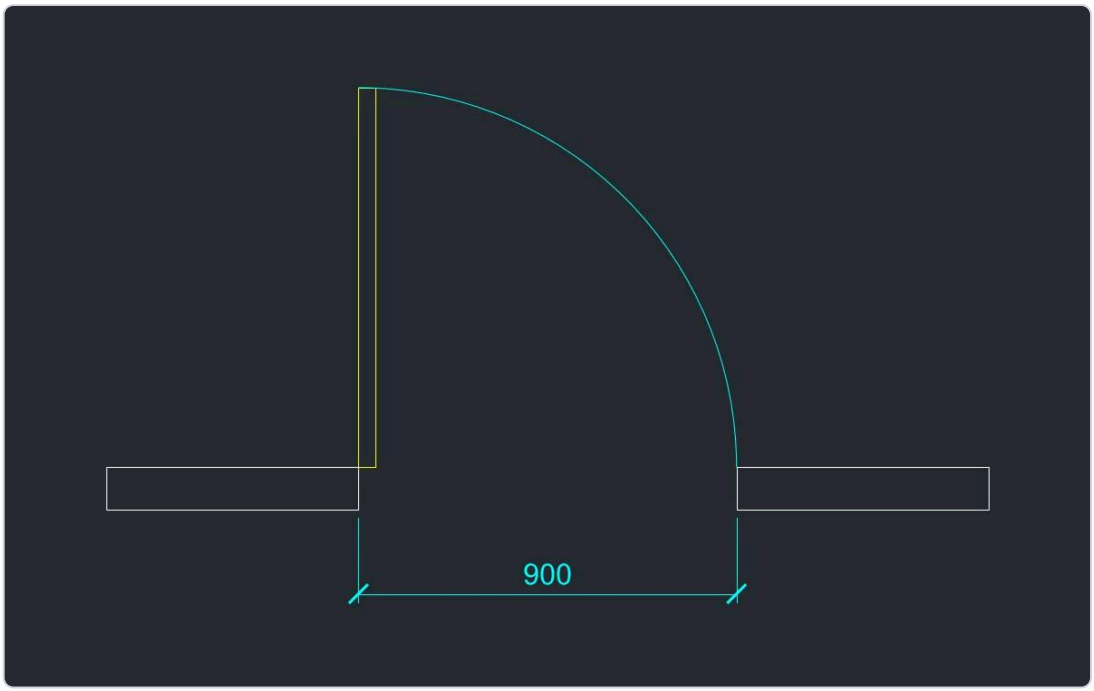


Tools menu

Try it: A door swing, starting every command from the Ribbon

A 900 door with its swing arc, the symbol on every floor plan. Practise finding the Ribbon buttons (you can still type the coordinates).

1. **Draw › Draw › Rectangle**, then `-600,-100` and `600` `Tab` `100` (left wall).
2. **Rectangle** again, `900,-100` and `600` `Tab` `100` (right wall, leaving a 900 opening).
3. Door leaf in the open position: **Rectangle**, `0,0` and `40` `Tab` `900`.
4. Swing arc: **Draw › Draw › Arc ▾ › 3-Point**, then `0,900`, `636.4,636.4`, `900,0` (the middle point is on the 900 radius at 45°).
5. Select the arc and change its colour or linetype in the **Properties** group on the Ribbon.



900 door with its swing arc

Challenge: Turn the four pieces into a block with **Draw › Block › Block** and place it in every room. See BLOCK.

See also

- Interface Components
- Command Index

Command Line, Aliases and Smart Input

Type commands and aliases, use command autocomplete and Smart Input, enter coordinates, and use CAL and M2P.

What it does

Every command in MCAD 2026 can be typed with its full AutoCAD-style name or a short alias (for example `L` = LINE, `C` = CIRCLE). You can type in two places:

- **Type directly in the drawing area** – the command box opens at the cursor position.
- **Type in the Command line** at the bottom of the window.

Entering a command

1. Type the command name or alias, for example `DIM`.
2. A list of suggestions appears above the Command line. Use the up and down arrows to choose, or press `Tab` to complete the name.
3. Press `Enter` or `Space` to run it. Press `Esc` to cancel.
4. Press `Enter` or `Space` when no command is active to repeat the last command.
5. Press the up and down arrows when no suggestions are shown to recall commands you typed before.



Command suggestions when you type DIM

Most commands accept values after the name on the same line, for example `OFFSET 10`, `FILLET 5`, `ARRAYPOLAR 8` or `HATCH ANSI31 2 45`. If you leave them out, the program asks in the prompt bar and shows the previous value in angle brackets `<...>`. Press Enter to use that value.

Transparent commands

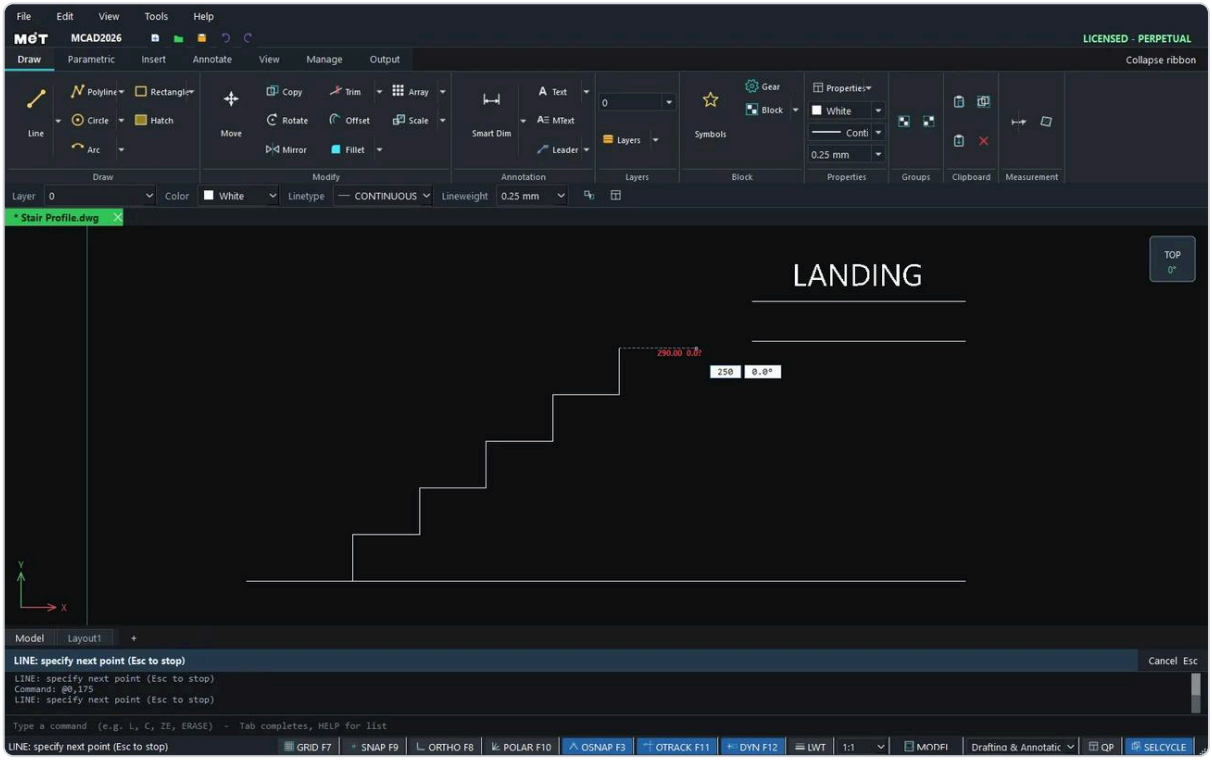
Type `'` before a command to run it in the middle of another command without cancelling it. The transparent commands are `'ZOOM`, `'PAN`, `'VIEWLOCK` and `'M2P`. Starting any other command ends the command that is waiting (as in AutoCAD).

Smart Input (Dynamic Input)

While a drawing command is waiting for a point, two number boxes appear at the cursor.

- Type a length → press `Tab` to go to the angle box → type an angle → press `Enter`.

- RECTANG shows W / H boxes, and CIRCLE shows the radius.
- Turn it on or off with `F12` (DYN on the Status bar).



Typing a length of 250 in the Smart Input box while drawing a LINE

Typing coordinates when the program asks for a point

When the prompt bar asks for a point (for example in ID, DIST, POINT or MOVE), you can type one of the following and press Enter.

Format	Meaning
`x,y`	An absolute coordinate, for example `100,50`.
`@dx,dy`	A relative distance from the previous point, for example `@40,0`.
`d<a`	A distance and angle, for example `50<30`.

CAL - Calculator

`CAL <expression>` calculates numbers on the command line. It supports `+ - * / % **`, `sqrt`, `sin`, `cos`, `tan` (in degrees), `atan`, `abs`, `round`, `min`, `max`, `pi` and `e`.

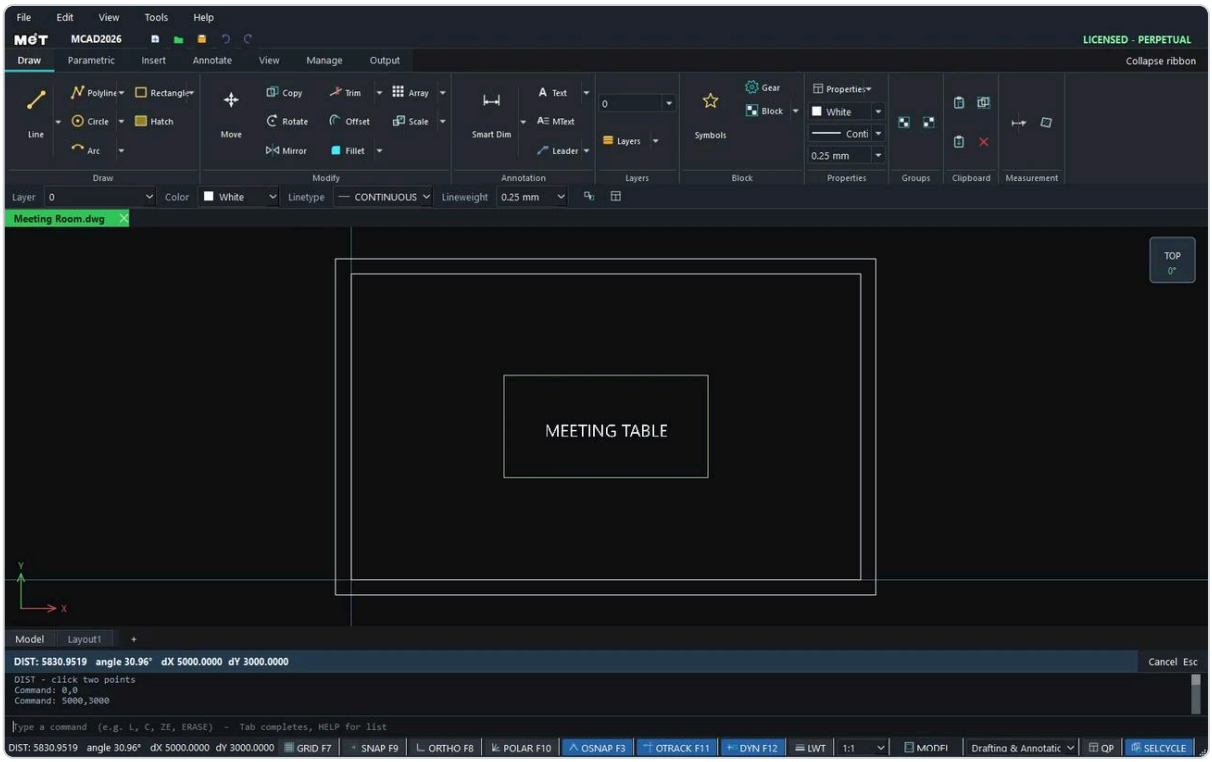
Examples: `CAL 25.4*3` → 76.2, `CAL sqrt(2)`, `CAL sin(30)`

M2P - Midpoint between two points

During a command that asks for a point, type `'M2P` (or `MTP`) and click two points. The midpoint between them is passed to the command.

Command output

Inquiry commands such as `DIST` show their result in the prompt bar and in the Command line history. Press `F2` to open the full command history. Press `Ctrl+9` to hide or show the Command line.

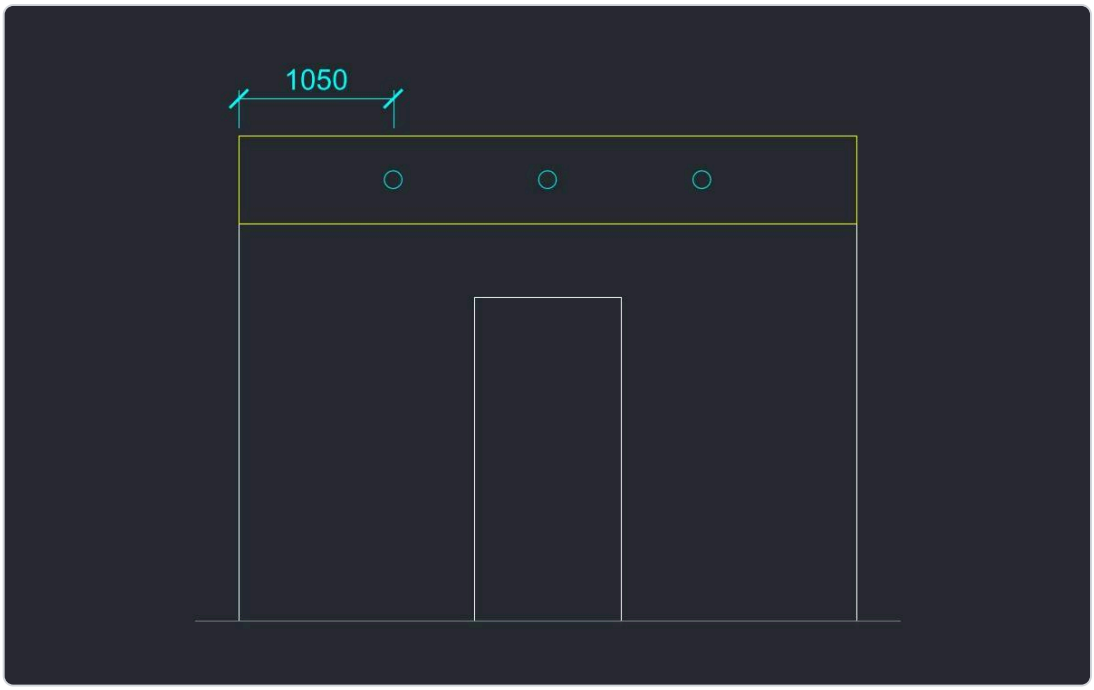


DIST result shown in the prompt bar

Try it: Storefront fascia with evenly spaced lamps

A 4.2 m shopfront with a 600 mm fascia and three sign lamps that must be exactly evenly spaced. You use CAL to do the maths on the command line and type coordinates instead of clicking.

1. Type `CAL 4200/4` and press Enter. The answer `1050` is the spacing for 3 lamps (4 gaps).
2. Shop opening: `REC` `0,0`, then `4200` `Tab` `2700` Enter.
3. Fascia: `REC` `0,2700`, then `4200` `Tab` `600` Enter.
4. Lamps: `C` `1050,3000` radius `60`, then repeat at `2100,3000` and `3150,3000` (press Enter on an empty line to repeat CIRCLE).
5. 1000 wide door: `L` `1600,0`, then `@0,2200` `@1000,0` `@0,-2200` (Enter after each), and `Esc`.



4.2 m storefront with a fascia and three lamps 1050 apart

Challenge: While LINE is asking for a point, type `'M2P` and click the two bottom corners of the shopfront. The line starts exactly in the middle.

See also

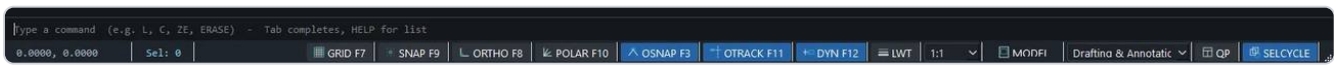
- Command Index
- Options, Shortcuts and Aliases
- Status Bar

Status Bar: Snap, Ortho, Polar, OSnap

Drafting aid buttons on the Status bar: GRID, SNAP, ORTHO, POLAR, OSNAP, OTRACK, DYN, and related commands.

What it does

The Status bar at the bottom of the window shows the cursor coordinates, the number of selected objects, and buttons that turn drafting aids on and off. A blue button means the aid is on.



Status bar

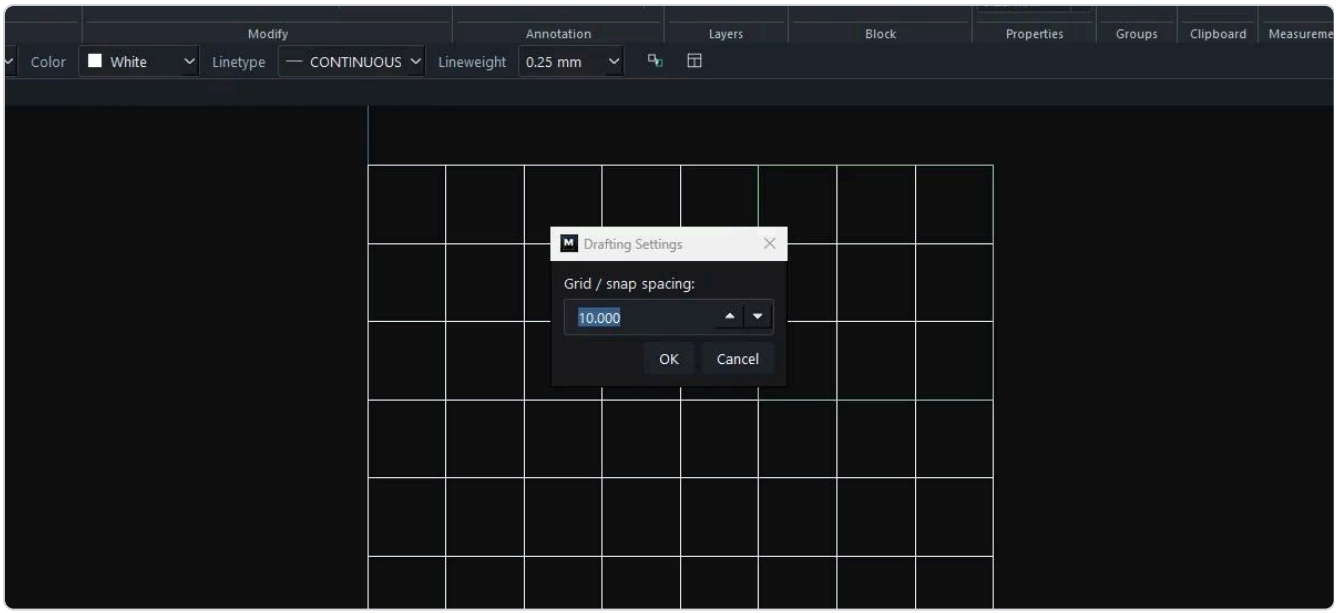
Status bar buttons

Button	Shortcut	Equivalent command	Meaning
GRID	`F7`	–	Shows the Grid
SNAP	`F9`	`SNAP`	Grid Snap: the cursor jumps by the Grid spacing

Button	Shortcut	Equivalent command	Meaning
ORTHO	`F8`	`ORTHO`	Forces horizontal/vertical lines
POLAR	`F10`	`POLAR`	Polar tracking at the set angles (default 45°)
OSNAP	`F3`	`OSNAP`	Object Snap: snaps to End, Mid, Center, Quadrant, Intersection, Nearest, and more
OTRACK	`F11`	–	Object Snap Tracking: draws tracking lines from snap points
DYN	`F12`	–	Smart Input at the cursor
LWT	–	–	Shows line weights
1:1 ▾	–	–	Annotation scale. See Annotation Scale
MODEL	–	–	Switches between Model / Layout
Drafting & Annotation ▾	–	`WSCURRENT`	Workspace. See Workspaces
QP	`Ctrl+Shift+P`	`QPMODE`	Quick Properties at the cursor
SELCYCLE	–	–	Click again at the same spot to cycle through overlapping objects

Related commands

Command	Alias	Format	Meaning
SNAP	`SN`	`SNAP <distance|ON|OFF>`	Sets the Grid Snap spacing (entering a number also turns Snap on)
OSNAP	`OS`, `DDOSNAP`	`OSNAP <ON|OFF>`	Turns Object Snap on/off
ORTHO	–	`ORTHO <ON|OFF>`	Turns Ortho on/off
POLAR	–	`POLAR <ON|OFF>`	Turns Polar tracking on/off (turning Polar on turns Ortho off)
APERTURE	–	`APERTURE <1-50>`	Size of the Object Snap target (pixels)
PICKBOX	`PICKBOXSIZE`	`PICKBOX <1-50>`	Size of the object selection box (pixels)
OSMODE	–	`OSMODE <value>`	Object Snap set as a bitmask (same as AutoCAD)
DSETTINGS	`SE2`, `DDRMODES`	–	Drafting Settings window: Grid/Snap spacing, followed by the Object snap aperture
ISOPLANE	`ISO`	`ISOPLANE <Left|Top|Right|OFF>`	Isometric drafting: sets Polar to the 30° axes of the selected plane
DIMSNAP	–	`DIMSNAP [ON|OFF]`	Lets Smart Dimension snap to End/Mid/Center at close range (OFF = measure objects only)



Drafting Settings window

Steps: set a 5 mm Grid Snap

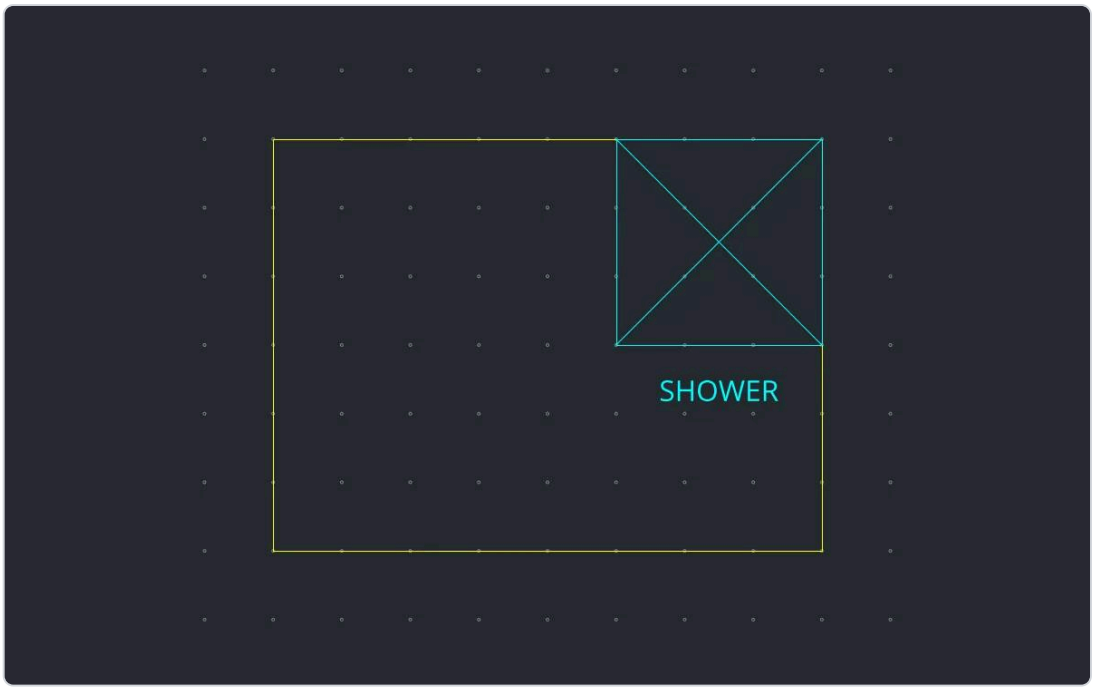
1. Type `SNAP 5` and press Enter (or open **Manage › Customization › Drafting Settings** and enter `5`).
2. Press `F7` to show the Grid.
3. Start drawing. The cursor jumps every 5 mm. Press `F9` to turn it off.

> **Note:** `ISOPLANE` does not tilt the grid. The drawing is still true 2D. The 30° Polar angles make the drawing look isometric. `ISOPLANE OFF` restores the 45° Polar angles.

Try it: Bathroom floor on a 300 mm tile grid

Set Grid Snap to the tile size and the cursor jumps one tile at a time. Draw a 2.4 x 1.8 m bathroom and a 900 x 900 shower tray without typing a single number.

1. Type `SNAP 300`, press Enter, then press `F7` to show the Grid.
2. Press `F8` to turn on ORTHO so lines stay horizontal or vertical.
3. `PL`, then click along the grid: `0,0` → `2400,0` → `2400,1800` → `0,1800` → `0,0` (watch the Status bar coordinates as you click), then Enter.
4. Shower tray: `REC`, click `1500,900`, then click `2400,1800` (3 x 3 tiles).
5. Press `F9` to turn Snap off when you draw fixtures that do not sit on the grid.



2400 x 1800 bathroom on a 300 mm grid with a shower tray

Challenge: Type `ISOPLANE Top` and sketch the shower tray as an isometric view with 30° Polar, then `ISOPLANE OFF`.

See also

- Command Line and Smart Input
- Units and System Variables

Zoom, Pan and View Controls

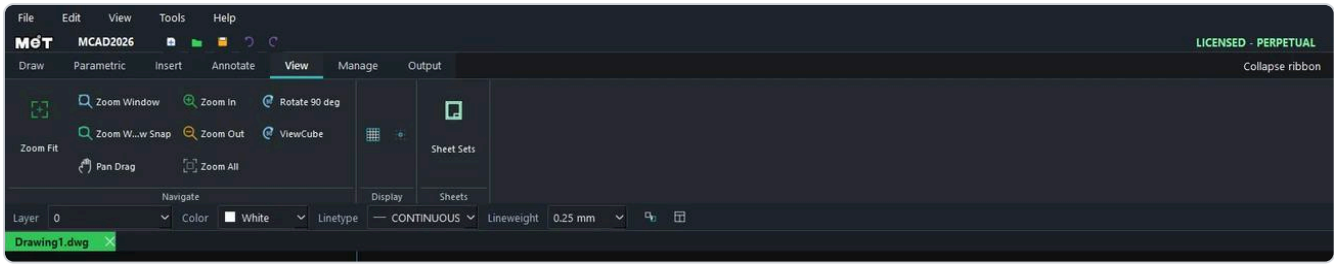
Zoom, pan, save views, lock the view, rotate the view, and the UCS coordinate system.

What it does

Controls the view in the drawing area without changing the drawing itself.

Where to find it

- Ribbon: **View › Navigate** and **Draw › (Zoom Fit ▾)**
- **View** menu
- Mouse: scroll the wheel = Zoom, press and drag the wheel = Pan



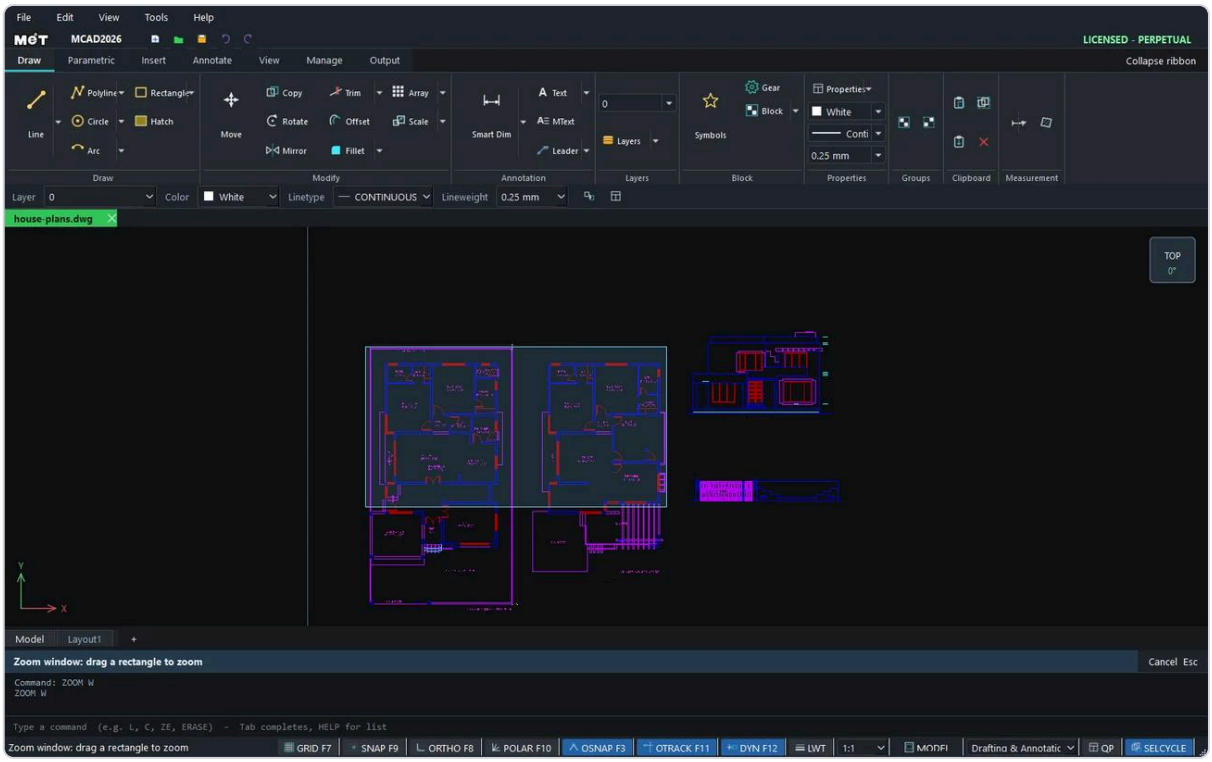
View tab, Navigate group

Commands

Command	Alias	Format	Meaning
ZOOM	`Z`, `ZE`, `ZA`, `ZW`	`ZOOM <E|A|W|I|O|ZWS>`	E = Extents (fit all objects), A = All, W = Window, ZWS = Window with Snap, I/O = in/out (can be used inside another command as `'ZOOM`)
PAN	`P`	–	Drag to pan. Press `Esc` to exit (can be used inside another command)
VIEW	`V`	`VIEW <Save|Restore|List|Delete> [name]`	Save/recall named views
VIEWLOCK	`VL`	`VIEWLOCK [ON|OFF]`	Locks the view center so it changes only with PAN/ZOOM
REGEN	`RE`, `REGENALL`, `REDRAW`, `REDRAWALL`	–	Redraws the screen from the drawing data
VIEWCUBE	`NAVVCUBE`	`VIEWCUBE <ON|OFF|RESET>`	Show/hide the ViewCube. RESET = return to the TOP 0° view (click the ViewCube to rotate the view by 90°)
UCS	–	`UCS <World|Origin x,y|Z angle|Previous|?>`	Moves the coordinate reference point or rotates the axes (does not change the drawing, only the coordinates you type and see)
UCSICON	–	`UCSICON <ON|OFF|ORIGIN|NOORIGIN>`	Shows the axis icon at the screen corner or at the Origin

Steps: Zoom Window

1. Type `ZW` (or `ZOOM W`) and press Enter.
2. The Prompt bar shows `Zoom window: drag a rectangle to zoom`.
3. Drag a window around the area you want.



Zoom Window: dragging a window around the floor plans (house-plans.dwg demo)

Shortcuts

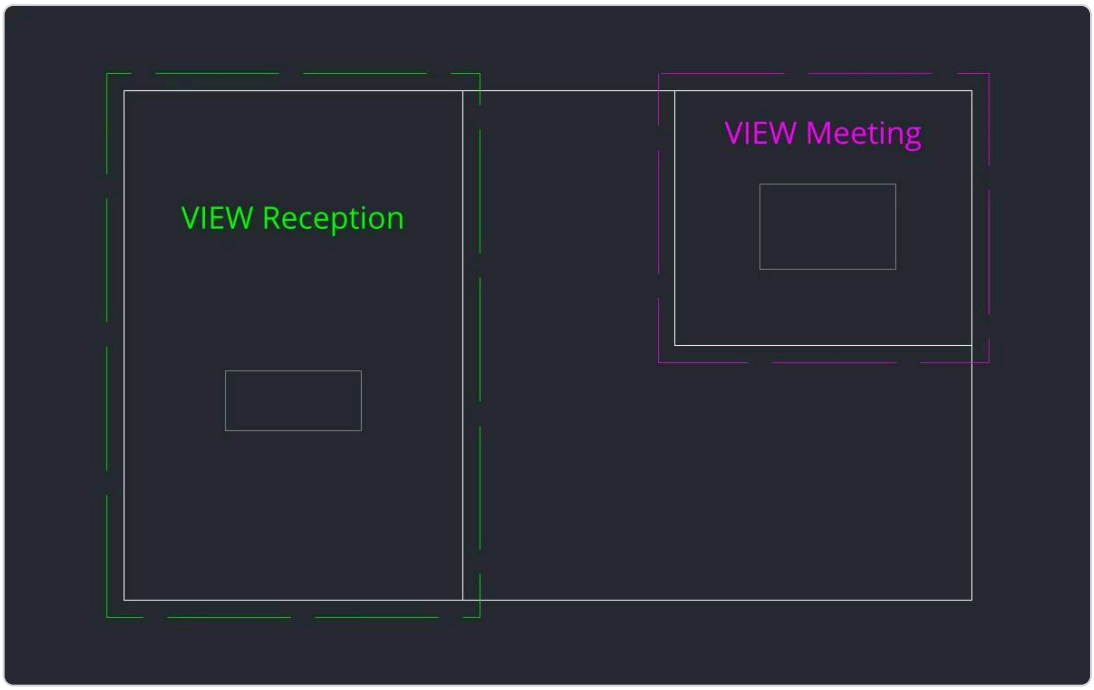
Shortcut	Meaning
`Ctrl++` / `Ctrl+-`	Zoom In / Zoom Out
`Ctrl+Shift+R`	Rotate the view 90° clockwise
`Ctrl+Tab` / `Ctrl+Shift+Tab`	Next / previous file tab
`Ctrl+PgDown` / `Ctrl+PgUp`	Next / previous Layout tab

> **Note:** XLINE and RAY are not counted in Zoom Extents, so they do not shrink your real drawing until it is too small to see.

Try it: Named views of a small office plan

A 10 x 6 m office with a reception area and a meeting room. Save named views, then jump between them with one command instead of zooming around every time.

1. Draw the plan: `REC` `0,0`, then `10000` `Tab` `6000`; the reception wall `L` `4000,0` `@0,6000`; and the meeting room `REC` `6500,3000`, then `3500` `Tab` `3000`.
2. Type `ZW`, drag a window around the reception area on the left, then type `VIEW Save Reception`.
3. Type `ZW`, drag around the meeting room, then type `VIEW Save Meeting`.
4. Type `ZE` to see the whole plan, then `VIEW Restore Meeting`. The screen jumps straight back to the meeting room.
5. Type `VIEW List` to see the saved views.



Office plan with the Reception and Meeting view frames

Challenge: Type `VIEWLOCK ON` before you edit the meeting room so the view stays put while you work.

See also

- Interface Components
- Layouts and Viewports

Options, Shortcuts and Aliases (OPTIONS / CUI)

The Options window: general settings, keyboard shortcuts, command aliases, and exporting/importing profiles.

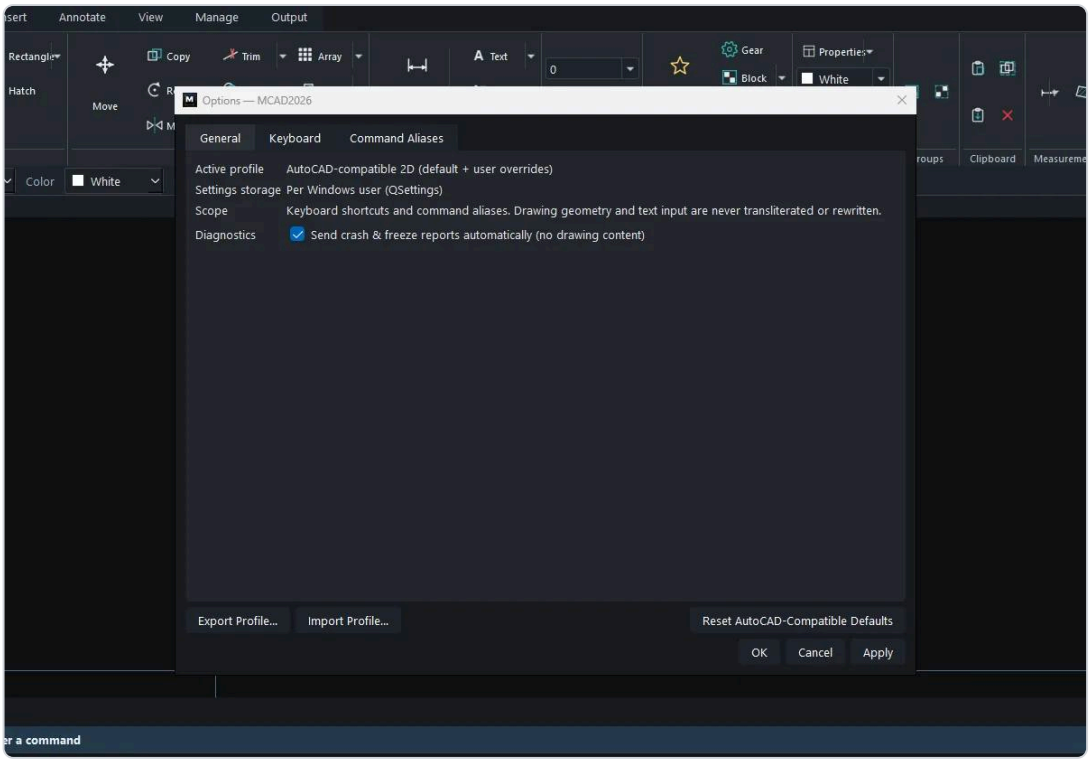
What it does

Customize shortcuts and aliases to suit your habits, and move your settings to another computer.

Where to find it

- Ribbon: **Manage › Customization › Options** or **Keyboard/Aliases**
- Menu **Tools › Options...** / **Tools › Customize Shortcuts and Aliases...**
- Command line: `OPTIONS` (`OP`), `CUI` , `ALIASEDIT`

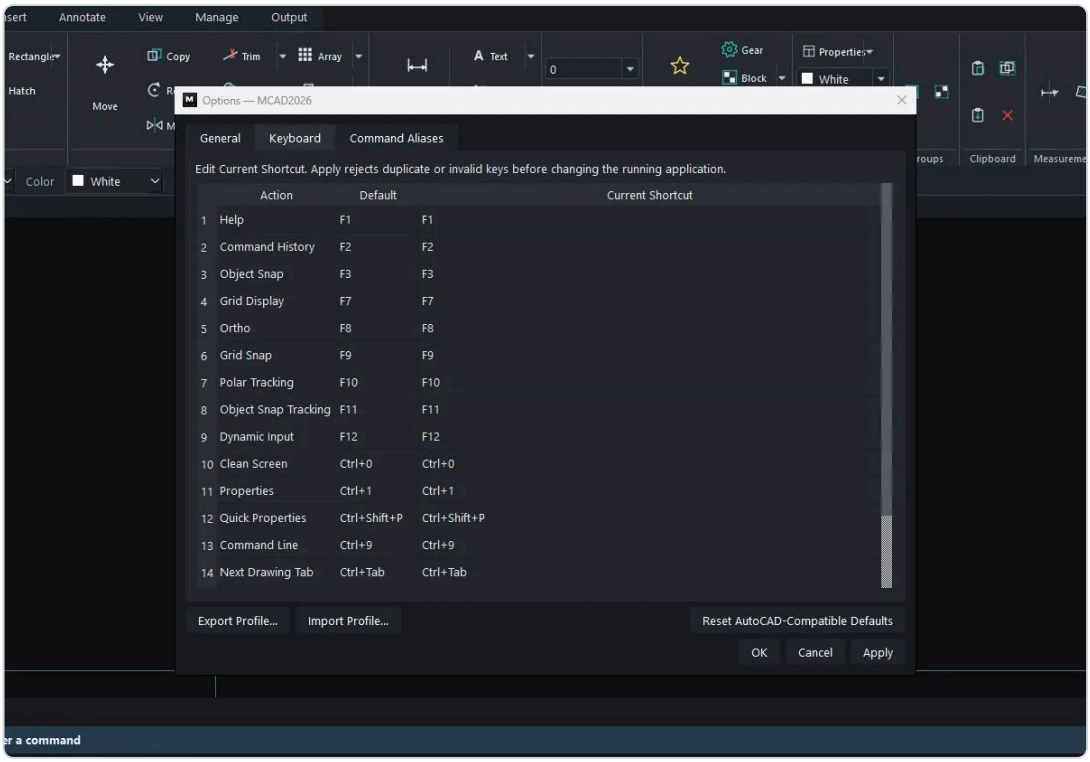
General tab



Options, General tab

Option	Meaning
Active profile	The AutoCAD-compatible 2D profile plus the user's changes
Settings storage	Stored separately for each Windows user
Diagnostics	Automatically sends reports when the program hangs or closes unexpectedly (no drawing data is sent)

Keyboard tab

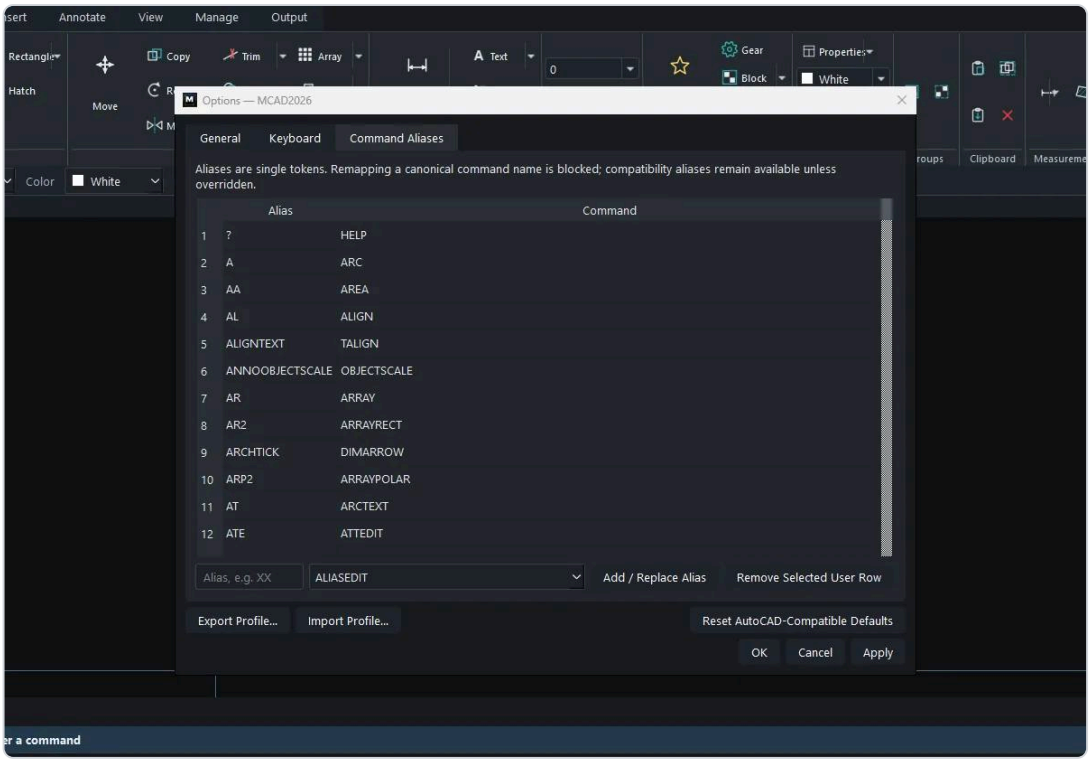


Options, Keyboard tab

Edit the **Current Shortcut** box of the command you want, then click **Apply**. The program rejects keys that are duplicated or invalid.

Action	Default
Help	F1
Command History	F2
Object Snap	F3
Grid Display	F7
Ortho	F8
Grid Snap	F9
Polar Tracking	F10
Object Snap Tracking	F11
Dynamic Input	F12
Clean Screen	Ctrl+0
Properties	Ctrl+1
Quick Properties	Ctrl+Shift+P
Command Line	Ctrl+9
Next / Previous Drawing Tab	Ctrl+Tab / Ctrl+Shift+Tab
Next / Previous Layout Tab	Ctrl+PgDown / Ctrl+PgUp

Command Aliases tab



Options, Command Aliases tab

1. Type the new alias in the `Alias, e.g. XX` box.
2. Choose the target command in the box next to it.
3. Click **Add / Replace Alias**, then **Apply** or **OK**.

Button	Meaning
Add / Replace Alias	Add or replace an alias
Remove Selected User Row	Delete an alias row that the user added
Export Profile... / Import Profile...	Save/import the Shortcut + Alias profile as a `.json` file
Reset AutoCAD-Compatible Defaults	Restore all defaults

> **Note:** An alias must be a single word, and you cannot use the real (canonical) command name as an alias.

Try it: A WL shortcut for partition walls

If you draw stud partitions all day, make your own `WL` alias and a 100 mm wall takes a few keystrokes. Try it on an office meeting room.

1. Type `ALIASEDIT` and press Enter (or **Manage › Customization › Keyboard/Aliases**), and go to the Command Aliases tab.
2. Type `WL` in the Alias box, choose the `DLINE` command, click **Add / Replace Alias**, then **OK**.
3. Type `WL 100 C`, Enter, then `0,0` Enter and `3000,0` Enter. You get a 100 thick, 3 m long wall.
4. Side wall: `WL 100 C`, then `3000,-50` Enter and `3000,2400` Enter.
5. Go back to Options and click **Export Profile...** to keep a `.json` file, so you can use the same aliases on another computer.



Meeting room partitions drawn with the WL alias

Challenge: Try to create an alias called `LINE`. The program refuses, because an alias cannot be a real command name. Make up your own short alias for `DIMSTYLE` instead.

See also

- Command Line and Aliases
- Workspaces

Workspaces, Palettes and Clean Screen

Save and switch Workspaces, show/hide palettes, Clean Screen, and Quick Properties.

What it does

A Workspace is a saved screen layout (which palettes are open, the Ribbon, and so on). You save it under a name so you can switch back to it later.

Where to find it

- Status bar: the **Drafting & Annotation** ▾ box (choose a Workspace or **Save Current As...**) and the **QP** button
- **View** menu: Properties Palette (`Ctrl+1`), Layers Palette, Symbols Palette, Quick Properties (`Ctrl+Shift+P`), Clean Screen (`Ctrl+0`)



Right side of the Status bar: Workspace and QP

Commands

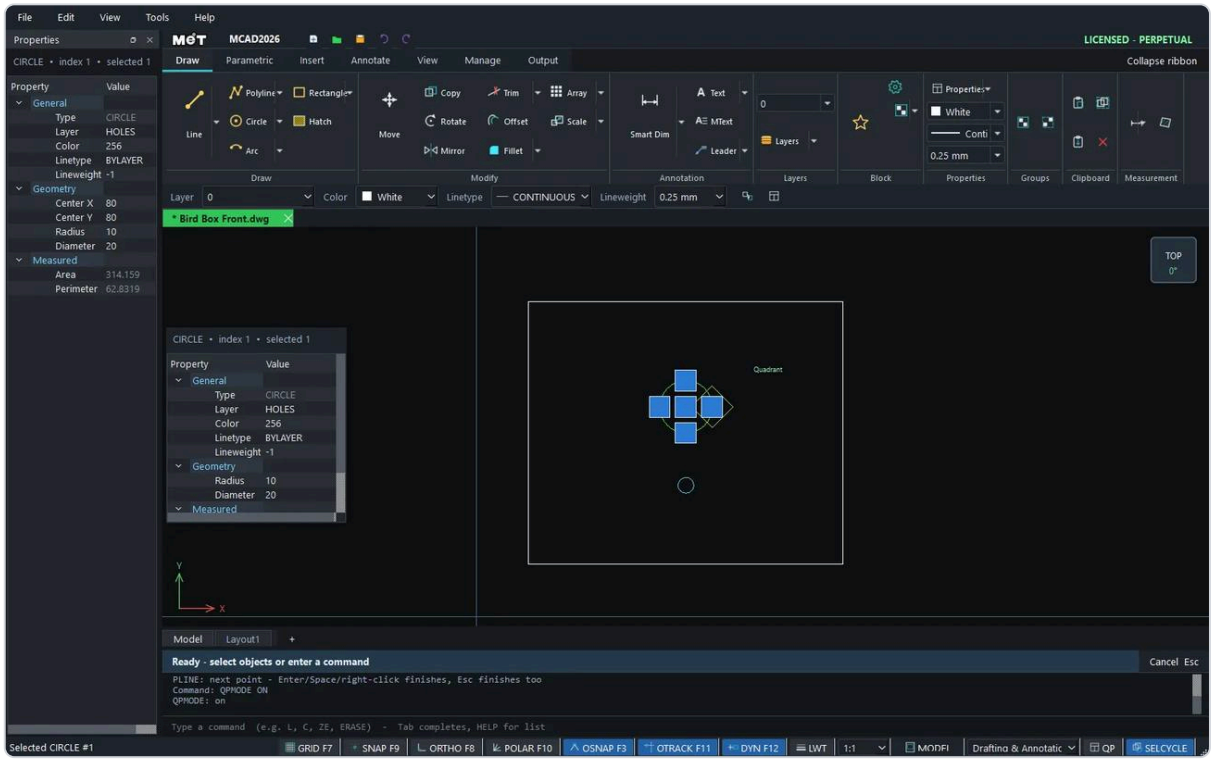
Command	Alias	Format	Meaning
WSCURRENT	`WORKSPACE`	`WSCURRENT [name]`	Shows the current Workspace, or switches to a named one
WSSAVE	–	`WSSAVE <name>`	Saves the current screen layout
WSDELETE	–	`WSDELETE <name>`	Deletes a Workspace
QPMODE	`QUICKPROPERTIES`, `QP`	`QPMODE <ON|OFF>`	Quick Properties box at the cursor when you select an object

Steps: save a Workspace

1. Show or hide the palettes you want from the **View** menu.
2. Type `WSSAVE Detailing` and press Enter.
3. Later, type `WSCURRENT Detailing` or choose it from the box on the Status bar.

Quick Properties

Turn on `QPMODE ON` and click an object. A small box with the main properties appears at the cursor, and you can edit the values right there.



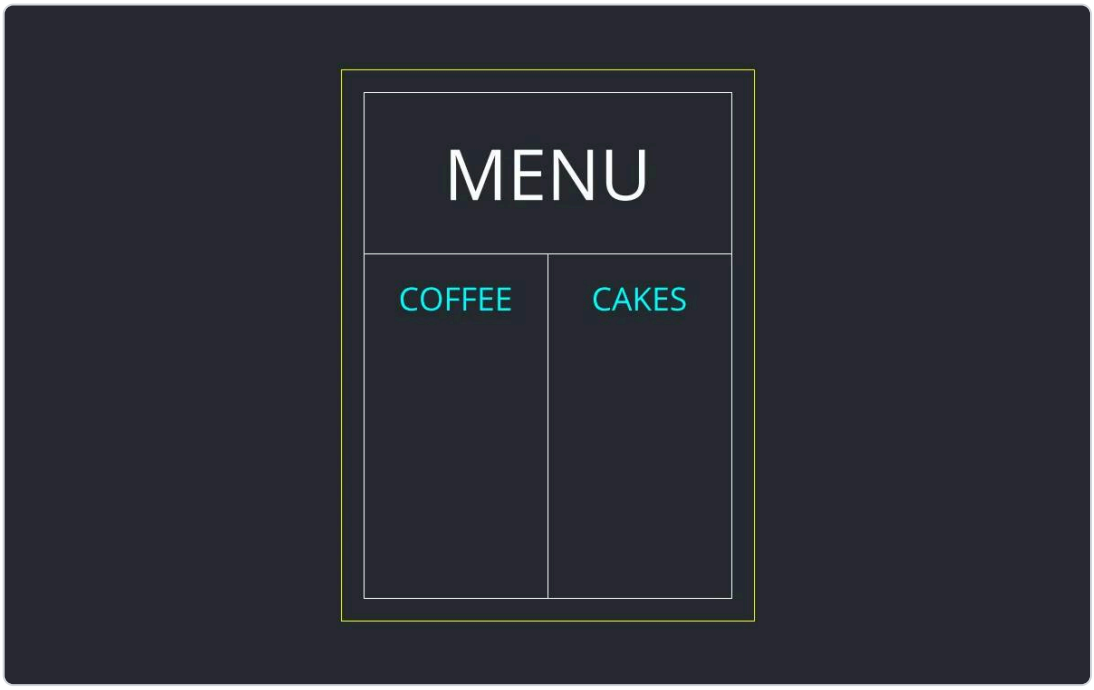
Quick Properties and the Properties palette for the selected circle

> **Note: Clean Screen** (`Ctrl+0`) hides the Ribbon and all palettes so the drawing area fills the screen. Press it again to restore them.

Try it: A "Signage" workspace for a café menu board

Sign work needs a big drawing area and no electrical symbol palette. Make a workspace just for signs and switch to it when you need it.

1. In the **View** menu, hide the Symbols Palette and the Layers Palette, and keep the Properties Palette open (`Ctrl+1`).
2. Type `WSSAVE Signage` and press Enter.
3. Draw the menu board: `REC` `0,0`, then `900` `Tab` `1200`; inner frame `REC` `50,50`, then `800` `Tab` `1100`.
4. Divide it: `L` `50,800` `@800,0` and `L` `450,50` `@0,750`, then add `TEXT` `MENU`, `COFFEE` and `CAKES`.
5. Press `Ctrl+0` (Clean Screen) to see the board full screen, and again to come back. Next time, just pick **Signage** from the Workspace box on the Status bar.



900 x 1200 menu board with COFFEE and CAKES panels

Challenge: Type `WSCURRENT` to list every workspace you have saved.

See also

- Interface Components
- PROPERTIES, MATCHPROP and CHPROP

Units (UNITS) and System Variables

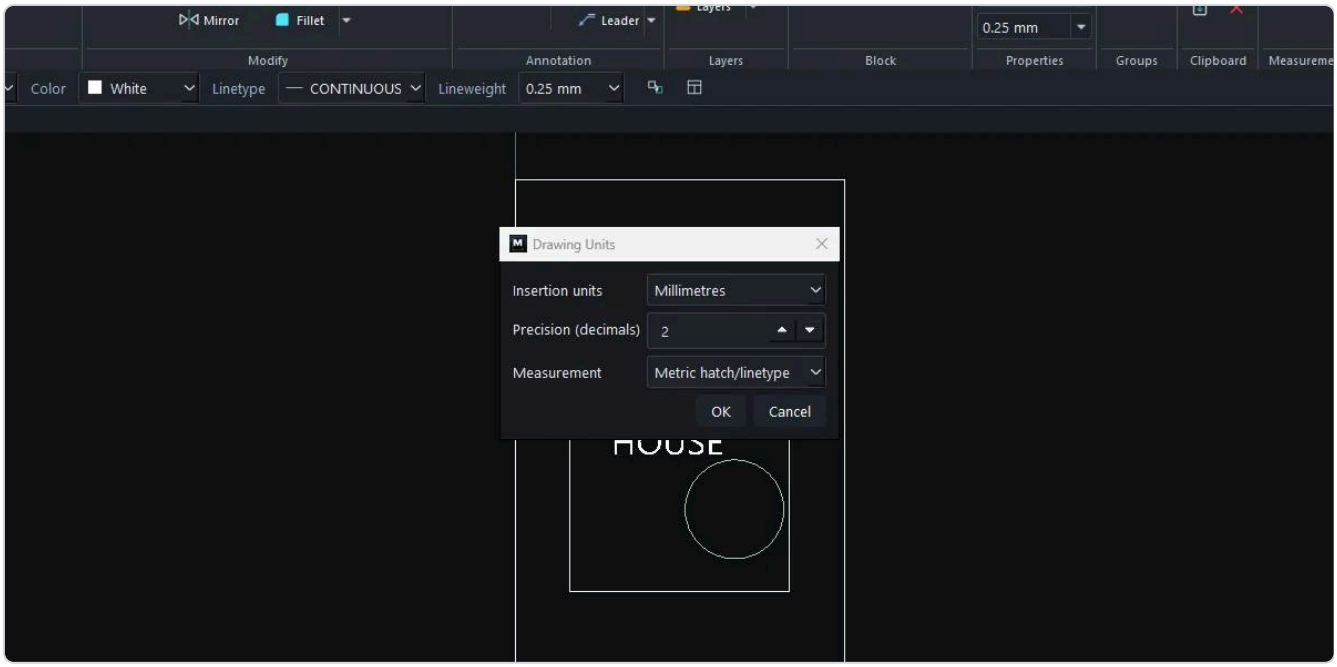
Set drawing units (UNITS), LIMITS, the BASE point, and System Variables with SETVAR.

What it does

Sets the units and settings of the drawing. They are saved with the file.

Where to find it

- Ribbon: **Manage › CAD Standards › Drawing Units**
- Command line: `UNITS` (`UN`, `DDUNITS`) reports the current values



Drawing Units window

Drawing Units window

Option	Meaning
Insertion units	Millimetres, Centimetres, Metres, Inches, Feet, Unitless (the `INSUNITS` value)
Precision (decimals)	Number of decimals shown, 0–8 (the `LUPREC` value)
Measurement	Metric or Imperial for Hatch/Linetype (the `MEASUREMENT` value)

Drawing setup commands

Command	Alias	Format	Meaning
UNITS	`UN`, `DDUNITS`	–	Reports the units, decimals, and measurement system
LIMITS	–	`LIMITS <x1> <y1> <x2> <y2>` or `LIMITS OFF`	The named paper boundary, used by ZOOM ALL and STATUS (it does not block clicks outside the boundary)
BASE	–	`BASE <x> <y>`	The base point when this drawing is inserted into another drawing
SETVAR	`SET`	`SETVAR <name> <value>` / `SETVAR ?` / `SETVAR ? DIM`	Sets or lists System variables (filter with a search word)
GETVAR	`GET`	`GETVAR <name>`	Reads a variable's value and description
LTSCALE	`LTS`	`LTSCALE <value>`	Linetype scale for the whole drawing
DIMSCALE	–	`DIMSCALE <value>`	Overall scale of Dimensions and the current Annotation scale
TEXTSIZE	–	`TEXTSIZE <value>`	Default height of new text
DDPTYPE	`PTYPE`	`DDPTYPE <size>`	POINT display size (pixels, 0–50)
QTEXT	–	`QTEXT <ON|OFF>`	Shows text as rectangles so panning is faster in drawings with a lot of text

Steps: change the drawing to metres with 3 decimals

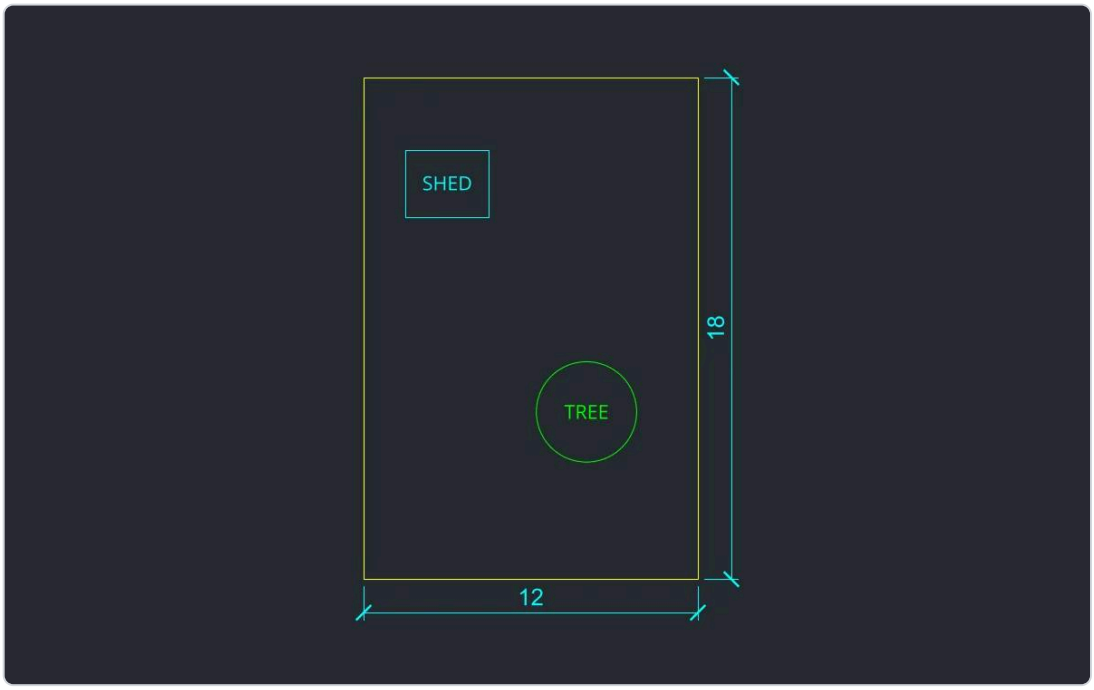
1. Choose **Manage › CAD Standards › Drawing Units**.
2. Set Insertion units = `Metres` and Precision = `3`.
3. Click **OK**. The Status bar shows `Units: metres, 3 decimals.`

> **Note:** Changing the units does not scale existing objects. To resize them, use SCALE.

Try it: A backyard plot in metres

Landscape plans are usually drawn in metres. Set the drawing to metres with 3 decimals, then lay out a 12 x 18 m plot, a shed and a tree canopy.

1. **Manage › CAD Standards › Drawing Units**: set Insertion units = `Metres` and Precision = `3`, then **OK**.
2. Plot: `REC` `0,0`, then `12` `Tab` `18` Enter.
3. A 3 x 2.4 m shed: `REC` `1.5,13`, then `3` `Tab` `2.4`.
4. Tree canopy: `C` `8,6` with radius `1.8`.
5. Type `LIMITS 0 0 12 18`, then `ZOOM A`. Type `UNITS` to confirm the drawing is in metres with 3 decimals.



12 x 18 m plot with a shed and a tree

Challenge: Set `LTSCALE 0.02` and change the fence line to DASHED to see why a drawing in metres needs a small LTSCALE.

See also

- Status Bar
- DIMSTYLE and STYLE

Measuring and inquiry (DIST, AREA, LIST)

Measure distance, angle, area, radius and point coordinates, and summarise a drawing with DIST, AREA, ID, LIST and STATUS.

What it does

These commands report sizes and information without changing the drawing. The results appear in the prompt bar and the command line history.

Where to find it

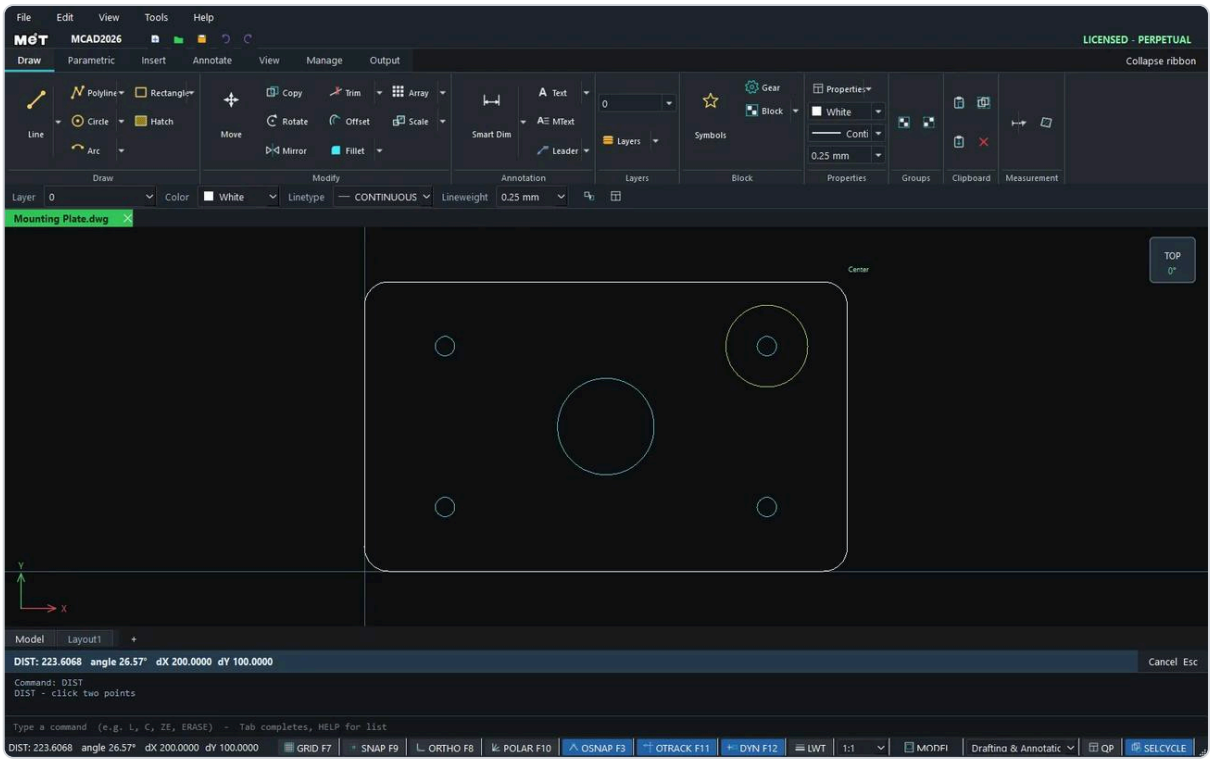
- Ribbon: **Draw › Measurement › Measure** ▾ (Distance, Angle, Radius, Area, Coordinates) and **Area**
- Ribbon: **Manage › Drawing Cleanup › Drawing Status**

Commands

Command	Alias	Meaning
DIST	`DI`	Click 2 points to see the distance, angle, dX and dY.
AREA	`AA`	The area and perimeter of the selected closed objects (you can combine several).
ID	–	Click a point to see its X and Y coordinates.
MEASUREGEOM	`MEA`	`MEASUREGEOM <Distance|Angle|Area|Radius>` combines the measure modes. Angle = click the vertex, then the two arms. Radius = the radius, diameter and circumference of the selected circle.
LIST	`LI`	The number and type of the selected objects.
DBLIST	–	Counts every object in the drawing by type, and the number of layers.
STATUS	–	A summary of the drawing: the number of objects of each type, the number of layers, dimensions and the current layer.
TIME	–	The time the drawing has been open, the time used in this session and the number of edits.

Steps: measure the distance between holes

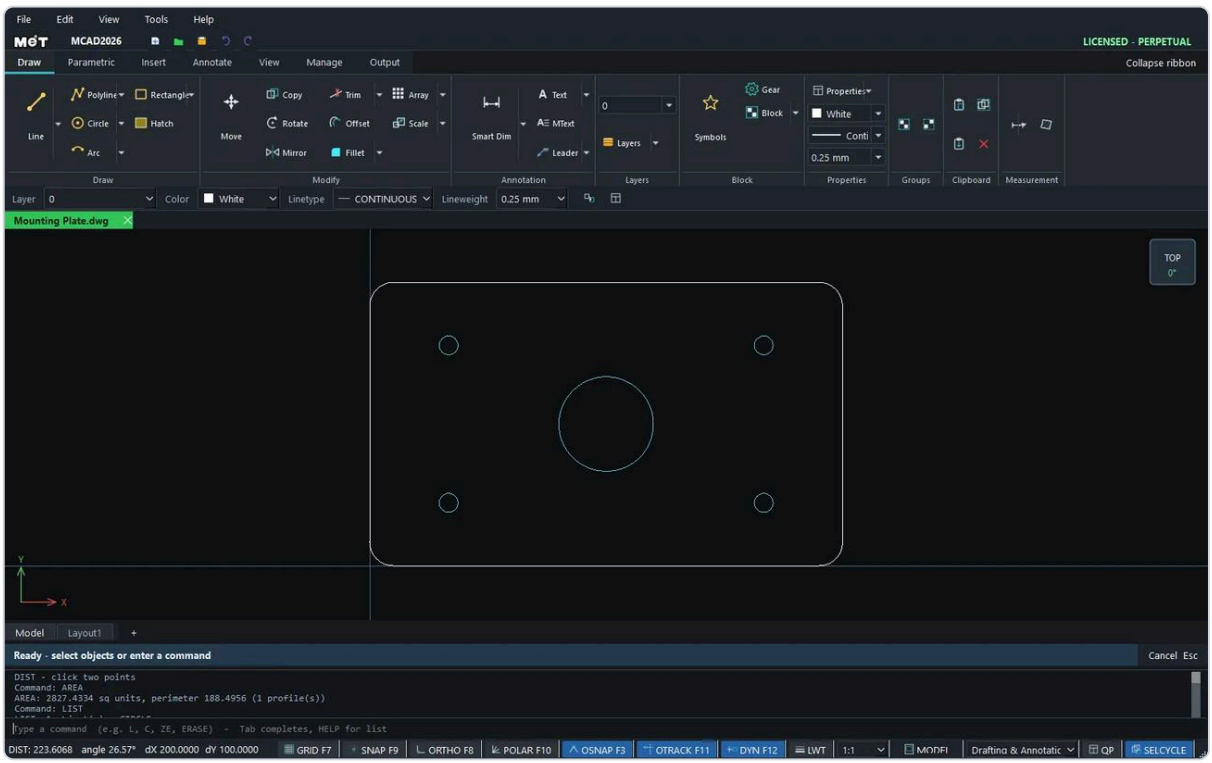
1. Type `DI` and press Enter.
2. Click the centre of the first hole (Object Snap picks the Center).
3. Click the centre of the second hole.
4. Read the result in the prompt bar, for example `DIST: 223.6068 angle 26.57° dX 200.0000 dY 100.0000`.



DIST result between two holes

Steps: find an area

1. Select a closed shape (rectangle, circle, closed polyline or region).
2. Type `AREA` and press Enter. The result reads, for example, `AREA: 1963.4954 sq units, perimeter 157.0796 (1 profile(s))`.



AREA, LIST and STATUS results on the command line

> **Note:** The Properties palette also shows the Area and Perimeter of the selected closed object, in the **Measured** section.

Try it: How much flooring for an L-shaped living room?

Before you order laminate, you need the floor area and the skirting length. Measure an L-shaped living room with AREA, DIST and CAL.

1. Draw the room: choose **Polyline - Closed Loop**, type `0,0` `5000,0` `5000,3000` `2500,3000` `2500,4500` `0,4500`, then press Enter.
2. Click the outline and type `AREA`. It reports `AREA: 18750000.0000 sq units, perimeter 19000.0000 (1 profile(s))`, which is 18.75 m² with a 19 m perimeter.
3. Add 10% for cutting waste: type `CAL 18.75*1.1`. The answer is `20.625` m² to order.
4. Will a 5.8 m sofa fit diagonally? Type `DI`, then `0,0` Enter and `5000,3000` Enter. The Prompt bar shows `DIST: 5830.9519 angle 30.96°`.



L-shaped living room, 18.75 m², with the 5830.95 diagonal

Challenge: The real skirting length is the perimeter minus the door. Try `CAL 19000-900` for a 900 door.

See also

- Selecting objects
- PROPERTIES

Chapter 2 quiz: Interface & settings

Choose the one best answer. Answers are in Appendix A

1. **What happens when you press Enter or Space while no command is active?**

   A. The drawing is saved
   B. The last selection is cleared
   C. The last command is repeated
   D. The Command line is hidden

2. **Which character do you type before ZOOM or PAN to run it transparently in the middle of another command?**

   A. @
   B. #
   C. !
   D. ' (apostrophe)

3. **Your clicks do not land exactly on line endpoints. Which key turns Object Snap (OSNAP) on or off?**

   A. F3
   B. F7
   C. F8
   D. F9

4. **What is the default Polar tracking angle (POLAR, F10)?**

   A. 15°
   B. 30°
   C. 45°
   D. 90°

5. **Which alias zooms so that every object in the drawing fits on the screen (Zoom Extents)?**

   A. ZW
   B. ZE
   C. VL
   D. RE

6. **You change the drawing units from millimetres to metres in Drawing Units. What happens to the objects already drawn?**

   A. They are scaled by 1/1000 automatically
   B. They are moved to a new layer
   C. They are converted to blocks
   D. They are not scaled; use SCALE if you need to resize them

CHAPTER 3

Draw

1. LINE

   Draw connected straight line segments by clicking, or by typing length and angle.

2. XLINE and RAY - Construction Lines

   Draw infinite construction lines (XLINE) and one-direction lines (RAY).

3. DLINE and MLINE - Parallel lines

   Draw a pair of parallel lines (DLINE) and several parallel lines at once (MLINE) for walls, trays or pipes.

4. PLINE - Polyline

   Draw a multi-segment polyline as a single object, open or closed.

5. CIRCLE and DONUT

   Draw a circle from its centre and radius, and a pair of concentric circles (DONUT).

6. ARC (3-point arc)

   Draw an arc through 3 points: the start point, a point on the arc and the end point.

7. RECTANG - Rectangle

   Draw a rectangle from corner to corner or from the center, with typed width and height.

8. POLYGON - Regular Polygon

   Draw a regular hexagon from a center point and a vertex.

9. ELLIPSE

   Draw an axis-aligned ellipse from its centre and a corner of its bounding box.

10. SLOT (Straight Slot)

    Draw a straight slot with rounded ends from two end centers and a width.

11. SPLINE, SPLINEDIT and SKETCH

    Draw free-form curves through points (SPLINE), edit their points (SPLINEDIT), and draw freehand (SKETCH).

12. POINT

    Place Point objects and set their display size with DDPTYPE.

13. HATCH and GRADIENT

    Fill closed areas with a pattern (HATCH), edit the hatch, apply a gradient fill (GRADIENT) and send hatches to the back.

14. SOLID, WIPEOUT and REVCLOUD

    Draw 2D filled areas (SOLID), masking areas (WIPEOUT), and revision clouds (REVCLOUD).

15. BOUNDARY, REGION and Booleans (UNION/SUBTRACT/INTERSECT)

Create a closed boundary (BOUNDARY) and a region, then combine, subtract or intersect closed shapes (UNION/SUBTRACT/INTERSECT).

16. Gear - Spur gear generator

    Create a closed involute spur gear profile, ready to use for DXF or G-code.

LINE

Draw connected straight line segments by clicking, or by typing length and angle.

What it does

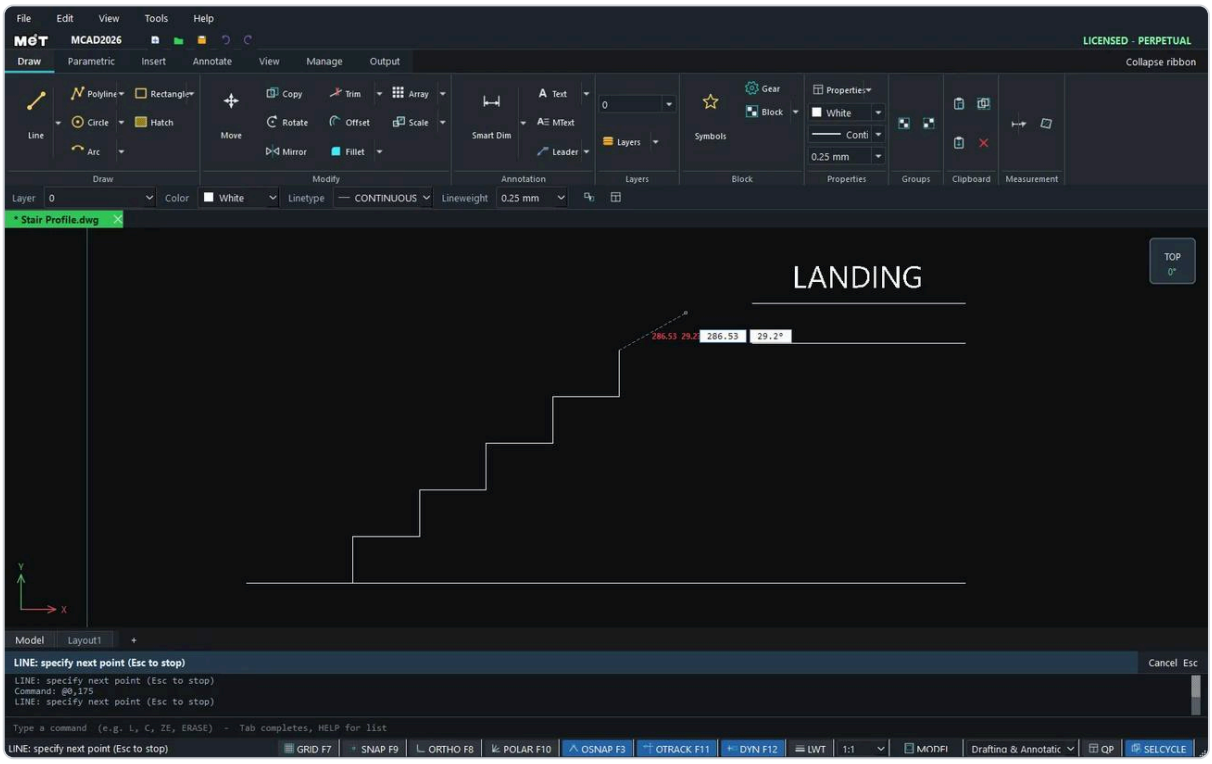
LINE draws straight segments one after another. Each segment is a separate Line object.

Where to find it

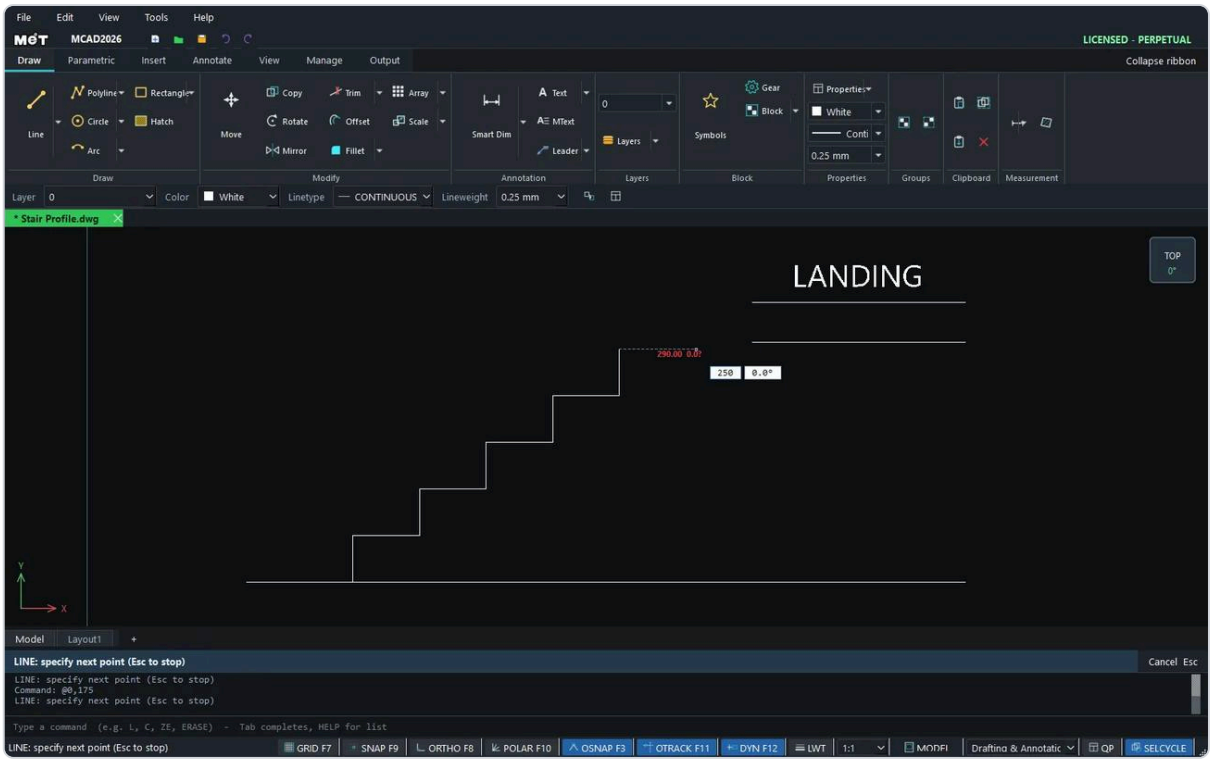
- Ribbon: **Draw › Draw › Line** (submenu: Line - 2-Point, Line - From Midpoint, Dashed Line)
- Command line: `LINE` (Alias `L`)

Steps

1. Type `L` and press Enter. The Prompt bar shows `LINE: specify first point`.
2. Click the start point (or type coordinates).
3. `LINE: specify next point (Esc to stop)` Click the next point, or type the length in the Smart Input box, press `Tab`, type the angle, and press Enter.
4. Repeat as many times as you need. Press `Esc` to finish.



Drawing continuous LINE segments, specifying the next point



Typing a tread length of 250 in the Smart Input box

Line submenu options

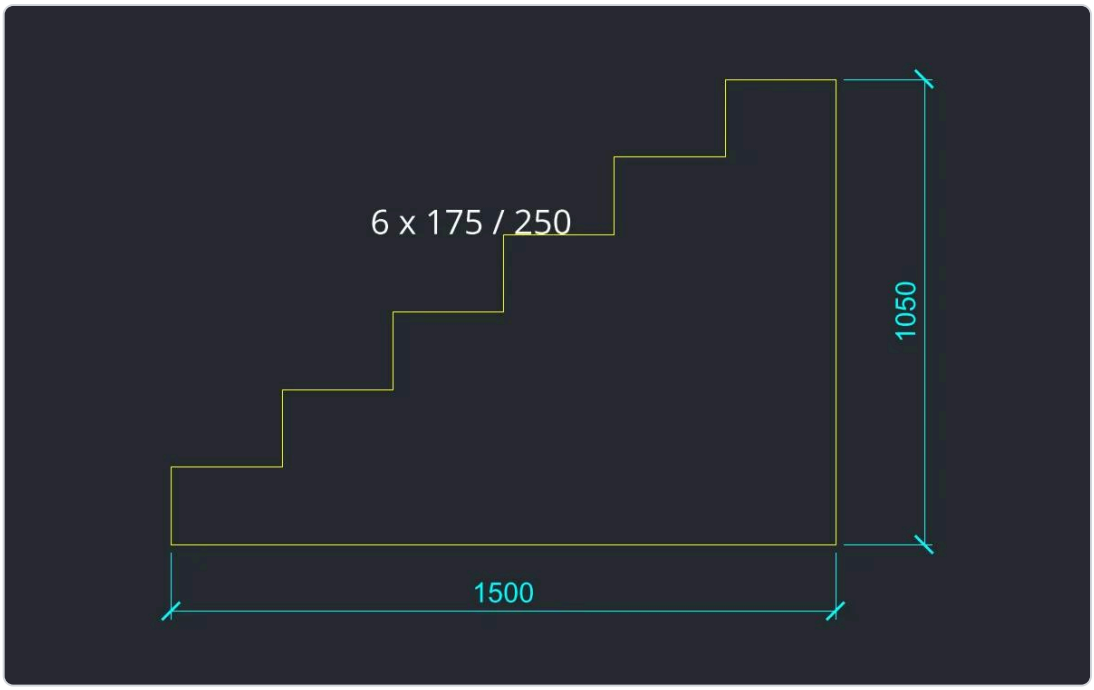
Option	Meaning
Line - 2-Point (home → end)	Draw from a start point to an end point (default)
Line - From Midpoint (center → end)	Click the midpoint first, then click one end. The line extends equally on both sides
Dashed Line (sets linetype)	Draw a Line with the Linetype set to DASHED
Dual Line / Construction Line / Ray / Multiline	See DLINE and MLINE and XLINE and RAY

> **Note:** Press `F8` (Ortho) to force horizontal/vertical, or `F10` (Polar) to snap to angles every 45°.

Try it: Side profile of a 6-step staircase

Risers of 175 and treads of 250 mm, which meets the comfortable-stair rule (2 x riser + tread = 600). One LINE command does it all with relative `@` coordinates.

1. Type `L`, press Enter, type `0,0` and press Enter.
2. Type `@0,175` Enter (riser), then `@250,0` Enter (tread).
3. Repeat step 2 five more times for 6 steps in total. The line now ends at `1500,1050`.
4. Type `@0,-1050` Enter, then `0,0` Enter to close the shape, and press `Esc`.



6-step staircase profile, 1050 high and 1500 long

Challenge: Try a gentler stair with `@0,150` risers and `@300,0` treads and compare the total length.

See also

- PLINE - Polyline
- Command Line and Smart Input
- Status Bar

XLINE and RAY - Construction Lines

Draw infinite construction lines (XLINE) and one-direction lines (RAY).

What it does

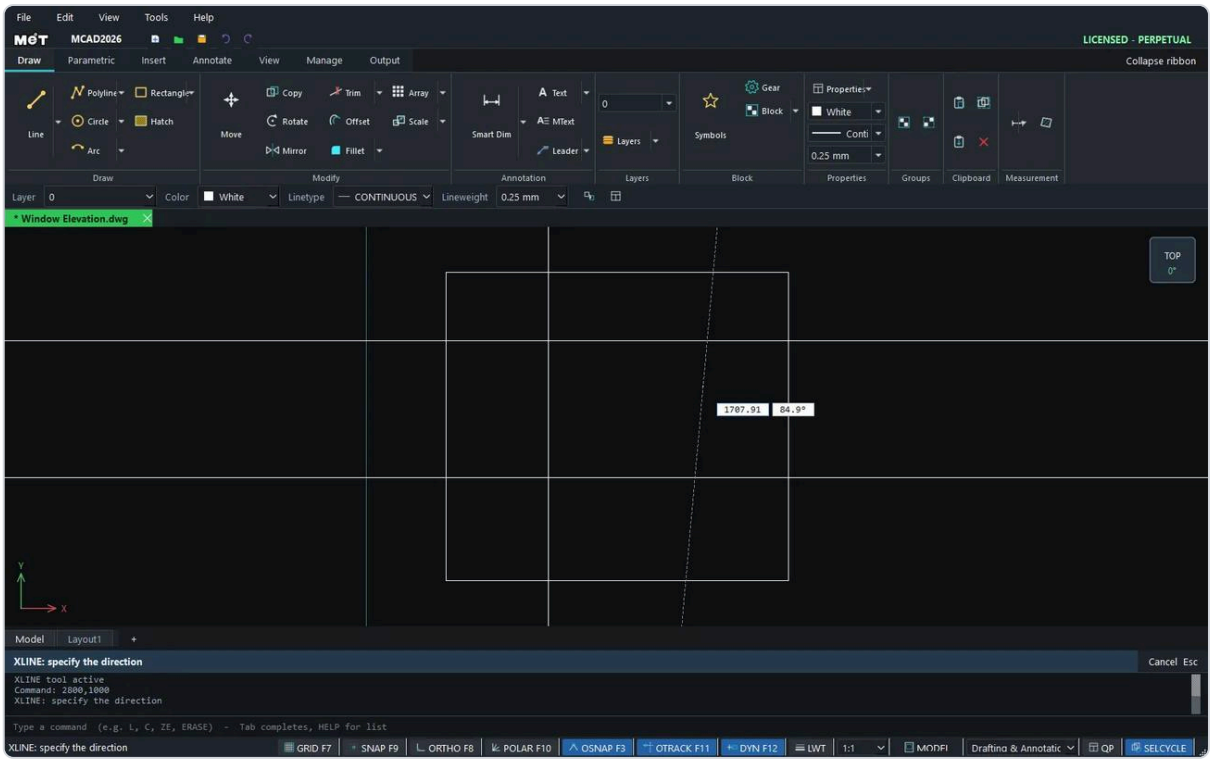
Construction lines are reference guides for drawing. XLINE extends in both directions. RAY extends from a start point in one direction. Neither is counted in Zoom Extents.

Where to find it

- Ribbon: **Draw › Draw › Line ▾ › Construction Line - Infinite (XLINE)** / **Ray - One Direction (RAY)**
- Command line: `XLINE` (Alias `XL`)
- Command line: `RAY`

XLINE steps

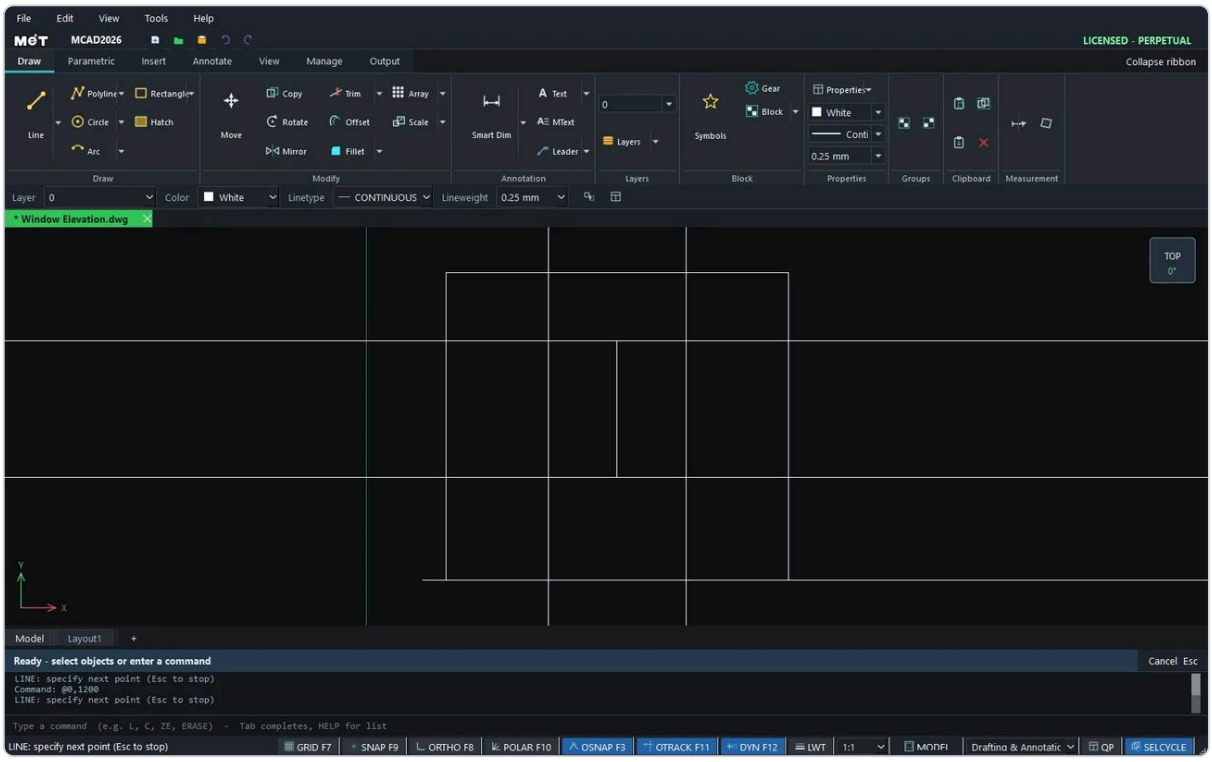
1. Type `XL` and press Enter.
2. `XLINE: specify a point on the line` Click a point the line passes through.
3. `XLINE: specify the direction` Click a second point to set the direction.



XLINE for the right window jamb, specifying the direction

RAY steps

1. Type `RAY` and press Enter.
2. `RAY: click the start point` Click the start point.
3. `RAY: click a point it passes through` Click a point the line passes through.



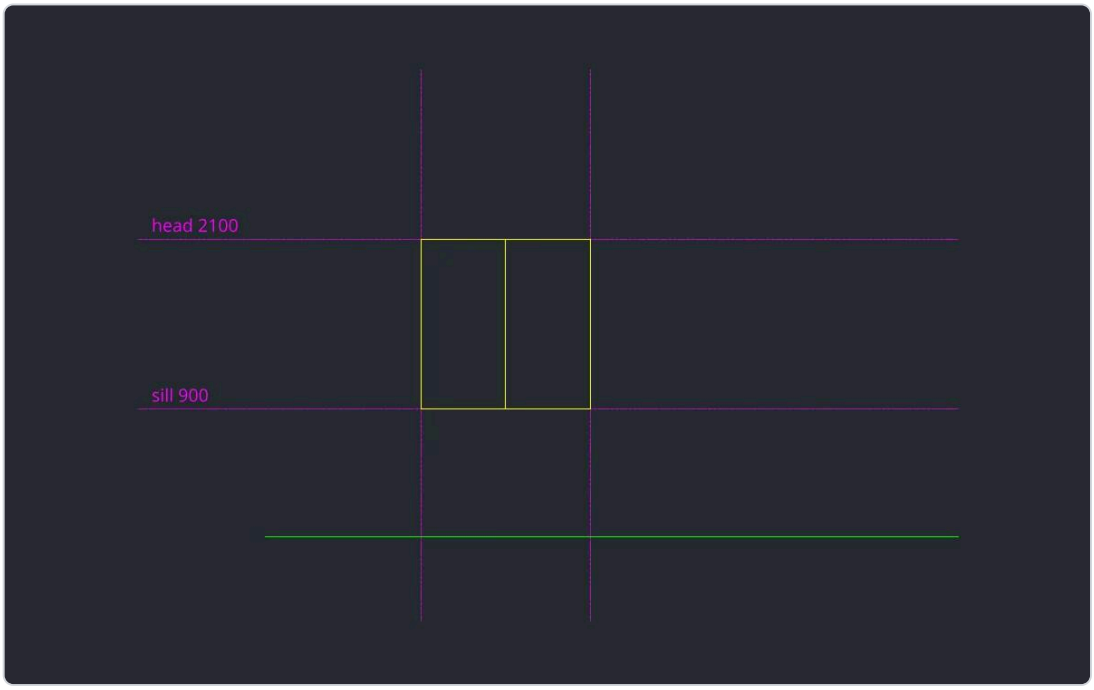
XLINE and RAY guides around a window elevation

> **Note:** Put construction lines on a separate Layer and turn that Layer off before plotting. See Layer Manager.

Try it: Set out a window elevation with construction lines

Architects often draw level lines before the openings. Set a 900 sill level and a 2100 head level, then drop a 1200 x 1200 window into the guide frame.

1. Sill level: type `XL`, Enter, `0,900`, Enter, then `@1000,0` and Enter.
2. Head level: `XL` `0,2100`, then `@1000,0`.
3. Left and right jambs: `XL` `600,0` `@0,1000` and `XL` `1800,0` `@0,1000`.
4. Floor level: type `RAY`, Enter, then `-500,0` Enter and `@1000,0` Enter.
5. Window: `REC` `600,900`, then `1200` `Tab` `1200` Enter, and a mullion with `L` `1200,900` `@0,1200`.



Window elevation inside XLINE and RAY guides

Challenge: Move all the guides to a layer called `CONSTRUCTION` and turn it off before you plot. See Layer Manager.

See also

- LINE
- Zoom, Pan and view controls

DLINE and MLINE - Parallel lines

Draw a pair of parallel lines (DLINE) and several parallel lines at once (MLINE) for walls, trays or pipes.

What it does

- **DLINE** (Dual line) – draws two parallel lines along the path of 2 clicked points.
- **MLINE** (Multiline) – draws 2 or more parallel lines along a multi-segment path, like a polyline.

Where to find it

- Ribbon: **Draw › Draw › Line ▾ › Dual Line - Two Parallels (DLINE)** / **Multiline - Parallel Runs (MLINE)**
- Command line: `DLINE` (alias `DLN`)
- Command line: `MLINE` (alias `ML`)

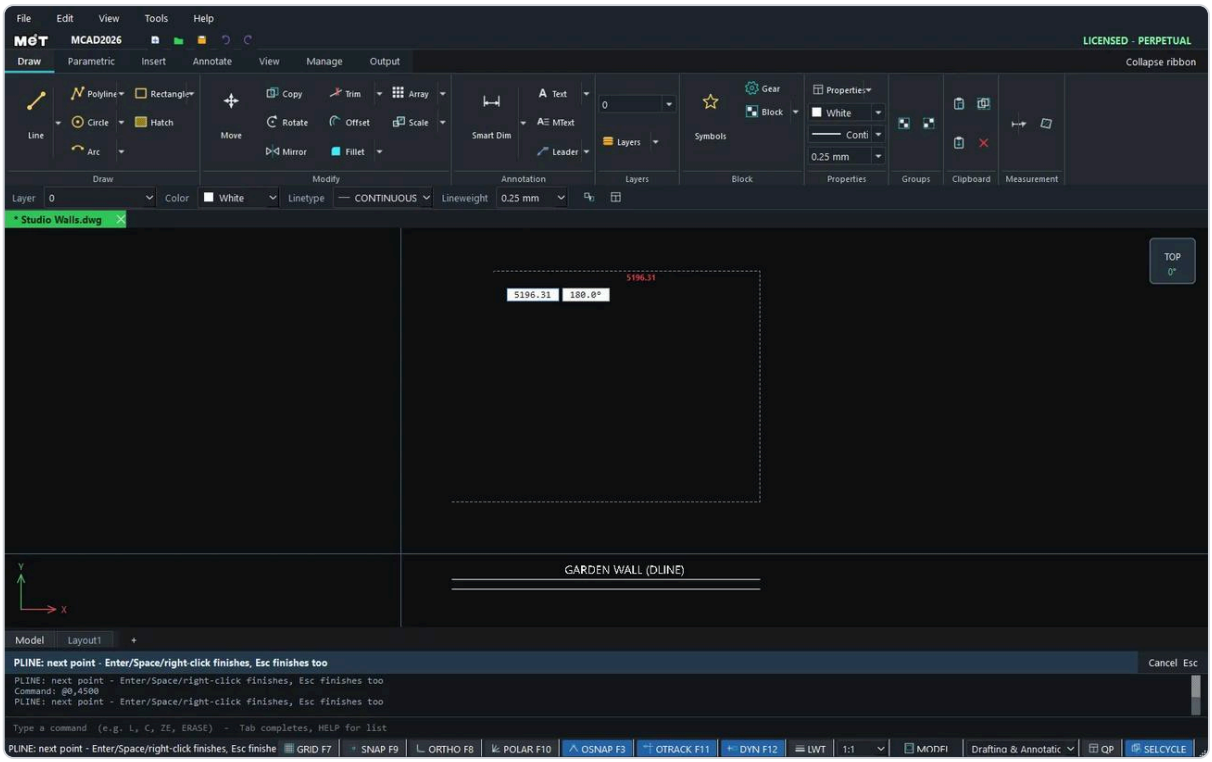
Steps for DLINE

1. Type `DLINE` and press Enter.
2. At `Dual line width <10>:`, type the spacing and press Enter.
3. At `Dual line justification [C=centre / E=edge]`, type `C` (the lines straddle the clicked path) or `E` (the clicked path is on the edge of one line).
4. Click the first and second points. You can type a length and angle as you do with LINE.

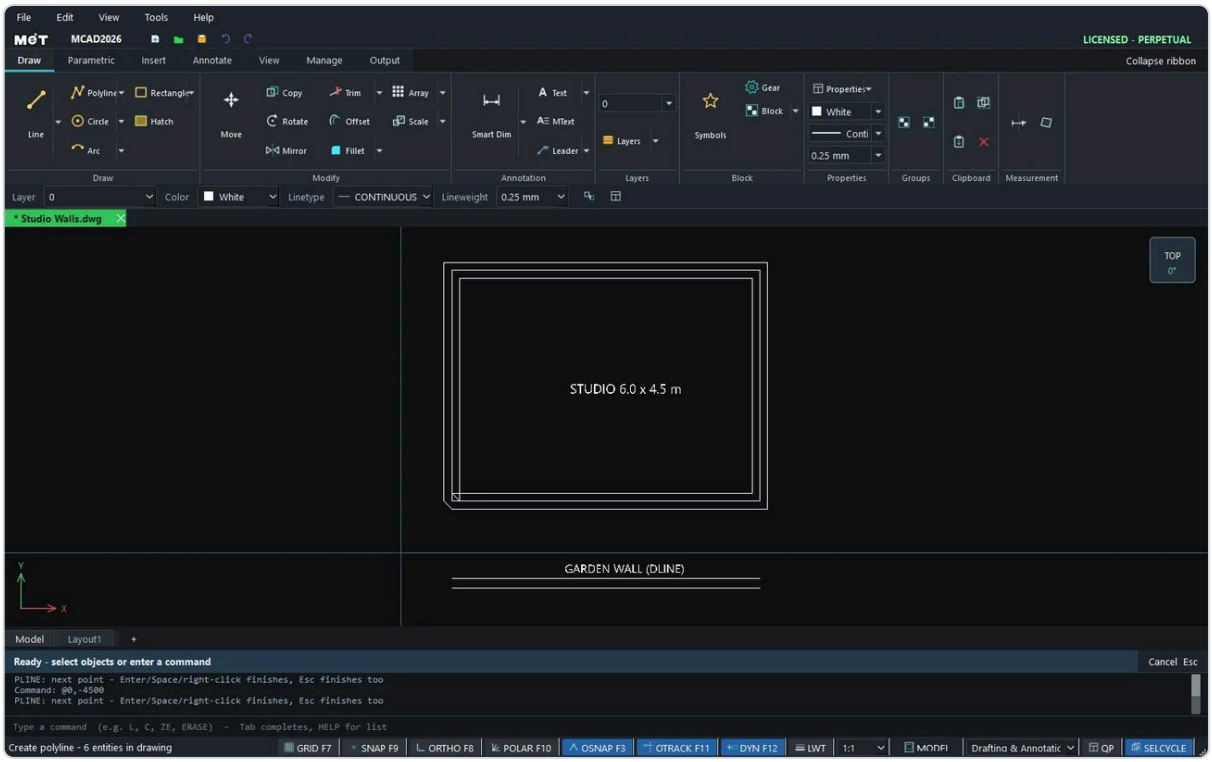
Shortcut: `DLINE 10 C` gives the width and justification on one line.

Steps for MLINE

1. Type `MLINE 3 8` and press Enter (3 lines, 8 apart).
2. Click points along the path.
3. Press Enter to finish.



MLINE drawing the wall path (the 3 lines appear when you press Enter to finish), and a finished DLINE below



Results of DLINE and MLINE

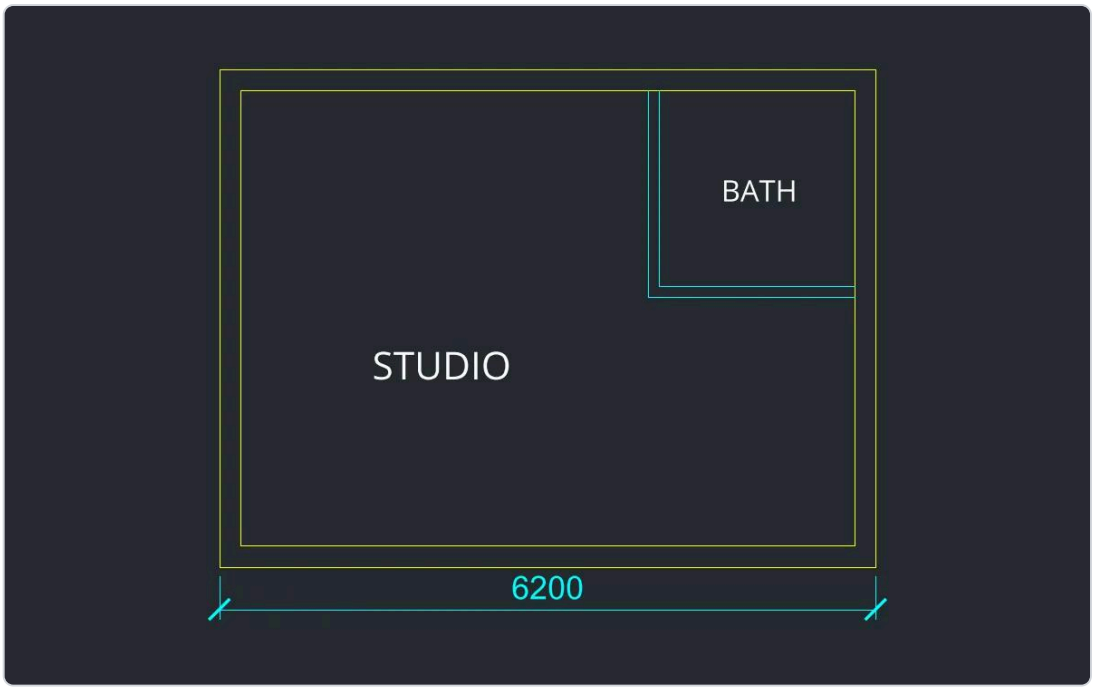
Options

Command	Syntax	Meaning
DLINE	`DLINE [width] [C|E]`	C = Centre, E = Edge.
MLINE	`MLINE <number of lines> <spacing> [Zero|Top|Bottom]`	Zero = the path is in the middle, Top = the path is on the first line, Bottom = the path is on the last line. You need at least 2 lines.

Try it: Walls of a tiny studio flat

A 6 x 4.5 m studio flat with 200 mm walls and a 2 x 2 m bathroom. The whole outer wall takes one command.

1. Type `MLINE 2 200` and press Enter (2 lines, 200 apart, path in the middle).
2. Type the points one by one, pressing Enter after each: `0,0` `6000,0` `6000,4500` `0,4500` `0,0`. Press Enter once more to finish.
3. Thin 100 mm bathroom wall: type `DLINE 100 C`, Enter, then `4000,4400` Enter and `4000,2500` Enter.
4. Type `DLINE 100 C` again, then `3950,2500` Enter and `5900,2500` Enter.
5. Tidy the wall junctions with TRIM.



Walls of a 6 x 4.5 m studio flat with a bathroom

Challenge: Try `MLINE 3 100` to draw a wall with a centre line (3 lines), then change the middle line to the CENTER linetype.

See also

- LINE
- OFFSET

PLINE - Polyline

Draw a multi-segment polyline as a single object, open or closed.

What it does

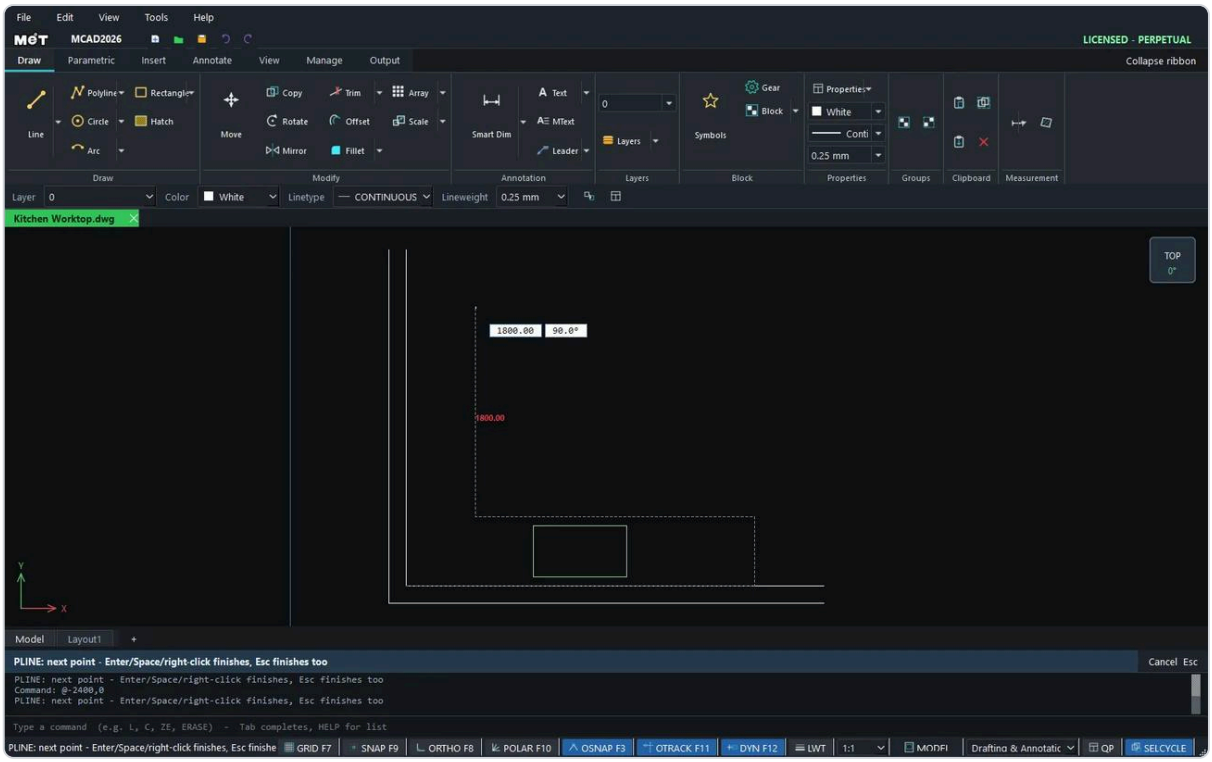
PLINE draws connected segments as a single object. Use it to make closed shapes for Hatch, Offset, Area, or G-Code.

Where to find it

- Ribbon: **Draw › Draw › Polyline** (submenu: Polyline - Open Loop, Polyline - Closed Loop, Spline, Freehand Sketch, Boundary)
- Command line: `PLINE` (Alias `PL`, `POLYLINE`)

Steps

1. Type `PL` and press Enter. `PLINE: specify the first point`
2. Click the first point.
3. `PLINE: next point - Enter/Space/right-click finishes, Esc finishes too` Keep clicking points (you can type a length/angle).
4. Press Enter, Space, right-click, or press Esc to finish.



Drawing a polyline, specifying the next point

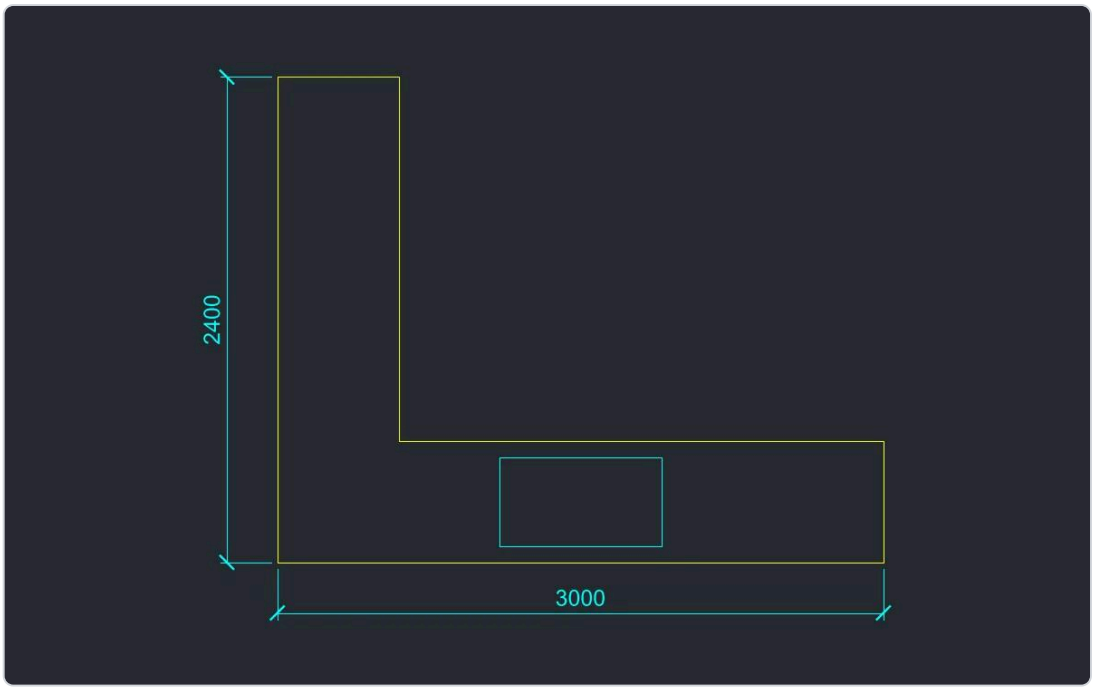
Option	Meaning
Polyline - Open Loop	Open polyline (default)
Polyline - Closed Loop	Closes the shape by joining the last point back to the first automatically

> **Note:** To join connected Line objects into a polyline, use `PEDIT`. See BREAK, JOIN and PEDIT.

Try it: L-shaped kitchen worktop

An L-shaped stone worktop, 600 mm deep, drawn as one closed object so you can measure it for the stone order straight away.

1. Choose **Draw › Draw › Polyline ▾ › Polyline - Closed Loop**.
2. Type the points, pressing Enter after each: `0,0` `@3000,0` `@0,600` `@-2400,0` `@0,1800` `@-600,0`.
3. Press Enter. The shape closes back to the first point by itself.
4. Sink cut-out: `REC` `1100,80`, then `800` `Tab` `440` Enter.
5. Click the L-shaped top and type `AREA`. It reports `2880000` sq units, which is 2.88 m^2 of stone.



3000 x 2400 L-shaped worktop with a sink cut-out

Challenge: OFFSET the outline by `20` outwards for a 20 mm overhanging nosing.

See also

- SPLINE, SPLINEDIT and SKETCH
- BOUNDARY and REGION

CIRCLE and DONUT

Draw a circle from its centre and radius, and a pair of concentric circles (DONUT).

What it does

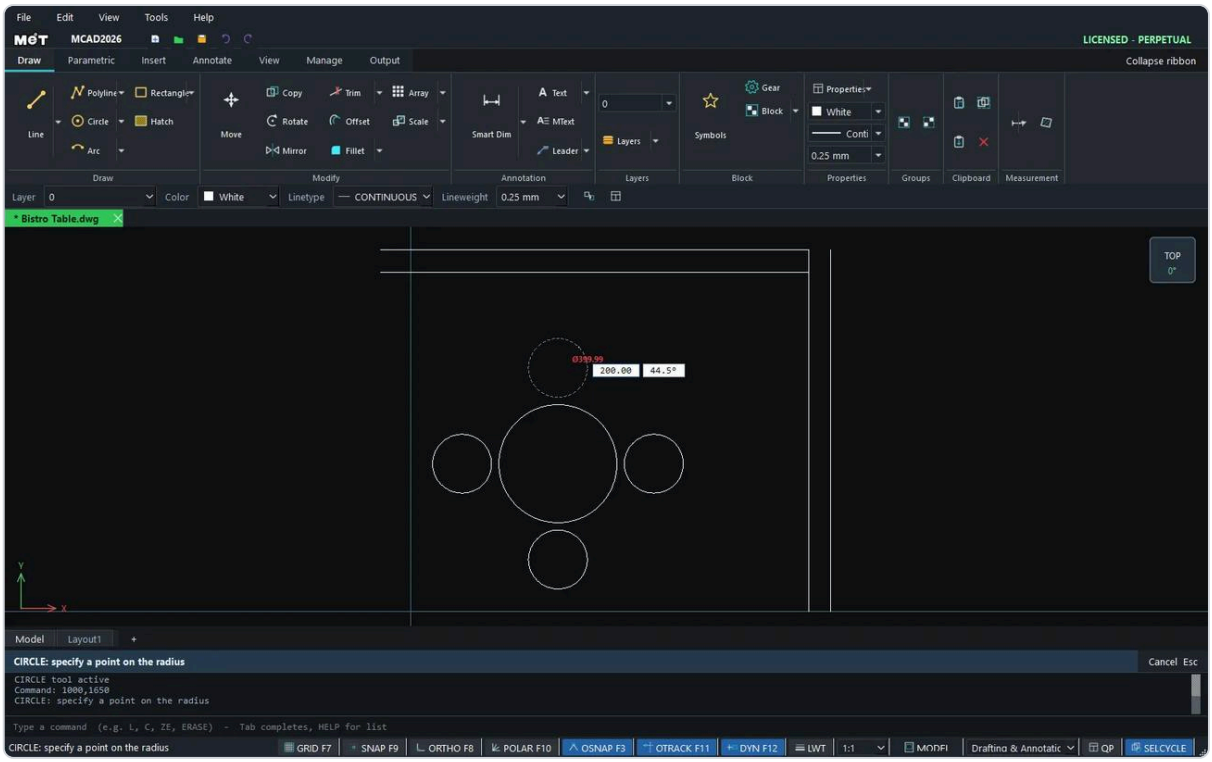
- **CIRCLE** – draws a circle from a centre point and a radius.
- **DONUT** – draws two concentric circles (inner radius / outer radius) with a single click.

Where to find it

- Ribbon: **Draw › Draw › Circle** (submenu: Center, Radius / Donut (DONUT) / Point (POINT))
- Command line: `CIRCLE` (alias `C`)
- Command line: `DONUT` (alias `DO`)

Steps for CIRCLE

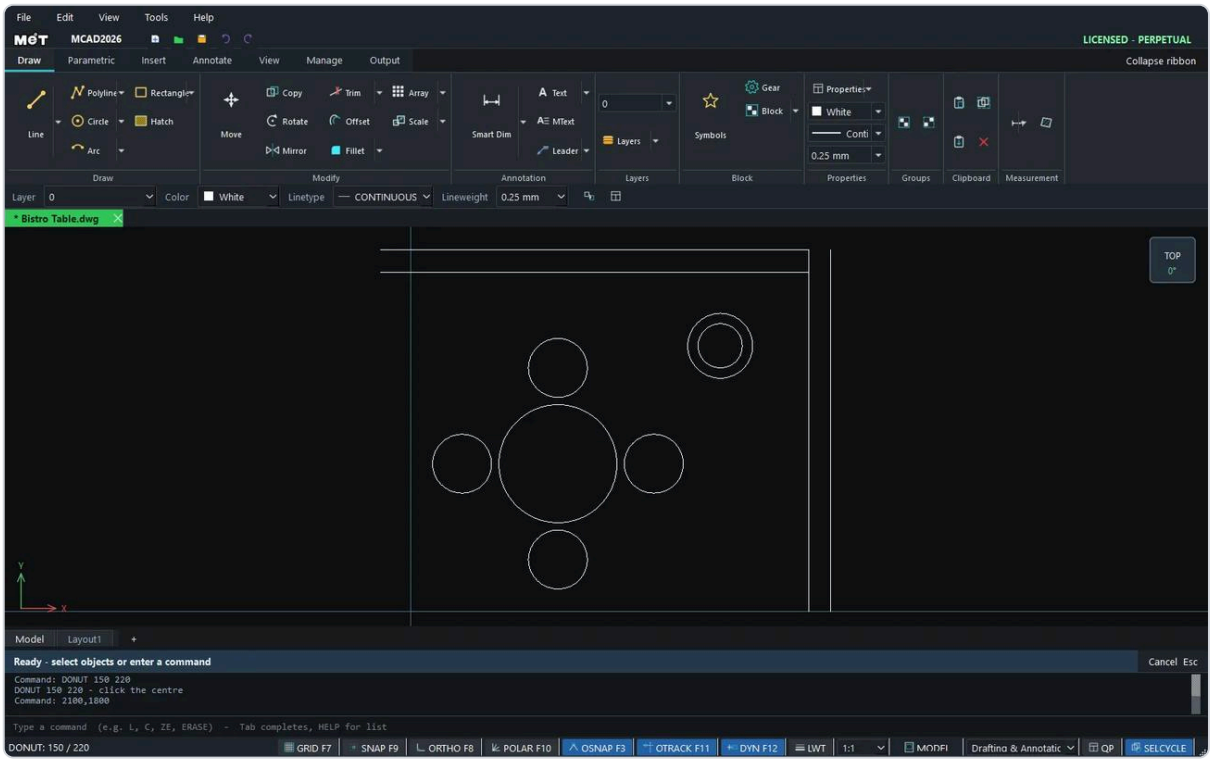
1. Type `C` and press Enter. The prompt reads `CIRCLE: specify the center point`.
2. Click the centre point.
3. At `CIRCLE: specify a point on the radius`, click a point on the circumference, or type the radius in the Smart Input box and press Enter.



CIRCLE while setting the radius

Steps for DONUT

1. Type `DONUT 20 40` and press Enter (inner radius 20, outer radius 40).
2. At `DONUT: click the centre`, click the centre point.



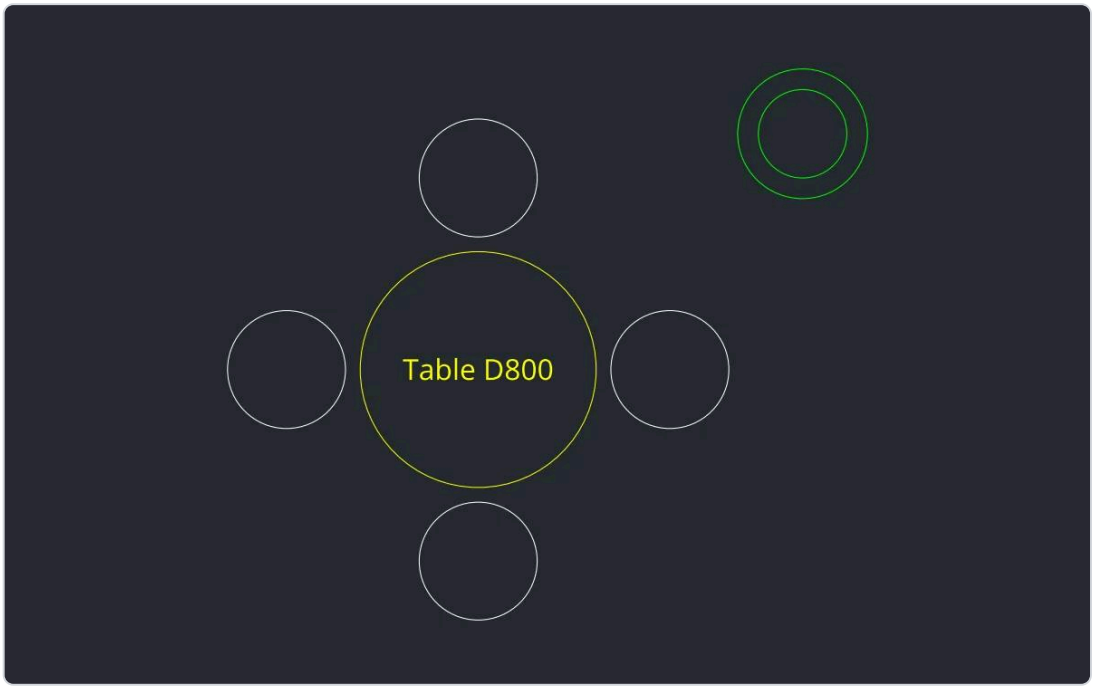
A circle and a DONUT

Command	Syntax	Meaning
DONUT	`DONUT [inner] [outer]`	The defaults are inner 10 and outer = 2 × inner. You need 0 < inner < outer.

Try it: Round bistro table with four stools

A café table layout you can drop straight into a floor plan, plus a plant pot in the corner. Only CIRCLE and DONUT are used.

1. Type `C`, Enter, `0,0`, Enter, then radius `400` and Enter. That is a Ø800 table top.
2. Draw the stools with radius `200`, one at a time: `C 650,0 200` / `C -650,0 200` / `C 0,650 200` / `C 0,-650 200` (press Enter after each value).
3. Add a plant pot: type `DONUT 150 220`, press Enter, then type `1100,800` and press Enter.
4. Type `ZE` to see the whole set. Each stool clears the table edge by 50 mm, enough to tuck it under.



Ø800 round table, four stools and a DONUT plant pot

Challenge: Delete three stools and run `ARRAYPOLAR 6` on the last one around `0,0` to make a six-seat table. See ARRAY.

See also

- ARC
- CENTERMARK and CENTERLINE

ARC (3-point arc)

Draw an arc through 3 points: the start point, a point on the arc and the end point.

What it does

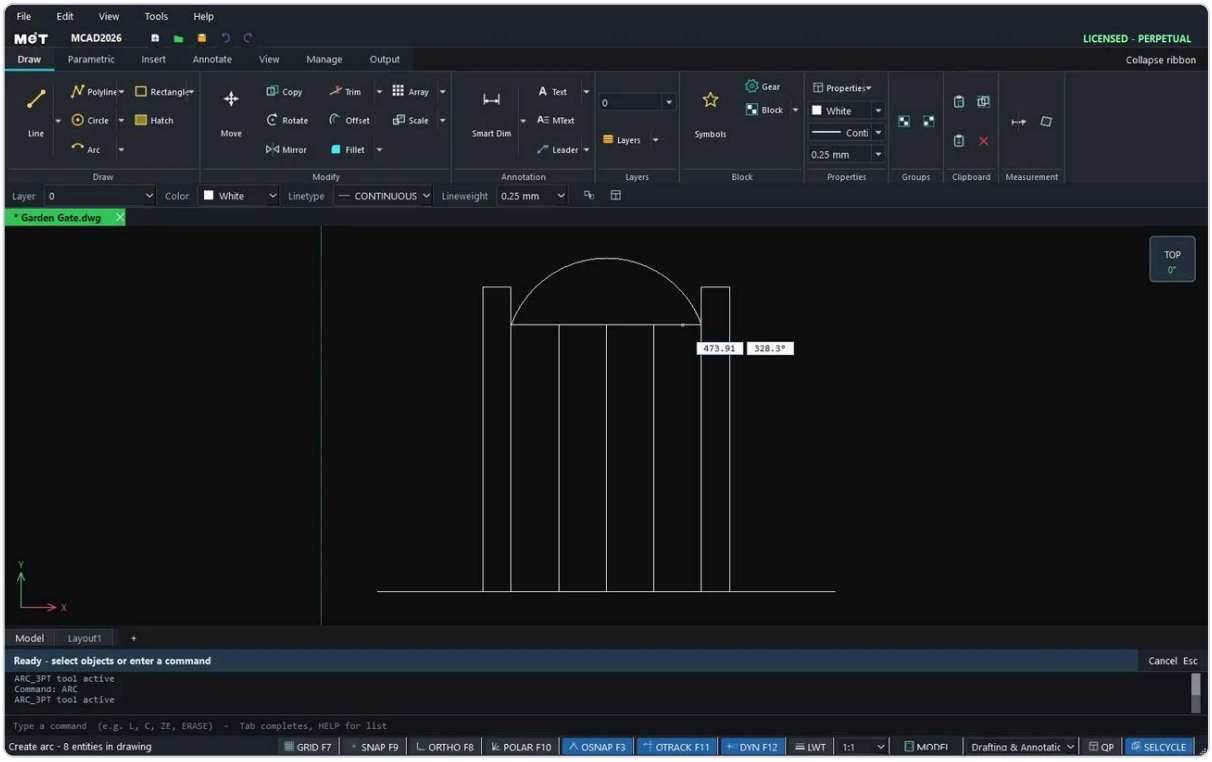
ARC draws an arc that passes through the 3 points you click.

Where to find it

- Ribbon: **Draw › Draw › Arc** (3-Point submenu)
- Command line: `ARC` (alias `A`)

Steps

1. Type `A` and press Enter.
2. Click the start point (ARC: specify the start point).
3. Click any point on the arc (ARC: specify a point on the arc).
4. Click the end point (ARC: specify the end point).



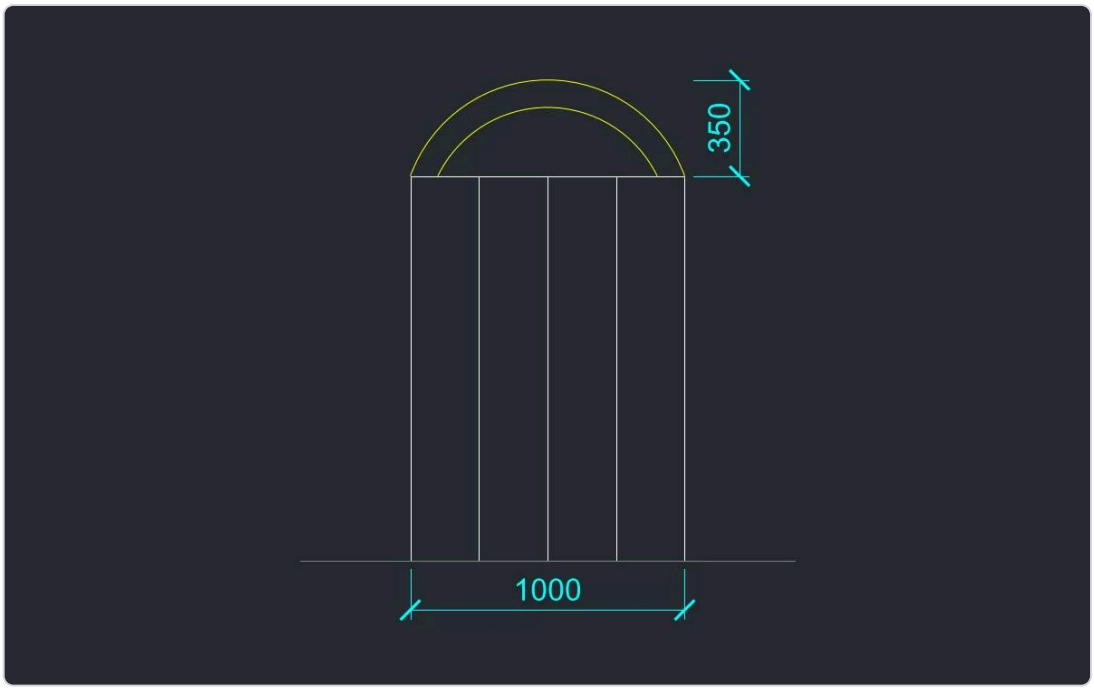
3-point arc

> **Note:** To round the corner between two lines, use FILLET. It creates a tangent arc exactly.

Try it: Arched garden gate

A cottage-style timber gate with an arched top. It takes less than 3 minutes and you practise ARC by typing all 3 points.

1. Type `REC` and press Enter. Type `0,0` and press Enter, then type `1000`, press `Tab`, type `1400` and press Enter. This is the gate leaf, 1000 wide and 1400 mm high.
2. Type `A` and press Enter. Type `0,1400` Enter (start point), `500,1750` Enter (point on the arc) and `1000,1400` Enter (end point). The arch rises 350 mm.
3. Repeat with the points `100,1400`, `500,1650` and `900,1400` for the inner edge of the arch.
4. Add the boards: type `L`, press Enter, type `250,0` Enter, `@0,1400` Enter, then press `Esc`. Repeat at `500,0` and `750,0`.



Arched garden gate, 1000 wide with a 350 mm arch

Challenge: Use MIRROR across the right edge to turn it into a double gate.

See also

- CIRCLE and DONUT
- DIMANGULAR, DIMRADIUS, DIMARC

RECTANG - Rectangle

Draw a rectangle from corner to corner or from the center, with typed width and height.

What it does

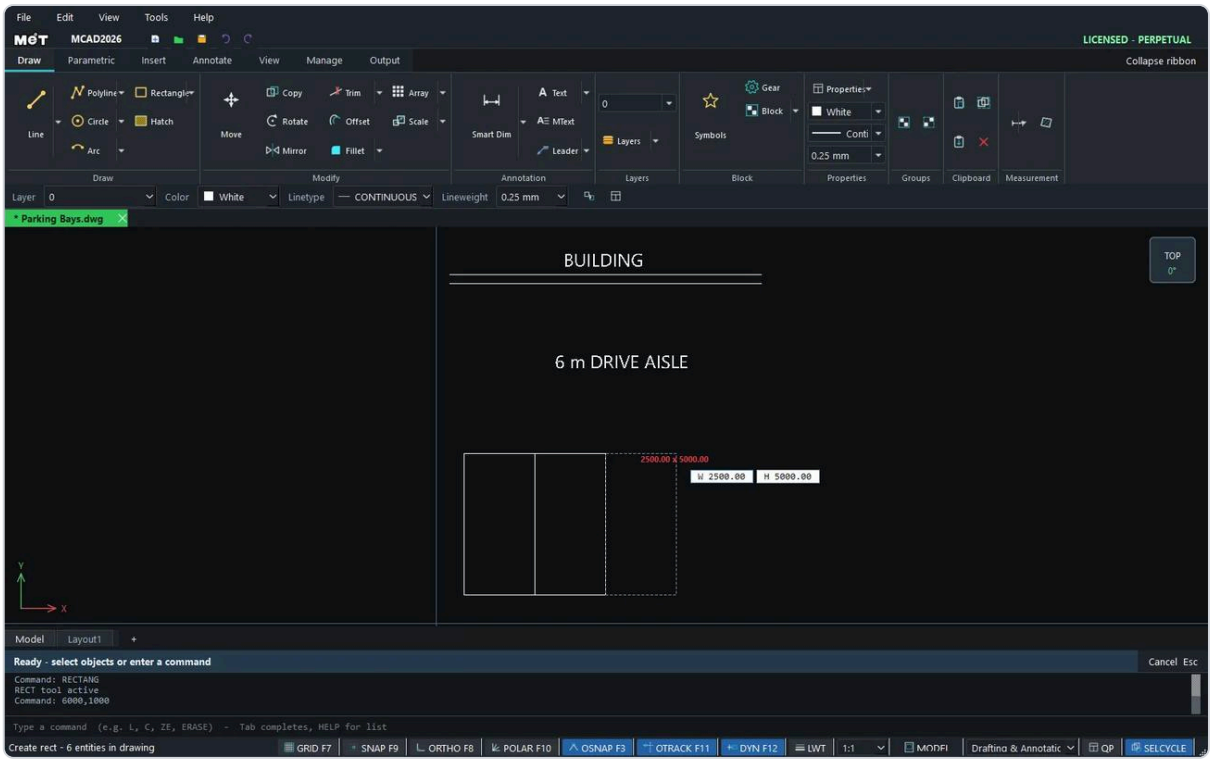
RECTANG draws a rectangle as a single closed object.

Where to find it

- Ribbon: **Draw › Draw › Rectangle** (submenu: Rectangle - Corner, Rectangle - Center, Revision Cloud, 2D Solid, Wipeout)
- Command line: `RECTANG` (Alias `REC`, `RECTANGLE`)

Steps (Corner)

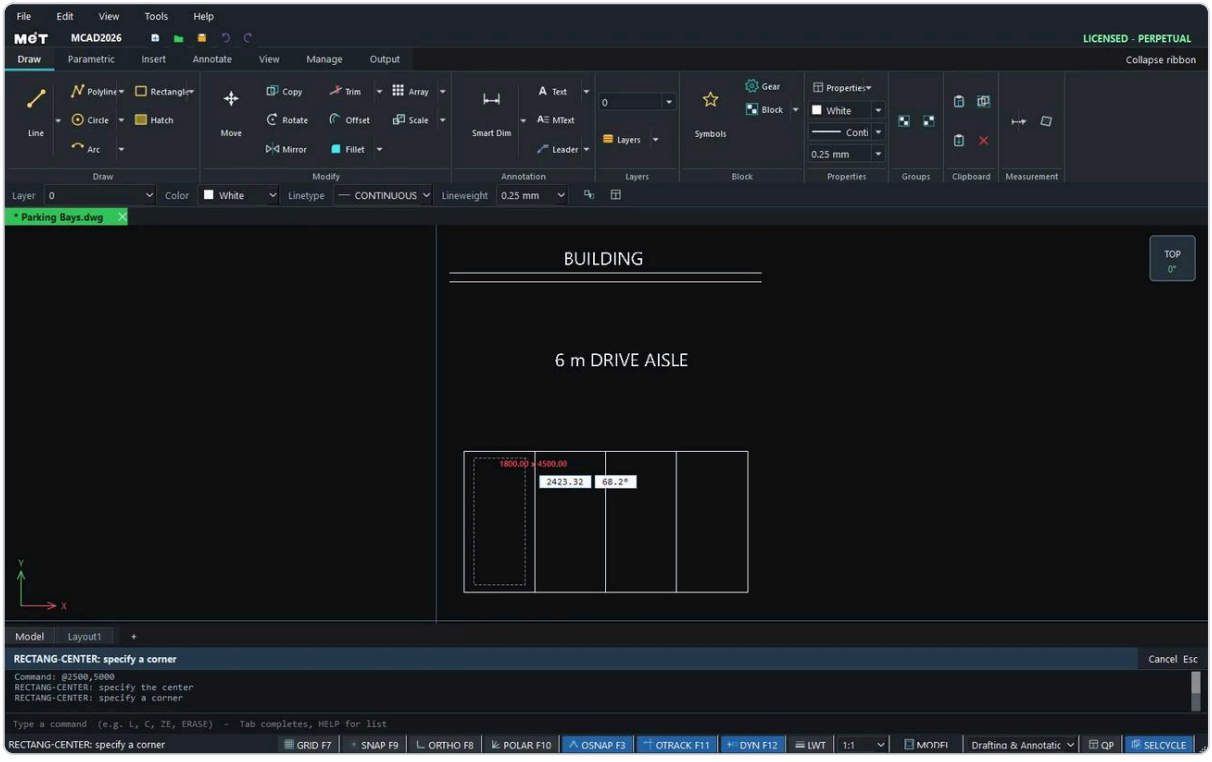
1. Type `REC` and press Enter.
2. Click the first corner.
3. Move the mouse. The Smart Input box shows **W** and **H**. Click the opposite corner, or type the width, press `Tab`, type the height, and press Enter.



RECTANG corner to corner, showing W and H

Steps (Center)

1. Choose **Rectangle ▾ › Rectangle - Center**.
2. `RECTANG-CENTER: specify the center` Click the center point.
3. `RECTANG-CENTER: specify a corner` Click a corner.



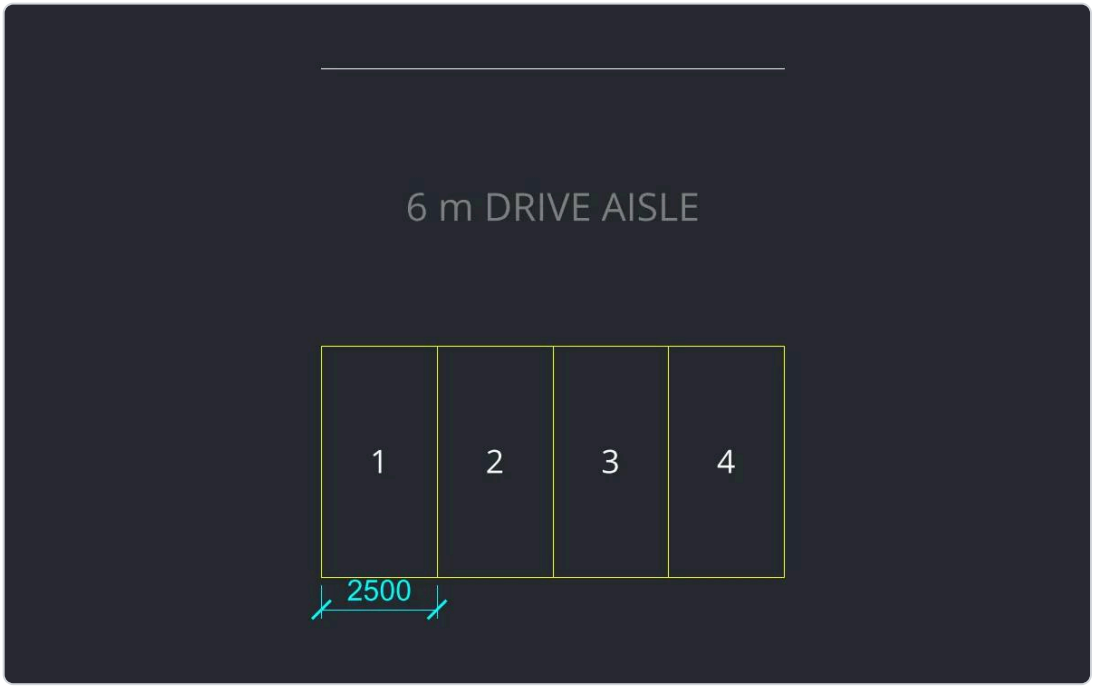
Rectangle - Center, specifying the corner

Option	Meaning
Rectangle - Corner	First corner → opposite corner
Rectangle - Center	Center point → corner (symmetric)

Try it: A row of four parking bays

Four standard 2.5 x 5 m parking bays in front of a building, with a 6 m drive aisle. Good practice for typing W/H in Smart Input.

1. Type `REC`, Enter, `0,0`, Enter, then `2500` `Tab` `5000` Enter.
2. Bays 2-4: `REC` at `2500,0`, `5000,0` and `7500,0`, each `2500` `Tab` `5000`.
3. Aisle edge: `L` `0,11000` Enter `@10000,0` Enter, then `Esc`. The aisle between the back of the bays and this line is 6 m wide.
4. Number the bays with `TEXT` at a height of `500`.



Four 2.5 x 5 m parking bays and a 6 m aisle

Challenge: Make the last bay an accessible bay `3600` wide and see how much longer the row gets.

See also

- POLYGON
- SOLID, WIPEOUT and REVCLOUD

POLYGON - Regular Polygon

Draw a regular hexagon from a center point and a vertex.

What it does

POLYGON draws a regular polygon, such as a hexagonal nut head.

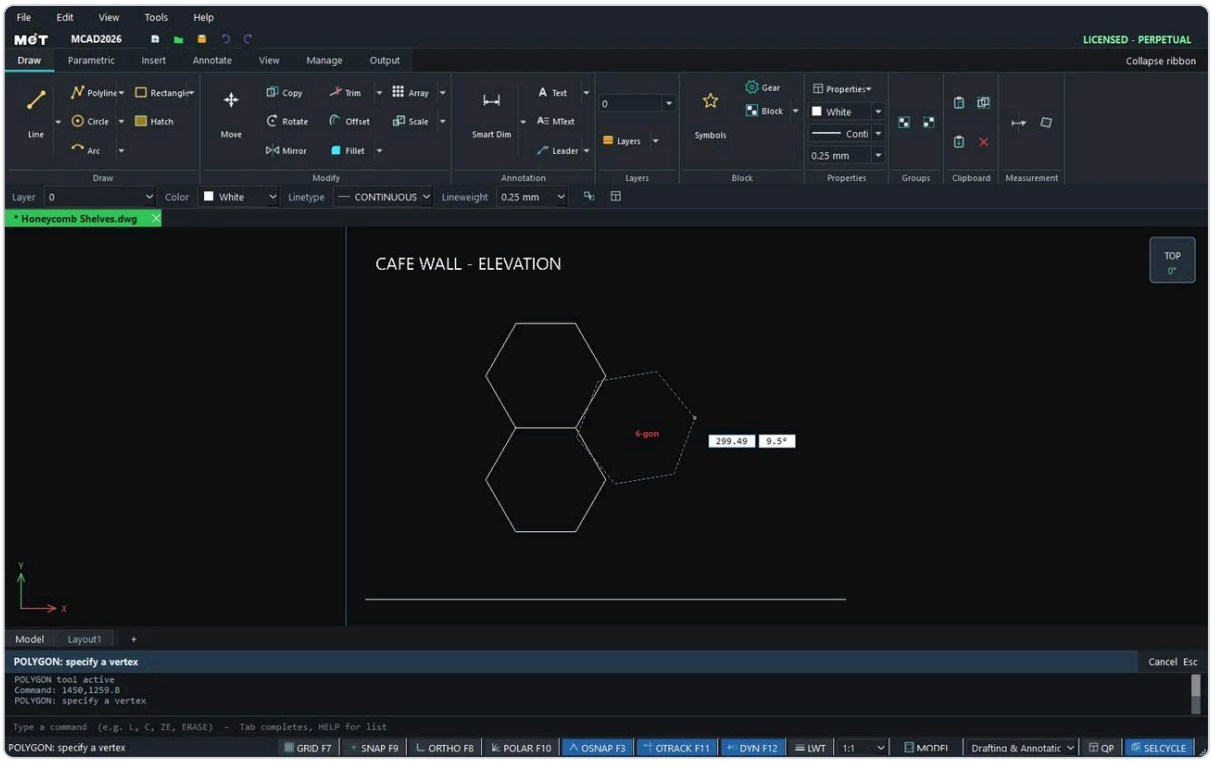
Where to find it

- Ribbon: there is no Ribbon button. Right-click in the drawing area › **Draw › Polygon**
- Command line: `POLYGON` (Alias `POL`)

> **Note:** In this version, POLYGON always draws a **6-sided** polygon (a hexagon). There is no command-line option to change the number of sides yet.

Steps

1. Type `POL` and press Enter. `POLYGON: specify the center`
2. Click the center point.
3. `POLYGON: specify a vertex` Click a vertex, or type the radius/angle in Smart Input.

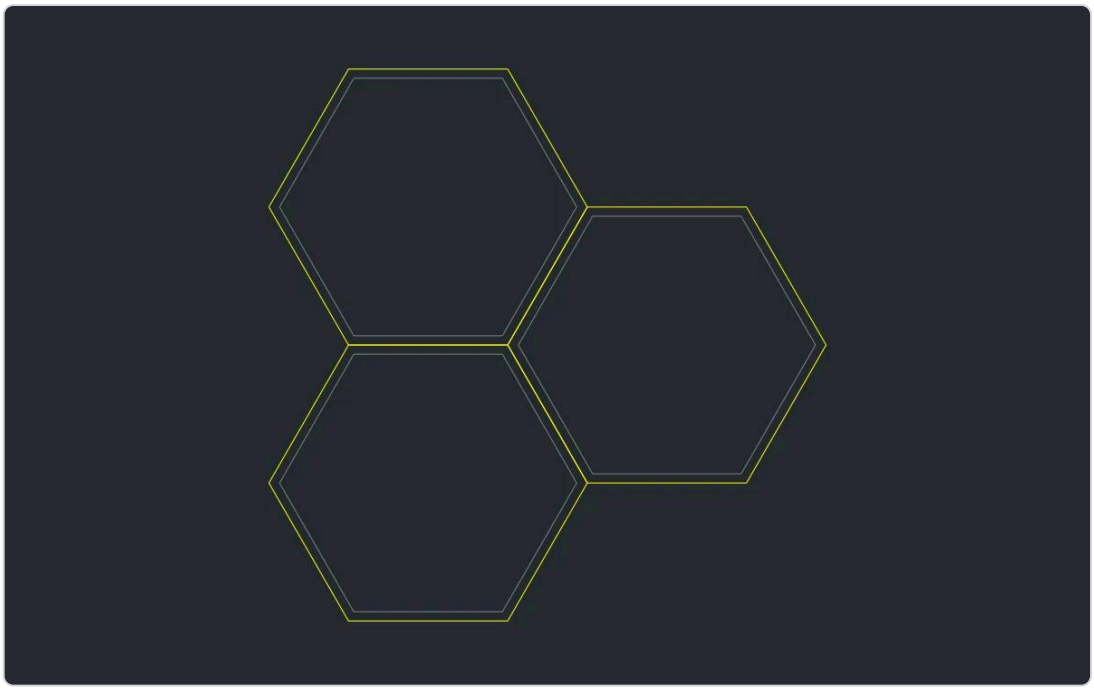


6-sided POLYGON, specifying the vertex

Try it: Honeycomb wall shelves

Three hexagonal shelves nested like a honeycomb, a popular café wall feature. Each one is 600 mm across the corners.

1. Type `POL`, Enter, `0,0`, Enter, then `@300<0` and Enter. That is a hexagon with a 300 radius.
2. Second shelf: `POL` `0,519.6`, then `@300<0` (519.6 = 2 x 300 x cos 30°, so the edges touch exactly).
3. Third shelf: `POL` `450,259.8`, then `@300<0`.
4. Board thickness: repeat at the same three centres with `@280<0`. The shelf walls are about 17 mm thick.



Three hexagonal honeycomb shelves

Challenge: Add a fourth shelf at `450,-259.8` and see how the honeycomb keeps tiling.

See also

- RECTANG
- CIRCLE

ELLIPSE

Draw an axis-aligned ellipse from its centre and a corner of its bounding box.

What it does

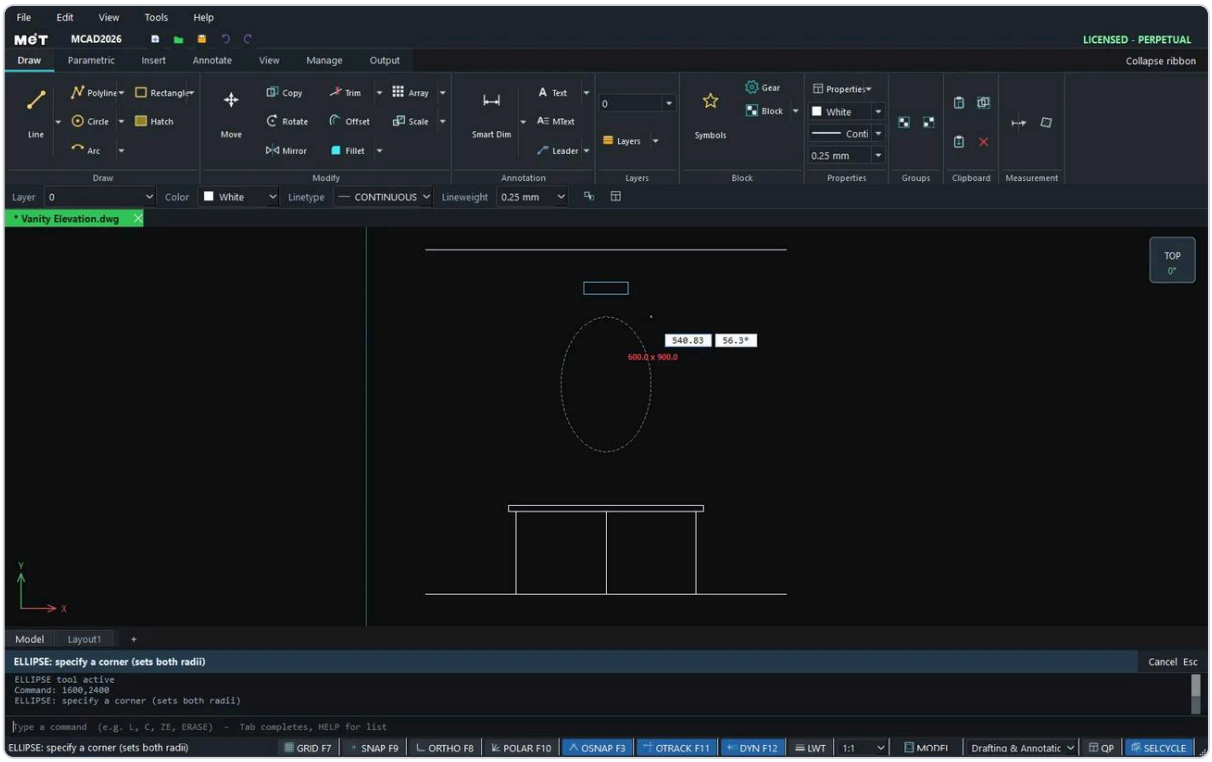
ELLIPSE draws an ellipse whose axes are parallel to the X and Y axes. Both radii come from the bounding box corner that you click.

Where to find it

- Ribbon: there is no Ribbon button. Right-click in the drawing area › **Draw › Ellipse**.
- Command line: `ELLIPSE` (alias `EL`)

Steps

1. Type `EL` and press Enter. The prompt reads `ELLIPSE: specify the center`.
2. Click the centre point.
3. At `ELLIPSE: specify a corner (sets both radii)`, click a corner of the box. The X distance is the horizontal radius and the Y distance is the vertical radius.



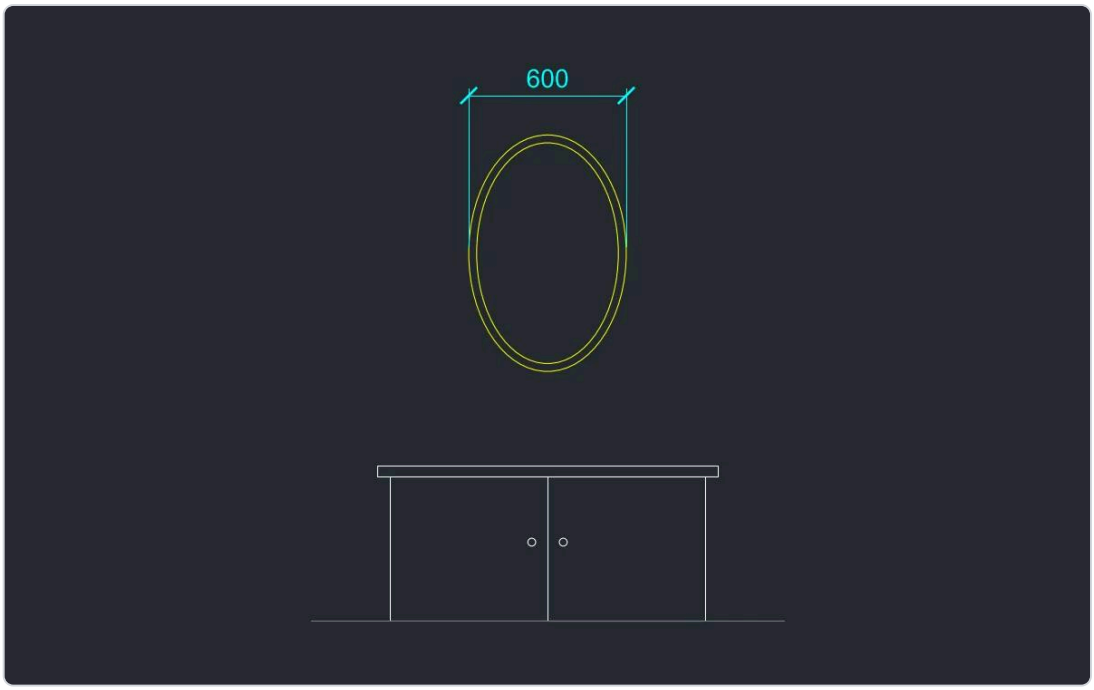
ELLIPSE showing a size of 600 x 900 for an oval mirror

> **Note:** For a tilted ellipse, draw it and then use ROTATE.

Try it: Oval mirror over a bathroom vanity

A bathroom elevation of the kind interior designers show clients: a vanity cabinet and an oval mirror with a double frame.

1. Vanity cabinet: `REC` `0,0`, then `1200` `Tab` `550` Enter.
2. Countertop: `REC` `-50,550`, then `1300` `Tab` `40` Enter.
3. Door split: `L` `600,0` Enter `@0,550` Enter, then `Esc`.
4. Mirror: type `EL`, Enter, `600,1400`, Enter (centre), then `@300,450` and Enter. The ellipse is 600 wide and 900 mm high.
5. Inner frame line: `EL` `600,1400`, then `@270,420`.



600 x 900 oval mirror over a vanity

Challenge: Draw a flat ellipse `EL` `600,570` `@200,120` as a countertop basin in a plan view.

See also

- CIRCLE

SLOT (Straight Slot)

Draw a straight slot with rounded ends from two end centers and a width.

What it does

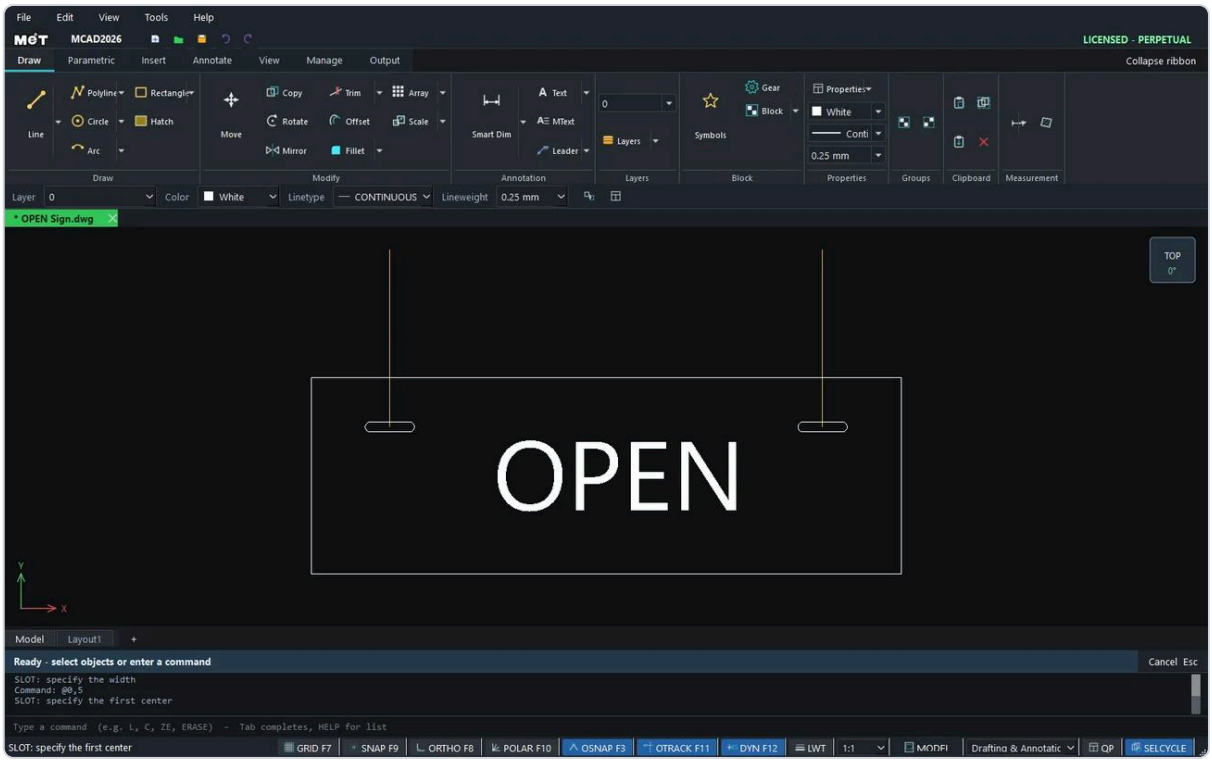
SLOT draws a slot with semicircular ends (2 straight lines + 2 end arcs). Use it for adjustment holes.

Where to find it

- Ribbon: type the command (there is no Ribbon button)
- Command line: `SLOT`

Steps

1. Type `SLOT` and press Enter.
2. `SLOT: specify the first center` Click the center of the first end.
3. `SLOT: specify the second center` Click the center of the second end.
4. `SLOT: specify the width` Click to set the width (measured from the centerline).

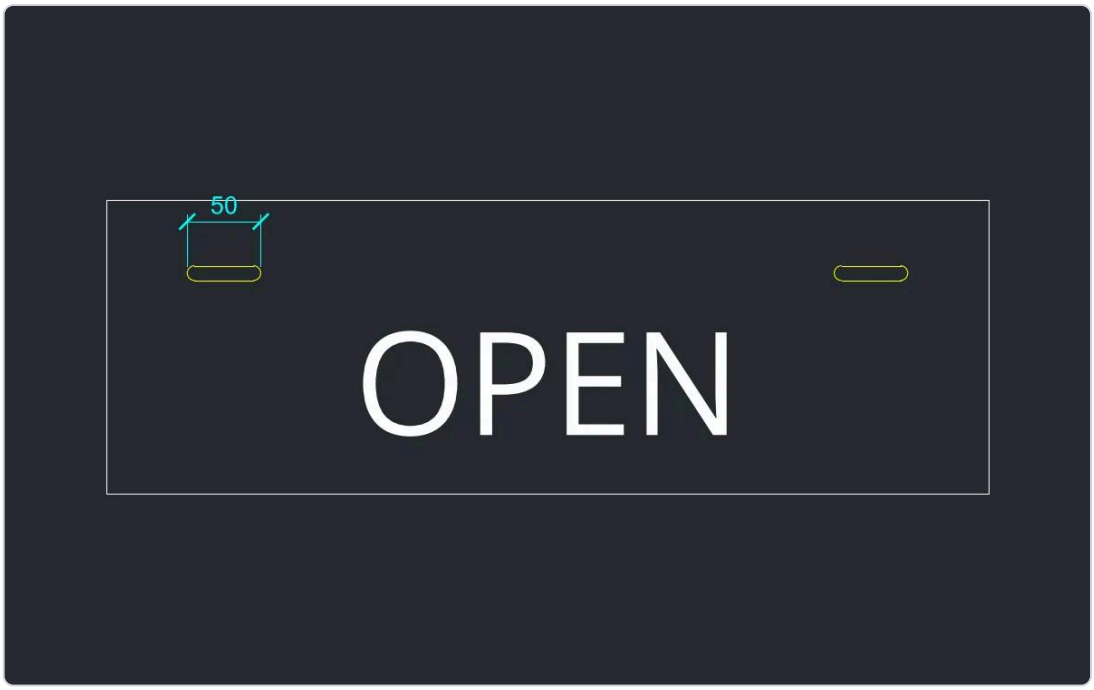


Finished SLOT

Try it: Slotted holes on a hanging shop sign

A 600 x 200 mm OPEN sign with two slots for the hanging chain. Sliding the chain in the slots lets you level the sign.

1. Sign plate: `REC` `0,0`, then `600` `Tab` `200` Enter.
2. Type `SLOT`, Enter, `60,150`, Enter (first end centre), `@40,0`, Enter (second end centre), then `@0,5` and Enter (5 from the centre line, so the slot is 10 wide).
3. Second slot: `SLOT` `500,150` `@40,0` `@0,5`.
4. Add `TEXT` `OPEN` at a height of `70` in the middle.



600 x 200 OPEN sign with two 10 x 50 slots

Challenge: Make vertical slots with `@0,40` instead, so you can adjust the sign height on the wall rather than its level.

See also

- DIMLINEAR
- GCODE

SPLINE, SPLINEDIT and SKETCH

Draw free-form curves through points (SPLINE), edit their points (SPLINEDIT), and draw freehand (SKETCH).

What it does

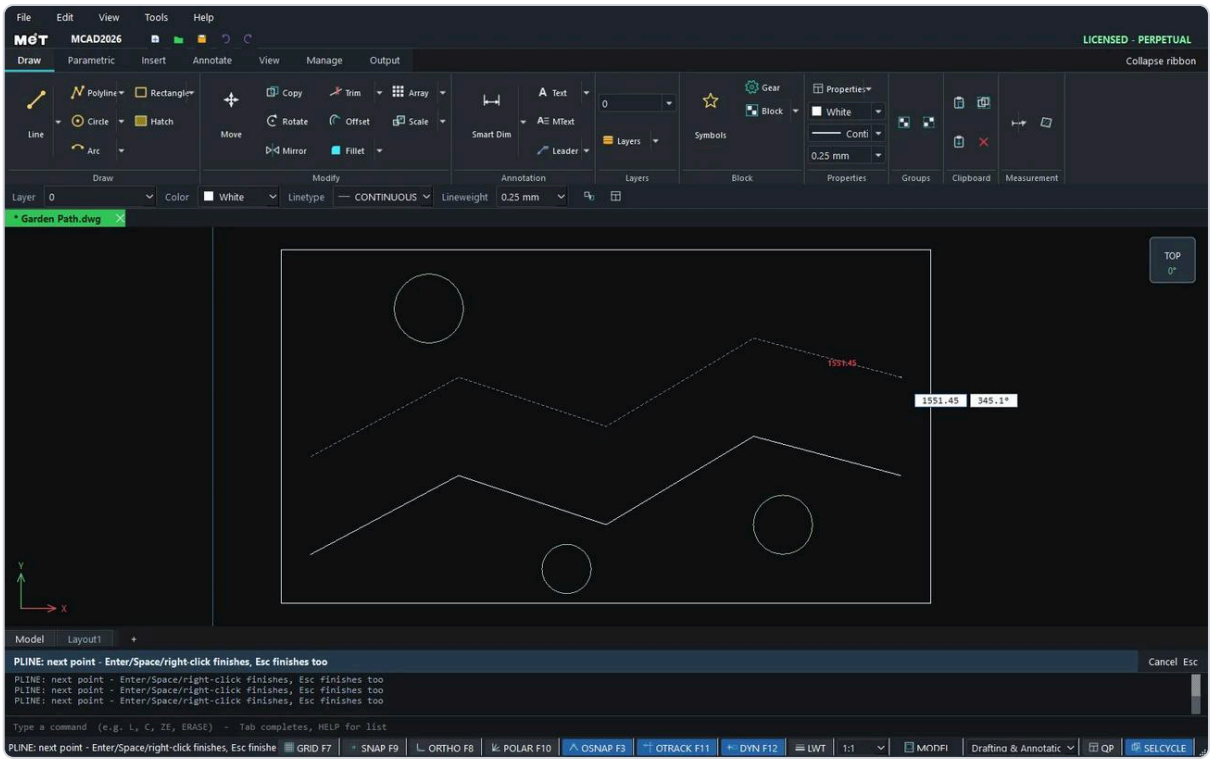
- **SPLINE** – a smooth curve through the points you click (Fit points).
- **SPLINEDIT** – close/open a Spline, and delete or add Fit points.
- **SKETCH** – draw freehand by dragging the mouse. The result is saved as a Polyline.

Where to find it

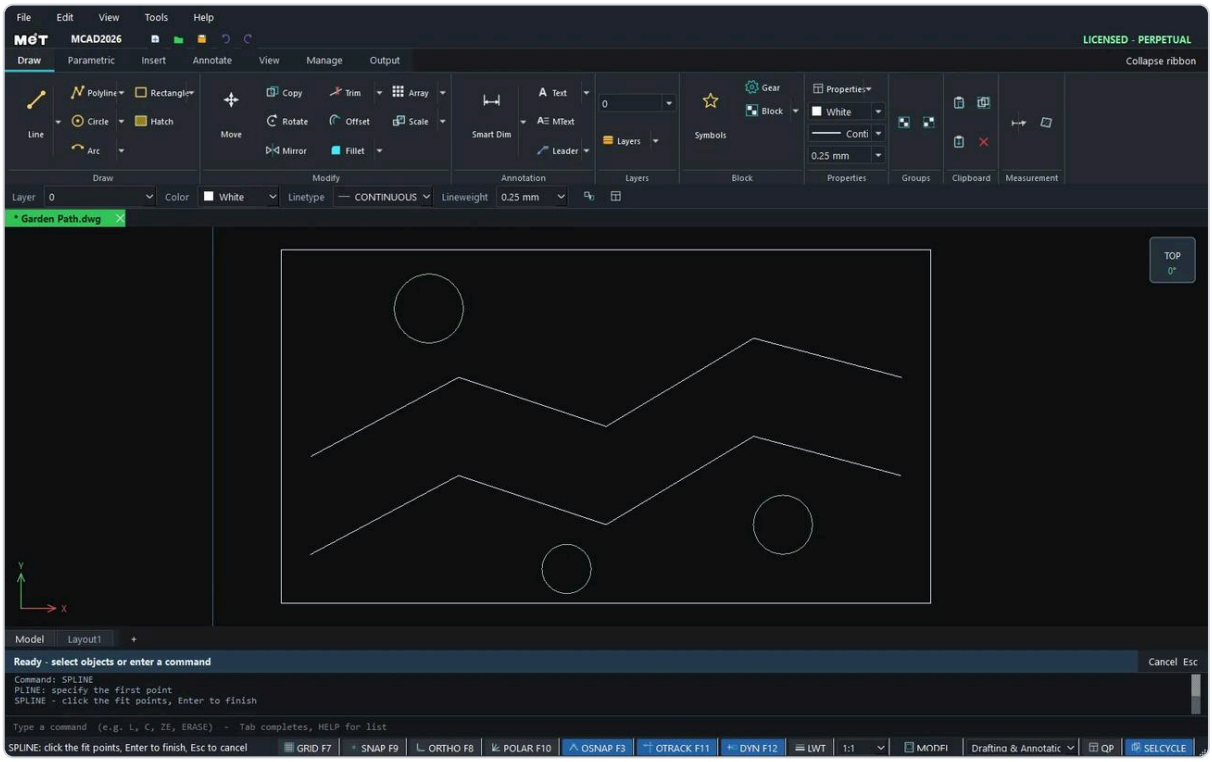
- Ribbon: **Draw › Draw › Polyline ▾ › Spline - Fit Points (SPLINE) / Freehand Sketch (SKETCH)**
- Command line: `SPLINE` (Alias `SPL`)
- Command line: `SPLINEDIT` (Alias `SPE`), `SKETCH`

SPLINE steps

1. Type `SPL` and press Enter. `SPLINE: click the fit points, Enter to finish, Esc to cancel`
2. Click each point the curve must pass through.
3. Press Enter to finish.



SPLINE, clicking Fit points



Finished Spline

SPLINEDIT

Select the Spline first, then type:

Format	Meaning
`SPLINEDIT Close` / `SPLINEDIT Open`	Close / open the curve
`SPLINEDIT Delete <n>`	Delete Fit point number n (starts at 0; at least 2 points must remain)

Format	Meaning
`SPLINEDIT Add <n> <x> <y>`	Add a point after point number n

SKETCH

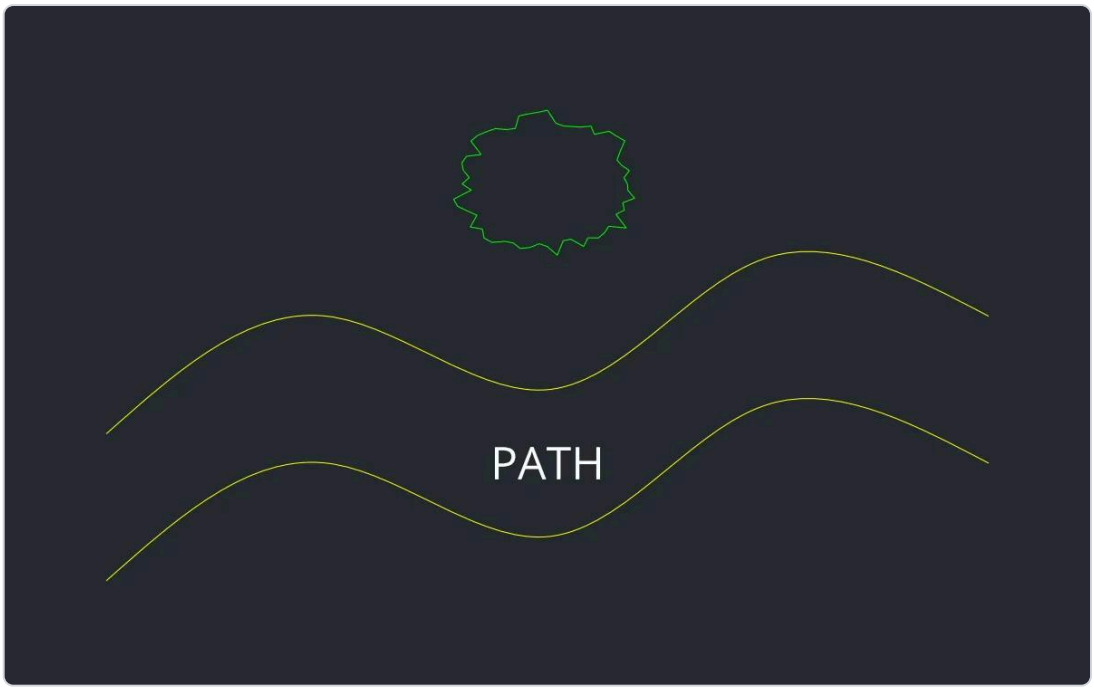
1. Type `SKETCH` or `SKETCH 2` (records a point every 2 units).
2. `SKETCH: drag to trace` Hold the mouse button down and drag.
3. Release the mouse button to finish.

> **Note:** You can break a Spline into smaller segments with EXPLODE.

Try it: Winding garden path

A 1 m wide path that curves naturally through a garden, with a hand-drawn shrub beside it. You use both SPLINE and SKETCH.

1. Left edge of the path: type `SPL`, Enter, then `0,0`, `1500,800`, `3000,300`, `4500,1200`, `6000,800` (Enter after each), and press Enter once more to finish.
2. Right edge: `SPL` with the same points moved up by 1000: `0,1000`, `1500,1800`, `3000,1300`, `4500,2200`, `6000,1800`, then Enter.
3. Shrub: type `SKETCH 20`, press Enter, hold the mouse button and drag a bumpy loop above the path near `3000,2700`. Release to finish.



SPLINE garden path with a SKETCH shrub

Challenge: Select the left edge and type `SPLINEDIT Add 1 2200 400` to add a fit point, and watch the path reshape.

See also

- PLINE

POINT

Place Point objects and set their display size with DDPTYPE.

What it does

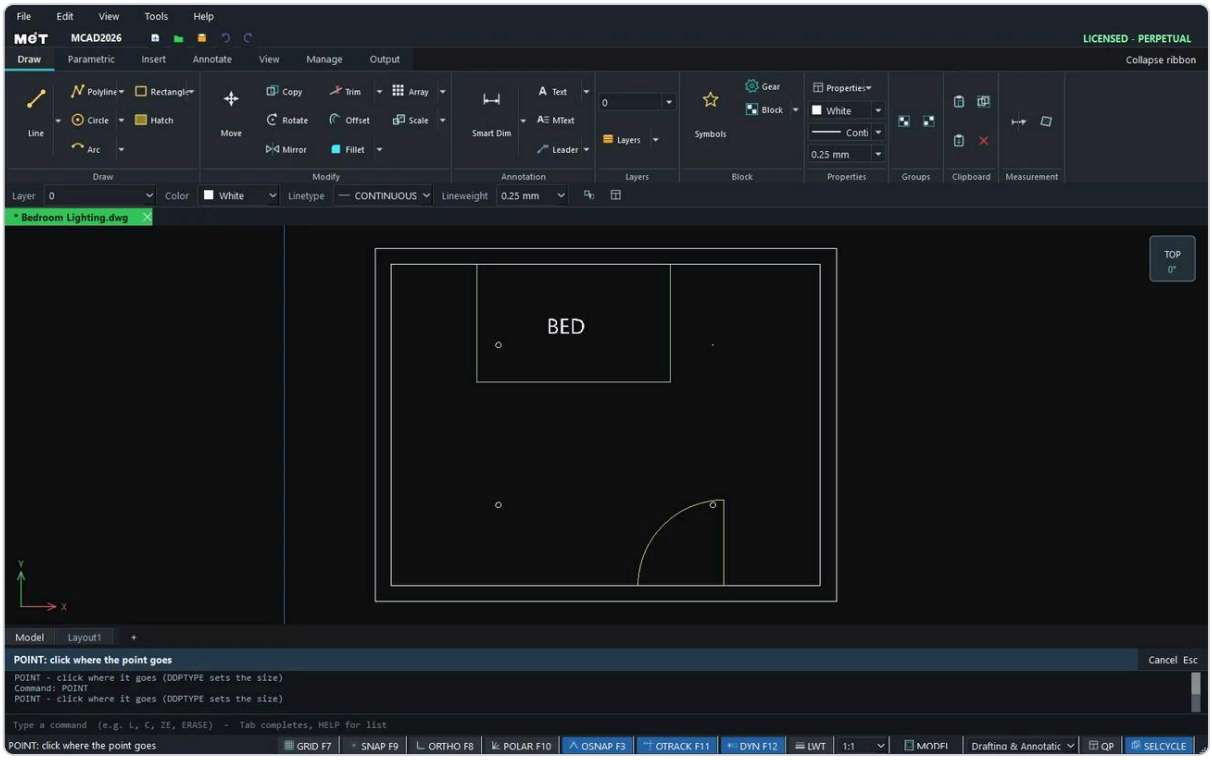
POINT places a reference point (for example, a drilling position). Object Snap can snap to it.

Where to find it

- Ribbon: **Draw › Draw › Circle ▾ › Point (POINT)**
- Command line: `POINT` (Alias `PO`)
- Display size: `DDPTYPE <size>` (Alias `PTYPE`)

Steps

1. Type `DDPTYPE 4` to set the point size to 4 pixels (you only do this once).
2. Type `PO` and press Enter. `POINT: click where the point goes`
3. Click a location (or type `x,y`). The Prompt bar shows the coordinates of the point you placed.
4. Press Enter to place the next point with the same command.



Three points placed with POINT

Try it: Downlight layout for a bedroom

Place four downlights and a central pendant in a 4 x 3 m bedroom, so the electrician can cut the ceiling in exactly the right spots.

1. Room: `REC 0,0`, then `4000 Tab 3000` Enter.
2. Type `DDPTYPE 6` and press Enter so the points are easy to see.
3. Type `PO`, press Enter, then type `1000,750` and press Enter.

4. Press Enter to repeat POINT and type `3000,750`. Do the same for `1000,2250` and `3000,2250`.
5. Pendant: a POINT at `2000,1500`. The Prompt bar shows the coordinates of every point you place, so you can check each one at once.



4 x 3 m bedroom with four downlights and a pendant

Challenge: Use `DIST` on two downlights to check they are 2000 mm apart.

See also

- ARRAY - DIVIDE and MEASURE
- Units and System Variables

HATCH and GRADIENT

Fill closed areas with a pattern (HATCH), edit the hatch, apply a gradient fill (GRADIENT) and send hatches to the back.

What it does

- **HATCH** – fills a closed shape with a hatch pattern. It supports inner holes (islands) and associative boundaries, so the hatch follows when you edit the boundary.
- **HATCHEDIT** – changes the pattern, scale or angle of the selected hatch.
- **GRADIENT** – applies a two-colour gradient fill.
- **HATCHTOBACK** – sends all hatches behind the lines.

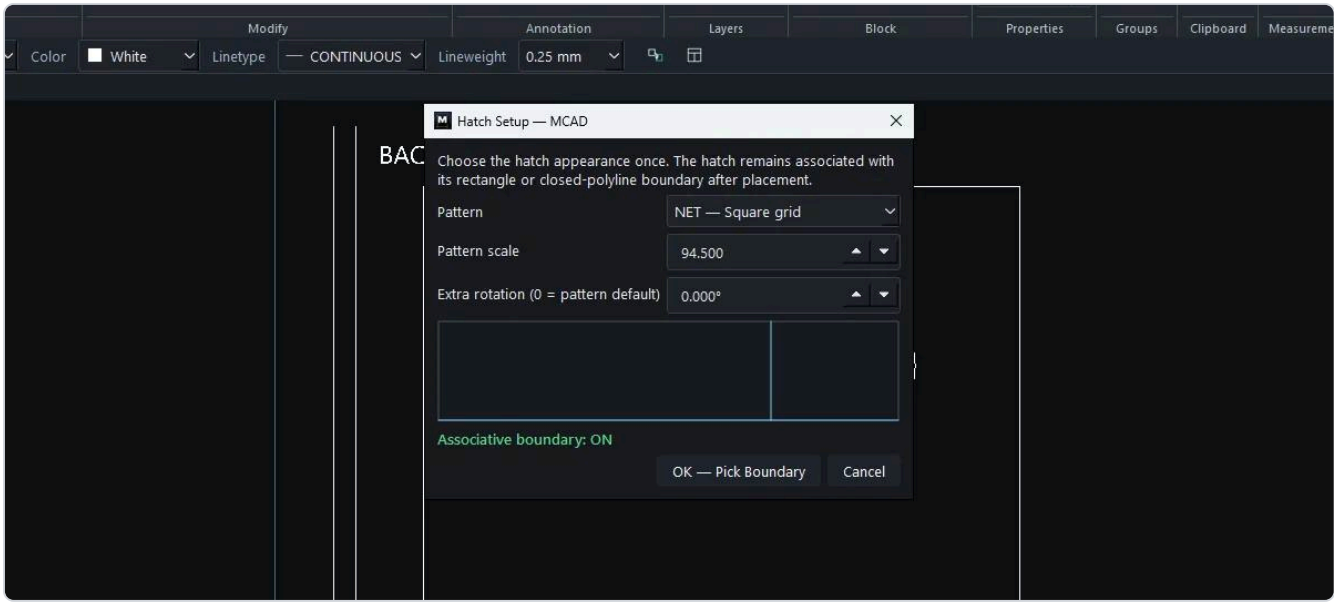
Where to find it

- Ribbon: **Draw › Draw › Hatch** (opens the Hatch settings dialog)
- Command line: `HATCH` (aliases `H`, `BH`, `H2`, `BHATCH`)
- Command line: `HATCHEDIT` (`HE`), `GRADIENT` (`GD`), `HATCHTOBACK` (`HTB`)

- The contextual **Hatch** tab appears when you select a hatch.

Steps from the Ribbon

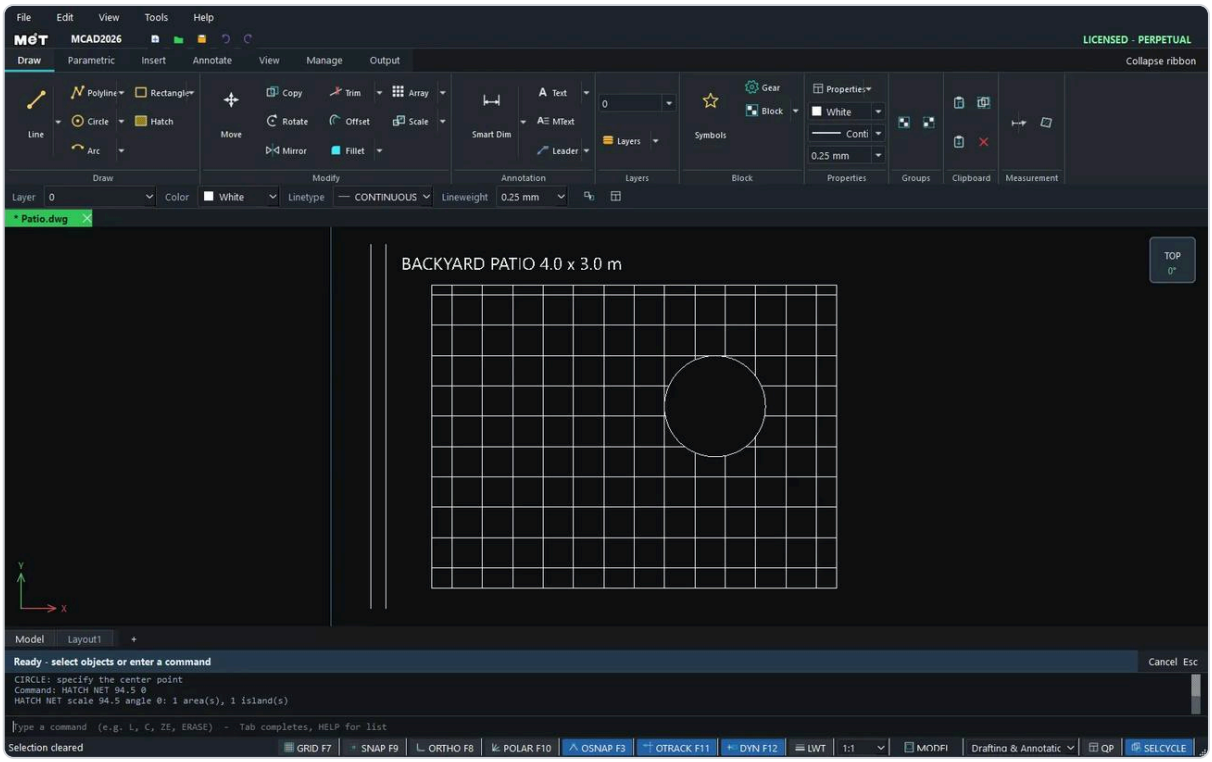
1. Click **Draw › Hatch**. The settings dialog opens.
2. Choose the **Pattern**, **Pattern scale** and **Extra rotation**. Check the preview below them.
3. Click **OK — Pick Boundary**, then click inside a closed shape (or select the closed shapes before you start the command).



Hatch settings dialog

Steps from the command line

1. Select the closed shapes, both the outer boundary and the inner holes (rectangle, circle, closed polyline).
2. Type `HATCH ANSI31 8 0` and press Enter (pattern, scale, angle). If nothing is selected, the program asks `HATCH: select closed profile(s)`. Click the shapes, then press Enter.
3. The result reads `HATCH ANSI31 scale 8 angle 0: 1 area(s), 1 island(s)`.



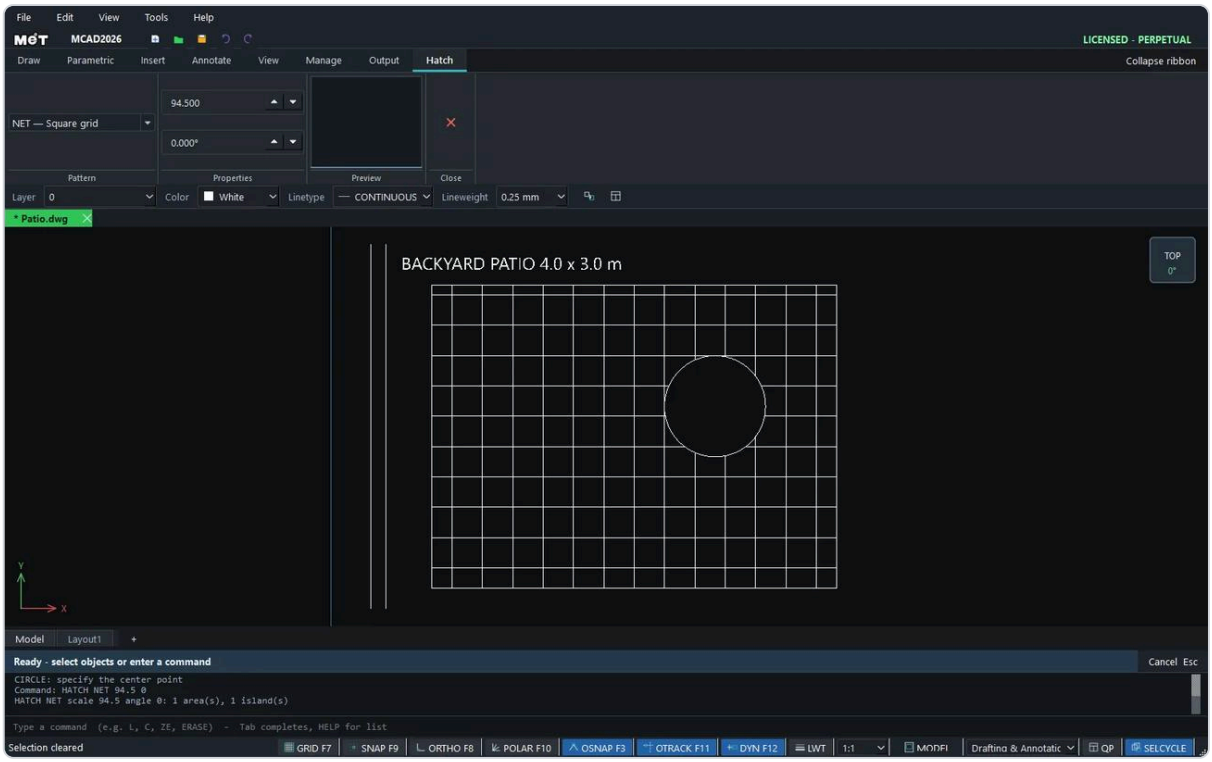
NET hatch on a tiled patio, leaving the tree pit empty

Options

Option	Meaning
Pattern	ANSI31, ANSI32, ANSI33, ANSI34, ANSI37, CONCRETE, CROSS, EARTH, LINE, NET, SOLID, STEEL (`HATCH ?` lists them).
Scale	The pattern size. If it is so large that no lines fall inside the area, the program warns `try a smaller scale`.
Angle	An extra rotation added to the pattern's own angle.
Associative boundary	The hatch follows its boundary when the boundary is edited.

Editing a hatch

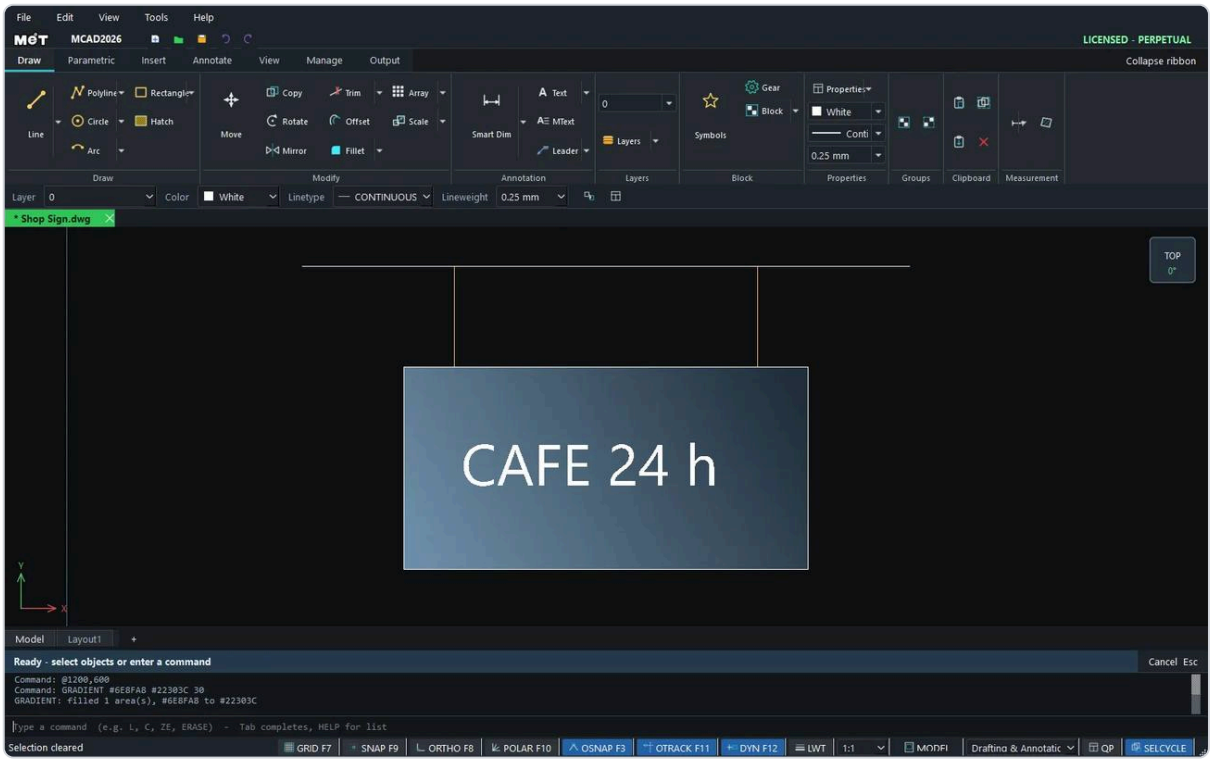
- Click a hatch. The **Hatch** tab appears on the Ribbon. Change the Pattern, Scale or Angle and see the result immediately. Click ✕ (Close Hatch Editor) to close the tab.
- Or select the hatch and type `HATCHEDIT ANSI37 4 45`.



Contextual Hatch tab

GRADIENT

1. Select a closed shape.
2. Type `GRADIENT #6E8FA8 #22303C 30` (colour 1, colour 2, angle).



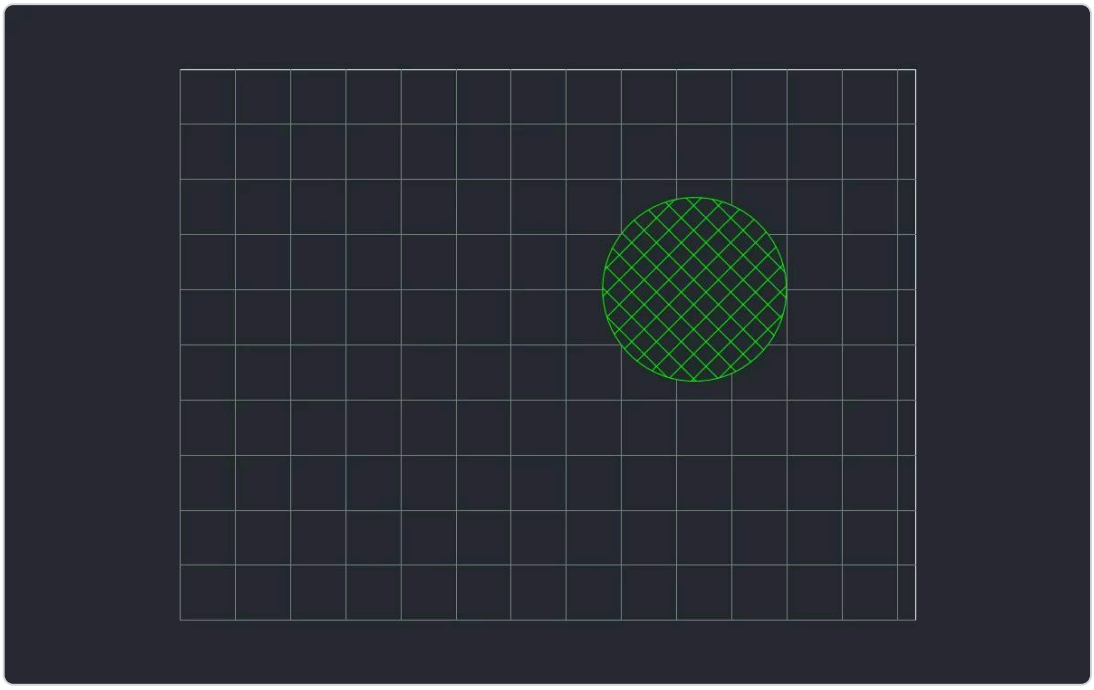
Gradient fill on a rectangle

Note: Use `HATCHTOBACK` when a hatch covers boundary lines or text. You can also use `TEXTTOFRONT` for text.

Try it: Tiled patio with a round tree pit

A 4 x 3 m backyard patio with 300 mm tiles and a round tree pit. You see how HATCH leaves an island empty by itself.

1. `REC` `0,0`, then `4000` `Tab` `3000` Enter.
2. Tree pit: `C` `2800,1800` with radius `500`.
3. Select the rectangle and the circle, type `HATCH NET 94.5 0` and press Enter. NET lines are 3.175 x 94.5 ≈ 300 mm apart, the size of one tile, and the tree pit stays empty.
4. Select only the circle, type `HATCH EARTH 15 0` and press Enter to fill the pit with soil.



300 mm tiled patio with an EARTH-hatched tree pit

Challenge: Click the tile hatch and set Angle to `45` on the **Hatch** tab for a diagonal layout.

See also

- BOUNDARY, REGION
- DRAWORDER

SOLID, WIPEOUT and REVCLOUD

Draw 2D filled areas (SOLID), masking areas (WIPEOUT), and revision clouds (REVCLOUD).

What it does

- **SOLID** – a filled triangle or quadrilateral.
- **WIPEOUT** – a background-colored area that hides what is underneath (for example, to make room for text).
- **REVCLOUD** – a revision cloud around the selected objects. Use it to mark changes in a revision.

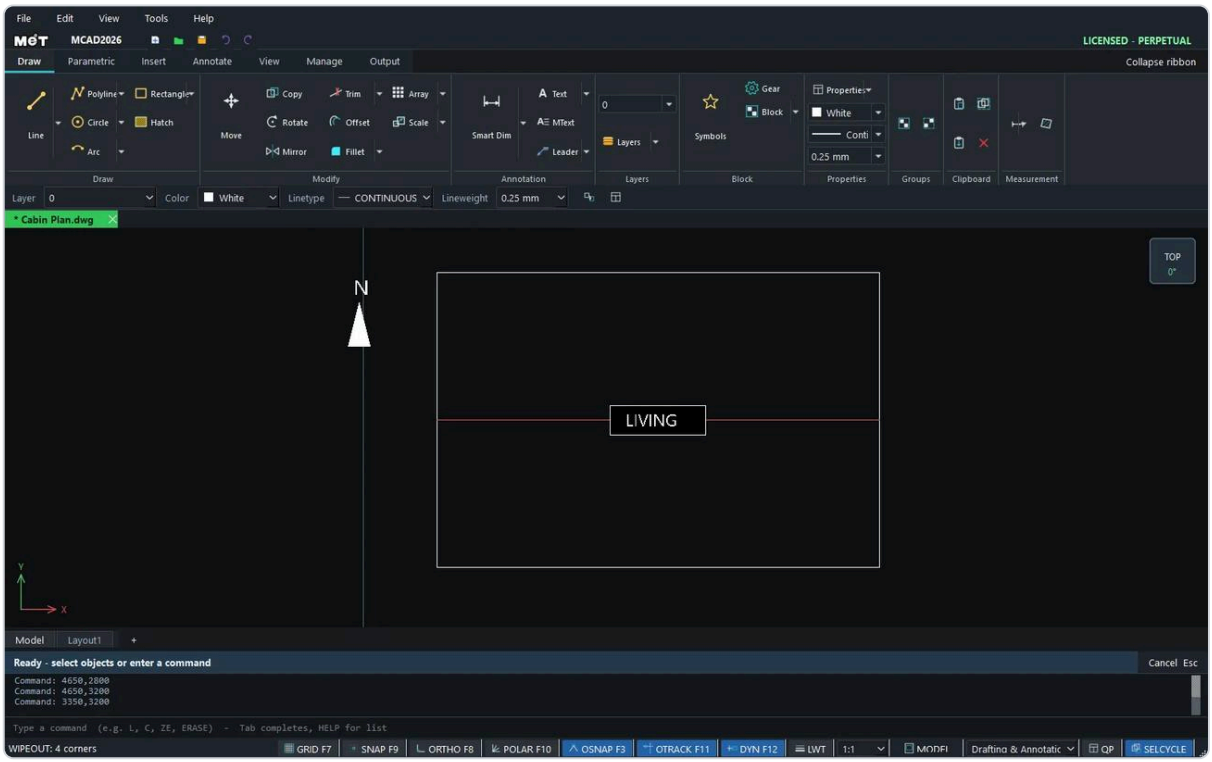
Where to find it

- Ribbon: **Draw › Draw › Rectangle ▾ › 2D Solid - Filled (SOLID)** / **Wipeout - Mask (WIPEOUT)** / **Revision Cloud (REVCLOUD)**
- Command line: `SOLID` (Alias `SO`)
- Command line: `WIPEOUT` (`WO`), `REVCLOUD` (`RC`)

SOLID / WIPEOUT steps

1. Type `SOLID` (or `WIPEOUT`) and press Enter.
2. Click corners 1, 2, and 3. A triangle is created as soon as you click corner 3.
3. Click corner 4 to make a quadrilateral, or press `Esc` to keep the triangle.

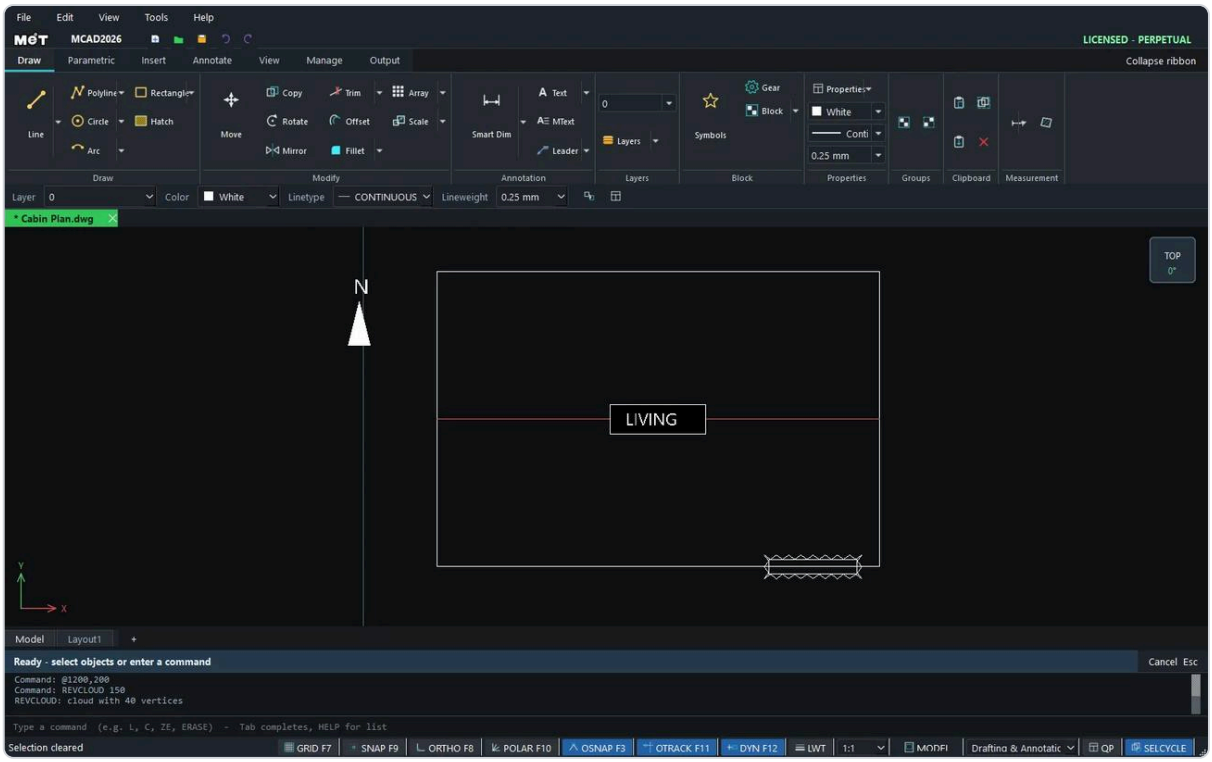
> **Note:** The corner order follows AutoCAD. Corners 1-2 form one edge, and 3-4 form the opposite edge in the same direction. If you click around the shape in a circle, you get a bow-tie shape (as in the left picture).



SOLID north arrow (left) and WIPEOUT under the LIVING label hiding the beam line (right)

REVCLOUD steps

1. Select the objects you want to surround with a cloud.
2. Type `REVCLOUD 10` (arc length 10) and press Enter.

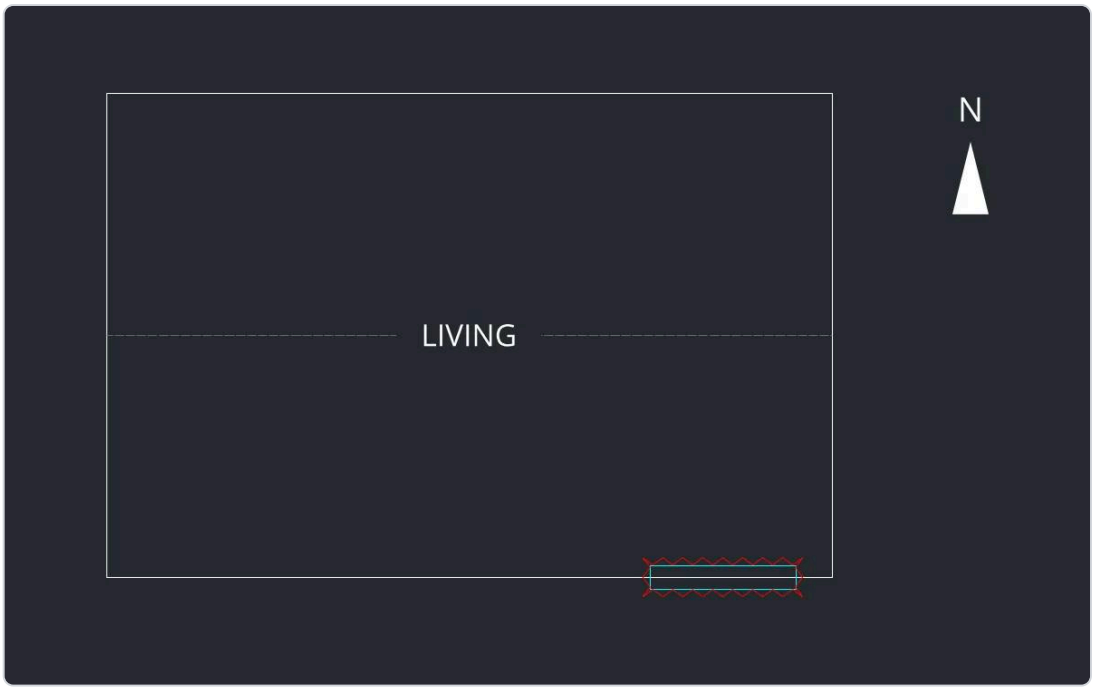


REVCLOUD around the window

Try it: North arrow, room label and revision cloud on a cabin plan

A small cabin plan with a filled north arrow, a room label that the beam line does not cross, and a revision cloud around a window that just moved. All three commands in one job.

1. Plan: `REC` `0,0`, then `6000` `Tab` `4000` Enter. Draw a beam across the room with `L` `0,2000` `@6000,0` (set its Linetype to DASHED).
2. North arrow: type `SOLID`, Enter, then `7000,3000`, `7300,3000`, `7150,3600` (Enter after each), and press `Esc` to keep the triangle.
3. Label background: type `WIPEOUT`, Enter, then `2400,1800`, `3600,1800`, `2400,2200`, `3600,2200` (AutoCAD order: edges 1-2 and 3-4 run the same way). The beam line under the label disappears.
4. Add `TEXT` `LIVING` in the middle of that area.
5. New window: `REC` `4500,-100`, then `1200` `Tab` `200` Enter. Select the window, type `REVCLOUD 150` and press Enter.



Cabin plan with a north arrow, LIVING label and revision cloud

Challenge: Add a `TEXT` `N` above the arrow, then ROTATE the arrow to the site's true north.

See also

- HATCH
- DRAWORDER

BOUNDARY, REGION and Booleans (UNION/SUBTRACT/INTERSECT)

Create a closed boundary (BOUNDARY) and a region, then combine, subtract or intersect closed shapes (UNION/SUBTRACT/INTERSECT).

What it does

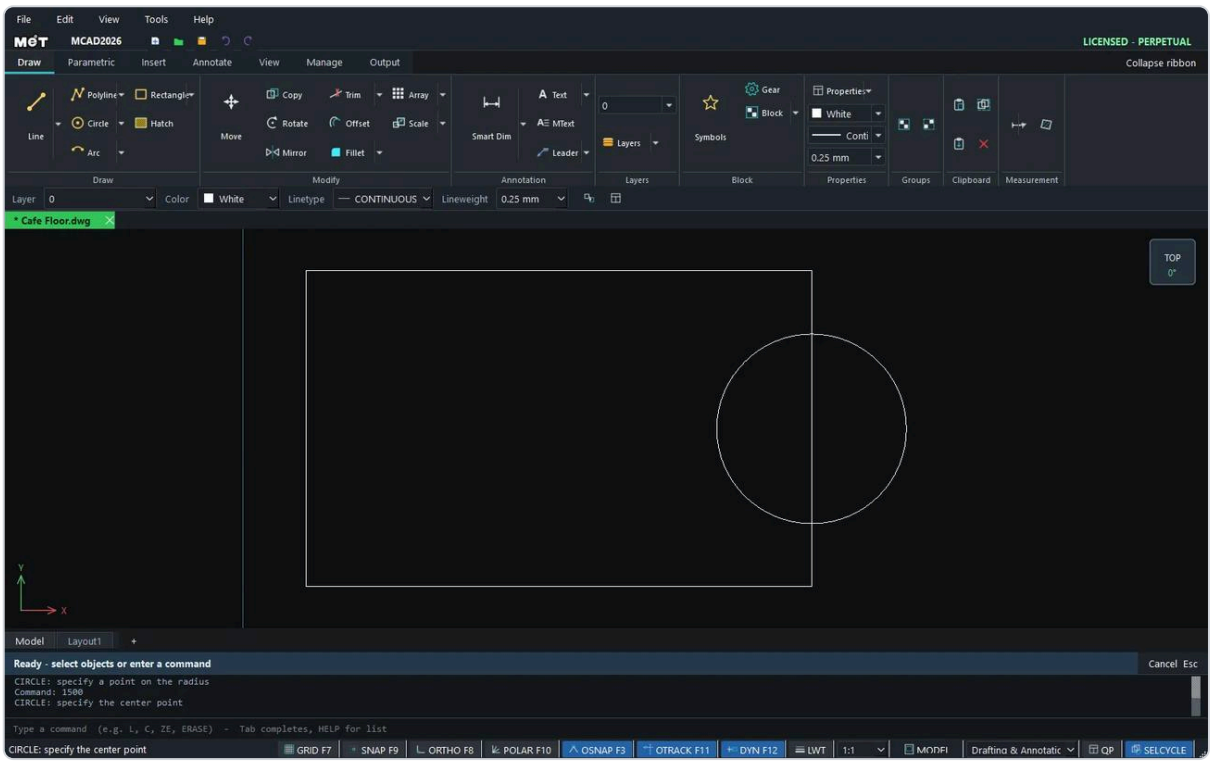
Command	Alias	Meaning
BOUNDARY	`BO`	Draws a closed polyline around the selected closed shapes (several shapes are merged into one outline).
REGION	`REG`	Converts closed shapes into a region (inner loops become holes).
UNION	`UNI`	Merges overlapping closed shapes into one outline.
SUBTRACT	`SU`	Subtracts the other shapes from the first shape.
INTERSECT	`IN`	Keeps only the overlapping area.

Where to find it

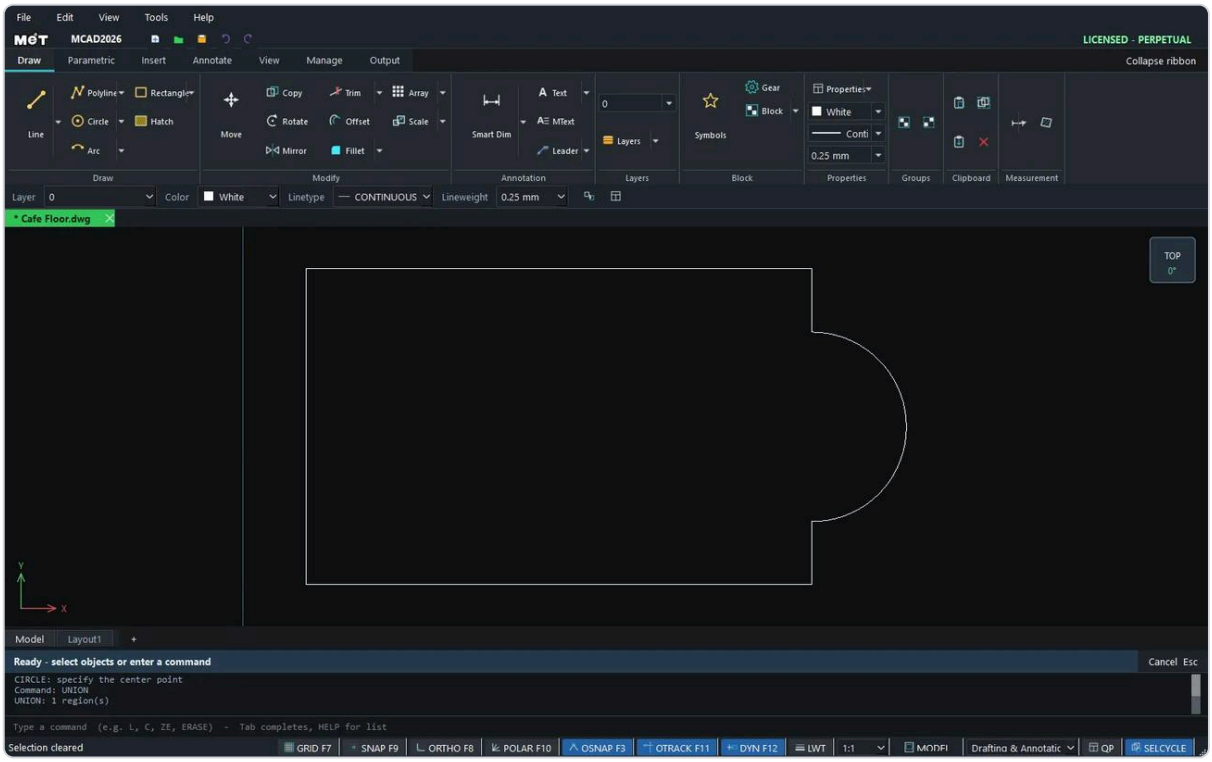
- Ribbon: **Draw › Modify › Region** ▾ (Region, Union, Subtract, Intersect) and **Draw › Polyline** ▾ **› Boundary from a Pick (BOUNDARY)**
- Command line: see the table above

Steps

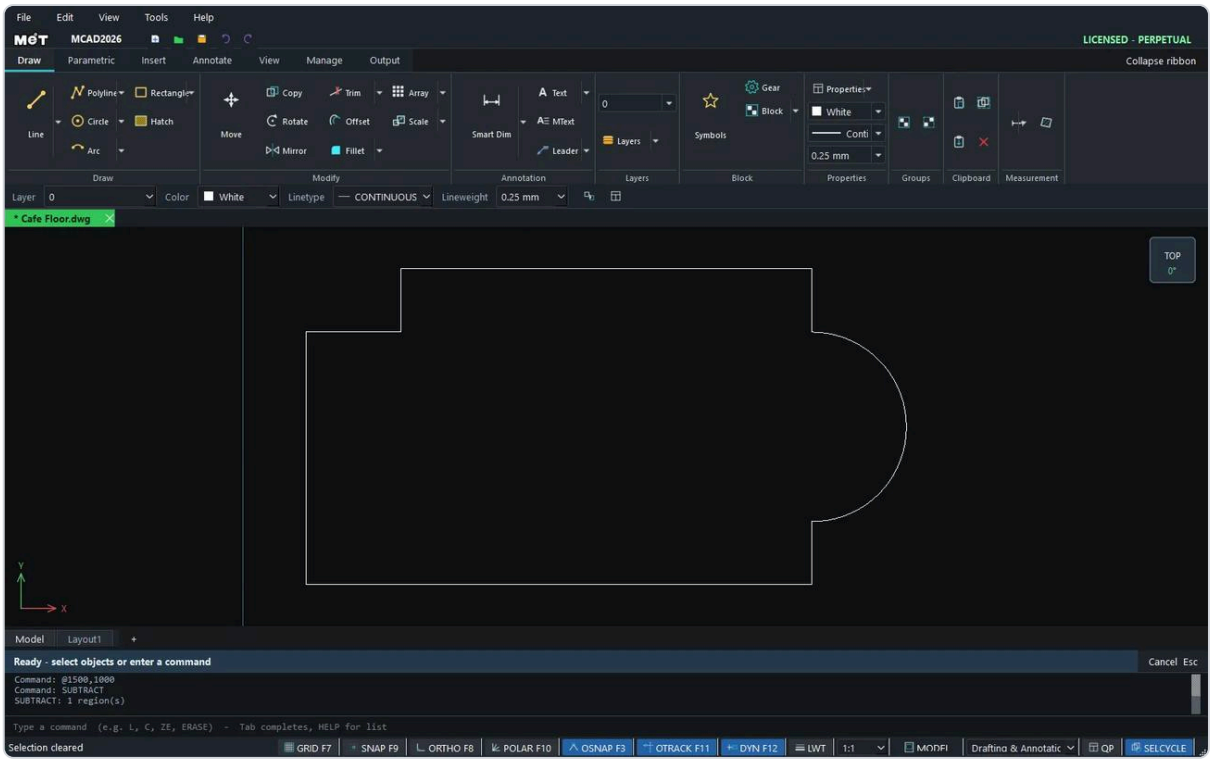
1. Draw overlapping closed shapes, for example a rectangle and a circle.
2. Select both shapes. For SUBTRACT, select the main shape first.
3. Type `UNION` (or `SUBTRACT`, `INTERSECT`) and press Enter.



Starting shapes: a rectangle and a circle that overlap



Result of UNION



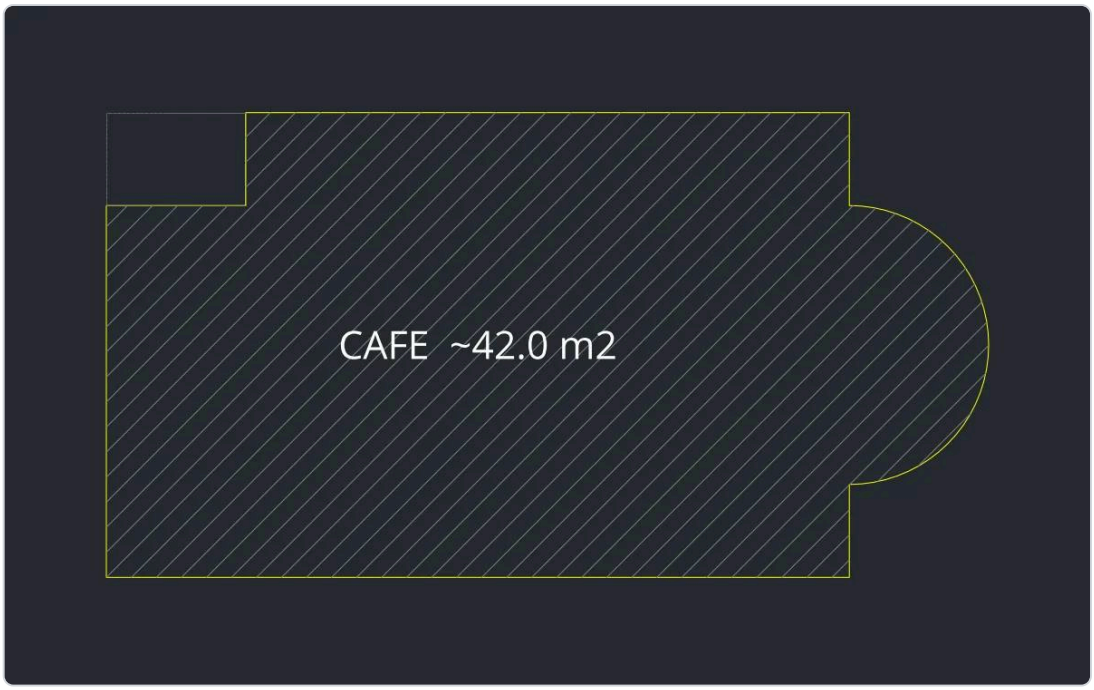
Result of SUBTRACT

> **Note:** A Boolean operation needs at least two overlapping closed shapes (rectangle, circle or closed polyline). Open lines are ignored.

Try it: Café floor with a round bay window

A small café with a curved bay window you can sit in, and a storeroom corner cut away. You use UNION and SUBTRACT, then measure the real floor area with AREA.

1. Type `REC`, Enter, `0,0`, Enter, then `8000` `Tab` `5000` Enter (an 8 x 5 m floor).
2. Type `C`, Enter, `8000,2500`, Enter, then radius `1500` and Enter (the bay window).
3. Select the rectangle and the circle, type `UNION` and press Enter. You now have one outline.
4. Draw the storeroom in the top-left corner: `REC`, `0,4000`, then `1500` `Tab` `1000` Enter.
5. Click the café outline first, then the storeroom. Type `SUBTRACT` and press Enter. The corner is cut away.
6. Select the result, type `AREA` and press Enter. It reports about `42,029,800` sq units, roughly 42 m². (The half circle is split into short straight segments, so the number is a little below the theoretical value.)



Café floor after UNION and SUBTRACT

Challenge: Try `INTERSECT` on a fresh rectangle and circle. Only the overlapping half-round remains, which makes a nice semicircular planter.

See also

- HATCH
- Measure area with AREA

Gear - Spur gear generator

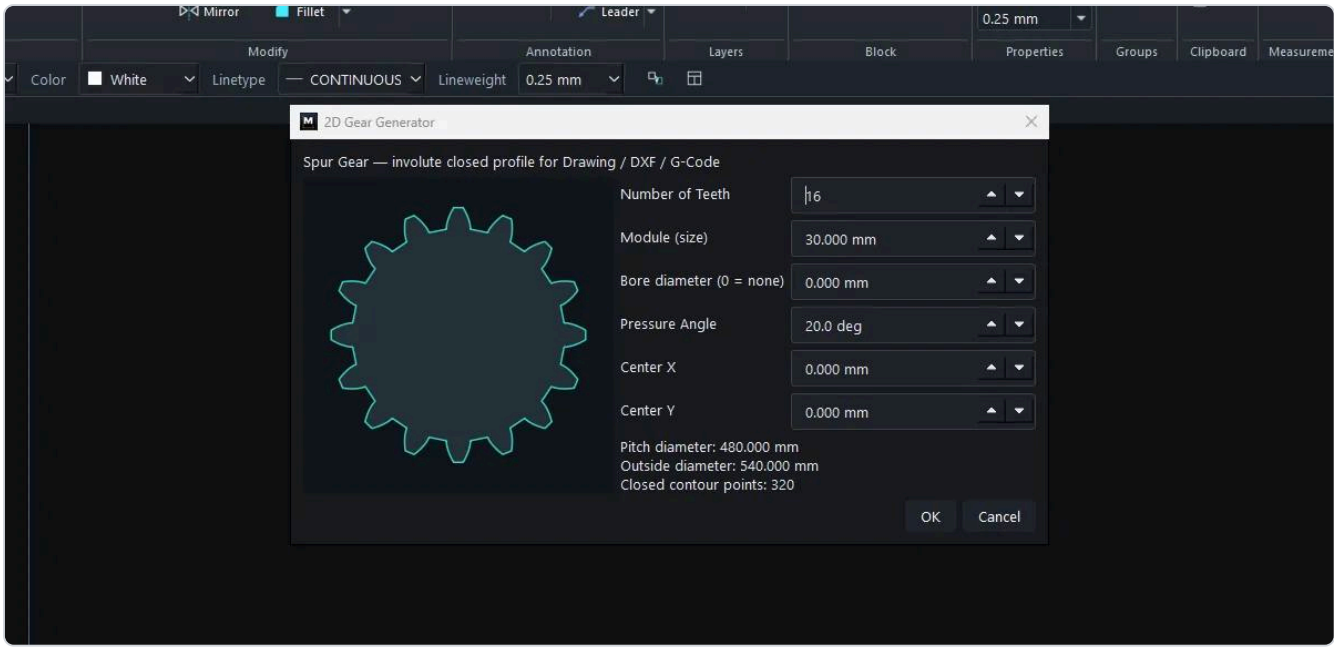
Create a closed involute spur gear profile, ready to use for DXF or G-code.

What it does

The Gear Generator creates an involute spur gear as a single closed shape. You can cut it directly with a laser or CNC machine.

Where to find it

- Ribbon: **Draw › Block › Gear**



2D Gear Generator dialog (Spur Gear)

Steps

1. Click **Draw › Block › Gear**.
2. Enter the values. The preview and the calculated diameters update as you type.
3. Click **OK**. The gear is placed at the Center X/Y you set.

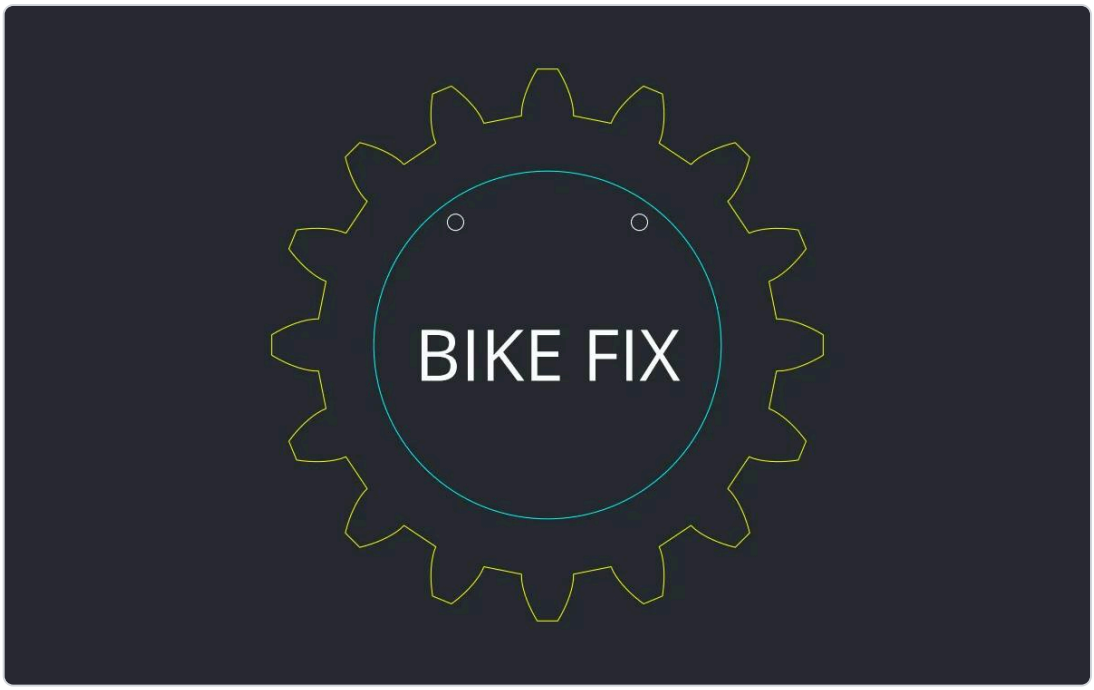
Options

Option	Meaning
Number of Teeth	The number of teeth.
Module (size)	The module in mm. It sets the tooth size.
Bore diameter (0 = none)	The shaft hole. 0 = no hole.
Pressure Angle	The pressure angle (normally 20°).
Center X / Center Y	The position of the gear centre.
Pitch diameter / Outside diameter	Calculated values (Pitch = Module × number of teeth).
Closed contour points	The number of points in the closed contour.

Try it: Gear-shaped sign for a bike workshop

A hanging shop sign shaped like a gear, about half a metre across. Because the gear is one closed outline, you can send it straight to a laser cutter or CNC router.

1. Click **Draw › Block › Gear**.
2. Enter Number of Teeth `16`, Module `30`, Bore diameter `0`, Pressure Angle `20`, Center X `0` and Center Y `0`, then click **OK**. The pitch diameter is 480 mm and the outside diameter 540 mm.
3. Inner ring: `C` `0,0` with radius `170`.
4. Two holes for the hanging chain: `C` `-90,120` radius `8` and `C` `90,120` radius `8`.
5. Type `TEXT`, set the height to `50`, click in the middle of the ring and type `BIKE FIX`.



16-tooth gear sign with the text BIKE FIX

Challenge: Set Bore diameter to `200` instead of drawing a circle. You get an open centre for an acrylic logo insert.

See also

- GCODE
- NEST

Chapter 3 quiz: Draw

Choose the one best answer. Answers are in Appendix A

1. **What does the Line - From Midpoint option do?**

   A. You click the midpoint first, then one end, and the line extends equally on both sides
   B. It draws a line from the midpoint of an existing line to a point you click
   C. It draws a line with the DASHED linetype
   D. It draws an infinite construction line

2. **In this version, how many sides does POLYGON draw?**

   A. You type the number of sides each time
   B. 4 sides
   C. Always 6 sides (a hexagon)
   D. 8 sides

3. **While drawing RECTANG, you type the width in the Smart Input box. Which key moves to the height (H) box?**

   A. Enter
   B. Tab
   C. Space
   D. Shift

4. **In the command `HATCH ANSI31 8 0`, what do the three values mean, in order?**

   A. Layer, colour, linetype
   B. Scale, angle, pattern
   C. Pattern, angle, scale
   D. Pattern, scale, angle

5. **A hatch covers the boundary lines and text. Which command sends all hatches behind the lines?**

   A. HATCHTOBACK
   B. HATCHEDIT
   C. GRADIENT
   D. BOUNDARY

6. **What is the difference between XLINE and RAY?**

   A. XLINE is a dashed line and RAY is a solid line
   B. XLINE is printed and RAY is not
   C. XLINE extends in both directions; RAY extends from a start point in one direction
   D. There is no difference; RAY is an alias of XLINE

CHAPTER 4

Modify

1. Selecting Objects (SELECT, QSELECT, FILTER)
   Select objects by clicking, with Window/Crossing boxes, QSELECT, FILTER, and SELECTSIMILAR, and hide or isolate objects.
2. ERASE, OOPS, UNDO and REDO
   Delete objects (ERASE), restore the last deleted objects (OOPS) and undo or redo (UNDO/REDO).
3. MOVE and COPY
   Move (MOVE) and copy (COPY) objects from a base point to a destination point. COPY can place multiple copies in one run.
4. ROTATE
   Rotate objects about a base point by dragging or typing an angle.
5. SCALE and SCALEREF - Resize Objects
   Enlarge or reduce objects by a scale factor (SCALE) or against a known real distance (SCALEREF).
6. MIRROR
   Mirror the selected objects across an axis line.
7. OFFSET - Parallel Copies
   Create parallel lines or shapes at a fixed distance from the original, on one side or both.
8. TRIM, EXTEND and EXTRIM
   Cut lines at intersections (TRIM), extend lines to an edge (EXTEND), and trim everything at a closed boundary (EXTRIM).
9. FILLET and CHAMFER - Corner rounding and bevelling
   Round a corner (FILLET) or bevel it (CHAMFER) between two lines.
10. STRETCH and LENGTHEN
    Stretch or shrink part of a shape with a Crossing window (STRETCH) and change the length of a line end (LENGTHEN).
11. BREAK, JOIN and PEDIT
    Cut a gap in a line (BREAK), join lines that line up (JOIN) and merge them into a polyline (PEDIT).
12. ALIGN
    Move and rotate objects using two pairs of source and destination points.
13. ARRAY - Rectangular / Polar / Path
    Copy objects in rows (ARRAY/ARRAYRECT), around a point (ARRAYPOLAR) and along a path (ARRAYPATH, MEASURE, DIVIDE).
14. EXPLODE and XPLODE
    Break blocks, tables, multileaders and splines into basic objects (EXPLODE/XPLODE).
15. DRAWORDER, OVERKILL, REVERSE and FLATTEN

Set the stacking order (DRAWORDER), delete duplicate objects (OVERKILL), reverse line direction (REVERSE) and set Z to zero (FLATTEN).

16. Clipboard: COPYCLIP, CUTCLIP, PASTECLIP, COPYBASE

    Copy, cut and paste through the Clipboard between tabs or programs, with a base point, and use the right-click menu.

17. GROUP, UNGROUP and GROUPEDIT

    Group objects (GROUP), ungroup them (UNGROUP) and add or remove members (GROUPEDIT).

18. Editing with grips and Quick Properties

    Reshape objects by dragging grips and edit properties from Quick Properties.

19. Geometric Constraints (Parametric)

    Add geometric constraints (horizontal, vertical, parallel, perpendicular, tangent and more) to a parametric 2D drawing.

20. Dimensional Constraints (DCLINEAR, DCRADIUS ...)

    Add constraints that hold a value (distance, radius, diameter, angle) and convert dimensions into constraints.

Selecting Objects (SELECT, QSELECT, FILTER)

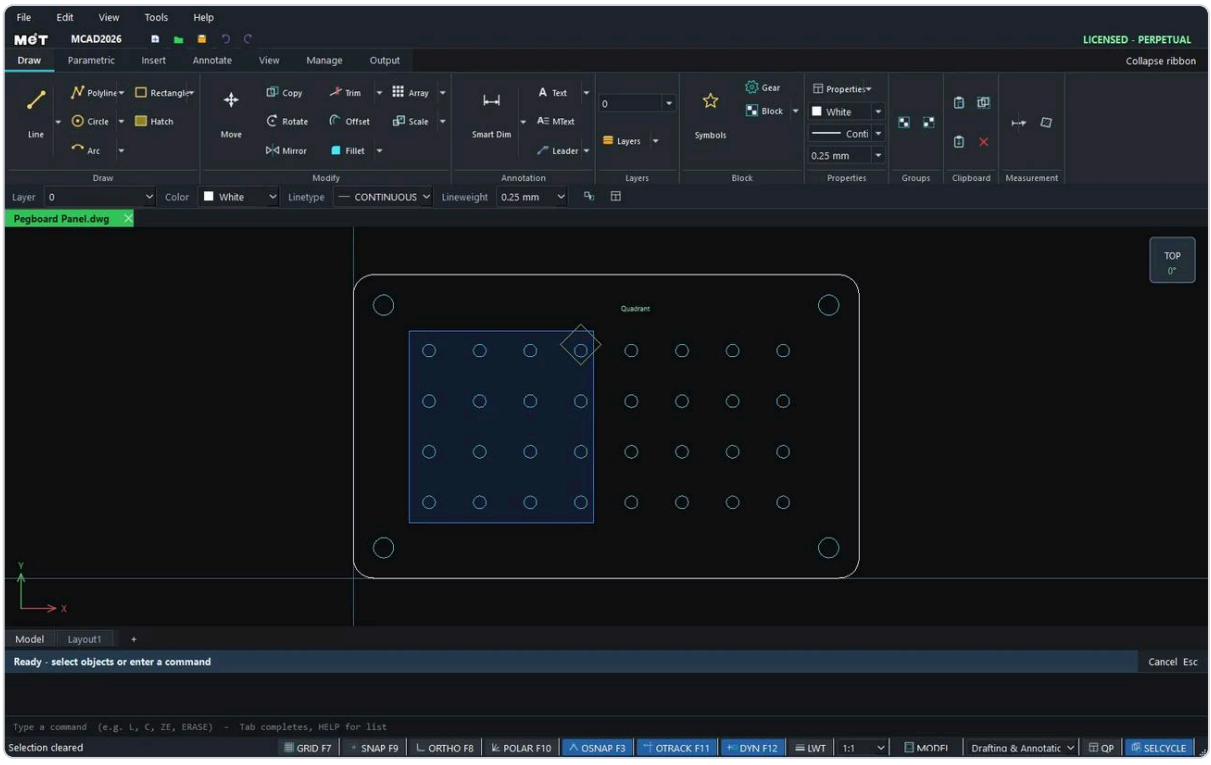
Select objects by clicking, with Window/Crossing boxes, QSELECT, FILTER, and SELECTSIMILAR, and hide or isolate objects.

What it does

Most Modify commands work in two ways: **select first, then run the command** (Pickfirst), or **run the command first, then select** (the program asks `select objects - Enter to continue, Esc to cancel`; select, then press Enter).

Selecting with the mouse

Method	Result
Click an object	Selects it. A click on another object replaces the selection; `Ctrl`+click adds or removes objects
Drag a box from left to right (Window, blue)	Selects only objects that are completely inside the box
Drag a box from right to left (Crossing)	Selects objects that cross or are inside the box
Click again at the same spot (SELCYCLE on)	Cycles through overlapping objects
`Esc`	Cancels the selection
`Ctrl+A` or right-click empty space › Select All	Selects everything



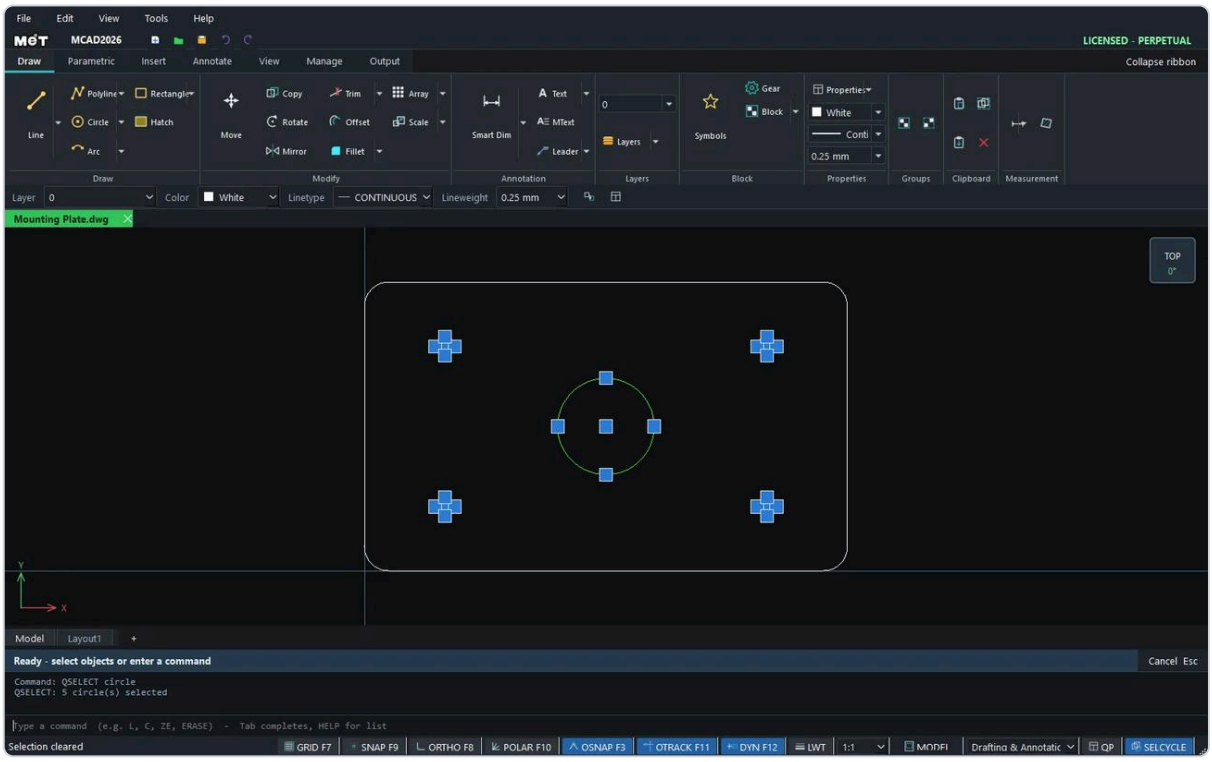
Dragging a Window box to select objects

Selection commands

Command	Alias	Format	Meaning
SELECT	`SE`	–	Returns to selection mode
QSELECT	`QSE`	`QSELECT <type> [layer]` / `QSELECT ?`	Selects every object of one type (for example circle, line, text). You can limit it to a Layer
FILTER	`FI`	`FILTER <prop>=<value> ...` / `FILTER ?`	Selects by any property, for example `FILTER kind=circle r=6.0` (multiple conditions are combined with AND; a value must match exactly as stored, so type `6.0`, not `6`)
SELECTSIMILAR	`SELSIM`	–	Extends the selection to all objects of the same type on the same Layer
SELECTPREVIOUS	`SELP`	–	Recalls the previous selection set
SELECTLAST	`SELL`	–	Selects the most recently created object
ISOLATEOBJECTS	`ISOLATE`	–	Hides everything except the selection
HIDEOBJECTS	`HIDEOBJ`	–	Hides the selected objects
UNISOLATEOBJECTS	`UNISOLATE`	–	Shows objects hidden by the two commands above

Example: select all circles

1. Type `QSELECT circle` and press Enter.
2. The result is `QSELECT: 5 circle(s) selected`. You can run a command right away, for example `CENTERMARK`.

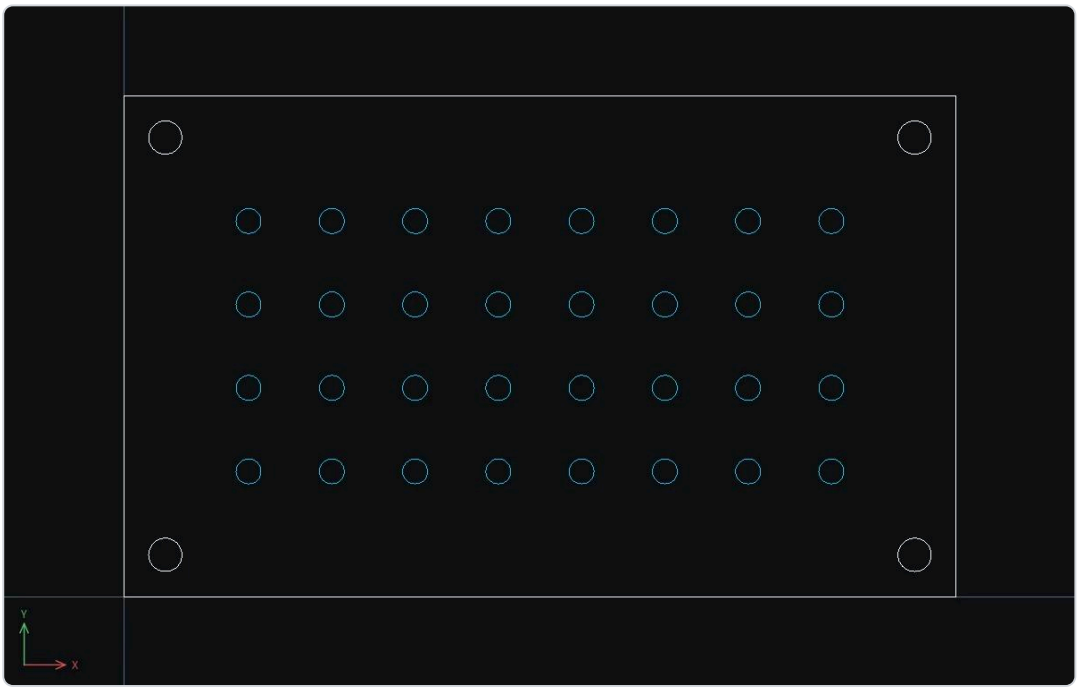


QSELECT selecting all 5 circles

Try it: Pegboard panel

A 200 x 120 mm pegboard for tools or craft supplies, with 32 peg holes and 4 mounting holes. You will select only the peg holes and move them to their own layer, even though every hole is a circle.

1. Board: `REC`, `0,0`, `@200,120`, Esc. Peg hole: `C`, `30,30`, `3`, Esc. Mounting holes: `C` `10,10` `4`, `C` `190,10` `4`, `C` `10,110` `4`, `C` `190,110` `4`. Press Esc after each one.
2. Click the peg hole and type `ARRAYRECT 8 20 0`. Drag a window around the 8 peg holes (from 20,20 to 180,40) and type `ARRAYRECT 4 0 20`. You now have 32 peg holes.
3. Type `QSELECT circle`. The result reads `QSELECT: 36 circle(s) selected`, which includes the mounting holes.
4. `Ctrl`+click each of the 4 mounting holes on its circle line, away from the blue grips. That removes them, so 32 objects stay selected.
5. Type `CHPROP layer=PEGS color=#00C8FF`. The peg holes turn cyan on the new PEGS layer.



Pegboard with 32 peg holes on the PEGS layer

Challenge: Click one mounting hole and type `SELSIM`. Exactly the 4 mounting holes are selected, because they are now the only circles left on layer 0.

See also

- Editing with Grips
- GROUP
- Layer Tools

ERASE, OOPS, UNDO and REDO

Delete objects (ERASE), restore the last deleted objects (OOPS) and undo or redo (UNDO/REDO).

What it does

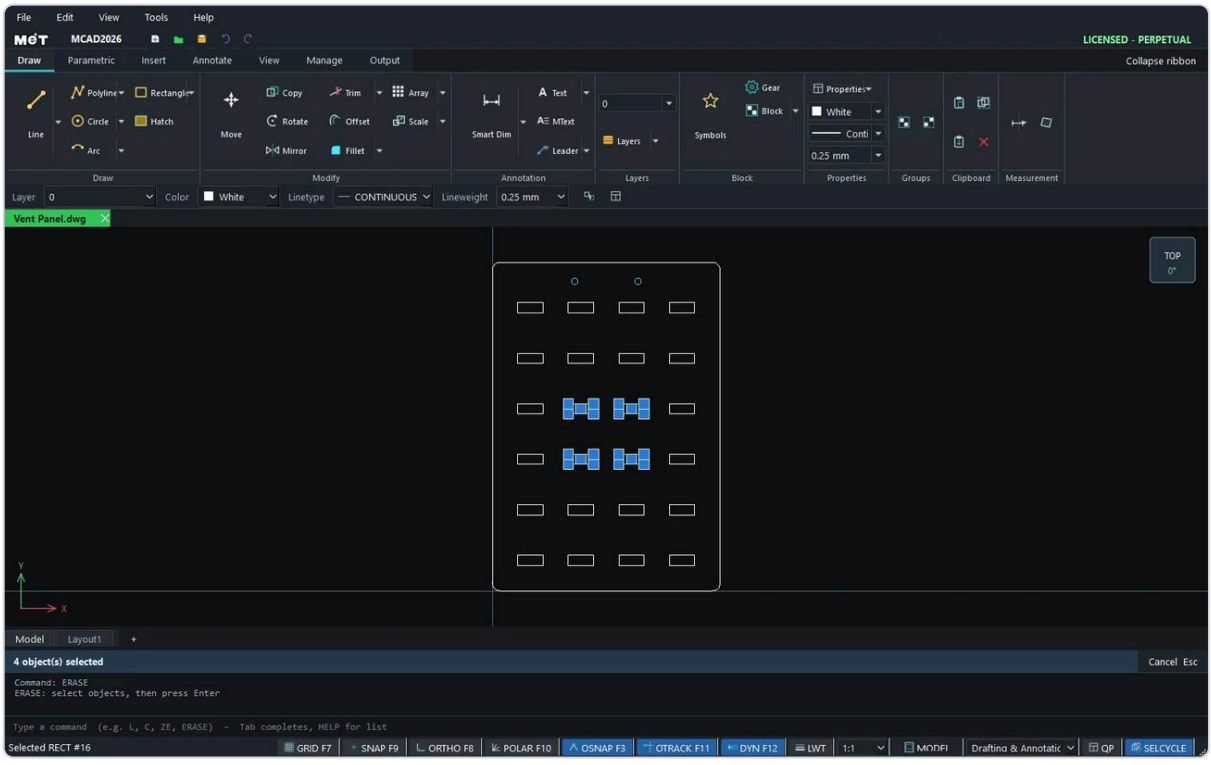
Command	Alias	Shortcut	Meaning
ERASE	`E`, `DELETE`, `DEL`	`Delete`	Deletes the selected objects.
OOPS	–	–	Restores the objects deleted by the last ERASE, without cancelling what you drew afterwards.
UNDO	`U`	`Ctrl+Z`	Undoes edits one step at a time.
REDO	–	`Ctrl+Y`	Redoes what you undid.

Where to find it

- Ribbon: **Draw › Clipboard › Delete**
- Quick Access Toolbar: Undo / Redo
- Menu: **Edit › Undo / Redo**

Steps for ERASE

1. Type `E` and press Enter. The prompt reads `ERASE: select objects - Enter to continue, Esc to cancel`.
2. Click the objects to delete. Selected objects show blue grips.
3. Press Enter.



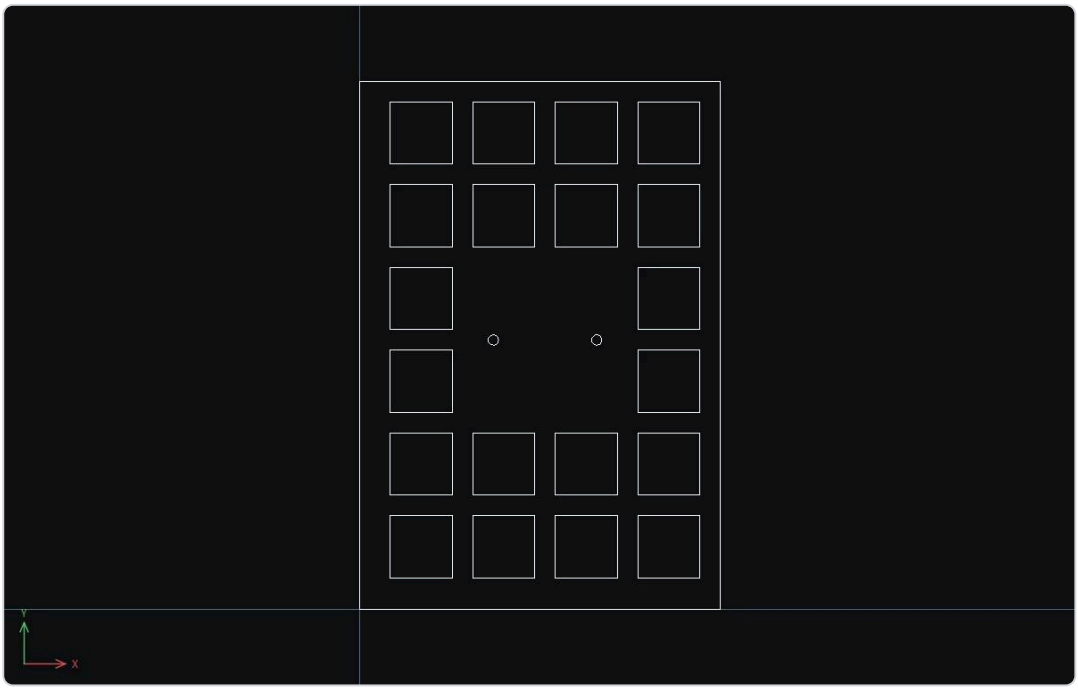
ERASE while selecting objects

> **Note:** Right-click an object (when nothing is selected) › **Delete (object under cursor)** to delete that object immediately.

Try it: Garden trellis panel with a name-plate area

A 175 x 255 mm plywood trellis for a climbing plant, with a solid patch in the middle for a plant name plate. Build a full grid of openings, then ERASE the four in the middle.

1. Frame: `REC`, `0,0`, `@175,255`, Esc. First opening: `REC`, `15,15`, `@30,30`, Esc.
2. Click the opening and type `ARRAYRECT 4 40 0`. You get a row of 4.
3. Drag a window around the 4 openings and type `ARRAYRECT 6 0 40`. You now have 24 openings.
4. Drag a window from about 50,90 to 130,170 so it holds the four middle openings (rows 3 and 4, columns 2 and 3). Type `E` and press Enter. The result reads `Erased`.
5. Screw holes for the name plate: `C`, `65,130`, `2.5`, Esc, and `C`, `115,130`, `2.5`, Esc.



Trellis panel with 20 openings and a solid centre for the name plate

Challenge: Erase one more opening by mistake, then type `OOPS`. It comes back, and your screw holes stay where they are (UNDO would have removed them first).

See also

- Selecting objects
- Clipboard

MOVE and COPY

Move (MOVE) and copy (COPY) objects from a base point to a destination point. COPY can place multiple copies in one run.

What it does

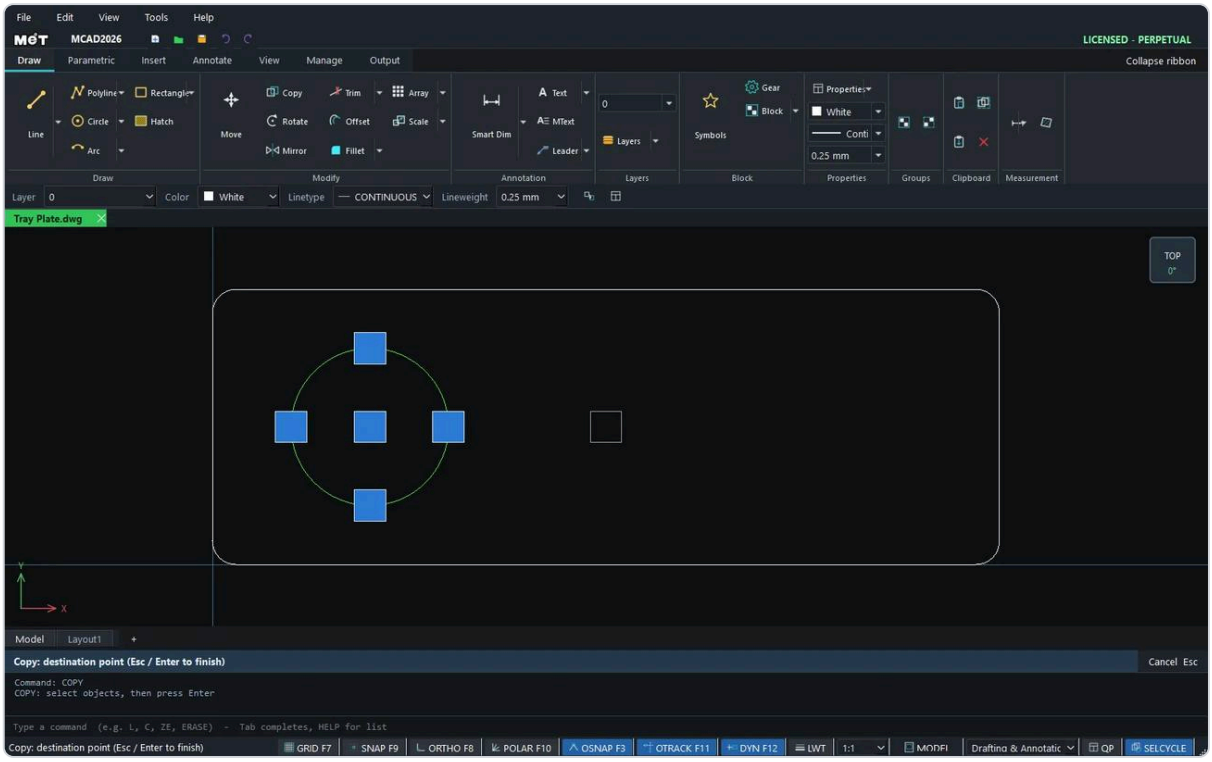
- **MOVE** – moves the selected objects.
- **COPY** – copies objects to several locations in one command.

Where to find it

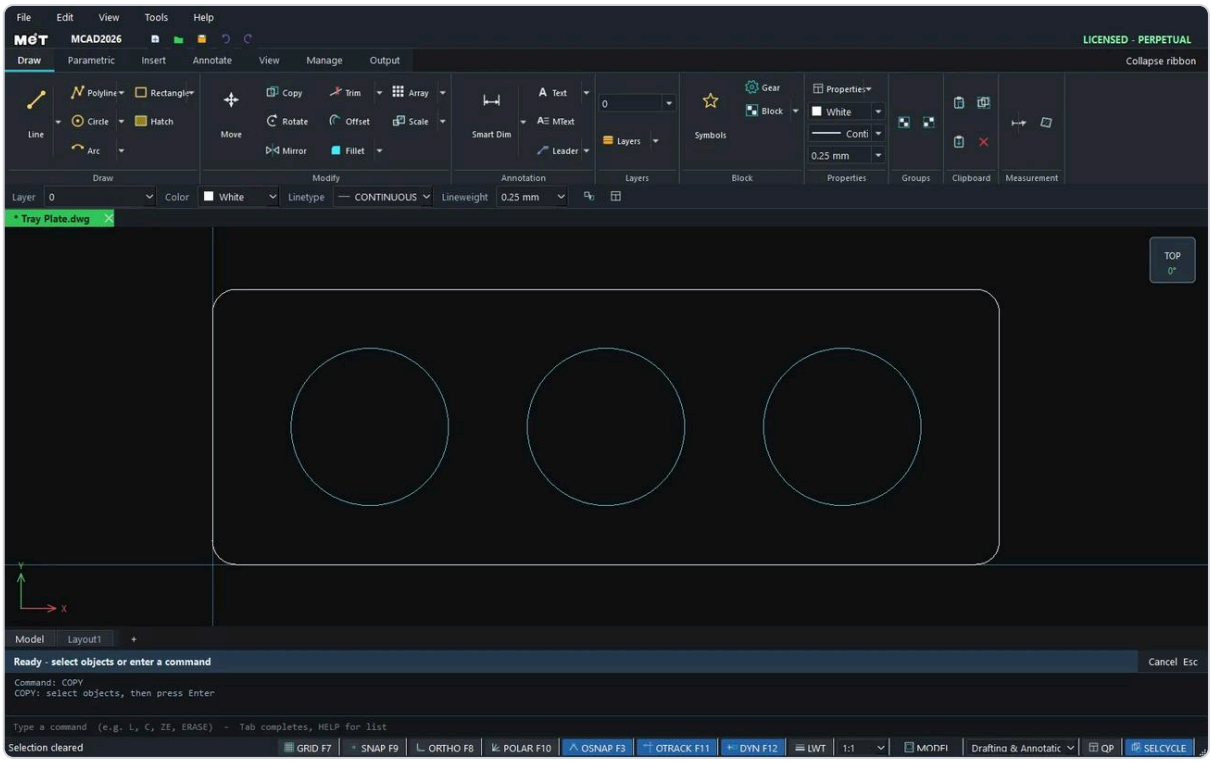
- Ribbon: **Draw › Modify › Move** / **Copy**
- Command line: `MOVE` (Alias `M`)
- Command line: `COPY` (Alias `CO`, `CP`)
- Right-click the selected objects › Move / Copy

COPY steps

1. Type `CO` and press Enter. `COPY: select objects - Enter to continue, Esc to cancel`
2. Click the objects, then press Enter.
3. `Copy: base point (snap to center/quadrant/end)` Click the base point.
4. `Copy: destination point (Esc / Enter to finish)` Click the destination point, or type a distance in Smart Input.
5. Keep clicking more locations. Press Enter or Esc to finish.



COPY, specifying the destination point



Mounting holes copied to 2 locations

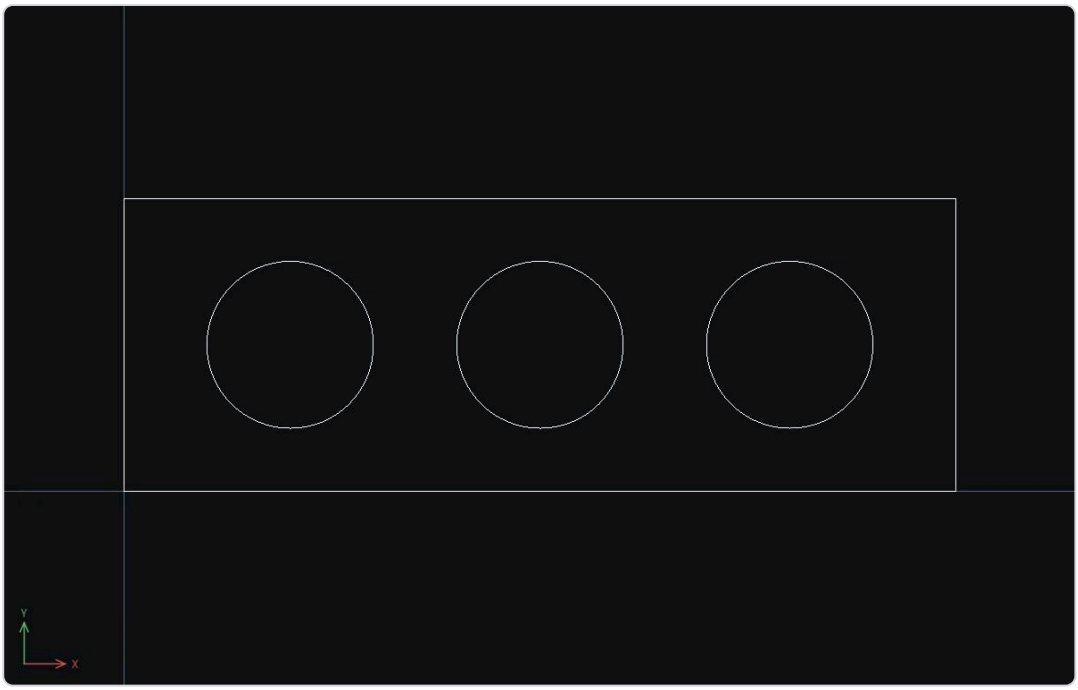
MOVE works the same way, but moves the objects once.

> **Note:** You can also drag already-selected objects with the mouse to move them directly.

Try it: Three-candle tea-light holder

A 200 x 70 mm wooden tray for three tea lights. Draw one Ø40 hole, and COPY places the other two exactly 60 mm apart.

1. Tray: `REC` , `0,0` , `@200,70` , Esc. First hole: `C` , `40,35` , `20` , Esc.
2. Type `CO` and press Enter. Click the circle and press Enter.
3. Base point: type `40,35` and press Enter. Destinations: `@60,0` , Enter, then `@120,0` , Enter. Press Enter again to finish. Each `@` distance is measured from the base point, so the holes land at 100 and 160.



Tea-light tray with three evenly spaced holes

Challenge: Make a five-candle tray: draw the tray as `@320,70` and copy to `@60,0`, `@120,0`, `@180,0` and `@240,0`.

See also

- ROTATE
- Clipboard
- ARRAY

ROTATE

Rotate objects about a base point by dragging or typing an angle.

What it does

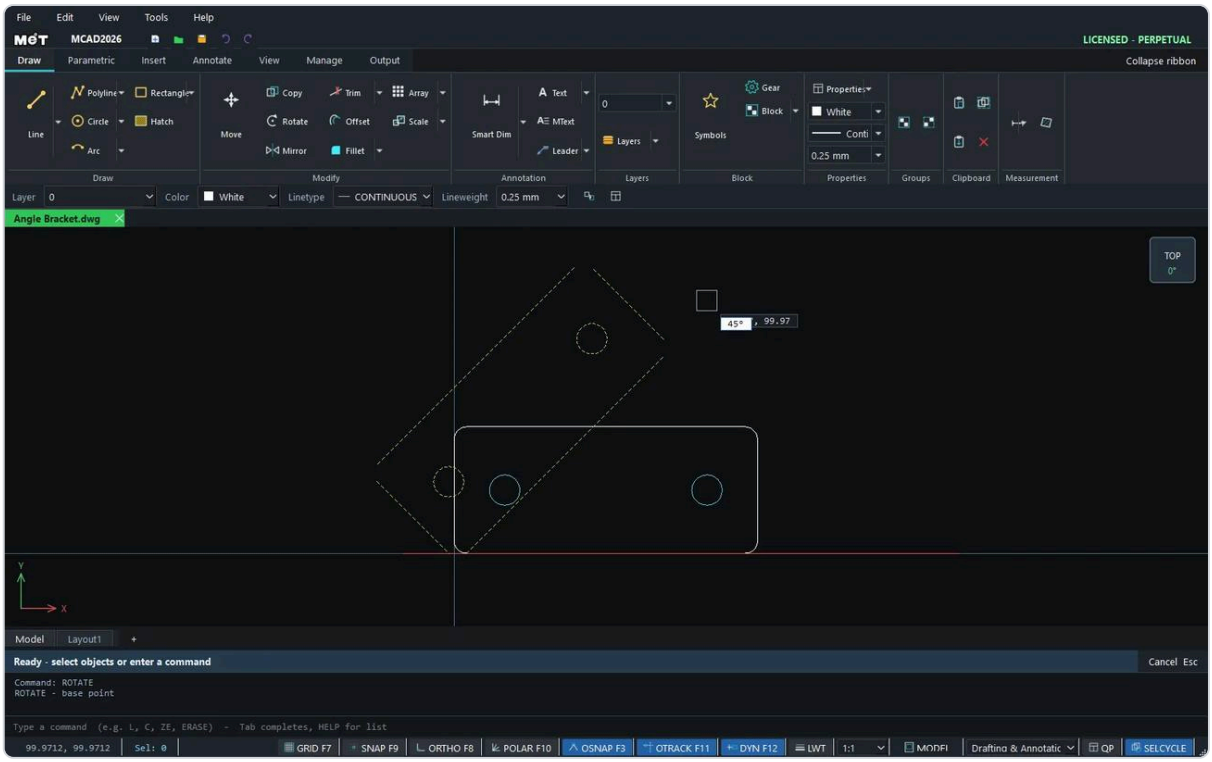
ROTATE rotates the selected objects about a base point.

Where to find it

- Ribbon: **Draw › Modify › Rotate**
- Command line: `ROTATE` (Alias `RO`)
- Right-click the selected objects › Rotate

Steps

1. Select the objects.
2. Type `RO` and press Enter (`ROTATE - base point`).
3. Click the base point.
4. Drag the mouse to rotate (the preview shows the angle) and click, or type the angle in the Smart Input box and press Enter.



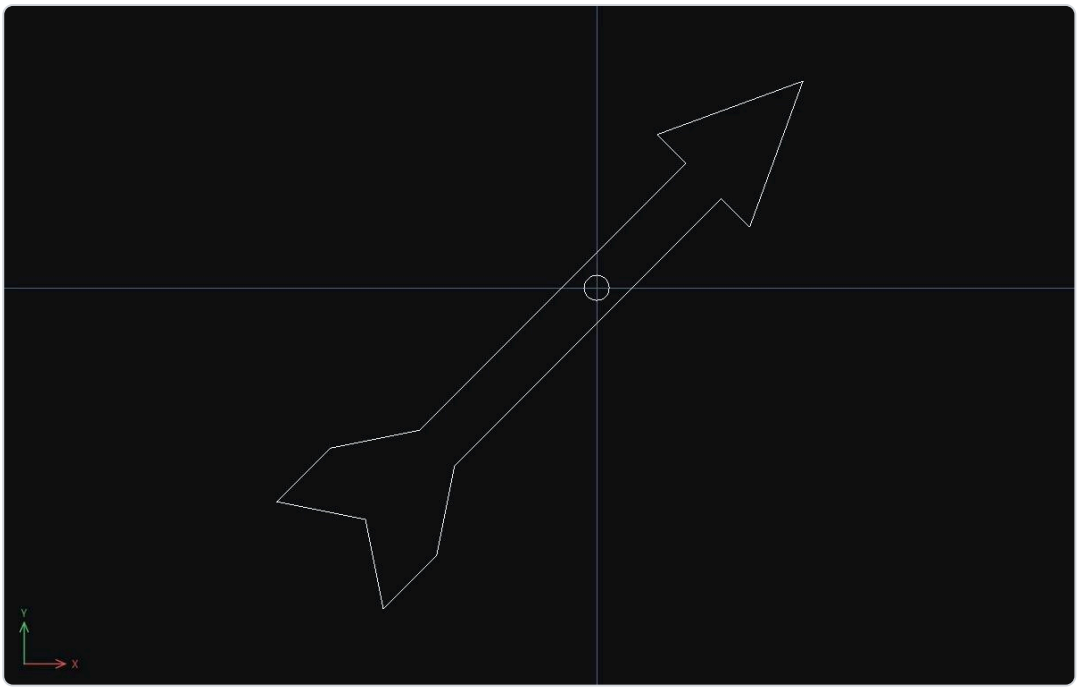
ROTATE rotating about the lower-left corner by 45°

> **Note:** A positive angle rotates counterclockwise.

Try it: Weather vane arrow

A garden weather vane arrow in 3 mm aluminium. Draw it pointing east, then ROTATE it 45° about the pivot hole to see it swing into a north-east wind.

1. Arrow: `PL`, then `-60,-15` `@-15,0` `@10,15` `@-10,15` `@15,0` `@15,-10` `@75,0` `@0,8` `@28,-13` `@-28,-13` `@0,8` `@-75,0` `-60,-15`, and press Enter.
2. Pivot hole: `C`, `0,0`, `2.5`, Esc.
3. Click the arrow, type `RO` and press Enter.
4. Click the centre of the pivot hole (the Center snap marker appears).
5. Type `45` and press Enter. The arrow now points north-east.



Weather vane arrow rotated 45° about its pivot hole

Challenge: Run `RO` again on the arrow, pick the same pivot and type `-90`. Now it points south-east.

See also

- ALIGN
- MOVE and COPY

SCALE and SCALEREF - Resize Objects

Enlarge or reduce objects by a scale factor (SCALE) or against a known real distance (SCALEREF).

What it does

- **SCALE** – enlarges or reduces objects about a base point by a scale factor.
- **SCALEREF** – scales the drawing so a measured distance equals its real distance. Good for drawings imported from PDF or images.

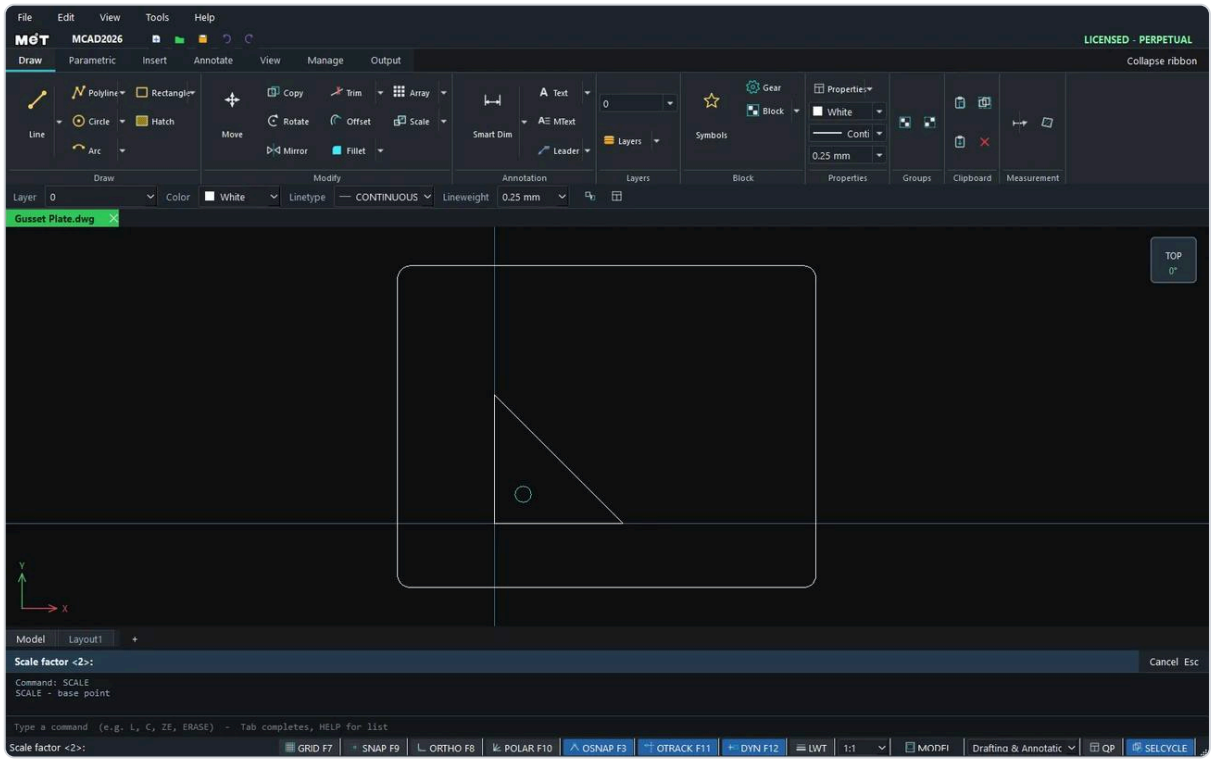
Where to find it

- Ribbon: **Draw › Modify › Scale** (submenu: Scale, Scale by Reference)
- Command line: `SCALE` (Alias `SC`)
- Command line: `SCALEREF` (Alias `SR`)

SCALE steps

1. Select the objects (or start the command first, then select and press Enter).
2. Type `SC` and press Enter. `Scale: pick base point using object snap.`
3. Click the base point.
4. `Scale factor <2>:` Type the factor (for example `0.5`) and press Enter.

Shortcut: `SCALE 2` enters the factor first. Then you only click the base point.



SCALE asking for the factor after the base point is clicked

SCALEREF steps

1. Select the whole drawing you want to adjust.
2. Type `SR` and press Enter.
3. Click the two ends of a distance whose real value you know.
4. The Scale by Reference window shows the measured value. Type the real distance and click OK.

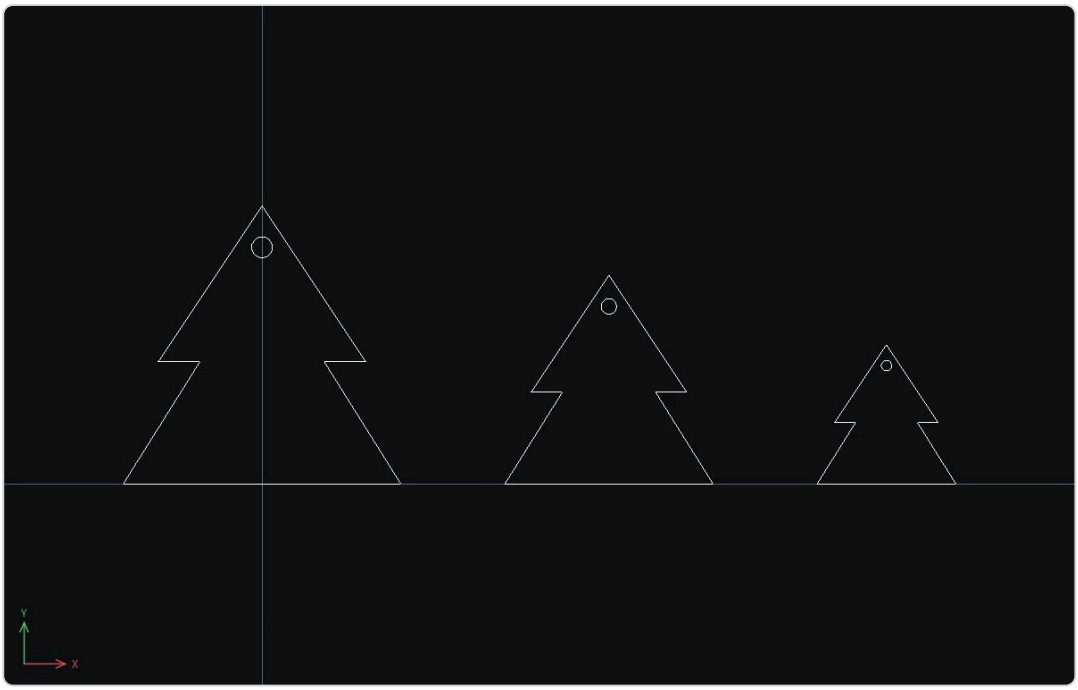
Shortcut: `SCALEREF 98.4 100` (measured value, real value).

Try it: Christmas tree ornaments in three sizes

A set of three plywood tree ornaments, 80, 60 and 40 mm wide. Draw the big one, copy it twice, and SCALE the copies down.

1. Tree: `PL`, then `-40,0` `@80,0` `@-22,35` `@12,0` `@-30,45` `@-30,-45` `@12,0` `-40,0`, and press Enter. Hanging hole: `C`, `0,68`, `3`, Esc.
2. Drag a window around the tree and its hole. Type `CO` and press Enter. Base point `0,0`, then destinations `@100,0` and `@180,0`. Press Enter.
3. Drag a window around the middle tree (from 50,-10 to 150,90). Type `SC 0.75` and press Enter, then click the midpoint of its bottom edge (Midpoint snap).
4. Drag a window around the right tree (from 130,-10 to 230,90). Type `SC 0.5` and press Enter, then click the midpoint of its bottom edge.

All three trees stand on the same line, because each one was scaled about its own base.



Three tree ornaments at 100%, 75% and 50%

Challenge: Copy the big tree once more and use `SC 1.25` for a tree-top ornament.

See also

- Measuring with DIST
- Import PDF

MIRROR

Mirror the selected objects across an axis line.

What it does

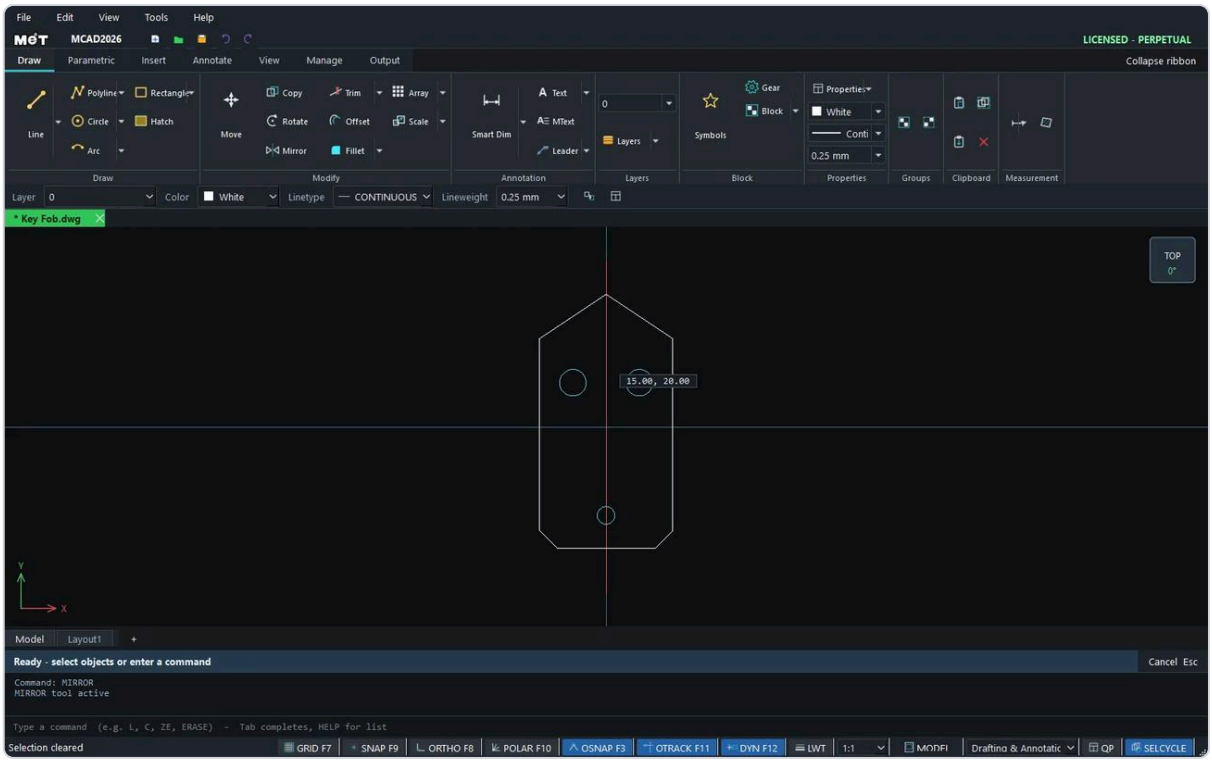
MIRROR creates a mirrored copy of the selected objects, using an existing line as the mirror axis. The original objects stay in place.

Where to find it

- Ribbon: **Draw › Modify › Mirror**
- Command line: `MIRROR` (Alias `MI`)
- Right-click the selected objects › Mirror

Steps

1. Draw the line to use as the mirror axis (if you do not have one yet).
2. Select the objects to mirror.
3. Type `MI` and press Enter.
4. Click the mirror axis line.

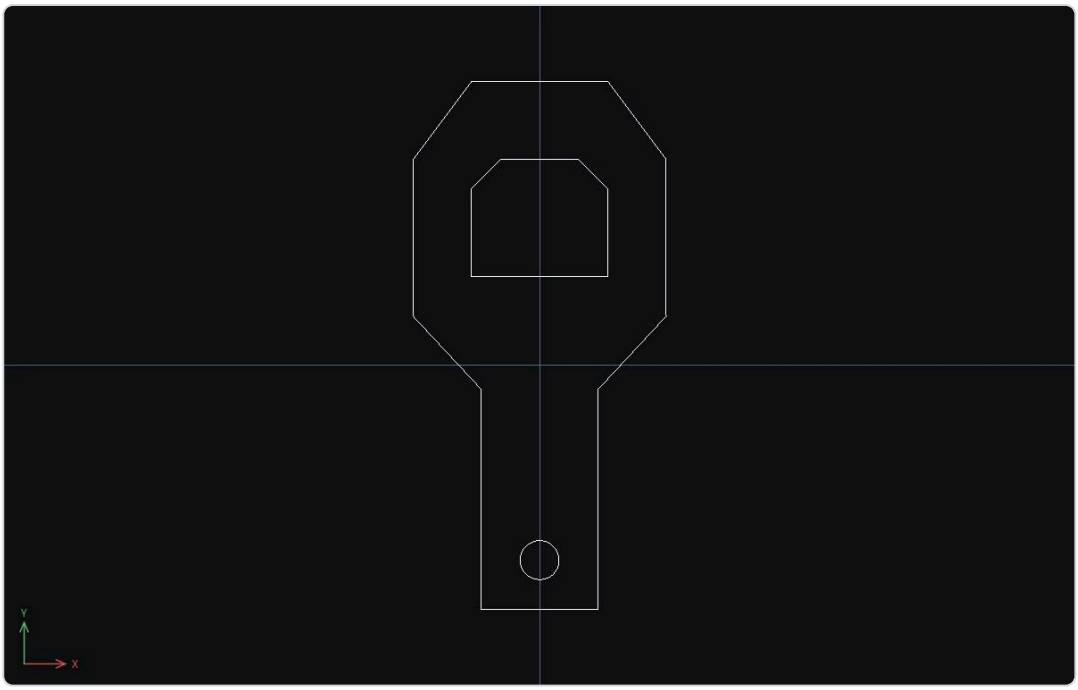


The shape on the left is mirrored across the vertical axis line

Try it: Bottle-opener keychain

A 2 mm steel keychain bottle opener. Draw only the right half, and MIRROR makes the other half, perfectly symmetric.

1. Mirror axis: `L`, `0,-60`, `0,70`, Esc.
2. Right half of the outline: `PL`, then `0,-50` `@12,0` `@0,45` `@14,15` `@0,32` `@-12,16` `@-14,0`, and press Enter.
3. Right half of the cap opening: `PL`, then `0,18` `@14,0` `@0,18` `@-6,6` `@-8,0`, and press Enter.
4. Key-ring hole on the axis: `C`, `0,-40`, `4`, Esc.
5. Drag a window from -1,-56 to 30,62. It takes the two half shapes but not the axis line or the hole.
6. Type `MI`, press Enter, and click the axis line. The left half appears.
7. Click the axis line and press Delete. It was only a helper.



Bottle-opener keychain mirrored from one half

Challenge: Change `@14,15` in step 2 to `@18,15` for a wider head and mirror again. The two sides always match.

See also

- GCSYMMETRIC
- COPY

OFFSET - Parallel Copies

Create parallel lines or shapes at a fixed distance from the original, on one side or both.

What it does

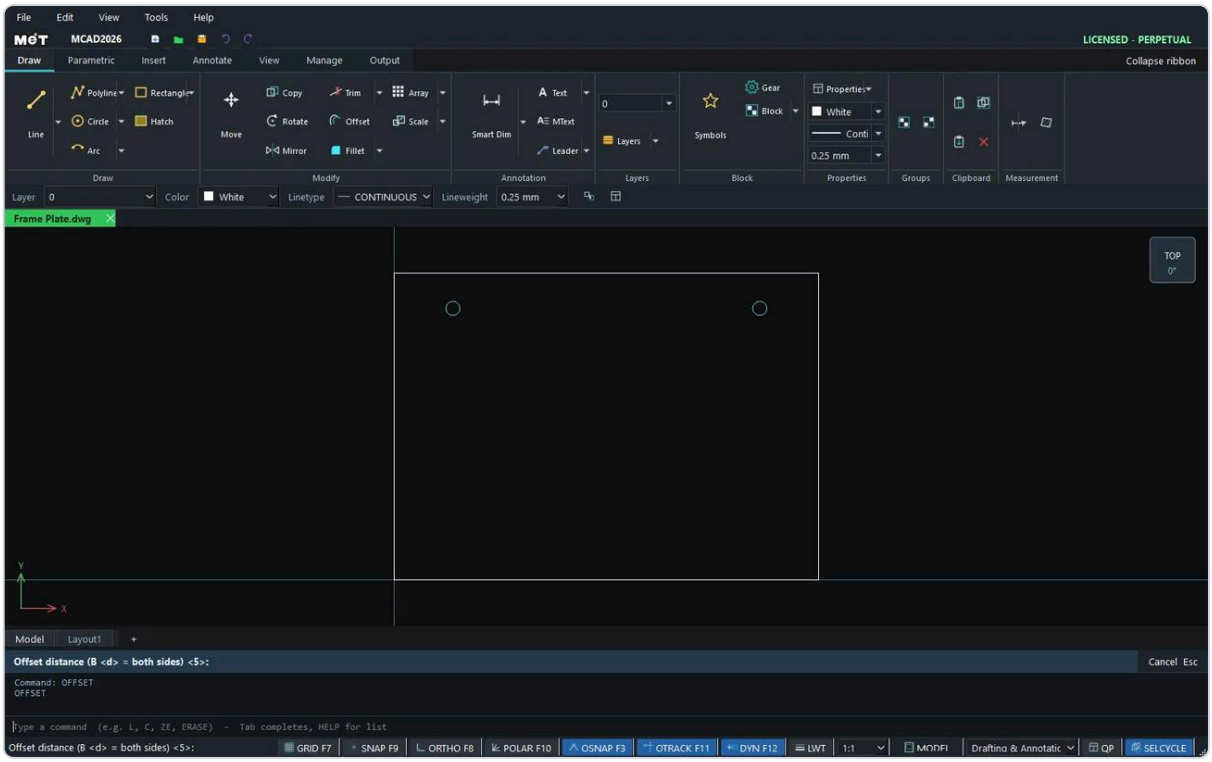
OFFSET creates a copy of a Line, Arc, Circle, or Polyline at a set distance. Polyline corners keep the same distance all the way around.

Where to find it

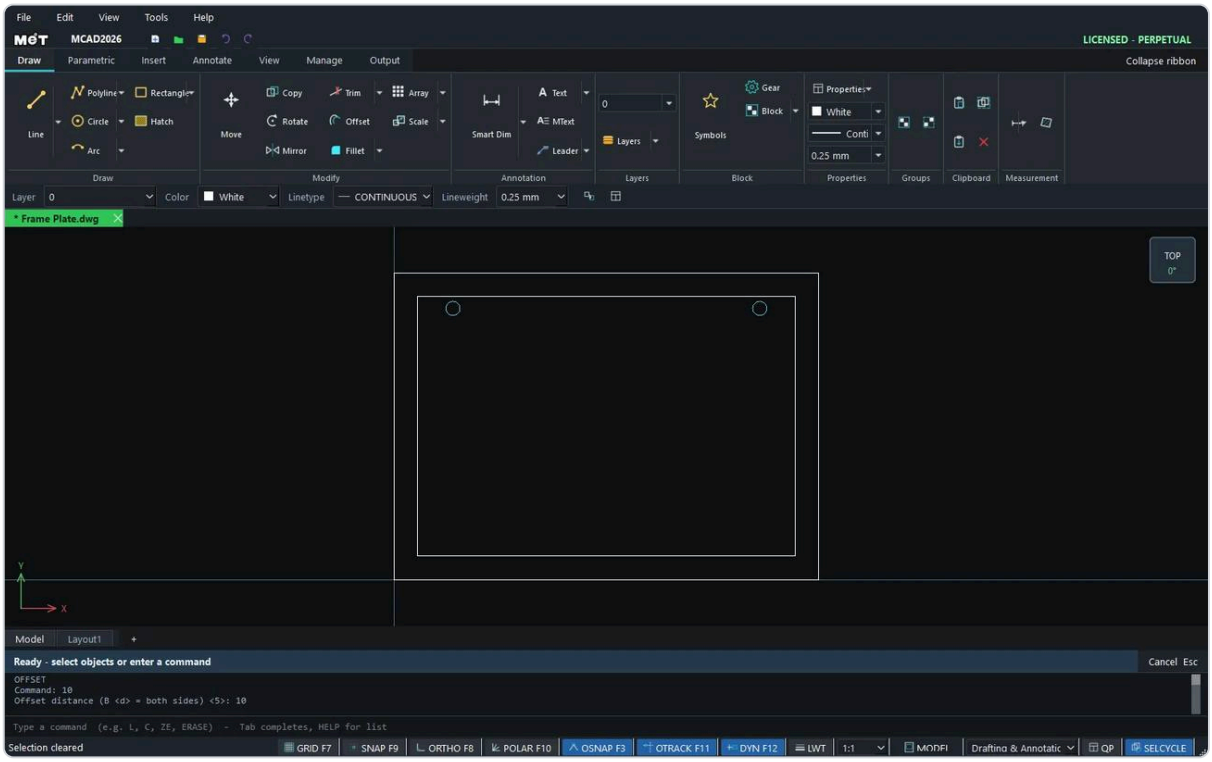
- Ribbon: **Draw › Modify › Offset**
- Command line: `OFFSET` (Alias `O`)

Steps

1. Type `O` and press Enter. `Offset distance (B <d> = both sides) <5>:`
2. Type a distance, for example `10`, and press Enter (or `B 10` to create both sides).
3. Click the object.
4. `Offset: click the side to offset toward.` Click the side you want (inside or outside).
5. Keep clicking more objects. Press Esc to finish.



OFFSET asking for the distance on the Command line



Polyline offset inward by 10 mm

Shortcut: `OFFSET 10` enters the distance first. Then select the object, press Enter, and click the side.

Option	Meaning
`<distance>`	Offset one side, toward the side you click
`B <distance>`	Both sides – creates both copies without clicking a side
Empty Enter	Use the previous distance

Try it: Acrylic photo frame for a 6 x 4 inch print

The front plate of a frame for a 152 x 102 mm (6 x 4 inch) photo. One rectangle and one OFFSET give you the window, with a 6 mm lip all round to hold the print.

1. Frame plate: `REC`, `0,0`, `@180,130`, Esc.
2. Type `O` and press Enter. Type `20` and press Enter.
3. Click the rectangle, then click anywhere inside it. A 140 x 90 window appears 20 mm inside. Press Esc.
4. Hanging holes: `C`, `25,120`, `3`, Esc, and `C`, `155,120`, `3`, Esc.

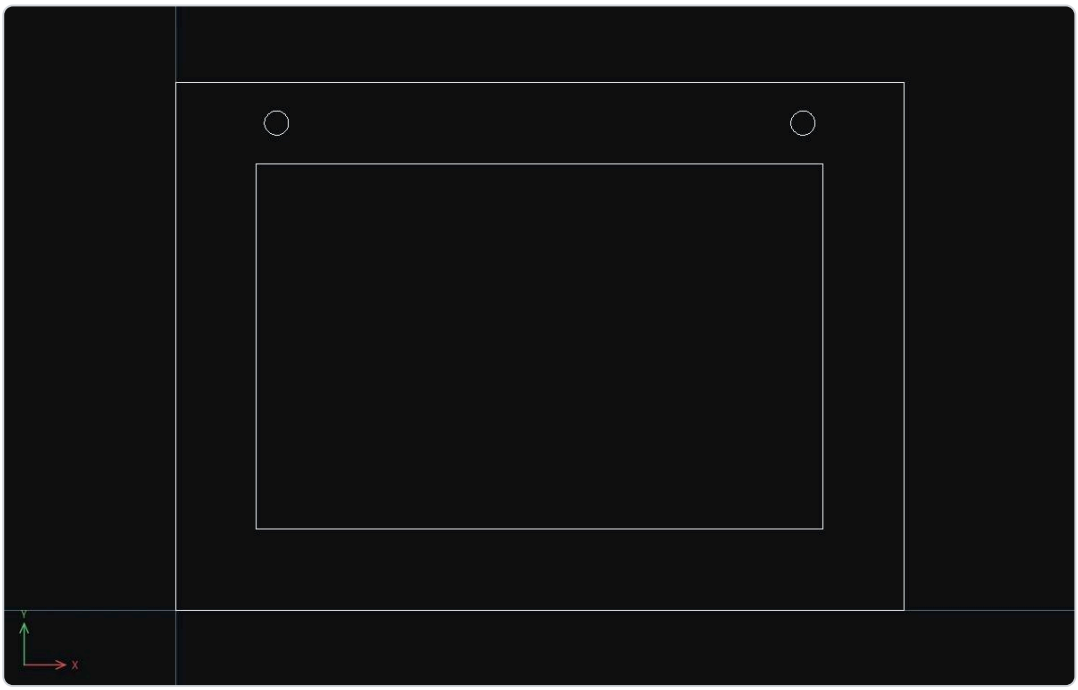


Photo frame plate with a window offset 20 mm inside

Challenge: Cut a second plate as a mat: offset the window 8 mm inwards for a smaller photo.

See also

- DLINE and MLINE
- TRIM and EXTEND

TRIM, EXTEND and EXTRIM

Cut lines at intersections (TRIM), extend lines to an edge (EXTEND), and trim everything at a closed boundary (EXTRIM).

What it does

Command	Alias	Meaning
TRIM	`TR`	Click the segment of a line you want to remove. The program cuts it automatically at the intersections with other lines
EXTEND	`EX`	Click near the end of a line. The line extends to the nearest edge

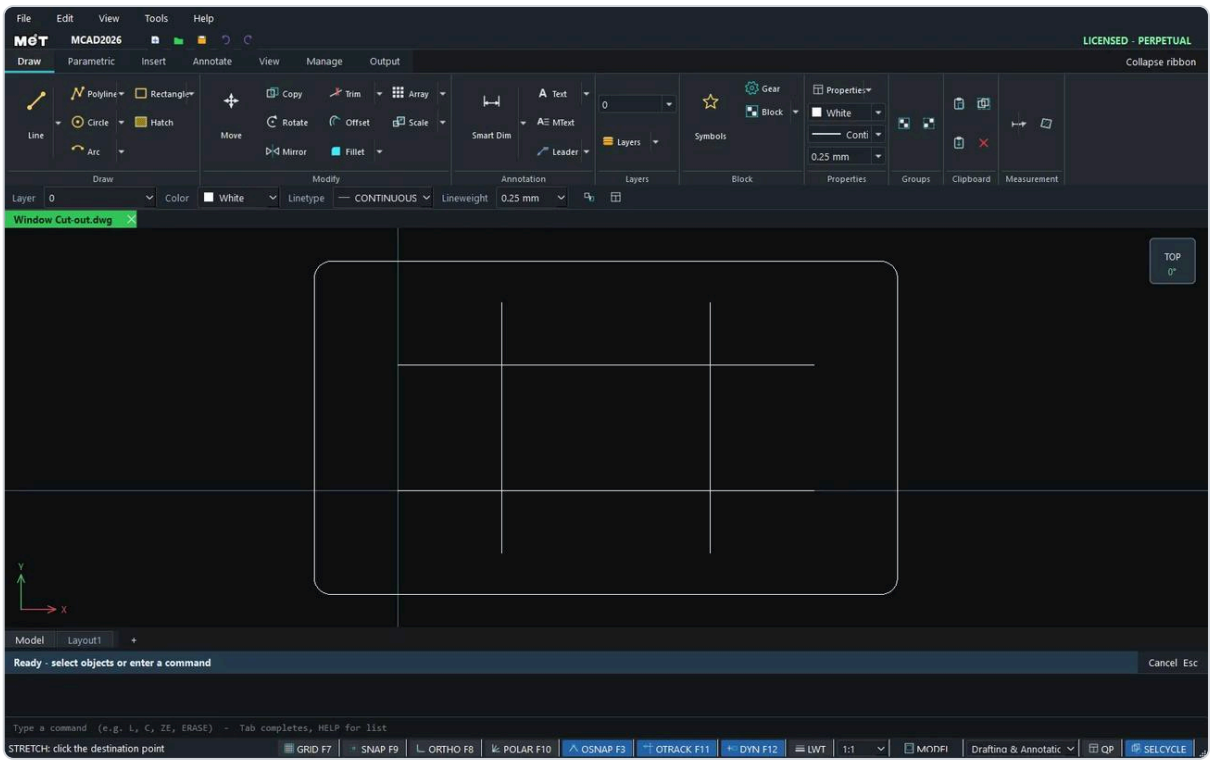
Command	Alias	Meaning
EXTRIM	–	Trims every line at the edge of a selected closed shape. `EXTRIM` = trim the inside, `EXTRIM Outside` = keep the inside and trim the outside

Where to find it

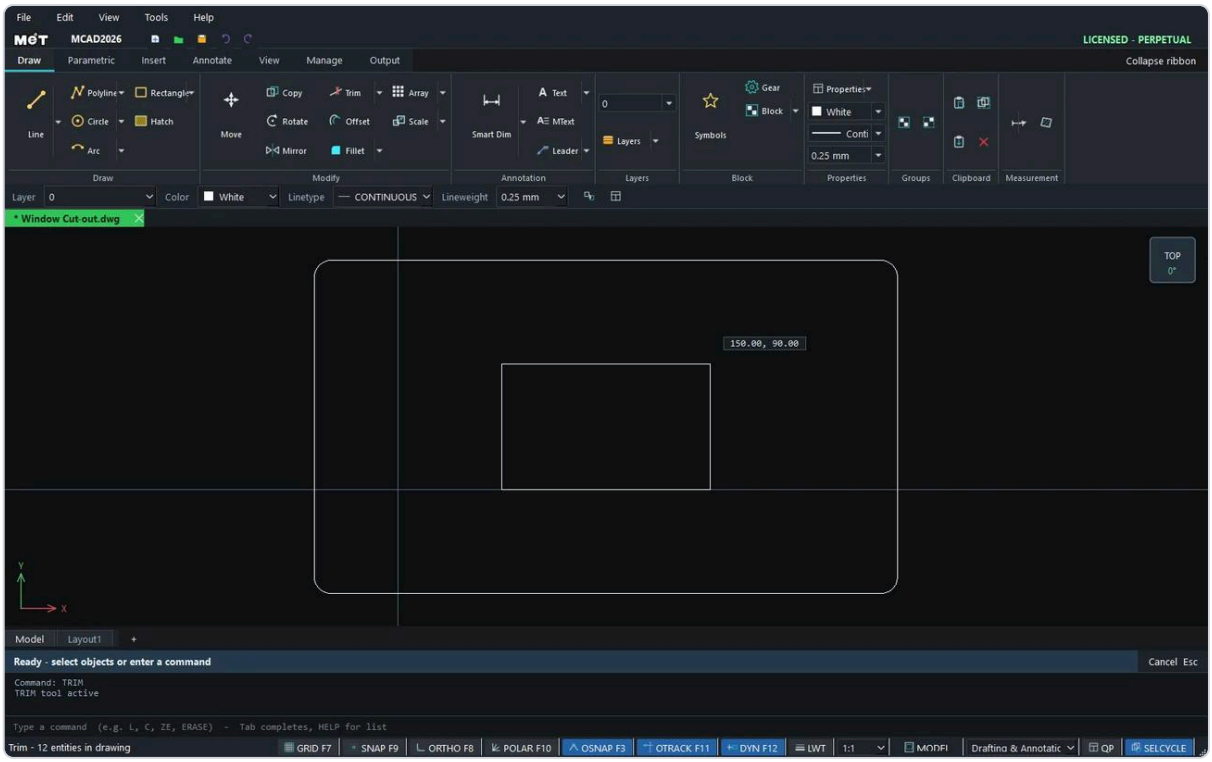
- Ribbon: **Draw › Modify › Trim** (submenu: Trim, Extend, Stretch, Trim at a Boundary (EXTRIM), Break, Join, Lengthen, Align, Explode, Bring to Front, Send to Back)
- Command line: `TRIM` (Alias `TR`)

TRIM steps

1. Type `TR` and press Enter.
2. Click each segment you want to remove (or hold the mouse button and drag across several segments, like a Fence).
3. Press Esc to finish.



Before TRIM: lines crossing in a grid



After TRIM: the overhanging ends are removed, leaving a rectangle

EXTEND steps

1. Type `EX` and press Enter.
2. Click near the end of the line you want to extend.
3. Press Esc to finish.

EXTRIM steps

1. Select the closed shape that acts as the boundary (rectangle, circle, closed Polyline).
2. Type `EXTRIM` (trim the inside) or `EXTRIM O` (trim the outside).

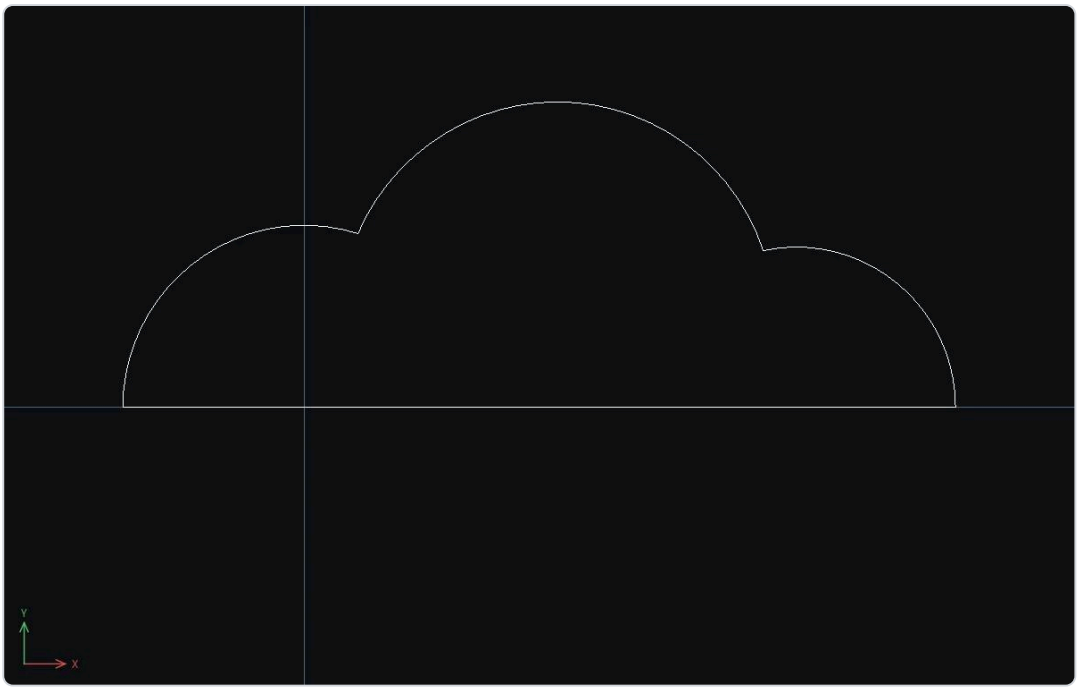
> **Note:** EXTRIM cuts Line/Polyline objects at the intersection. Circle, Arc, and Text objects are deleted only if they lie entirely on the side being removed.

Try it: Cloud night-light panel

The front panel of a cloud-shaped night light. Three circles and one line become a soft cloud outline with nothing but TRIM clicks.

1. Circles: `C` `0,0` `25`, `C` `35,12` `30`, `C` `68,0` `22`. Press Esc after each one.
2. Base line: `L`, `-25,0`, `90,0`, Esc.
3. Type `TR` and press Enter. Click every arc piece below the base line (7 pieces). You can also hold the mouse button and drag across several pieces to trim them in one stroke.
4. Click the 4 short arcs inside the cloud, where the circles overlap. Press Esc.

One clean cloud outline remains: three bumps on a flat base.



Cloud outline trimmed from three circles and a line

Challenge: Add a crescent moon cut-out: `C` `38,22` `6` and `C` `41,24` `5`. Then use `TR` to click the part of the big circle inside the small one, and the part of the small circle outside the big one.

See also

- BREAK and JOIN
- STRETCH and LENGTHEN

FILLET and CHAMFER - Corner rounding and bevelling

Round a corner (FILLET) or bevel it (CHAMFER) between two lines.

What it does

- **FILLET** – makes a rounded corner with the radius you set.
- **CHAMFER** – cuts a bevelled corner with the distance you set.

The radius and distance are remembered for next time.

Where to find it

- Ribbon: **Draw › Modify › Fillet** (submenu: Fillet, Chamfer)
- Command line: `FILLET` (alias `F`)
- Command line: `CHAMFER` (alias `CHA`)

Steps for FILLET

1. Type `F` and press Enter. The prompt reads `FILLET: radius = 3 - pick the first curve, or type a number to change it.`
2. Type a new radius, for example `20`, and press Enter. Skip this step to keep the current value.

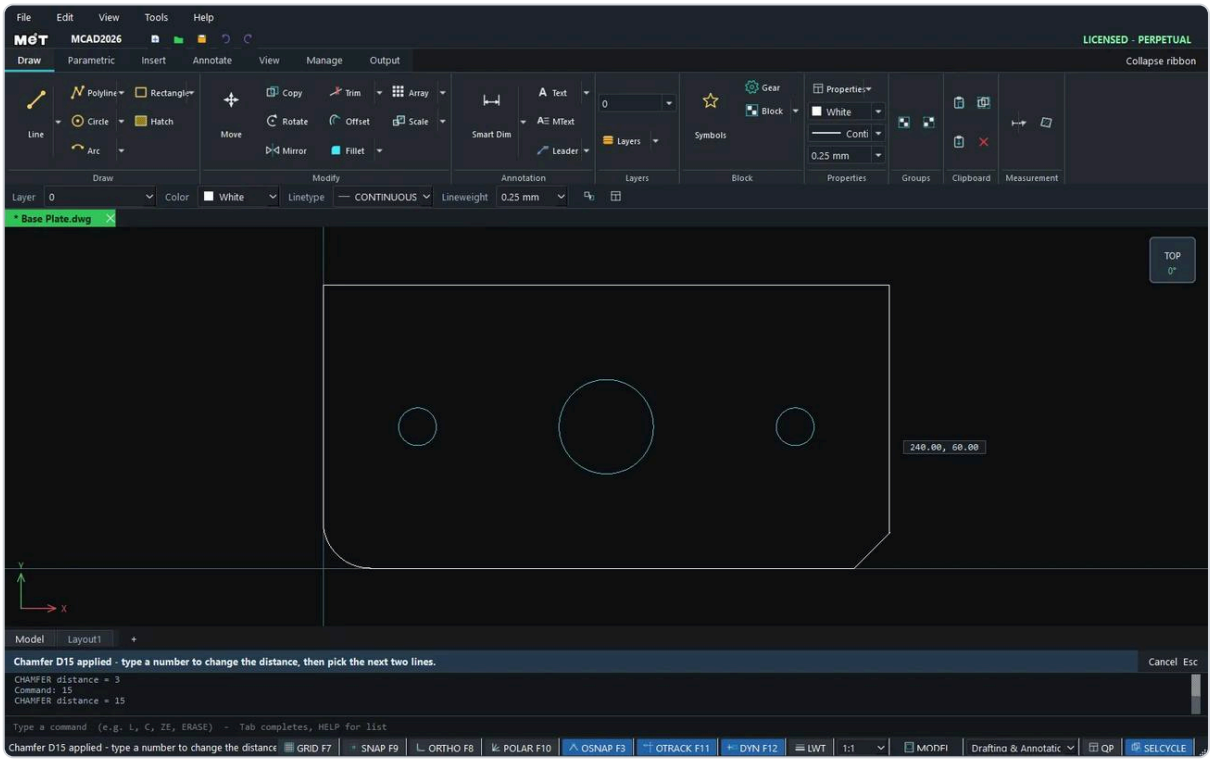
3. Click the first line. At `Fillet: now click the second curve.`, click the second line.
4. You can continue with the next corner. Press Esc to finish.

Shortcut: `F 20` or `FILLET R 20`

Steps for CHAMFER

1. Type `CHA` and press Enter. Type a distance, for example `15`, and press Enter.
2. Click the first line and the second line.

Shortcut: `CHA 15` or `CHAMFER D 15`



FILLET R20 (left) and CHAMFER 15 (right)

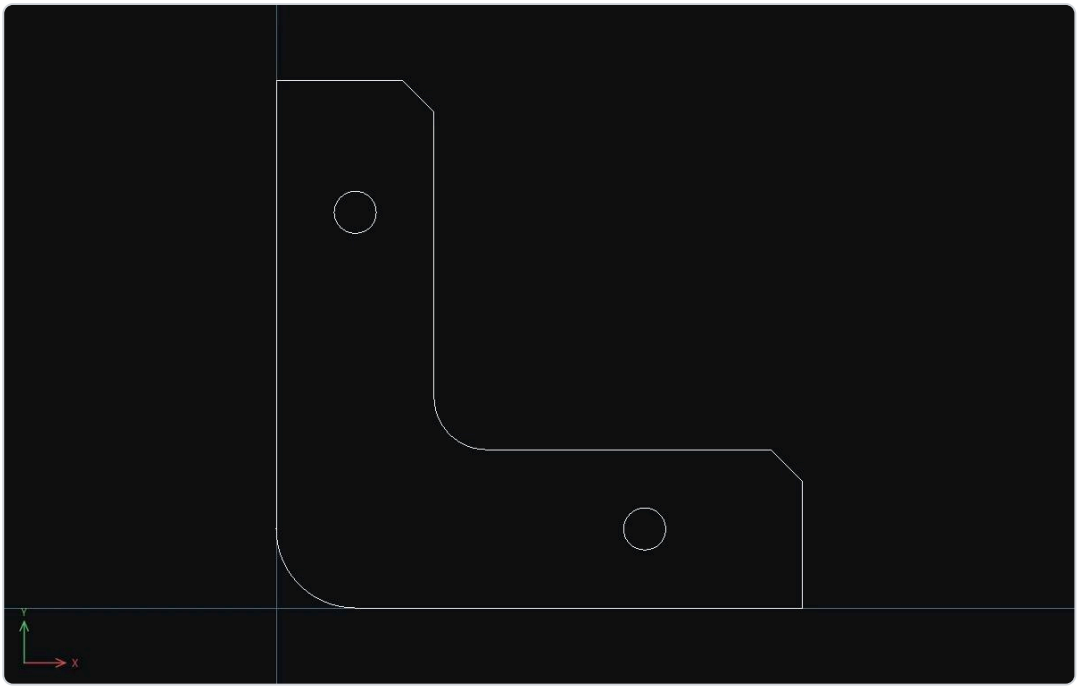
Option	Meaning
`FILLET [R] <radius>`	Radius 0 = join the lines at a sharp corner.
`CHAMFER [D] <distance>`	The same distance on both sides.

Try it: Steel corner bracket

An L-shaped corner bracket cut from 2 mm steel to strengthen a shelf or a frame. Rounded corners are safer to handle and stronger, and chamfered tips look neat.

1. Draw the outline with LINE: `L`, then `0,0` `@100,0` `@0,30` `@-70,0` `@0,70` `@-30,0` `0,0`, and press Esc. FILLET rounds Lines (and a Line with an Arc or Circle) and CHAMFER works on two Lines, so a RECTANG or PLINE outline will not take a fillet.
2. Type `F 15` and press Enter. Click the bottom edge, then the left edge. The outer corner becomes R15.
3. While FILLET is still running, type `10` and press Enter. Click the short inside horizontal edge, then the inside vertical edge. The inside corner becomes R10. Press Esc.

4. Type `CHA 6` and press Enter. Click the right edge of the short leg, then its top edge. Then click the right edge of the tall leg and its top edge. Press Esc.
5. Screw holes: `C`, `70,15`, `4`, Esc, and `C`, `15,75`, `4`, Esc.



Corner bracket with R15 and R10 fillets and 6 mm chamfers

Challenge: Round the two corners that are still sharp (bottom right and top left) with `F 5`.

See also

- TRIM and EXTEND
- ARC

STRETCH and LENGTHEN

Stretch or shrink part of a shape with a Crossing window (STRETCH) and change the length of a line end (LENGTHEN).

What it does

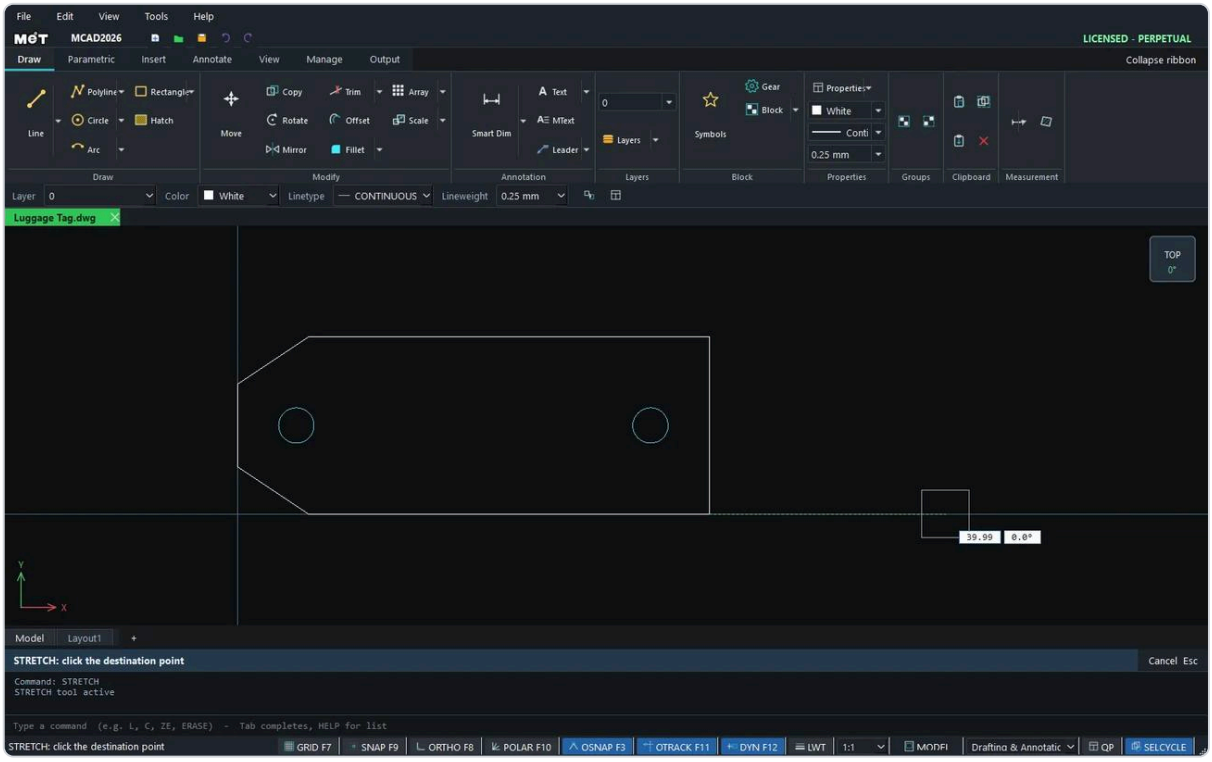
- **STRETCH** – moves only the vertices inside the Crossing window, so the shape stretches or shrinks. Objects that lie entirely inside the window are moved as a whole.
- **LENGTHEN** – adds length to the end of a line by a given distance (use a negative value to shorten it).

Where to find it

- Ribbon: **Draw › Modify › Trim ▾ › Stretch** / **Lengthen**
- Command line: `STRETCH` (Alias `S`)
- Command line: `LENGTHEN` (Alias `LEN`)

STRETCH steps

1. Type `S` and press Enter.
2. Click 2 corners of a Crossing window so it crosses the side you want to stretch.
3. `STRETCH: click the base point` Click the base point.
4. `STRETCH: click the destination point` Click a point or type a distance. The result is `Stretched 2 object(s)`.



STRETCH stretching the right side of a plate with holes

LENGTHEN steps

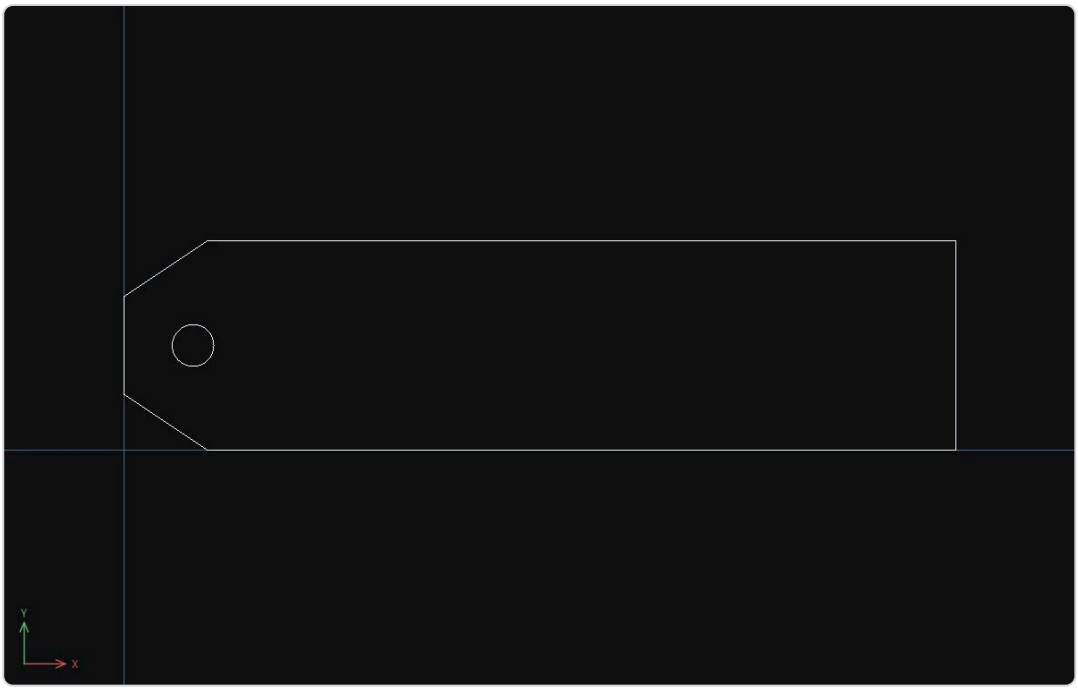
1. Type `LEN` and press Enter. `Lengthen delta (negative shortens) <10>:` Type a distance, for example `30`.
2. `LENGTHEN: click near a line end to extend by +30 (Esc to finish)` Click near the end you want to change.
3. Keep clicking other lines. Press Esc to finish.

Shortcut: `LEN 30` or `LEN -5` (shortens by 5).

Try it: Luggage tag that fits a longer name

A plywood luggage tag, 80 mm long. When the name does not fit, STRETCH makes the tag longer without moving the strap hole.

1. Tag: `PL`, then `0,8` `@12,-8` `@68,0` `@0,30` `@-68,0` `@-12,-8` `0,8`, and press Enter. Strap hole: `C`, `10,15`, `3`, Esc. Use PLINE for the outline: a RECTANG rectangle is not stretched, it only moves when it lies completely inside the window.
2. Type `S` and press Enter. Type the crossing-box corners `95,40` and `60,-5` so the box covers only the right end.
3. Base point `80,0`, destination `@40,0`. The result reads `Stretched 1 object(s)`. The tag is now 120 mm long, and the hole has not moved.



Luggage tag stretched from 80 to 120 mm

Challenge: Add an engraving guideline with `L`, `20,15`, `@60,0`. Then type `LEN 20` and click near its right end. The line grows to 80 mm.

See also

- TRIM and EXTEND
- Editing with Grips

BREAK, JOIN and PEDIT

Cut a gap in a line (BREAK), join lines that line up (JOIN) and merge them into a polyline (PEDIT).

What it does

Command	Alias	Meaning
BREAK	`BR`	Click 2 points on an object. The part between the two points is deleted.
JOIN	`J`	Click 2 objects that are collinear or touch end to end. They become one object.
PEDIT	`PE`	Merges the selected lines and polylines whose ends meet into one polyline.

Where to find it

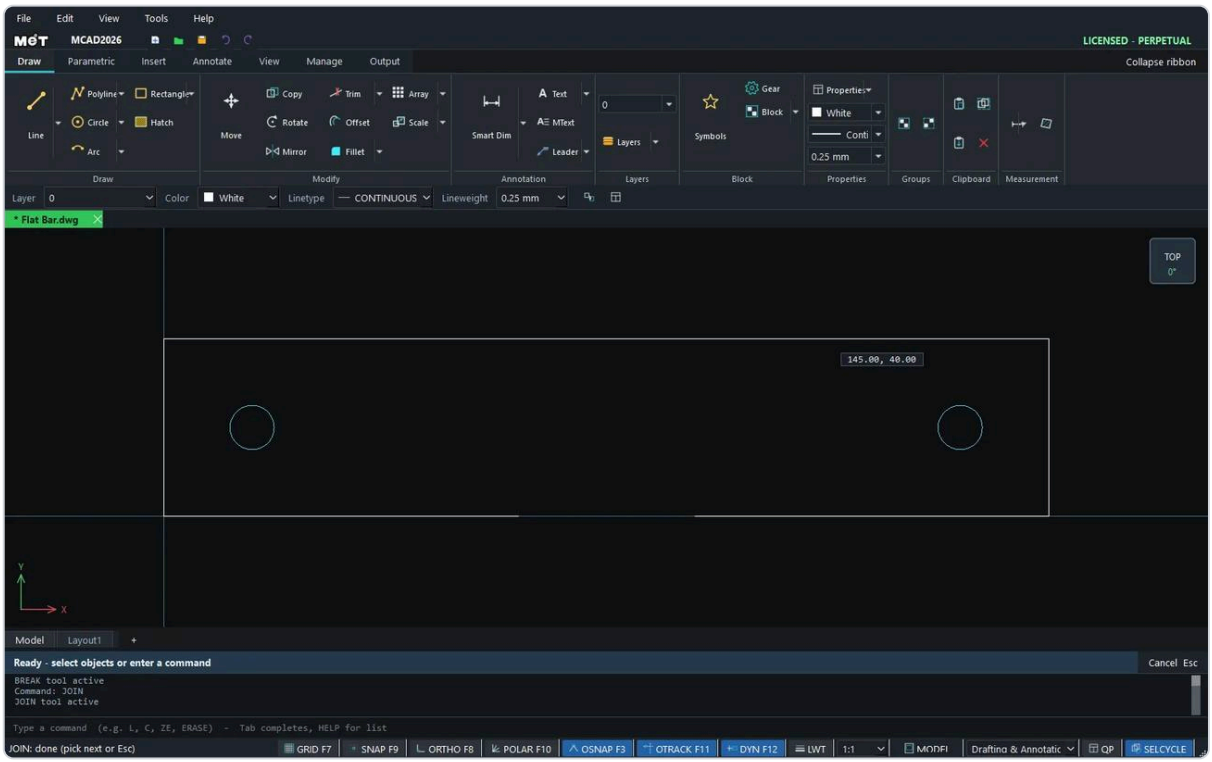
- Ribbon: **Draw › Modify › Trim ▾ › Break / Join**
- Command line: `BREAK` (alias `BR`)
- Command line: `JOIN` (`J`), `PEDIT` (`PE`)

Steps for BREAK

1. Type `BR` and press Enter.
2. Click the first point on the line, then click the second point. The middle part disappears.

Steps for JOIN

1. Type `J` and press Enter.
2. Click the first line, then click the second line.



The lower line is broken into a gap; the two upper segments are joined into one line

Steps for PEDIT

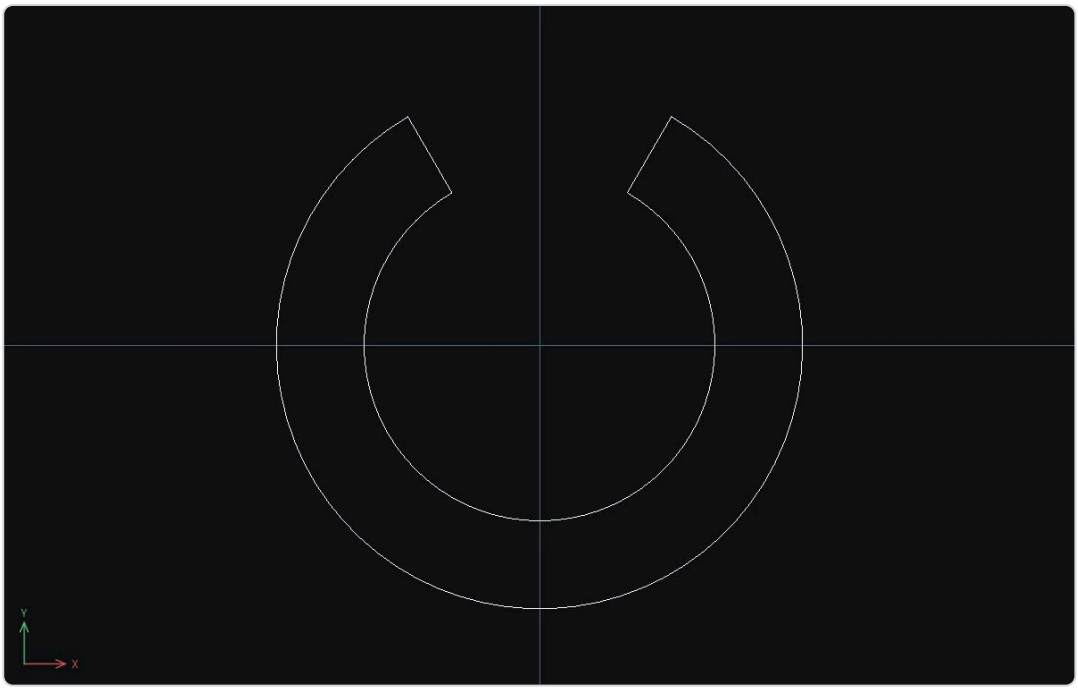
1. Select all the lines whose ends meet.
2. Type `PE` and press Enter. The Status bar shows `PEDIT: joined N segment(s) into polyline(s).`

Try it: Snap-in cable clip

A C-shaped clip cut from 3 mm acrylic. Push a USB cable through the gap and it stays put on your desk. BREAK opens the gap with two clicks.

1. Ring: `C`, `0,0`, `12`, then Esc. Then `C`, `0,0`, `8`, then Esc.
2. Type `BR` and press Enter. Click the outer circle at about 2 o'clock (upper right), then at about 10 o'clock (upper left). The top piece between the clicks disappears. BREAK removes the part that runs counterclockwise from the first click to the second, so click the right side first.
3. Do the same on the inner circle: the right point first, then the left one. Press Esc.
4. Close the ends: type `L`, click the end of the outer arc (the Endpoint snap marker appears), click the end of the inner arc, and press Esc. Repeat on the other side.

You now have one closed C outline, ready to cut.



Cable clip with a gap opened by BREAK

Challenge: Draw a line with `L`, `0,-30`, `@60,0`, use `BR` to break a gap out of the middle, then type `J` and click the two pieces. JOIN closes the gap again because the pieces are collinear.

See also

- PLINE
- TRIM

ALIGN

Move and rotate objects using two pairs of source and destination points.

What it does

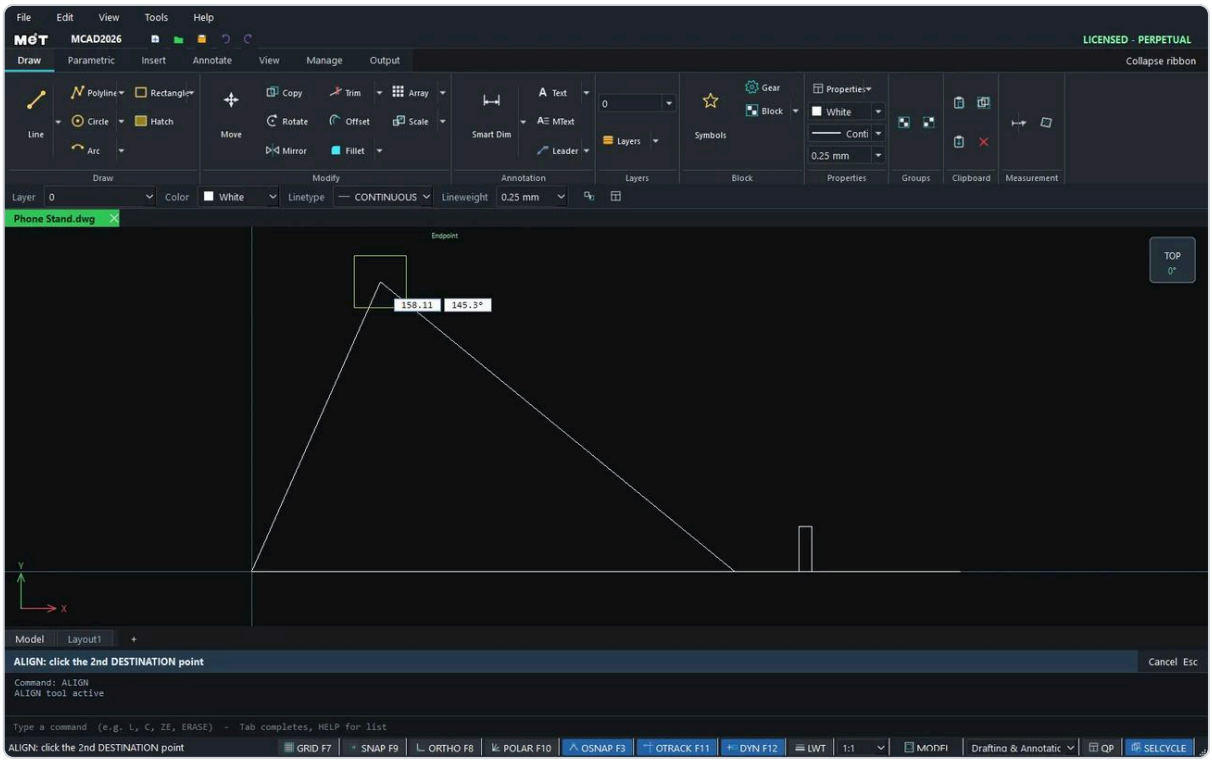
ALIGN moves and rotates the selected objects so that two source points land on two destination points. For example, you can fit a part flush against a slanted edge.

Where to find it

- Ribbon: **Draw › Modify › Trim ▾ › Align**
- Command line: `ALIGN` (alias `AL`)

Steps

1. Select the objects.
2. Type `AL` and press Enter.
3. Click source point 1 → destination point 1 → source point 2 → destination point 2. The prompt bar guides you through each step, for example `ALIGN: click the 2nd DESTINATION point`.



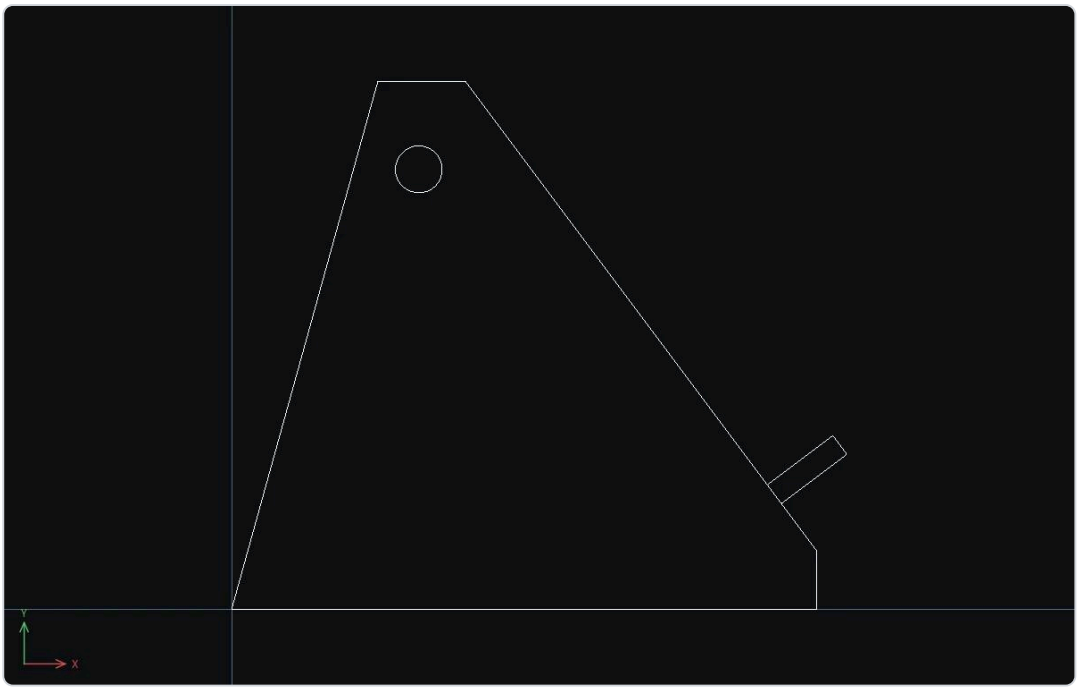
ALIGN picking the second destination point on a slanted line

Try it: Acrylic phone stand with a phone lip

A side panel for a phone stand cut from 5 mm acrylic. The phone leans on the sloped face, and a small lip at the bottom stops it from sliding off. You draw the lip lying flat anywhere, and ALIGN stands it up square to the slope in one go.

1. Side panel: type `PL` and press Enter, then type these points, pressing Enter after each: `0,0` `@100,0` `@0,10` `40,90` `25,90` `0,0`. Press Enter to finish.
2. Charging-cable hole: `C`, Enter, `32,75`, Enter, `4`, Enter, then Esc.
3. The lip, drawn flat away from the panel: `REC`, Enter, `150,0`, Enter, `@4,14`, Enter, then Esc.
4. Click the lip to select it, type `AL` and press Enter.
5. Type the four points, pressing Enter after each: `154,0` (source 1, the lip's bottom-right corner) → `94,18` (destination 1, 10 mm up the slope) → `150,0` (source 2, the bottom-left corner) → `40,90` (destination 2, the top of the slope). The status bar shows `Aligned 1 object(s)`.

The lip now sticks out 14 mm, exactly square to the sloped face, ready to catch the phone.



Phone stand side panel with the lip aligned to the slope

Challenge: For a phone in a thick case, draw the lip as `@6,14` instead and align it again using the source points `156,0` and `150,0`.

See also

- ROTATE
- MOVE

ARRAY - Rectangular / Polar / Path

Copy objects in rows (ARRAY/ARRAYRECT), around a point (ARRAYPOLAR) and along a path (ARRAYPATH, MEASURE, DIVIDE).

What it does

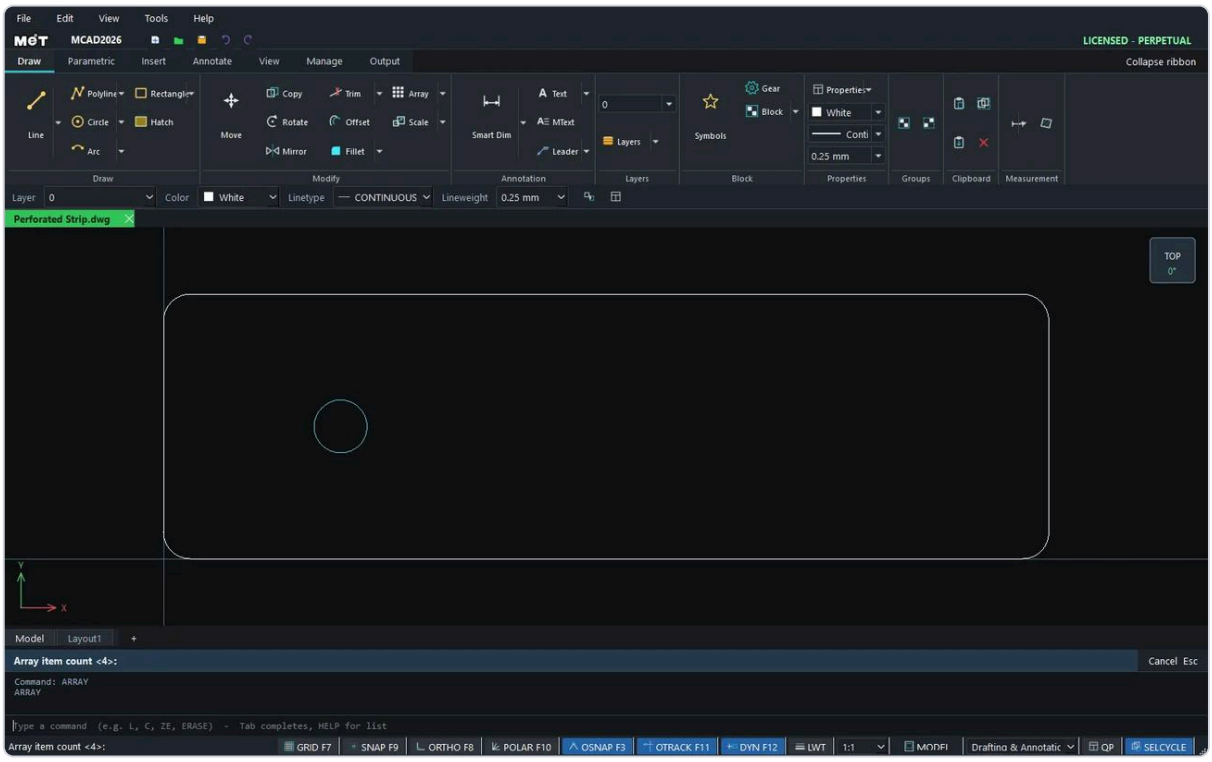
Command	Alias	Syntax	Meaning
ARRAY	`AR`	–	Asks for each value on the command line: count → X spacing → Y spacing.
ARRAYRECT	`AR2`	`ARRAYRECT <count> [dx] [dy]`	Copies objects in a row, with spacing dx, dy per copy.
ARRAYPOLAR	`ARP2`	`ARRAYPOLAR <count> [total angle]`	Copies objects around the centre point you click (the default total angle is 360°).
ARRAYPATH	–	`ARRAYPATH <count>`	Distributes N copies along a path (same as DIVIDE).
DIVIDE	`DIV`	`DIVIDE <count>`	Places copies at the points that divide a path into N equal parts.
MEASURE	`ME`	`MEASURE <distance>`	Places a copy every given distance along a path, rotated to the tangent.

Where to find it

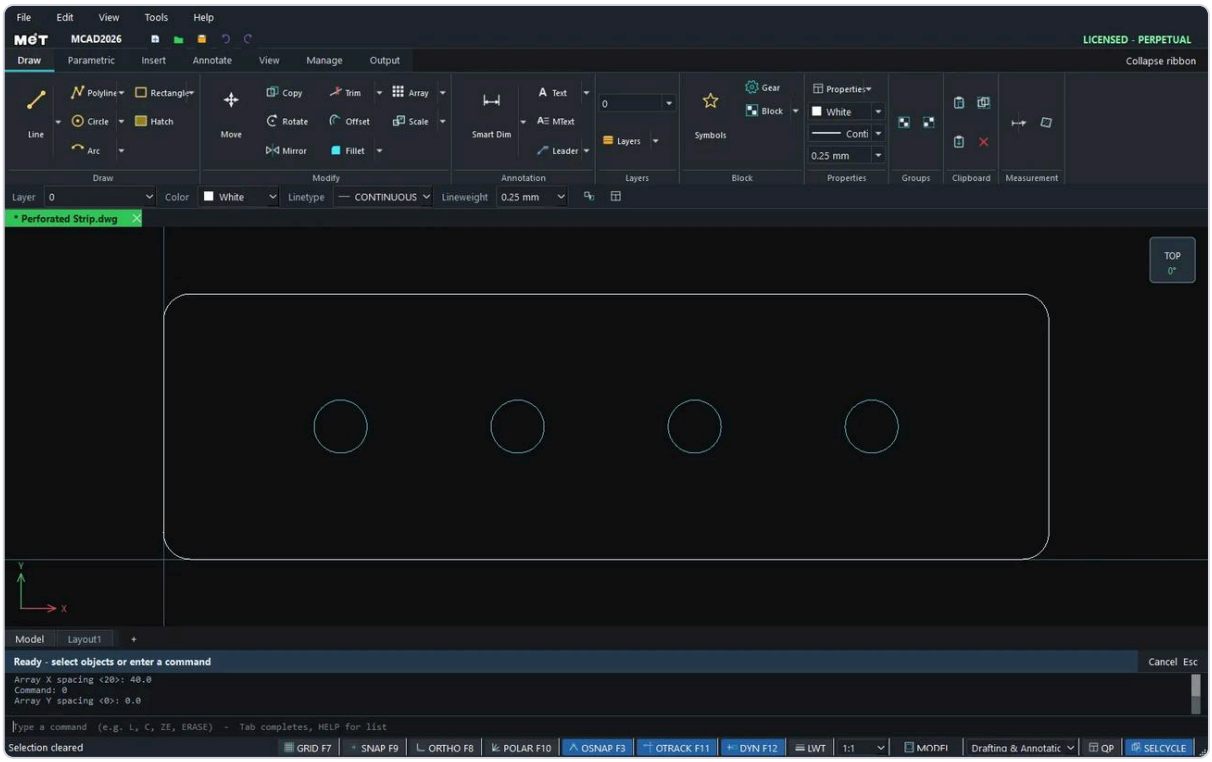
- Ribbon: **Draw › Modify › Array** (submenu: Rectangular Array, Polar Array, Array along Curve (MEASURE))
- Command line: see the table above

Steps for ARRAY (prompts for each value)

1. Select the objects.
2. Type `AR` and press Enter. At `Array item count <4>:`, type the count and press Enter.
3. At `Array X spacing <20>:`, type the X spacing and press Enter.
4. At `Array Y spacing <0>:`, type the Y spacing and press Enter.



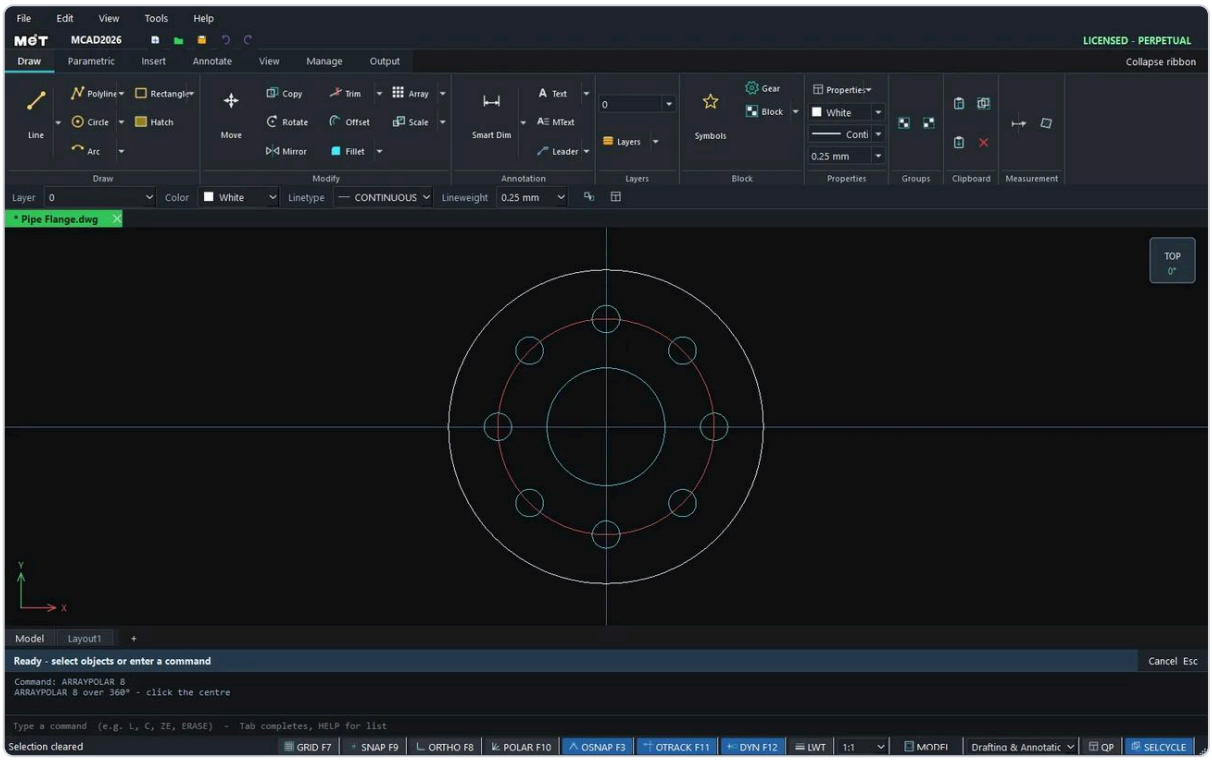
ARRAY asking for the count on the command line



4 holes at 40 spacing

Steps for ARRAYPOLAR

1. Select the hole.
2. Type `ARRAYPOLAR 8` and press Enter.
3. At `ARRAYPOLAR: click the centre`, click the centre point.



ARRAYPOLAR with 8 copies around a circle

Steps for MEASURE / DIVIDE

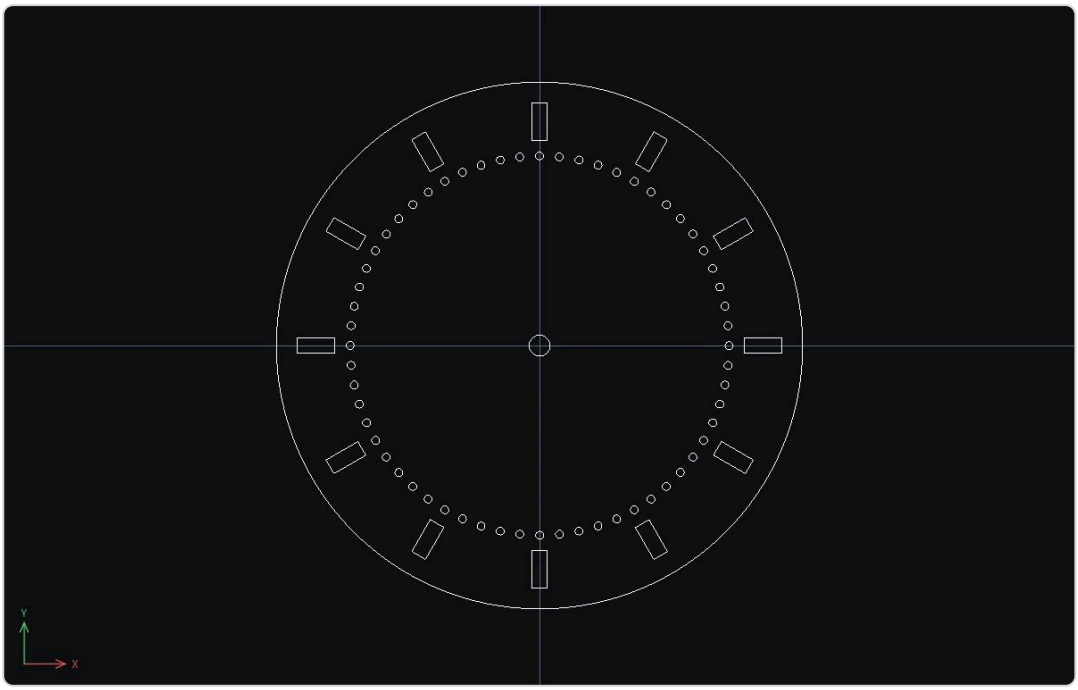
1. Select the object to copy.
2. Type `ME 20` (every 20 mm) or `DIV 5` (divide into 5 parts).
3. Click the path (Line, Polyline, Arc, Circle).

> **Note: Polar Array** on the Ribbon asks for the count → total angle → centre point. Press Enter at any prompt to accept the default value.

Try it: Wall clock face

A 200 mm laser-cut clock face with 12 hour slots and 60 minute dots, all made from one slot and one dot. The 8 mm centre hole fits a standard quartz clock movement.

1. Dial: `C`, Enter, `0,0`, Enter, `100`, Enter, then Esc.
2. Shaft hole: `C`, `0,0`, `4`, then Esc.
3. The 12 o'clock slot: `REC`, `-3,78`, `@6,14`, then Esc.
4. Click the slot, type `ARRAYPOLAR 12` and press Enter. At `ARRAYPOLAR: click the centre`, type `0,0` and press Enter (or click the centre). Twelve slots appear around the dial.
5. Minute dot: `C`, `0,72`, `1.5`, then Esc. Click the dot, type `ARRAYPOLAR 60`, then `0,0`.



Clock face with 12 hour slots and 60 minute dots

Challenge: Make a 24-hour dial: run `ARRAYPOLAR 24` on the first slot instead of 12.

See also

- COPY
- POINT

EXPLODE and XPLODE

Break blocks, tables, multileaders and splines into basic objects (EXPLODE/XPLODE).

What it does

- **EXPLODE** – breaks compound objects (block references, tables, multileaders, splines, blocks that came from a DWG) into plain lines and text. Nested blocks are broken one level at a time.
- **XPLODE** – explodes and also puts the resulting pieces on one layer or in one colour.

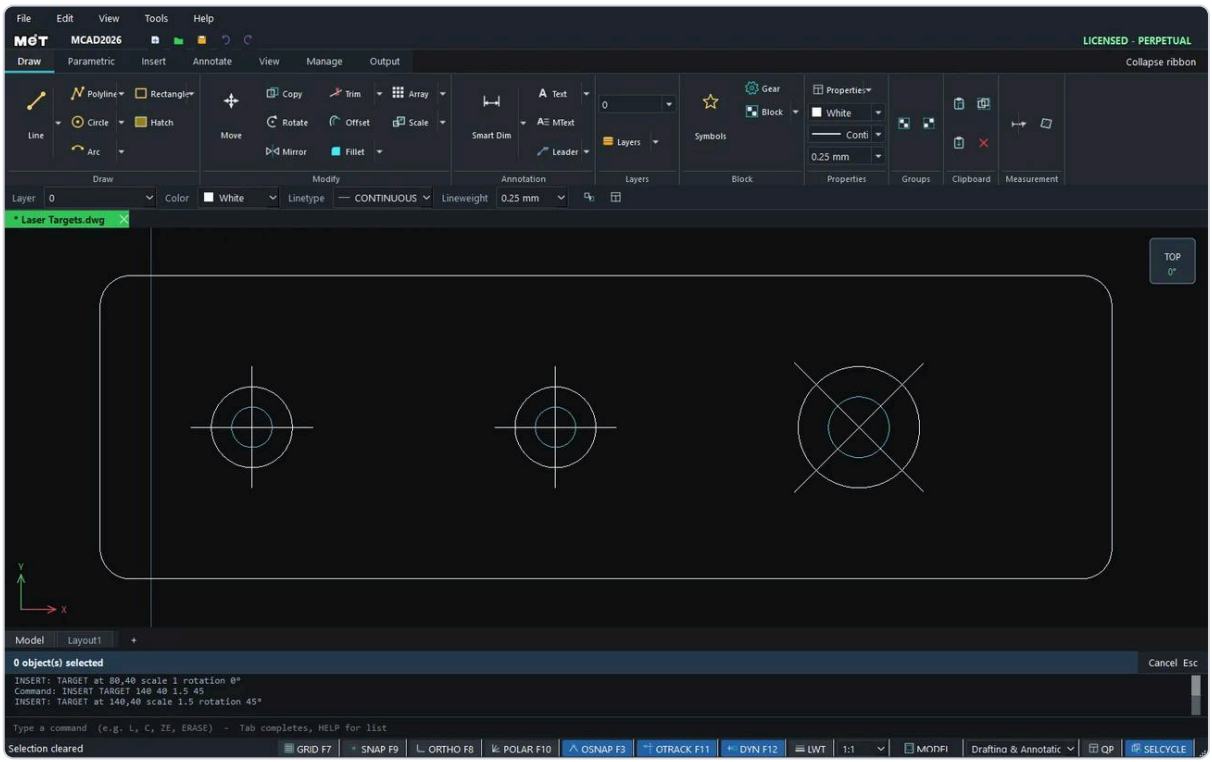
Where to find it

- Ribbon: **Draw › Modify › Trim ▾ › Explode**
- Command line: `EXPLODE` (alias `X`)
- Command line: `XPLODE` (alias `XP`)
- Right-click the selected object › Ungroup (Explode)

Steps

1. Select a block or compound object.
2. Type `X` and press Enter. The result reads, for example, `EXPLODE: 1 object(s) -> 3 primitives`.

XPLODE syntax	Meaning
`XPLODE Layer <name>`	The resulting pieces go on the layer you name.
`XPLODE Color #RRGGBB`	The resulting pieces use the colour you give.



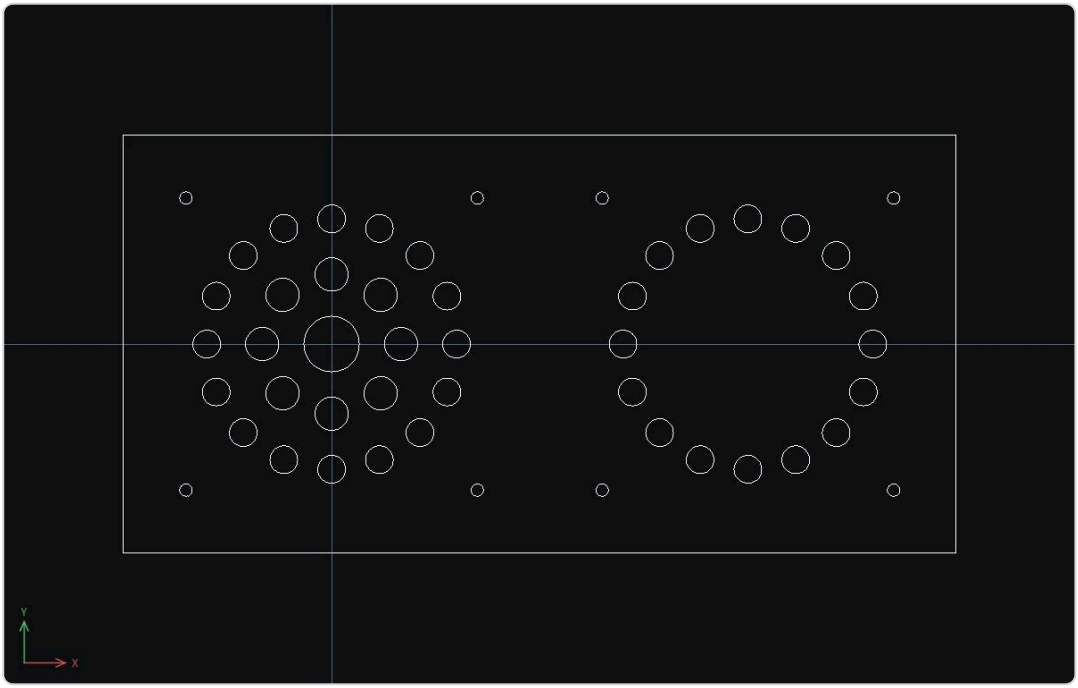
Three TARGET blocks before exploding (each block appears as one object)

> **Note:** A block that was stretched with a dynamic parameter explodes into the shape currently shown, not the block's original shape.

Try it: PC case side panel with two fan grilles

An acrylic side panel with two 120 mm fan grilles. Make the grille a block so both copies stay identical, then EXPLODE the second one when it has to be different: here, an open centre for a logo badge.

1. Grille holes: centre `C`, `0,0`, `10`. Inner ring hole `C`, `0,25`, `6`. Outer ring hole `C`, `0,45`, `5`. Mounting hole `C`, `52.5,52.5`, `2.25`. Press Esc after each one.
2. Click the inner ring hole, type `ARRAYPOLAR 8`, then `0,0`. Do the same for the outer ring hole with `ARRAYPOLAR 16` and for the mounting hole with `ARRAYPOLAR 4`.
3. Drag a window around the whole grille (from -60,-60 to 60,60) and type `BLOCK FAN 0 0`. The result reads `BLOCK: FAN defined from 29 object(s) ...`.
4. Second fan: type `INSERT FAN 150 0`.
5. Click the second grille, type `X` and press Enter. The result reads `EXPLODE: 1 object(s) -> 29 primitives`.
6. Drag a window around its centre (from 117,-33 to 183,33) and press Delete. Only this copy changes.
7. Panel outline: `REC`, `-75,-75`, `@300,150`.



Side panel with one full grille and one with an open centre

Challenge: Try to delete one hole from the first fan without exploding it. A click selects the whole block, which is exactly why EXPLODE exists.

See also

- BLOCK, INSERT
- GROUP

DRAWORDER, OVERKILL, REVERSE and FLATTEN

Set the stacking order (DRAWORDER), delete duplicate objects (OVERKILL), reverse line direction (REVERSE) and set Z to zero (FLATTEN).

What it does

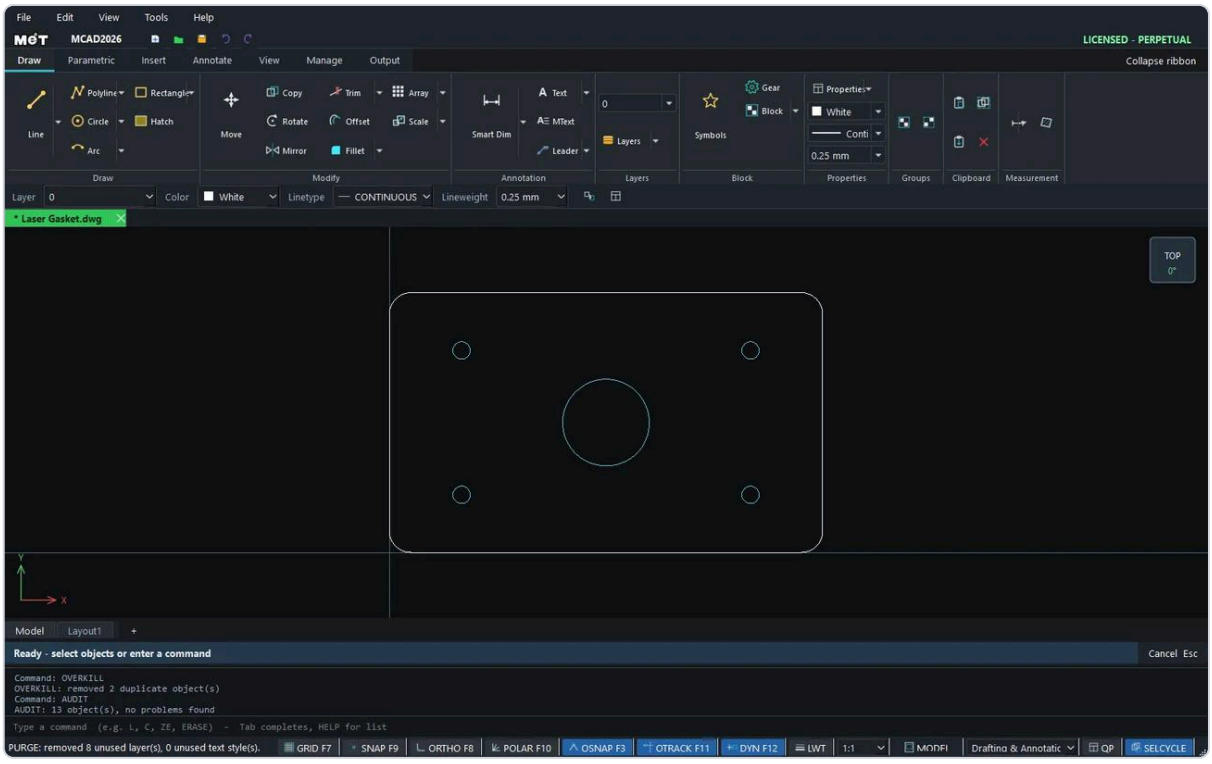
Command	Alias	Syntax	Meaning
DRAWORDER	`DR`	`DRAWORDER <Front|Back>`	Moves the selected objects to the front or to the back.
OVERKILL	`OV`	–	Deletes duplicate and overlapping lines in the whole drawing.
REVERSE	`REV`	–	Reverses the direction of the selected lines, polylines and splines. This matters for the cutting direction in G-code.
FLATTEN	–	–	Sets the Z value of every object to 0. It fixes snap and area problems caused by DWG files that have Z levels.

Where to find it

- Ribbon: **Draw › Modify › Trim ▾ › Bring to Front (DRAWORDER)** / **Send to Back (DRAWORDER)**
- Ribbon: **Manage › Drawing Cleanup › Delete Duplicates** (OVERKILL)

Steps for OVERKILL

1. Type `OV` and press Enter.
2. The result reads `OVERKILL: removed 2 duplicate object(s)` or `no duplicates found`.



OVERKILL removes 2 overlapping lines

Steps for DRAWORDER

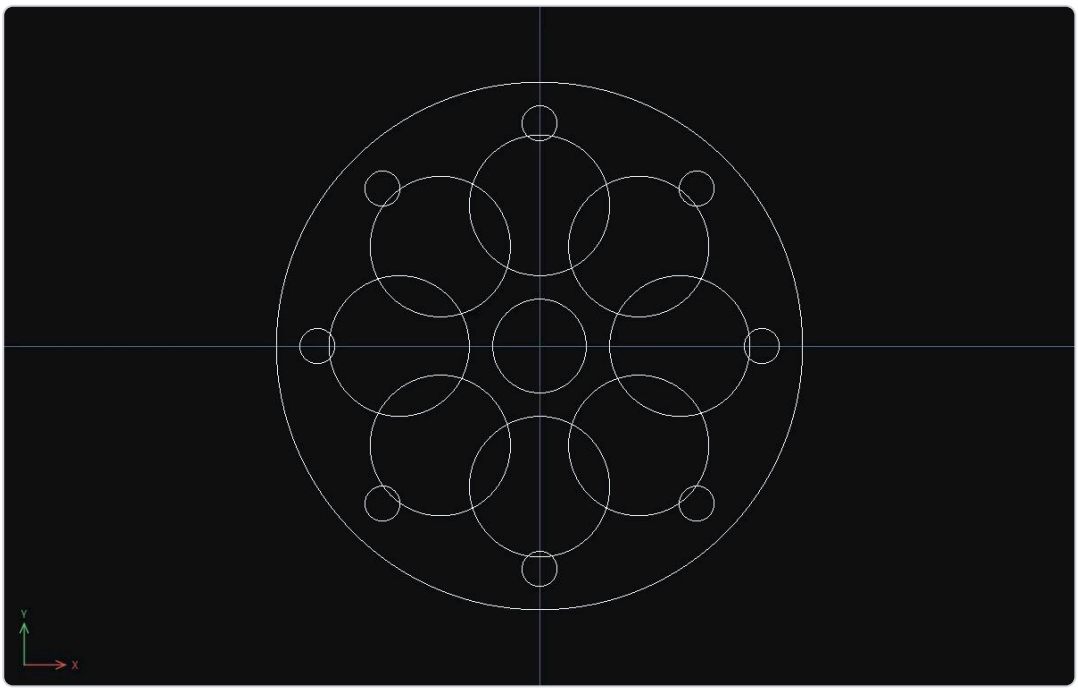
1. Select the objects.
2. Type `DR F` (to the front) or `DR B` (to the back).

> **Note:** Use `TEXTTOFRONT` to bring all text and dimensions to the front, and `HATCHTOBACK` to send all hatches to the back.

Try it: Mandala coaster without double cuts

A 90 mm mandala coaster for the laser. If the same petal is copied twice, the laser cuts it twice and scorches the wood. OVERKILL finds those hidden duplicates and deletes them.

1. Coaster: `C`, `0,0`, `45`, Esc. Centre: `C`, `0,0`, `8`, Esc. Petal: `C`, `0,24`, `12`, Esc. Dot: `C`, `0,38`, `3`, Esc.
2. Click the petal, `Ctrl`+click the dot, type `ARRAYPOLAR 8` and then `0,0`. Eight overlapping petals form the flower.
3. Now make the classic mistake: click the first petal, `Ctrl`+click its dot again, and run `ARRAYPOLAR 8`, `0,0` once more. Nothing looks different, but 14 extra circles now sit exactly on top of the others.
4. Type `OV` and press Enter. The result reads `OVERKILL: removed 14 duplicate object(s)`.



Mandala coaster with 8 petals and dots

Challenge: Run `OV` again. This time it reports that there are no duplicates, so the file is safe to send to the laser.

See also

- AUDIT, RECOVER and PURGE
- HATCH

Clipboard: COPYCLIP, CUTCLIP, PASTECLIP, COPYBASE

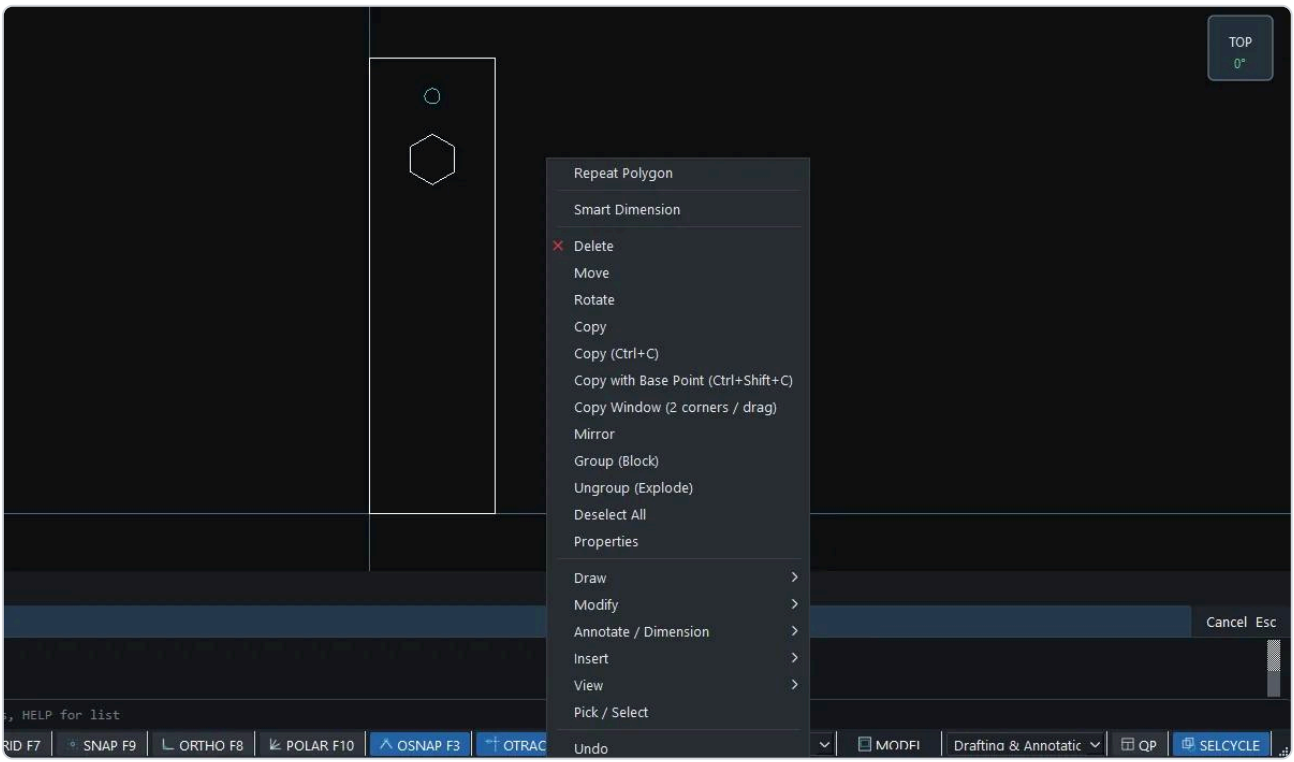
Copy, cut and paste through the Clipboard between tabs or programs, with a base point, and use the right-click menu.

What it does

Command	Alias	Shortcut	Meaning
COPYCLIP	`CC`	`Ctrl+C`	Copies the selected objects to the Clipboard.
CUTCLIP	`CX`	`Ctrl+X`	Cuts the selected objects to the Clipboard.
COPYBASE	–	`Ctrl+Shift+C`	Copies with a base point that you pick.
PASTECLIP	`PASTE` , `PA`	`Ctrl+V`	Pastes from the Clipboard. Click the position.

Where to find it

- Ribbon: **Draw › Clipboard** (Copy, Copy Window, Paste, Delete)
- Right-click in the drawing area



Right-click menu when objects are selected

Steps: copy to another drawing and keep a reference point

1. Select the objects, then press `Ctrl+Shift+C` (COPYBASE).
2. At `COPYBASE - pick base point` , click the base point.
3. Switch to another file tab and press `Ctrl+V` .
4. Click where to paste. The base point lands at the point you click.

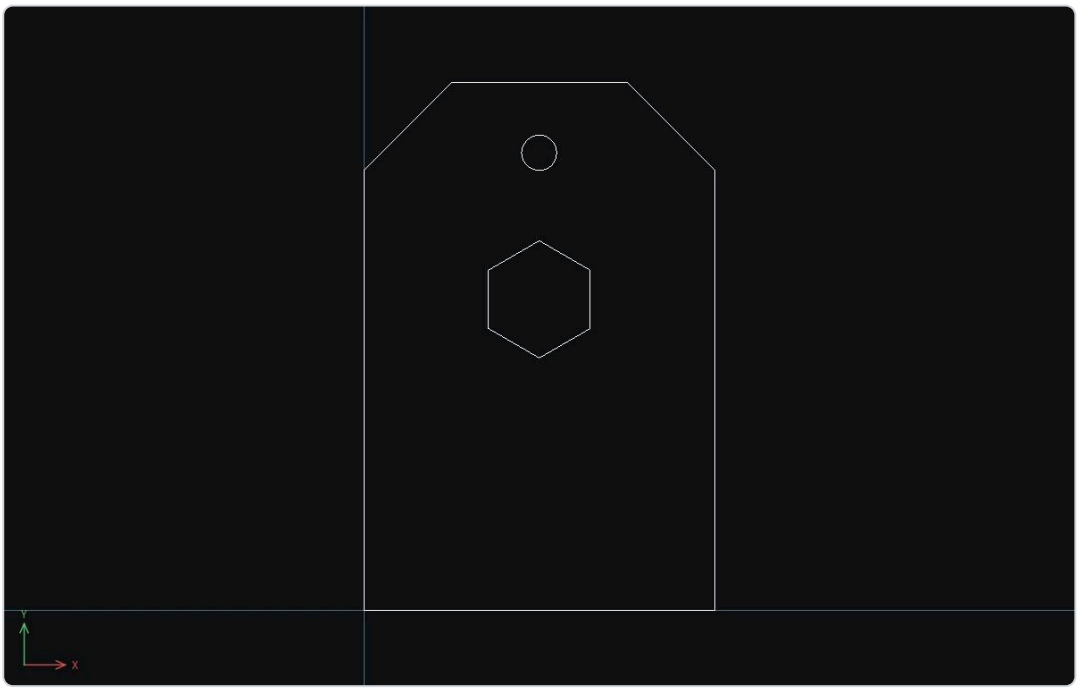
Right-click menu

Item	Meaning
Repeat ...	Repeats the last command.
Smart Dimension	Starts Smart Dimension.
Delete / Move / Rotate / Copy / Mirror	Edits the selected objects.
Copy Window (2 corners / drag)	Copies everything inside the window to the Clipboard.
Group (Block) / Ungroup (Explode)	Groups / ungroups objects.
Deselect All / Properties	Clears the selection / opens Properties.
Draw, Modify, Annotate / Dimension, Insert, View	Submenus that gather the tools (View includes Print Window (PDF)...).
Pick / Select, Undo, Redo	Returns to select mode, undoes, redoes.

Try it: Bookmark ornament reused on a gift tag

You already have a bookmark with a ribbon hole and a hexagon cut-out. Reuse both on a new gift tag, landing exactly where you want them, with COPYBASE.

1. Bookmark: `REC`, `0,0`, `@50,180`, then Esc. Ribbon hole: `C`, `25,165`, `3`, then Esc. Hexagon: `POL`, `25,140`, `@10<90`.
2. Drag a window around the hole and the hexagon only (not the outline). Press `Ctrl+Shift+C`, then type `25,165` (the hole centre) as the base point and press Enter. The status bar shows `Copied 2 object(s) - Ctrl+V to paste (any tab or window)`.
3. Press `Ctrl+N` for a new drawing. Gift tag: `PL`, then `0,0` `@60,0` `@0,75` `@-15,15` `@-30,0` `@-15,-15` `0,0`, and press Enter.
4. Type `SNAP 5`, press `Ctrl+V` and click at 30,78. The hole centre lands exactly there, with the hexagon 25 mm below it. Type `SNAP OFF`.



Gift tag with the ribbon hole and hexagon pasted from the bookmark

Challenge: Open a third drawing and press `Ctrl+V` again. The clipboard still holds the copy, so you can stamp the same ornament on a whole set of tags.

See also

- MOVE and COPY
- ERASE

GROUP, UNGROUP and GROUPEDIT

Group objects (GROUP), ungroup them (UNGROUP) and add or remove members (GROUPEDIT).

What it does

A group lets one click select every object in the group, without creating a block definition.

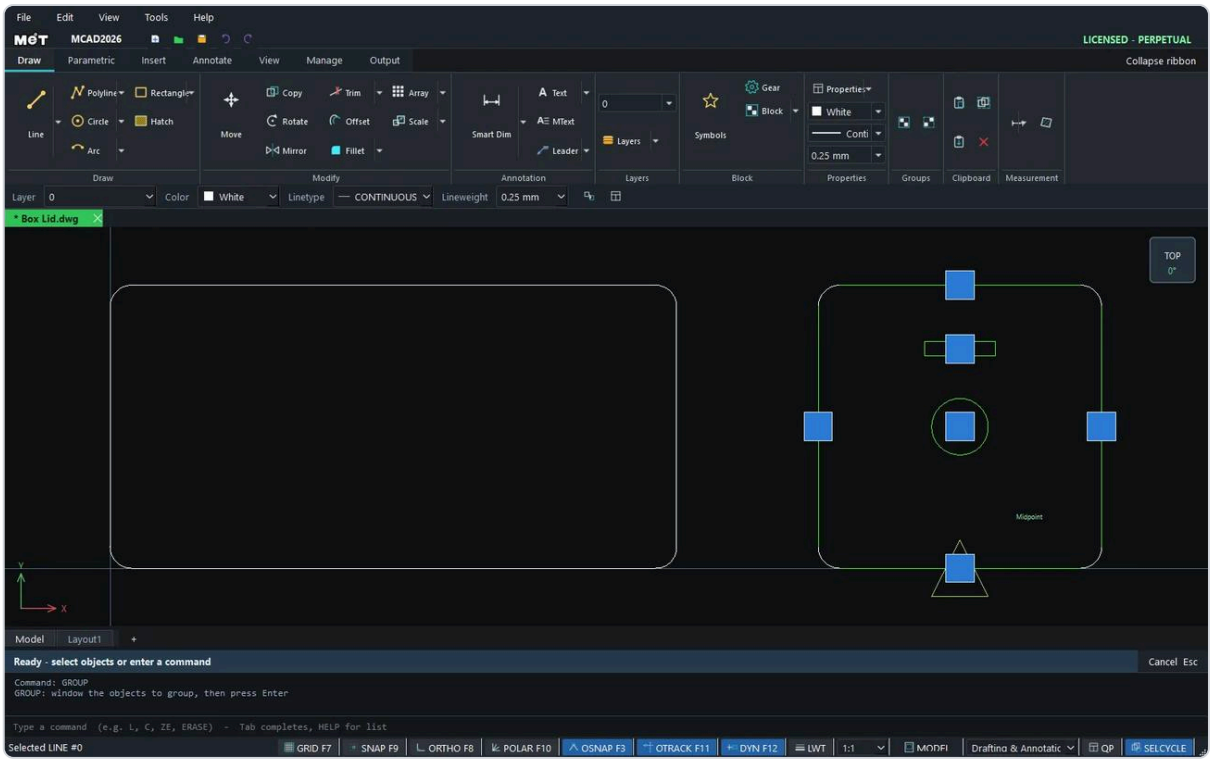
Command	Alias	Meaning
GROUP	`G`	Combines the selected objects (at least 2) into a group.
UNGROUP	`UNG`	Ungroups the selected group.
GROUPEDIT	`GROUPED`	`GROUPEDIT Add` adds the selected objects to the group. `GROUPEDIT Remove` removes the selected objects from it.

Where to find it

- Ribbon: **Draw › Groups › Group / Ungroup**
- Right-click the selected objects › Group (Block) / Ungroup (Explode)

Steps

1. Type `G` and press Enter. The prompt reads `GROUP: window the objects, then press Enter`.
2. Drag a window around the objects, then press Enter.
3. Click any one of them. The whole group is selected.

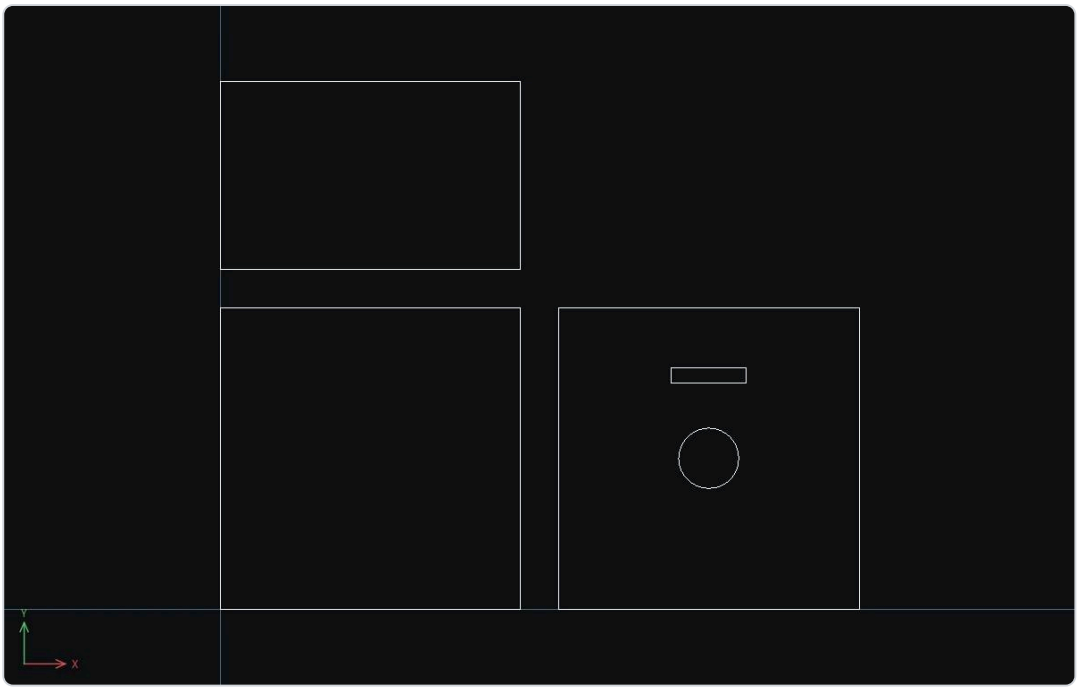


Clicking the plate edge selects the whole group

Try it: Wooden puzzle box panels

Three panels for a small puzzle box: the base, an end panel and a lid with a finger hole and a key slot. GROUP keeps the lid's pieces together, so one click moves all of them.

1. Base: `REC`, `0,0`, `@80,80`, Esc. End panel: `REC`, `0,90`, `@80,50`, Esc. Lid: `REC`, `200,0`, `@80,80`, Esc.
2. Finger hole: `C`, `240,40`, `8`, Esc. Key slot: `REC`, `230,60`, `@20,4`, Esc.
3. Type `G` and press Enter. Drag a window around the lid (from 195,-5 to 285,85) and press Enter.
4. Click the edge of the lid. The outline, the hole and the slot are all selected.
5. Type `M` and press Enter. Type the base point `200,0`, then the destination `@-110,0`. The whole lid moves next to the base, ready to nest on the sheet.



Base, end panel and grouped lid lined up for cutting

Challenge: Type `UNG` with the lid selected to ungroup it, then move only the key slot 10 mm higher.

See also

- BLOCK
- Selecting objects

Editing with grips and Quick Properties

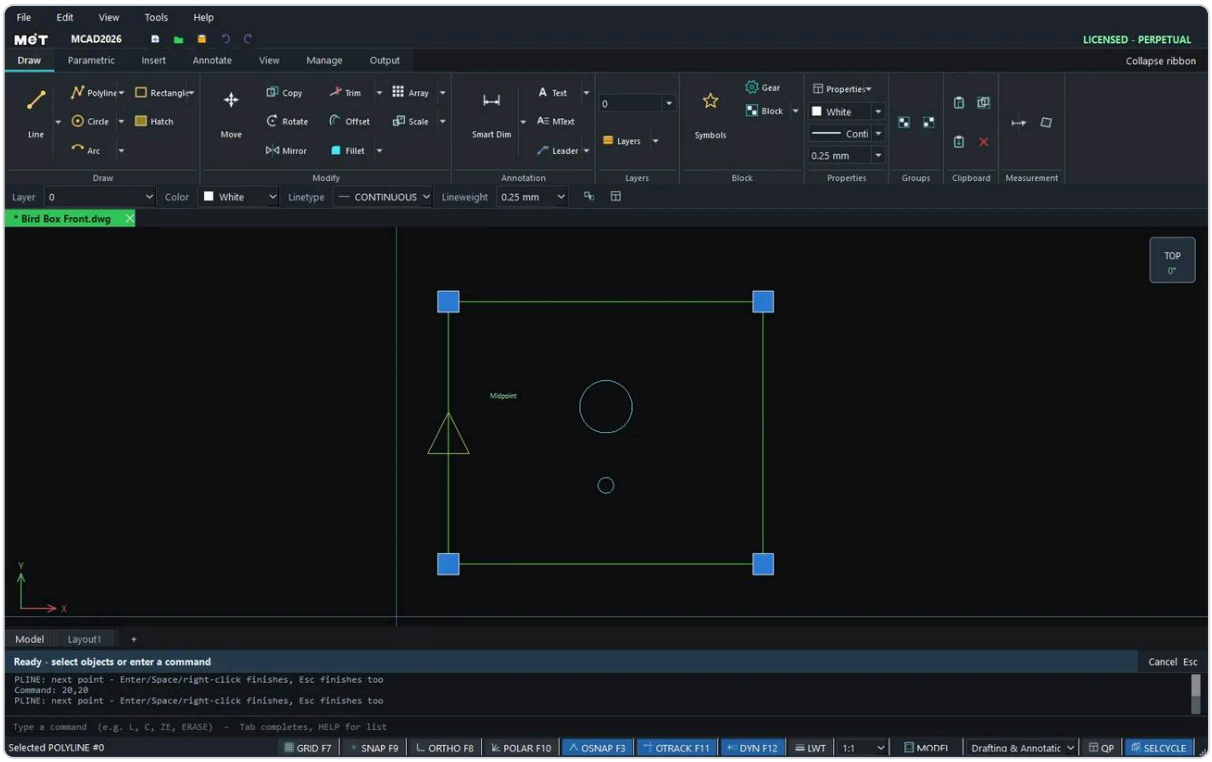
Reshape objects by dragging grips and edit properties from Quick Properties.

What it does

When you click an object, you see grips (blue squares) at its key points. Drag a grip to reshape the object without calling a command.

Steps

1. Click an object. The grips appear.
2. Click the grip you want and drag it. Object Snap works while you drag.
3. Click the new position, or type a distance in Smart Input.
4. While you drag a grip, press `Space` to switch the grip mode.
5. Press `Esc` to clear the selection.

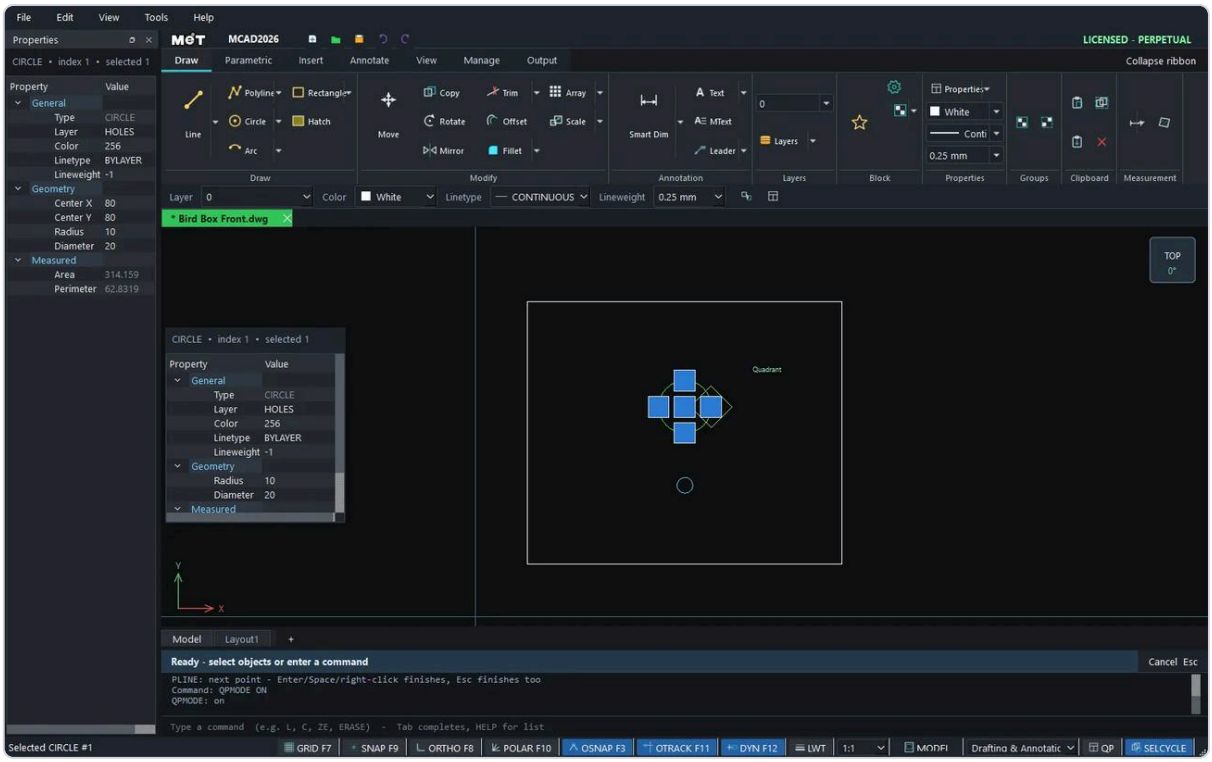


Grips on the selected outline

Object	Grips
Line	Both ends and the midpoint.
Circle	The centre and 4 quadrant points (dragging a quadrant changes the radius).
Polyline / Rectangle	Every vertex.
Table	Drag a grip to resize rows and columns.
Dimension	Drag the dimension line and text positions.

Quick Properties

Turn on `QPMODE ON` or press `Ctrl+Shift+P`, then click an object. A properties box appears at the cursor (Layer, Color, radius, length and more).

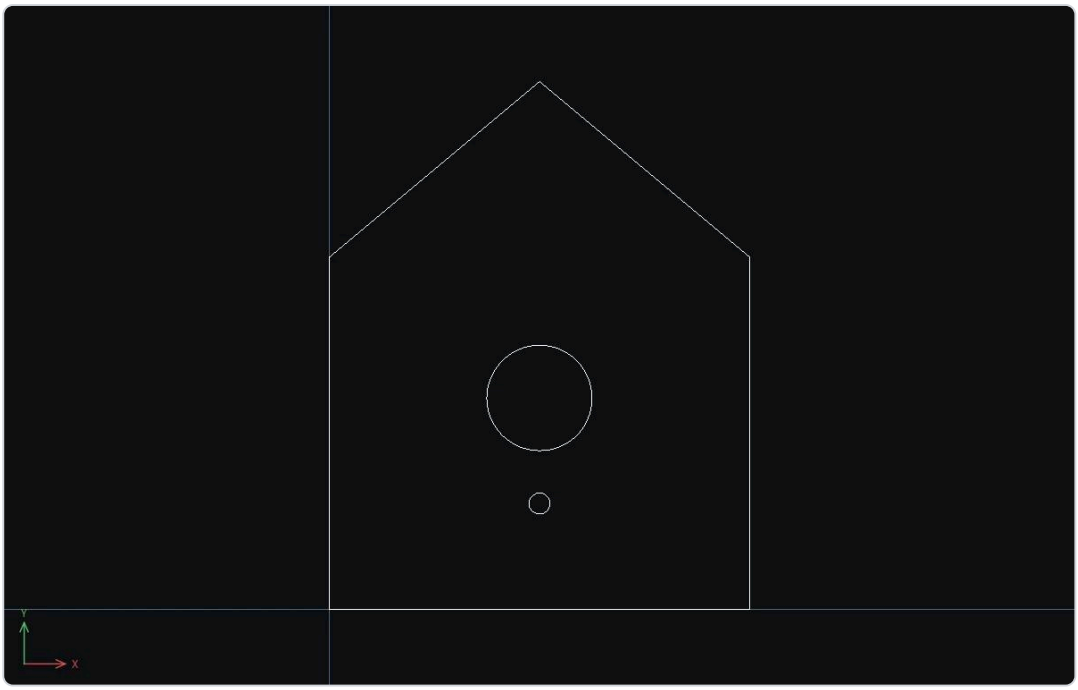


Quick Properties and the Properties palette for a circle

Try it: Birdhouse front panel

The front panel of a small birdhouse in 6 mm plywood. Start with a plain box, then pull the roof peak up and size the entrance hole by dragging grips. No commands needed.

1. Type `SNAP 5` so the grip drags land on a 5 mm grid.
2. Outline: `PL`, then `0,0` `@120,0` `@0,100` `@-60,0` `@-60,0` `0,0`, and press Enter. The extra point at 60,100 will become the roof peak.
3. Entrance hole: `C`, `60,60`, `10`, Esc. Perch hole: `C`, `60,30`, `3`, Esc.
4. Click the outline. Drag the grip in the middle of the top edge (at 60,100) up to 60,150. The box gets a gable roof.
5. Press Esc, click the entrance hole, and drag its right quadrant grip from 70,60 to 75,60. The hole is now Ø30.
6. Type `SNAP OFF`.



Birdhouse front panel with a gable roof and a Ø30 entrance

Challenge: With the entrance hole selected, press `Ctrl+1` and type a Radius of `16` in the Properties palette for a Ø32 entrance, the size sparrows like.

See also

- PROPERTIES
- STRETCH

Geometric Constraints (Parametric)

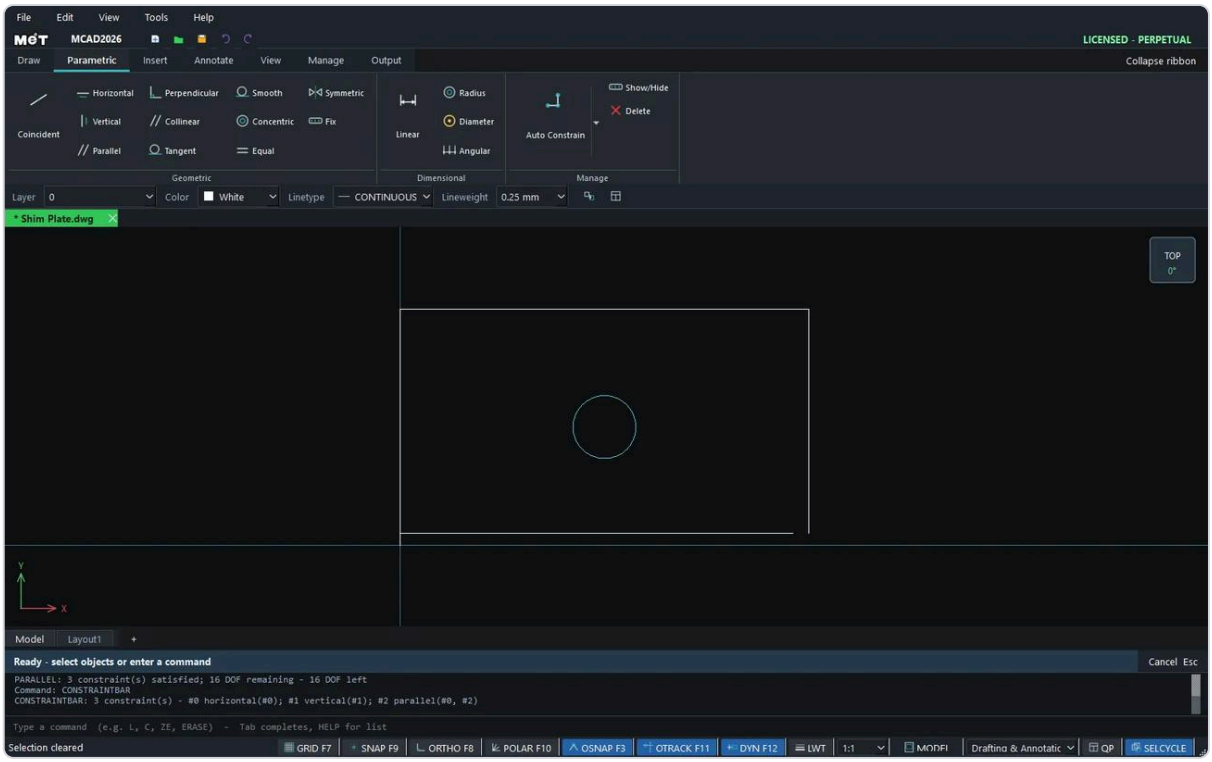
Add geometric constraints (horizontal, vertical, parallel, perpendicular, tangent and more) to a parametric 2D drawing.

What it does

A constraint forces a geometric relationship between objects. When you edit the drawing, the solver keeps that relationship. The result reports the remaining DOF (degrees of freedom), or `fully constrained`.

Where to find it

- Ribbon: **Parametric › Geometric** and **Parametric › Manage**



Parametric tab

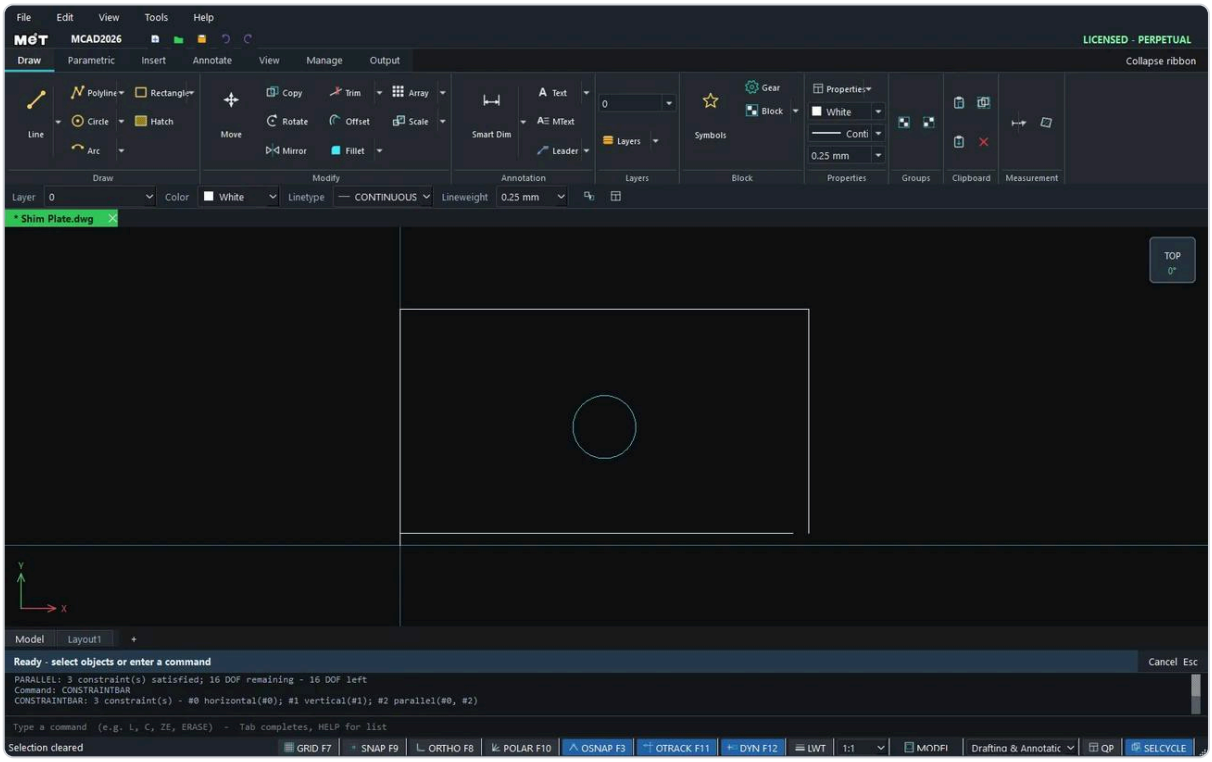
Commands

Command	Alias	Button	Meaning
GCHORIZONTAL	`GCH`	Horizontal	The selected line is horizontal.
GCVERTICAL	`GCV`	Vertical	The selected line is vertical.
GCPARALLEL	`GCPA`	Parallel	Two lines are parallel.
GCPERPENDICULAR	`GCPE`	Perpendicular	Two lines are perpendicular.
GCCOLLINEAR	`GCCOL`	Collinear	Two lines lie on the same line.
GCEQUAL	`GCE`	Equal	Equal length (lines) or equal radius (circles).
GCCOINCIDENT	`GCC`	Coincident	The start points of two objects coincide.
GCCONCENTRIC	`GCCO`	Concentric	Two circles share the same centre.
GCTANGENT	`GCT`	Tangent	Two objects are tangent.
GCSMOOTH	`GCSM`	Smooth	A smooth continuation between line and circle, or circle and circle.
GCSYMMETRIC	`GCSYM`	Symmetric	Two objects are symmetric about the selected axis line.
GCFIX	`GCF` , `FIXCONSTRAINT`	Fix	Fixes a line or circle in place.
GEOMCONSTRAINT	`GEOMCON`	–	`GEOMCONSTRAINT <type>` applies any of the types above.
AUTOCONSTRAIN	`AUTOC`	Auto Constrain	Automatically applies the constraints that are already true.

Command	Alias	Button	Meaning
CONSTRAINTBAR	`CBAR`	Show/Hide	Lists all constraints with their numbers (#0, #1, ...).
DELCONSTRAINT	`DELC`	Delete	`DELCONSTRAINT [n|all]` deletes number n, all of them, or the latest one.
CONSTRAINTSETTINGS	`CSETTINGS`	Auto Constrain ▾ › Inference ON/OFF	`CONSTRAINTSETTINGS INFER ON|OFF`

Steps

1. Select the objects (1 or 2, depending on the constraint type).
2. Click the button on the Parametric tab, or type the command, for example `GCH`.
3. The result reads, for example, `HORIZONTAL: 1 constraint(s) satisfied; 15 DOF remaining`.
4. If a constraint conflicts with an existing one, the program does not keep it (`constraint not kept`).



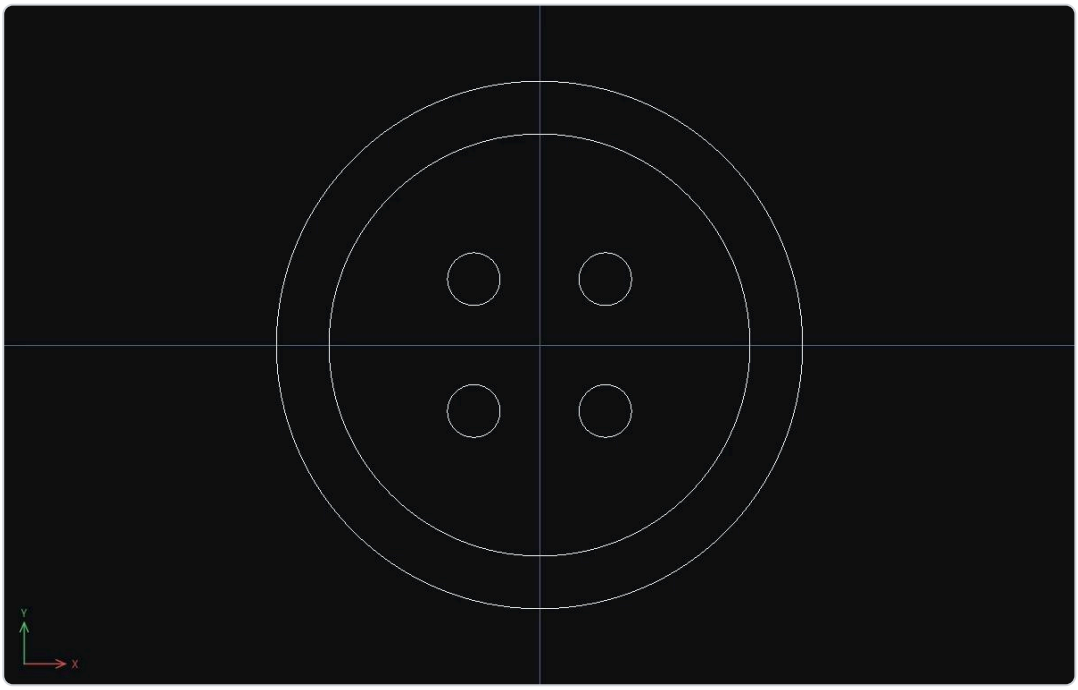
Constraint list after applying Horizontal, Vertical and Parallel

Try it: Big wooden coat button

A 40 mm plywood button for a chunky knitted coat. Draw it sloppily on purpose, then let the constraints line everything up.

1. Rim: `C`, `0,0`, `20`, Esc. Engraved ring, slightly off-centre: `C`, `1,0.5`, `16`, Esc.
2. Four holes in slightly different sizes: `C` `-5,5` `2`, `C` `5,5` `2.6`, `C` `-5,-5` `1.8`, `C` `5,-5` `2.3`. Press Esc after each one.
3. Click the rim and type `GCF`. The rim is fixed in place.
4. Click the rim, `Ctrl`+click the ring (click on the circle line, away from the blue grips), and type `GCCONCENTRIC`. The ring jumps to the centre.
5. Click the top-left hole, `Ctrl`+click the top-right hole, and type `GCE`. Repeat for the top-left hole with the bottom-left hole, and the top-left hole with the bottom-right hole. All four holes now have the same size.

6. Type `CBAR`. The list shows 5 constraints: fixed, concentric and three equal.



Coat button with a centred ring and four equal holes

Challenge: Click any hole and type `DCRADIUS 2`. All four holes follow, because they are constrained equal.

See also

- Dimensional Constraints

Dimensional Constraints (DCLINEAR, DCRADIUS ...)

Add constraints that hold a value (distance, radius, diameter, angle) and convert dimensions into constraints.

What it does

A dimensional constraint forces a size to a value you set. When you change the value, the shape changes with it.

Where to find it

- Ribbon: **Parametric › Dimensional** (Linear, Radius, Diameter, Angular) and **Auto Constrain ▾ › Convert dimensions (DCCONVERT)**

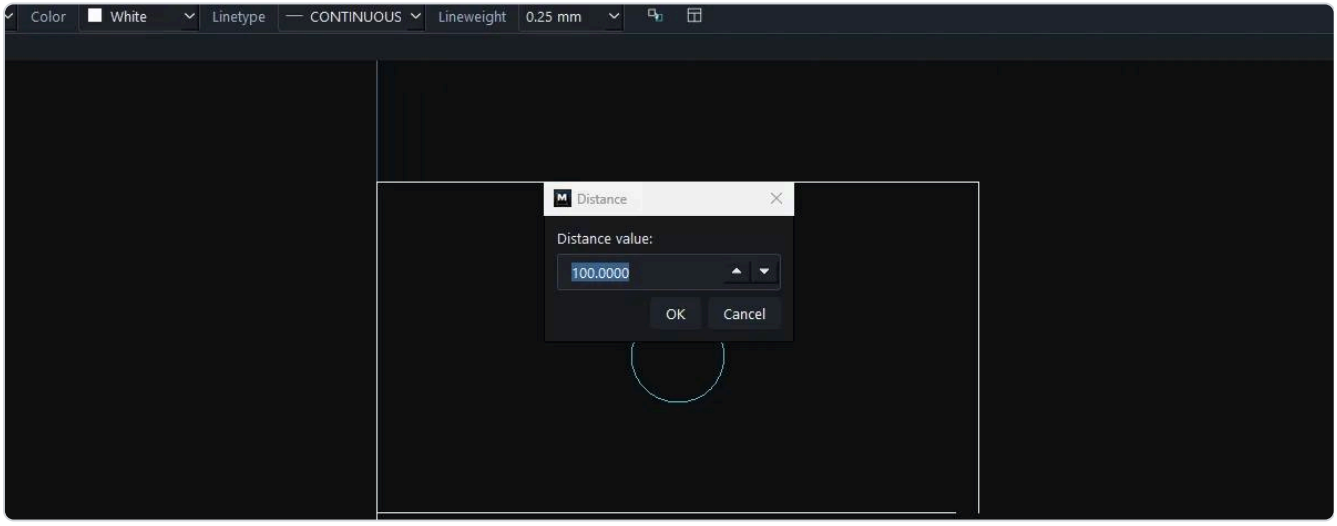
Commands

Command	Alias	Meaning
DCLINEAR	`DCL`	The distance between the start points of two lines. Select exactly 2 lines; it does not set the length of a single line.
DCRADIUS	`DCR`	The radius of a circle.
DCDIAMETER	`DCD`	The diameter of a circle.
DCANGULAR	`DCA`	The angle between two lines.

Command	Alias	Meaning
DIMCONSTRAINT	`DIMCON`	`DIMCONSTRAINT <distance|radius|diameter|angle> [value]`
DCCONVERT	–	Converts dimensions that are attached to objects into constraints. Free dimensions placed by clicking points cannot be converted.

Steps

1. Select the objects.
2. Type `DCRADIUS 25` and press Enter (give the value after the command), or click the button on the Ribbon and enter the value in the dialog that opens.
3. The solver adjusts the shape and reports the remaining DOF.

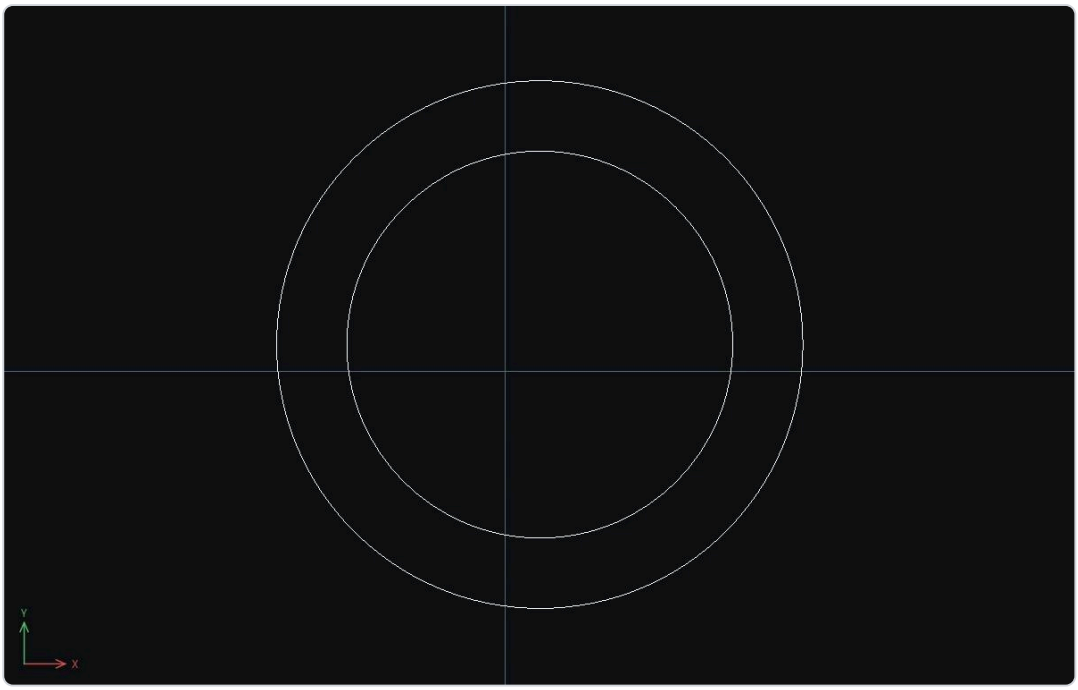


Distance value dialog for DCLINEAR

Try it: Desk cable grommet ring

An acrylic ring that dresses up the cable hole in a desk: 60 mm outside, with an inner hole sized to your cables. Sketch two circles at any size, then let the constraints set the exact numbers, and change them later.

1. Draw two circles that are deliberately a bit off: `C`, `0,0`, `27`, Esc. Then `C`, `4,3`, `17`, Esc.
2. Click the outer circle and type `DCDIAMETER 60`. The result reads `DIAMETER: 1 constraint(s) satisfied; 5 DOF remaining`.
3. Click the inner circle and type `DCDIAMETER 40`.
4. Click the outer circle, `Ctrl`+click the inner circle, and type `GCCONCENTRIC`. Both circles now share one centre.
5. The power plug needs a bigger hole: type `CBAR` to see the list (`#0 diameter(#0) = 60; #1 diameter(#1) = 40; #2 concentric(#0, #1)`), then `DELCONSTRAINT 1`. Click the inner circle and type `DCDIAMETER 44`. The ring updates and stays centred.



Grommet ring with Ø60 outer and Ø44 inner circles

Challenge: Make the ring 70 mm outside the same way: `CBAR` to find the number, `DELCONSTRAINT` it, then `DCDIAMETER 70` on the outer circle.

See also

- Geometric Constraints
- Smart Dimension

Chapter 4 quiz: Modify

Choose the one best answer. Answers are in Appendix A

1. **In OFFSET, what does typing `B 10` do?**

   A. Offsets 10 units toward the side you click
   B. Creates copies 10 units away on both sides without clicking a side
   C. Creates 10 offset copies
   D. Uses the previous distance

2. **What happens when you FILLET two lines with a radius of 0?**

   A. Nothing happens
   B. Both lines are deleted
   C. A 45° bevel is cut
   D. The lines are joined at a sharp corner

3. **What does dragging a selection box from left to right (Window, blue) select?**

   A. Only objects that are completely inside the box
   B. Objects that cross or are inside the box
   C. Every object on the current layer
   D. Only the most recently created object

4. **Which command brings back the objects removed by the last ERASE without undoing what you drew afterwards?**

   A. UNDO
   B. REDO
   C. OOPS
   D. SELECTPREVIOUS

5. **You select a closed rectangle as a boundary and want to keep what is inside it and trim everything outside. Which command do you use?**

   A. EXTRIM
   B. EXTRIM Outside (EXTRIM O)
   C. EXTEND
   D. BREAK

6. **In `ARRAYPOLAR 8`, if you do not enter a total angle, what angle is used?**

   A. 90°
   B. 180°
   C. 270°
   D. 360°

CHAPTER 5

Dimension & Annotation

1. Smart Dimension (DIM)
   Add smart dimensions with a single command: edge length, distance between edges, angle, diameter, and point-to-point distance.
2. DIMLINEAR, DIMALIGNED and DIMVERTICAL
   Horizontal dimensions (DIMLINEAR), vertical dimensions (DIMVERTICAL) and dimensions parallel to the line between two points (DIMALIGNED).
3. DIMANGULAR, DIMRADIUS, DIMDIAMETER, DIMARC, DIMJOGGED
   Dimension angles (DIMANGULAR), radii (DIMRADIUS), diameters (DIMDIAMETER), arc lengths (DIMARC) and jogged radii (DIMJOGGED).
4. DIMORDINATE - Ordinate Dimension
   Ordinate dimension - show the X or Y coordinate of a point relative to a datum.
5. QDIM, DIMCONTINUE and DIMBASELINE
   Add a whole row of dimensions at once: continuous, baseline, staggered, ordinate, radius, or diameter.
6. Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)
   Edit dimension text, position, arrowheads, prefix/suffix, break gaps and spacing.
7. DIMSTYLE and STYLE - Style Manager
   Manage text styles and dimension styles - font, height, arrows, decimals, units and colour.
8. CENTERMARK and CENTERLINE
   Draw a centre mark (CENTERMARK) on circles and arcs, and a centreline (CENTERLINE) between two lines.
9. QLEADER, LEADER, MLEADER and Callout
   Arrows with notes: QLEADER/LEADER, Multileader (MLEADER), and Callout boxes.
10. TOLERANCE - GD&T Feature Control Frame
    Place a GD&T Feature Control Frame with symbol, tolerance value, and datums.
11. TEXT, MTEXT and TEXTEDIT
    Type single-line text (TEXT) and multiline text (MTEXT) directly in the drawing area, and edit text (TEXTEDIT).
12. Text Tools (ARCTEXT, TALIGN, TCOUNT, FIND ...)
    Arc-shaped text, align text to a line, number text, justify, find/replace, and spell check.
13. TABLE and DATALINK - Tables
    Create tables (TABLE), fill in cells, export to CSV, and link to Excel files (DATALINK).
14. FIELD and UPDATEFIELD
    Text that calculates its own value (FIELD), such as the date, file name or area, and how to update it (UPDATEFIELD).

15. Annotation Scale (OBJECTSCALE, ANNOALLVISIBLE)

Set the annotation scale so text and dimensions display at the right size for the drawing scale (OBJECTSCALE, ANNOALLVISIBLE).

Smart Dimension (DIM)

Add smart dimensions with a single command: edge length, distance between edges, angle, diameter, and point-to-point distance.

What it does

Smart Dimension (`DIM`) picks the dimension type automatically from what you click, so you do not need to switch commands.

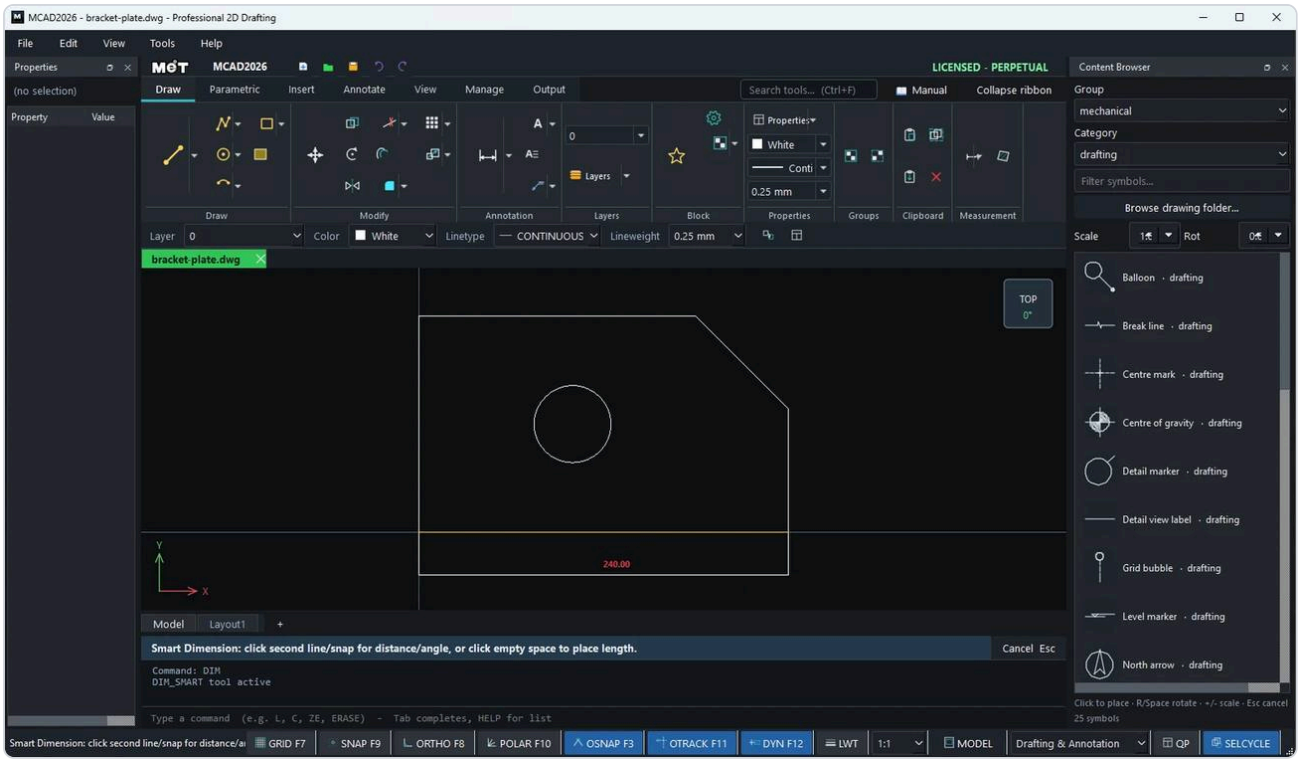
You click	You get
One edge, then empty space	The length of that edge
One edge, then a parallel edge	The distance between the edges
One edge, then an edge at an angle	The angle
The circumference of a circle/arc	Diameter/radius
An endpoint/center, then another point or edge	Point-to-point or point-to-edge distance (move up/down = X, left/right = Y, diagonal = Aligned)

Where to find it

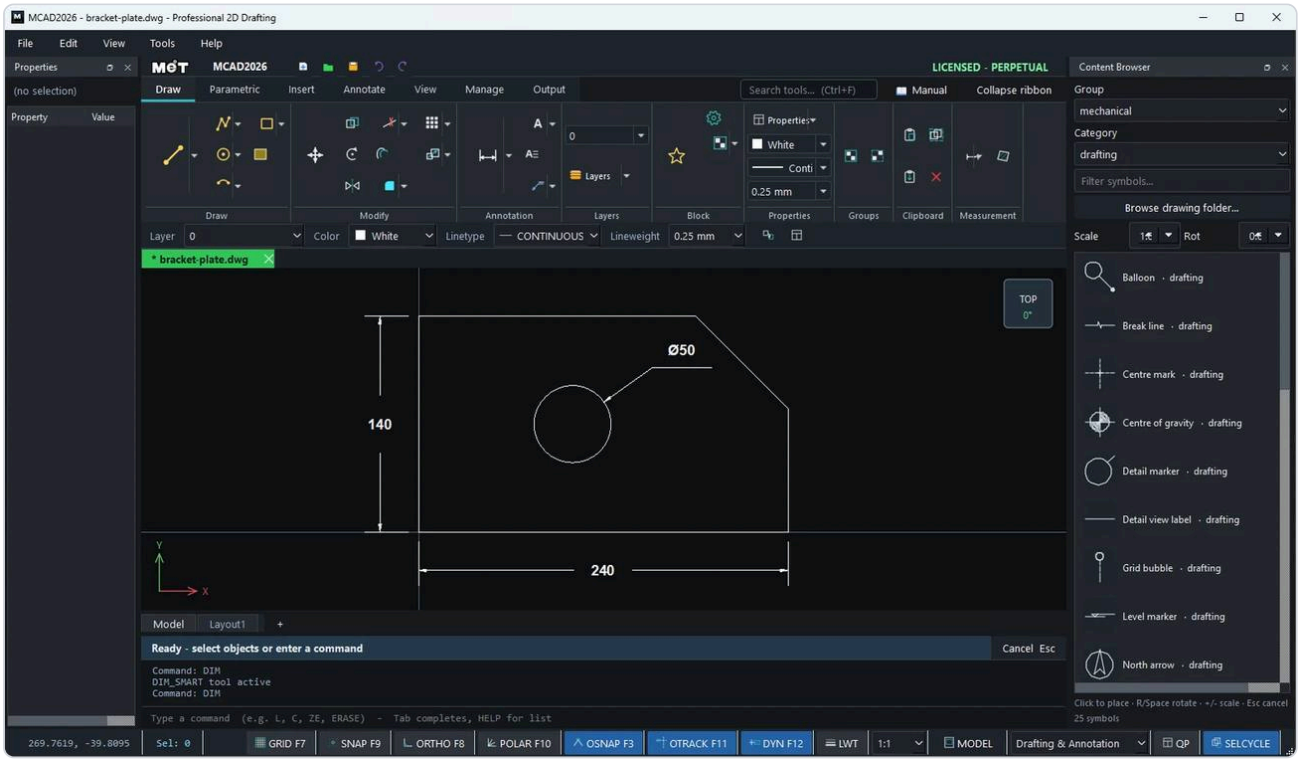
- Ribbon: **Draw › Annotation › Smart Dim** or **Annotate › Dimensions › Dimension**
- Command line: `DIM` (Alias `DIMENSION`)
- Right-click › Smart Dimension

Steps

1. Type `DIM` and press Enter.
2. Click the first edge. The Prompt bar shows: `Smart Dimension: click second line/snap for distance/angle, or click empty space to place length.`
3. Click the second edge, or click empty space to measure the length of a single edge.
4. The dimension attaches to the mouse: `move text/dim line, type a value to override, left-click or Enter to place, Esc to cancel.` Move it to the position you want and click.
5. You can start the next dimension right away. Press Esc to finish.



After clicking the bottom edge, the length of the edge is shown



Dimensions 240, 140, and Ø50 from Smart Dimension

Options while placing

Action	Result
Type a number	Replaces the displayed value (Override)
`R` (angle dimension)	Switches to the reflex angle
Move inside / outside the angle	Choose the interior angle or the supplementary angle

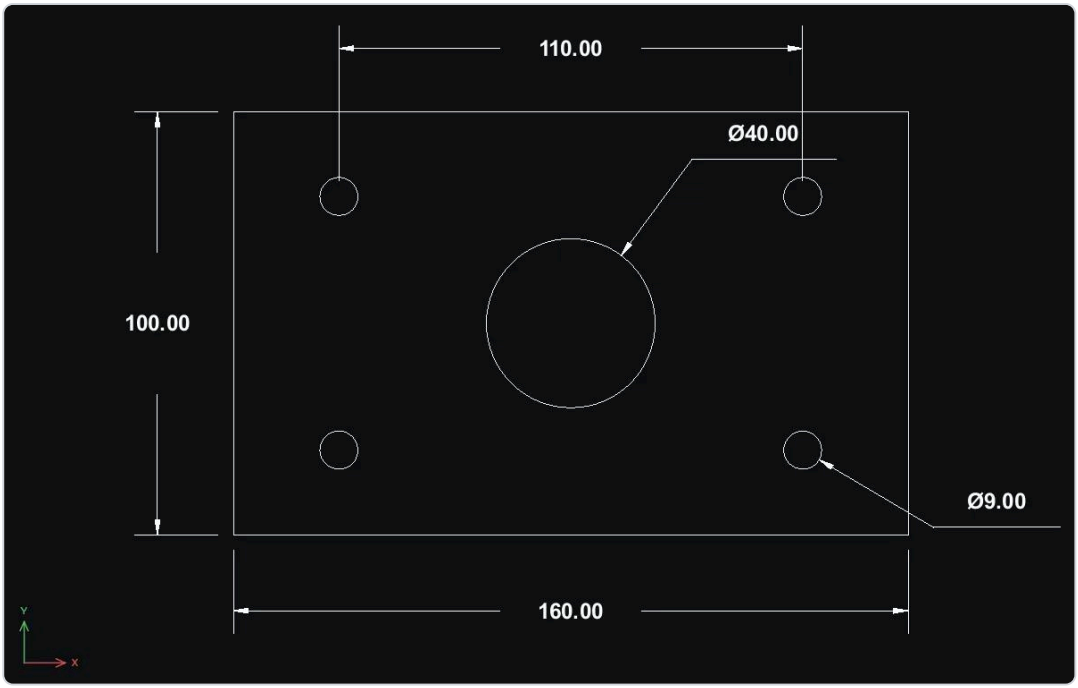
Action	Result
Enter / click	Place
Esc	Cancel

> **Note:** `DIMSNAP OFF` makes Smart Dimension measure objects only, without snapping to End/Mid/Center points. The default is ON, and it works at close range, so zoom in to snap to points. You can move a dimension later by clicking it and dragging.

Try it: Motor mounting plate

A 160 x 100 steel plate that carries a small gear motor: a Ø40 bore for the shaft boss and four Ø9 bolt holes. One `DIM` command handles lengths, diameters and centre distances without switching tools.

1. On **Annotate › Dim Defaults**, set **Dim H** `6` and **Arrow** `3.5` so the numbers suit a part this size.
2. Plate: `REC` `0,0` → `160` `Tab` `100`.
3. Shaft bore: `C` `80,50` radius `20`. Bolt holes: `C` radius `4.5` at `25,20`, `135,20`, `25,80` and `135,80`.
4. Type `DIM`. Click the bottom edge, click empty space, then move the dimension below the plate and click: `160.00`.
5. Click the left edge, click empty space, and place it on the left: `100.00`.
6. Click the edge of the bore and place the label up and to the right: `Ø40.00`. Do the same on the lower-right hole: `Ø9.00`.
7. Zoom in on the top holes. Click the centre of the top-left hole (wait for the Center snap marker), then the centre of the top-right hole, and move the mouse up before you click: `110.00` between the bolt centres. Press Esc to finish.



Motor plate dimensioned with Smart Dimension: 160, 100, 110, Ø40 and Ø9

Challenge: Add the vertical distance between the hole centres the same way, but move the mouse sideways this time. You should get `60.00`.

See also

- DIMLINEAR, DIMALIGNED, DIMVERTICAL
- DIMSTYLE
- Editing Dimensions

DIMLINEAR, DIMALIGNED and DIMVERTICAL

Horizontal dimensions (DIMLINEAR), vertical dimensions (DIMVERTICAL) and dimensions parallel to the line between two points (DIMALIGNED).

What it does

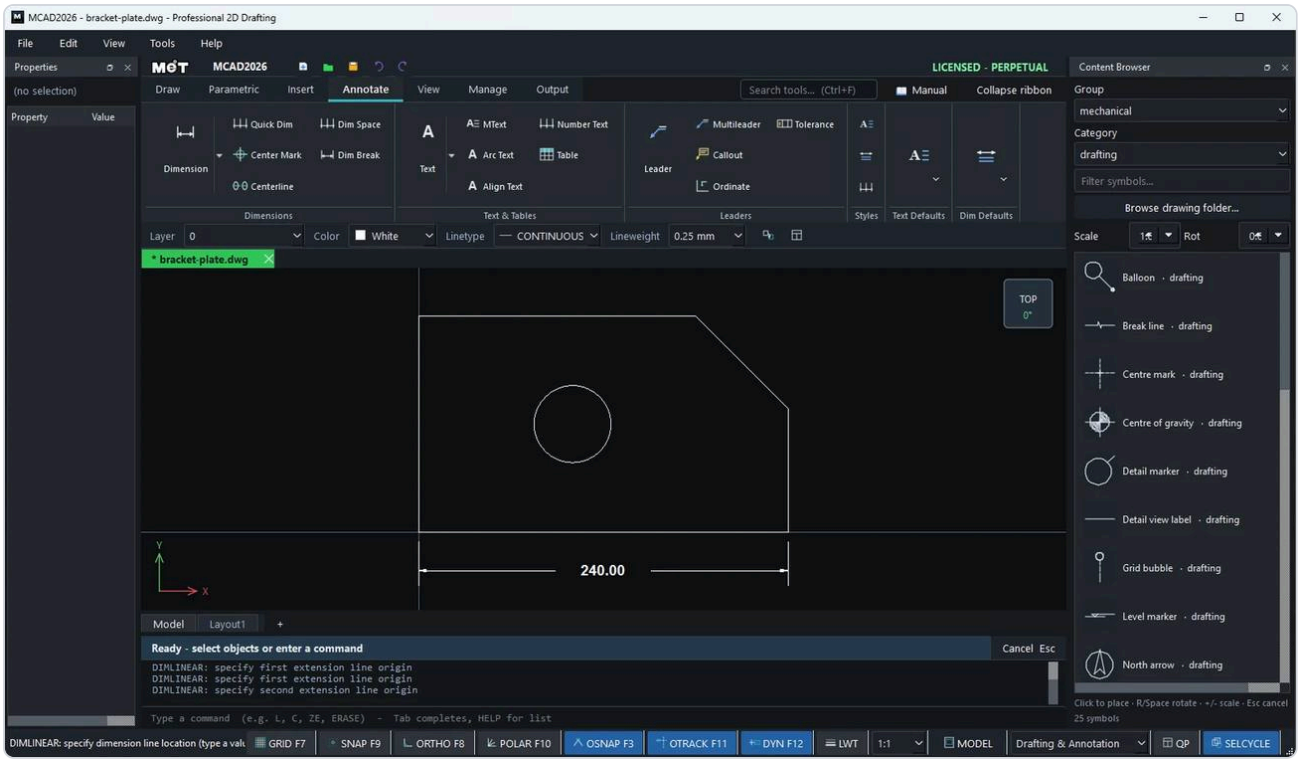
Command	Alias	Meaning
DIMLINEAR	`DLI`	The horizontal distance between two points.
DIMVERTICAL	`DVE`	The vertical distance between two points.
DIMALIGNED	`DAL`	The true distance along the line that joins two points.

Where to find it

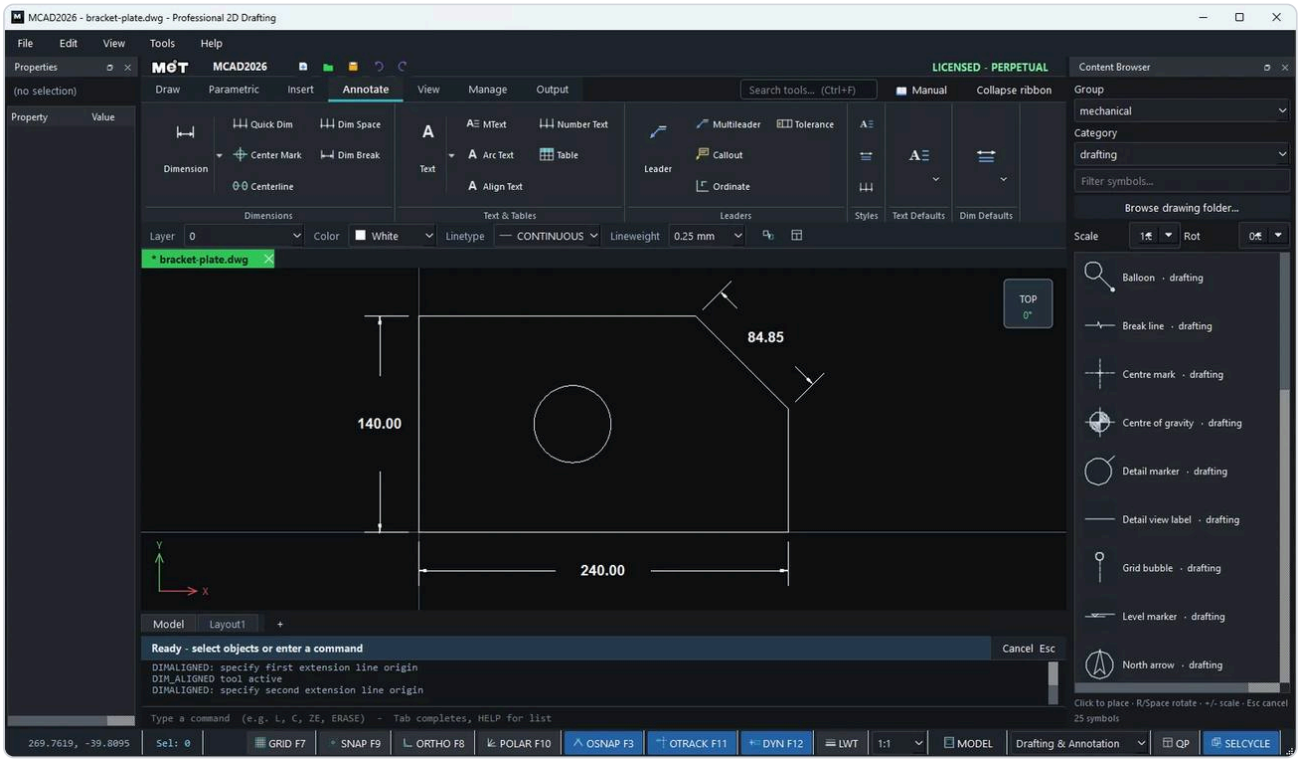
- Command line: see the table above. On the Ribbon, use Smart Dim, which chooses the type for you.

Steps

1. Type `DLI` and press Enter.
2. Click the first point (Object Snap picks End/Mid/Center).
3. Click the second point.
4. Move the mouse and click the position of the dimension line.



DIMLINEAR 240.00 placed below the plate by the third click (the dimension line position)



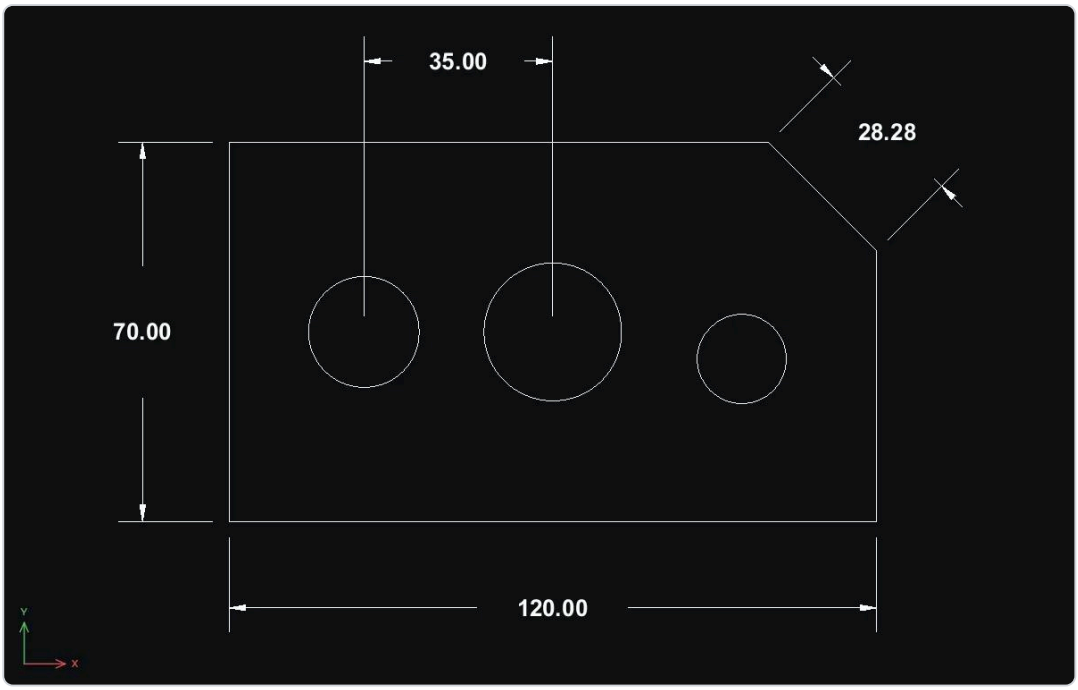
DIMLINEAR 240, DIMVERTICAL 140 and DIMALIGNED 84.85

Try it: Cable gland plate with a chamfered corner

A gland plate for the bottom of a control enclosure: 120 x 70 with one corner cut at 45° and holes for M20, M25 and M16 cable glands. DIMLINEAR, DIMVERTICAL and DIMALIGNED each get a turn.

1. On **Annotate › Dim Defaults**, set **Dim H** `5` and **Arrow** `3` so the numbers suit a part this size.
2. Outline: `PL`, then `0,0` `120,0` `120,50` `100,70` `0,70` `0,0` (Enter after each) and Enter to finish.

3. Gland holes: `C` `25,35` radius `10.25`, `C` `60,35` radius `12.75`, `C` `95,30` radius `8.25`.
4. `DLI`, click the corners `0,0` and `120,0`, then click below the plate: `120.00`.
5. `DVE`, click `0,0` and `0,70`, then click to the left: `70.00`.
6. `DAL`, click both ends of the chamfer and click outside the corner: `28.28`, the true length of the 45° cut.
7. `DLI`, click the centres of the M20 and M25 holes and click above the plate: `35.00`.



Gland plate dimensioned with DIMLINEAR 120 and 35, DIMVERTICAL 70 and DIMALIGNED 28.28

Challenge: Dimension the chamfer with `DLI` instead of `DAL`. You only get its horizontal part, `20.00`.

See also

- Smart Dimension
- QDIM, DIMCONTINUE, DIMBASELINE

DIMANGULAR, DIMRADIUS, DIMDIAMETER, DIMARC, DIMJOGGED

Dimension angles (DIMANGULAR), radii (DIMRADIUS), diameters (DIMDIAMETER), arc lengths (DIMARC) and jogged radii (DIMJOGGED).

What it does

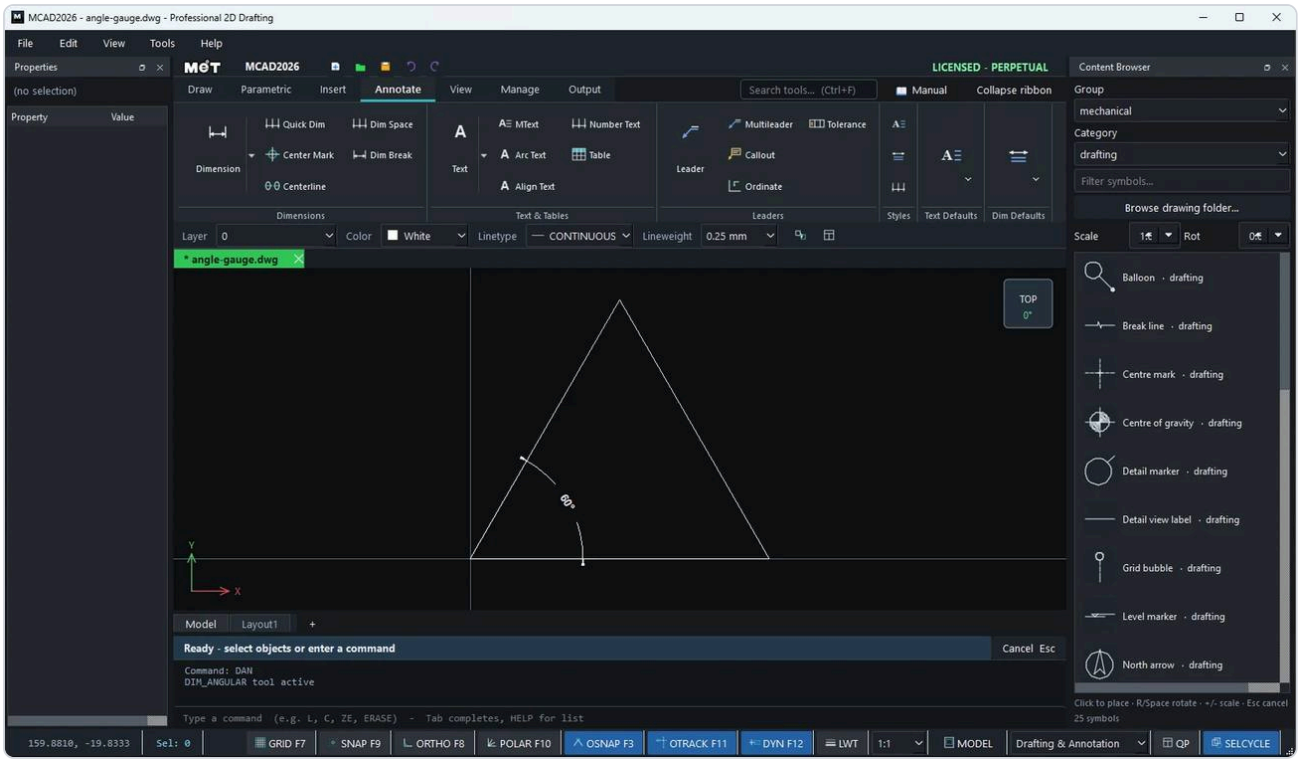
Command	Alias	Steps
DIMANGULAR	`DAN`	Click the angle vertex → a point on arm 1 → a point on arm 2 (the dimension is placed at once).
DIMRADIUS	`DRA`	Click a circle or arc → the position.
DIMDIAMETER	`DDI`	Click a circle or arc → the position.
DIMARC	`DAR`	Click an arc → the position (shows the arc length).
DIMJOGGED	`DJO`, `JOG`	Adds a jog symbol (zigzag) to the selected radius dimension (or to all of them). `DIMJOGGED` `Remove` removes it.

Where to find it

- Ribbon: **Draw › Annotation › Smart Dim ▾ › Angle between 2 lines**
- Command line: see the table above

Steps for DIMANGULAR

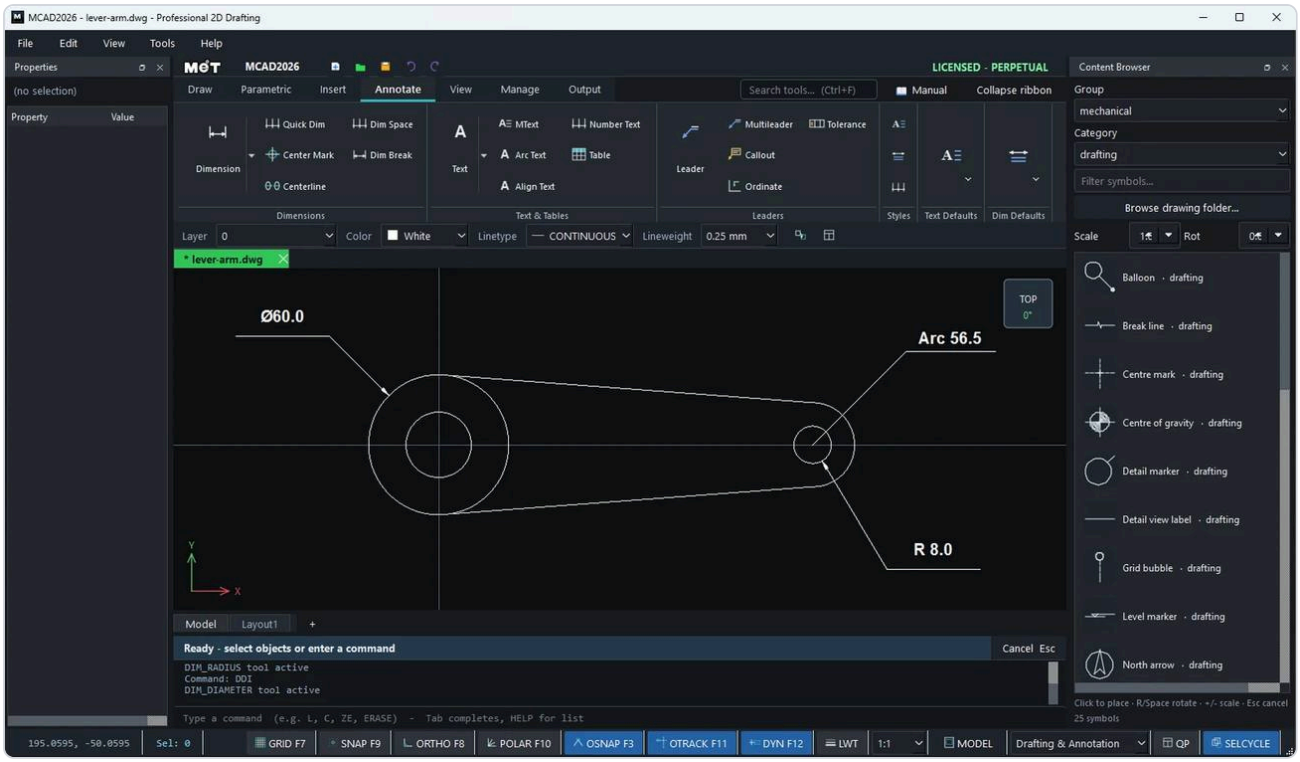
1. Type `DAN` and press Enter.
2. Click the angle vertex, then click on each arm.
3. The dimension is placed as soon as you click the second arm. To move the text, drag it with the mouse.



DIMANGULAR between two lines

Steps for DIMRADIUS / DIMDIAMETER / DIMARC

1. Type the command and press Enter.
2. Click the circumference or the arc.
3. At `Pick where to place the dimension.`, click the position.



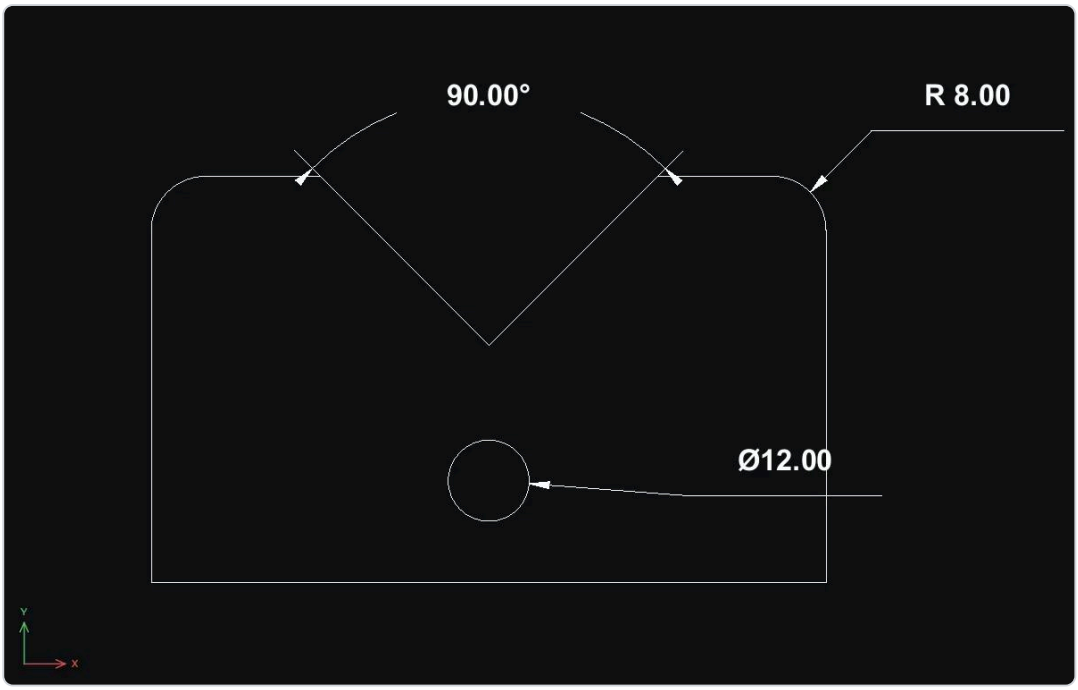
DIMARC on an arc, DIMRADIUS and DIMDIAMETER on circles

> **Note:** With **Angle between 2 lines** on the Ribbon, you click the two lines anywhere. You do not need to click the vertex.

Try it: V-block with a cross hole

A 100 x 60 V-block for holding round bar on the drill press: a 90° V, two R8 top corners and a Ø12 cross hole. One angle, one diameter and one radius say it all.

1. On **Annotate › Dim Defaults**, set **Dim H** `5` and **Arrow** `3` so the numbers suit a part this size.
2. Outline with `L`: `0,0` `100,0` `100,60` `75,60` `50,35` `25,60` `0,60` `0,0`, then Esc.
3. `F` `8` Enter, then round the two top outer corners (click the top line and the side line at each corner).
4. Cross hole: `C` `50,15` radius `6`.
5. `DAN`, click the bottom of the V (`50,35`), click a point on each face of the V: `90.00°`.
6. `DDI`, click the cross hole and place the label to the right: `Ø12.00`.
7. `DRA`, click the top-right fillet and place it outside the block: `R 8.00`.



V-block with the 90° V, the R8 fillet and the Ø12 cross hole dimensioned

Challenge: Run `DAR` on the same fillet. It shows the arc length instead, 12.57 for a quarter circle of R8.

See also

- Smart Dimension
- CENTERMARK

DIMORDINATE - Ordinate Dimension

Ordinate dimension - show the X or Y coordinate of a point relative to a datum.

What it does

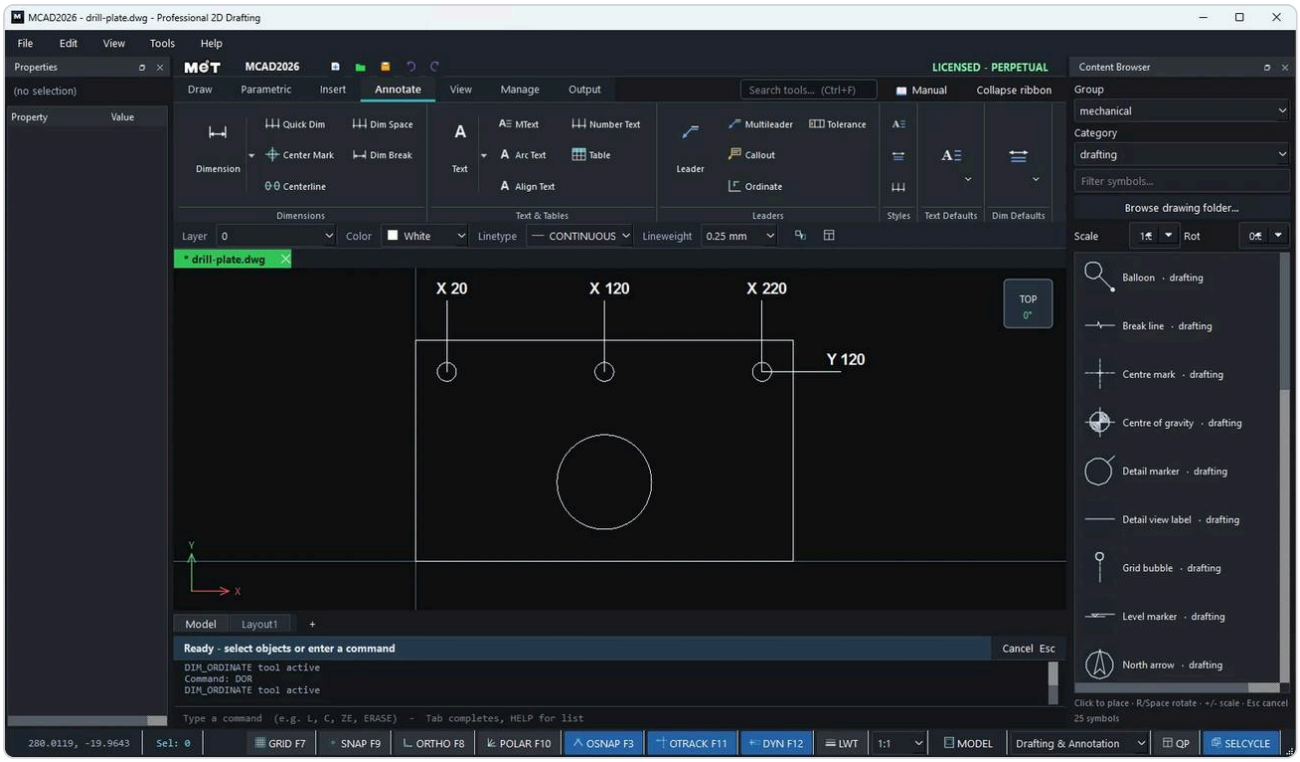
DIMORDINATE shows the X or Y coordinate of a point (for example, the centre of a hole) relative to the origin. It is often used on plates with many holes.

Where to find it

- Ribbon: **Draw › Annotation › Leader ▾ › Ordinate** or **Annotate › Leaders › Ordinate**
- Command line: `DIMORDINATE` (alias `DOR`)

Steps

1. If needed, move the origin to the corner of the part with `UCS ORIGIN x,y`.
2. Type `DOR` and press Enter.
3. Click the point you want to dimension.
4. Drag down or up, then click to show the **X** value. Drag left or right, then click to show the **Y** value.
5. Repeat the command (press Enter) for the next point.

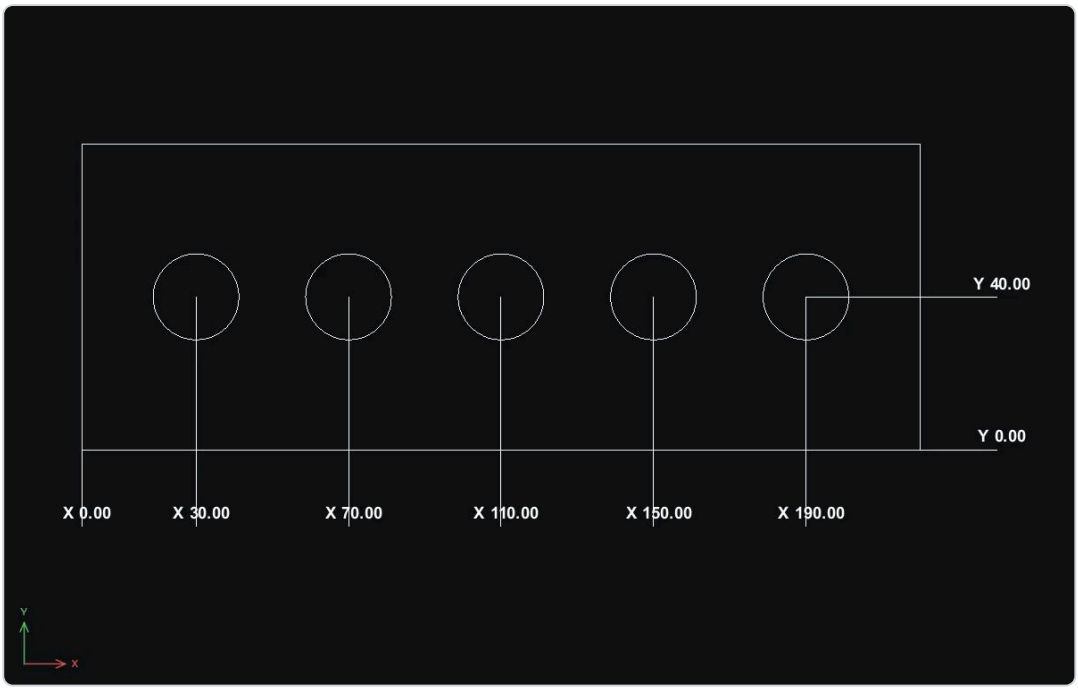


Ordinate X 20, X 120, X 220 and Y 120

Try it: Push-button panel insert

A 220 x 80 insert plate for a control desk with five Ø22.5 holes for 22 mm push-buttons. Ordinate dimensions give every hole centre straight from the zero corner, which is how a CNC operator wants to read it.

1. On **Annotate › Dim Defaults**, set **Dim H** `5` and **Arrow** `3` so the numbers suit a part this size.
2. `REC` `0,0` → `220` `Tab` `80`.
3. First hole: `C` `30,40` radius `11.25`. Select it and type `ARRAYRECT 5 40 0` for holes at X 30, 70, 110, 150 and 190.
4. `DOR`, click the corner `0,0`, drag down and click: `X 0.00`. Press Enter to repeat the command and do the same on the centre of each hole.
5. `DOR` on the centre of the last hole, drag to the right and click: `Y 40.00`. Do the same on the corner `220,0`: `Y 0.00`.



Push-button insert with ordinate dimensions X 0 to X 190 and Y 0, Y 40

Challenge: Add two Ø5 fixing holes at `10,10` and `210,70` and give them X and Y ordinates as well.

See also

- QDIM Ordinate
- Zoom and UCS

QDIM, DIMCONTINUE and DIMBASELINE

Add a whole row of dimensions at once: continuous, baseline, staggered, ordinate, radius, or diameter.

What it does

QDIM creates several dimensions at once from the selected objects.

Command	Alias	Meaning
QDIM	–	`QDIM [C|S|B|O|R|D]`
DIMCONTINUE	`DCO`	Same as `QDIM C` (continuous)
DIMBASELINE	`DBA`	Same as `QDIM B` (baseline)

Option	Meaning
C	Continuous: dimensions chained end to end (default)
S	Staggered
B	Baseline: all dimensions measured from the same base
O	Ordinate
R	Radius for every selected circle

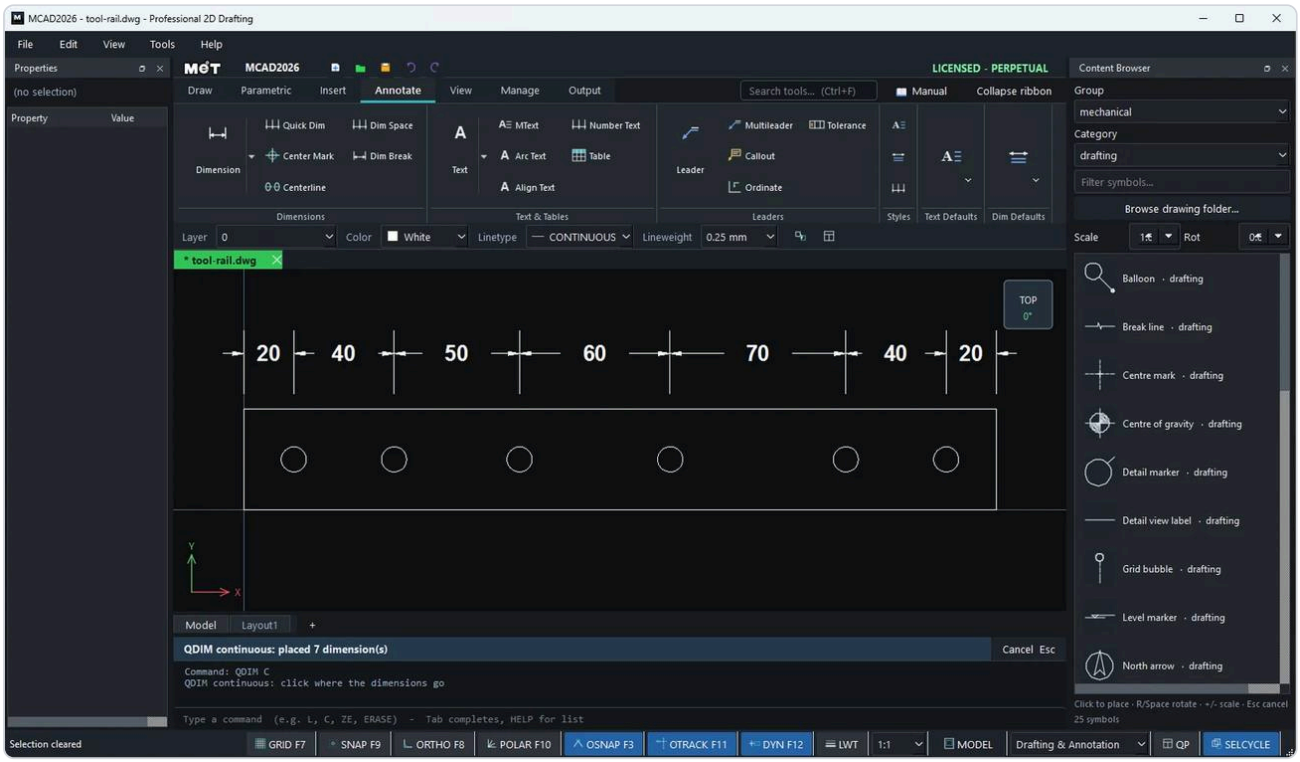
Option	Meaning
D	Diameter for every selected circle

Where to find it

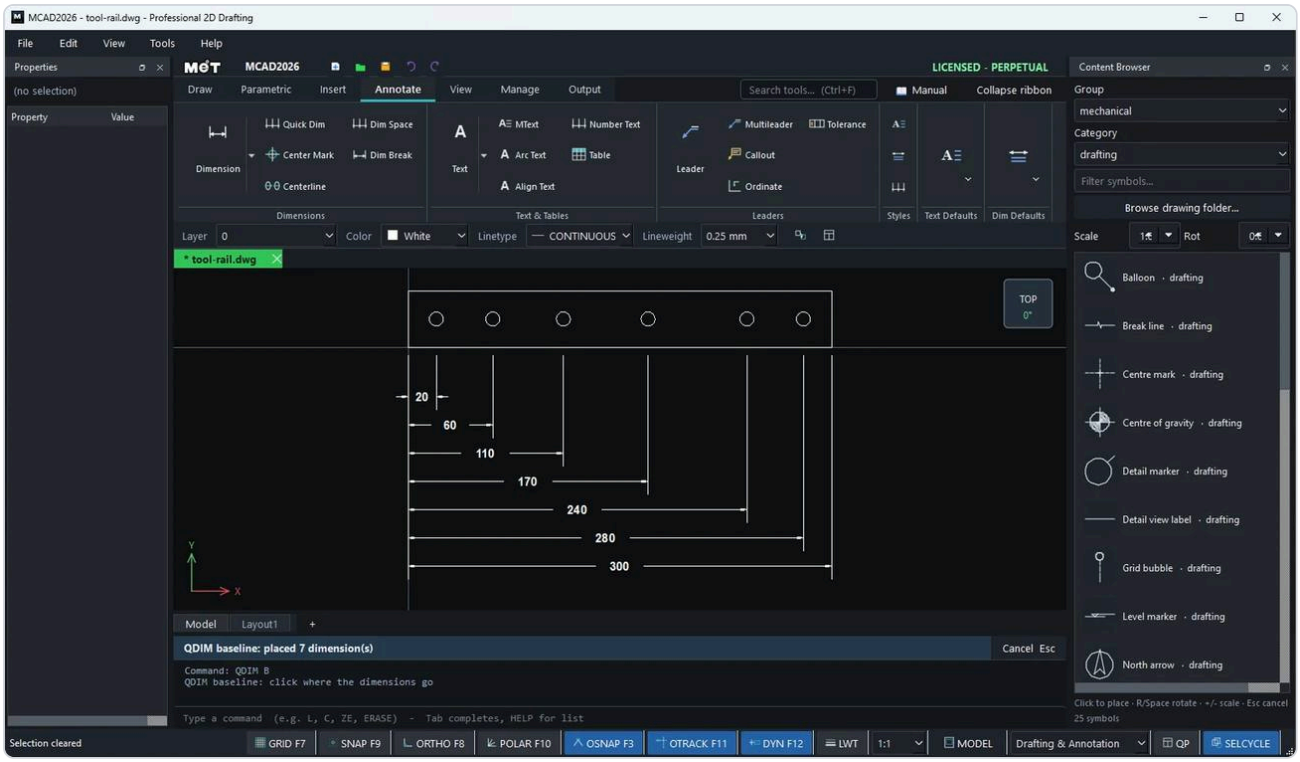
- Ribbon: **Annotate › Dimensions › Quick Dim**
- Command line: `QDIM`

Steps

1. Select the lines or objects you want to dimension.
2. Type `QDIM C` (or `DCO`) and press Enter.
3. `QDIM continuous: click the side where the dimensions go` Click the side where you want to place them.



QDIM Continuous

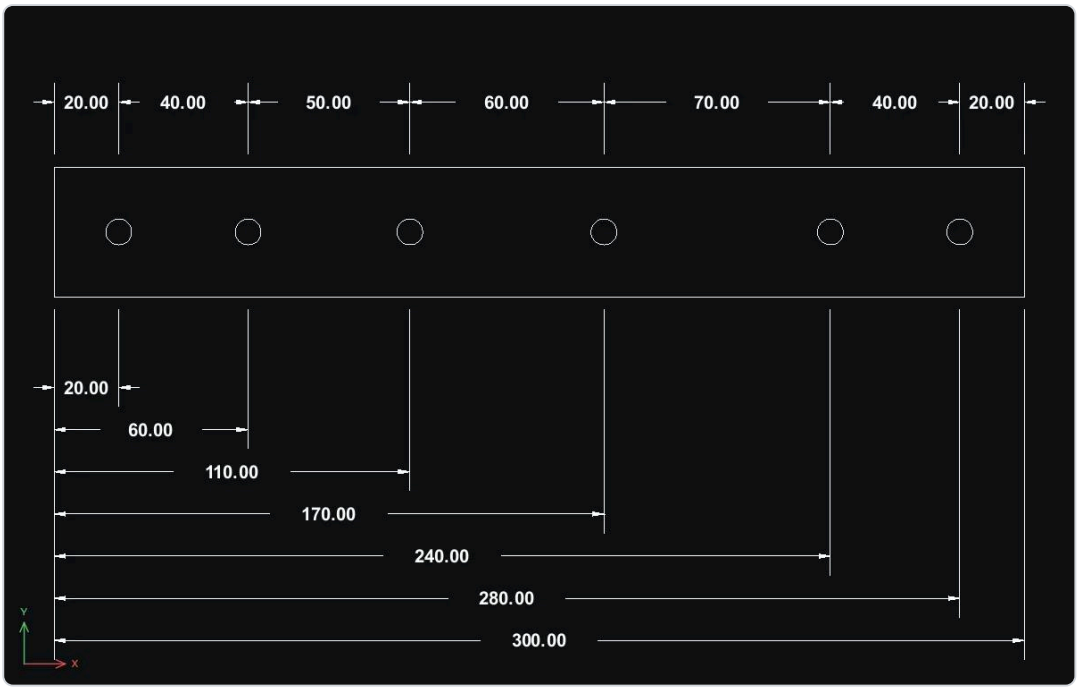


QDIM Baseline

Try it: Pegboard tool strip

A 300 x 40 steel strip for hanging tools on the workshop wall, with six Ø8 holes at uneven spacing (big spanners need more room). QDIM dimensions the whole row twice: once as a chain, once from a single base.

1. On **Annotate › Dim Defaults**, set **Dim H** `6.5` and **Arrow** `3.5` so the numbers suit a part this size.
2. `REC` `0,0` → `300` `Tab` `40`.
3. Holes: `C` radius `4` at `20,20`, `60,20`, `110,20`, `170,20`, `240,20` and `280,20`.
4. Drag a window around the strip so the strip and all six holes are selected.
5. Type `QDIM C` and click above the strip. One click places the chain 20, 40, 50, 60, 70, 40, 20.
6. Select everything again, type `QDIM B` and click below the strip. Every hole is now measured from the left end: 20, 60, 110 ... 300.



Tool strip with a continuous chain above and baseline dimensions below

Challenge: Try `QDIM S` (staggered) on the same selection, then even out the baseline stack with DIMSPACE.

See also

- DIMSPACE
- DIMLINEAR

Editing dimensions (DIMEDIT, DIMTEDIT, DIMBREAK ...)

Edit dimension text, position, arrowheads, prefix/suffix, break gaps and spacing.

What it does

These commands work on the **selected dimensions**. If nothing is selected, they work on **every dimension** in the drawing.

Command	Alias	Syntax	Meaning
DIMEDIT	`DED`	`DIMEDIT <Home|New <text>|Rotate <degrees>>`	Restores the text position, replaces the text, or rotates the text.
DIMTEDIT	`DIMTED`	`DIMTEDIT <Left|Right|Center|Home|Angle <degrees>>`	Moves the text to the left, right or centre, or rotates it.
DIMOVERRIDE	`DOV`	`DIMOVERRIDE <precision|text> <value>`	Changes the decimals (0–8) or the text without editing the style.
DIMPOST	`DP`	`DIMPOST <format>`	Sets a prefix/suffix for all dimensions. `<>` = the measured value, for example `DIMPOST <> mm` or `DIMPOST %%c<>`. With no value, it clears the setting.

Command	Alias	Syntax	Meaning
DIMARROW	`ARCHTICK`	`DIMARROW [arrow|tick]`	Switches between arrowheads and architectural ticks. With no value, it toggles.
DIMBREAK	–	`DIMBREAK [size|REMOVE]`	Leaves a gap in a dimension line where it crosses other objects.
DIMSPACE	–	`DIMSPACE [distance]`	Evens out the spacing of parallel dimensions (0 = align them on one line).
DIMJOGLINE	`DJL`	`DIMJOGLINE [0-1|REMOVE]`	Adds a not-to-scale jog symbol on the dimension line, at a position from 0 to 1.
DIMREGEN	–	–	Re-measures every dimension from its objects.

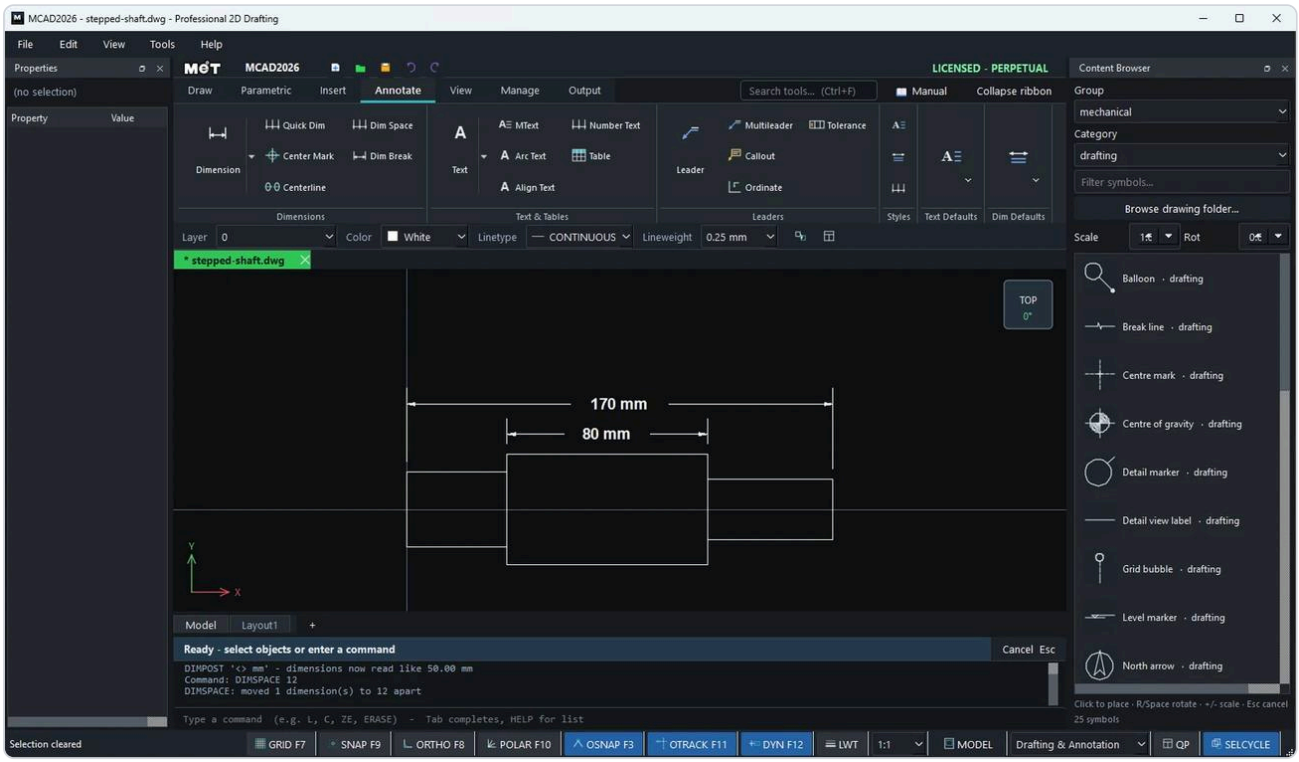
Special codes in DIMPOST: `%%c` = Ø, `%%d` = °, `%%p` = ±. Plain text, Multileaders and text typed over a dimension keep these codes as typed.

Where to find it

- Ribbon: **Annotate › Dimensions › Dim Space / Dim Break**
- Command line: see the table above
- You can also drag the dimension text directly with the mouse.

Steps: add the mm unit and even out the spacing

1. Type `DIMPOST <> mm` and press Enter. Every dimension now ends with mm.
2. Type `DIMSPACE 12` and press Enter. Dimensions on the same side are 12 mm apart.

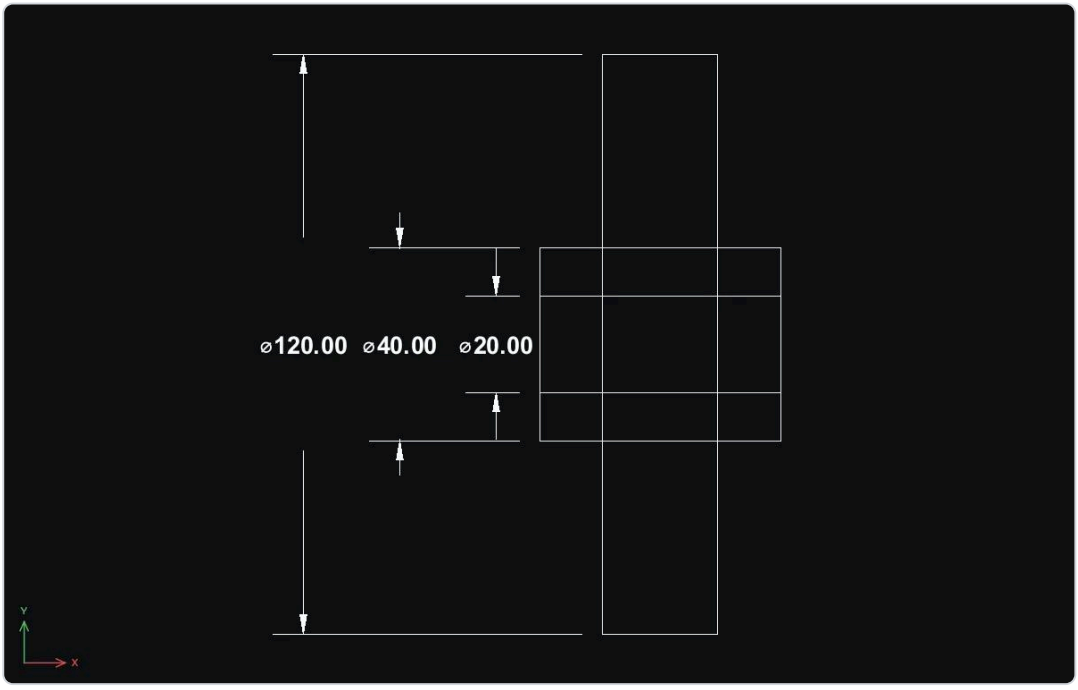


DIMPOST adds mm and DIMSPACE spaces two rows of dimensions

Try it: Belt pulley diameters

A side view of a belt pulley: Ø120 rim, Ø40 hub and a Ø20 bore. You place three rough vertical dimensions, then two edit commands turn them into a tidy diameter stack.

1. On **Annotate › Dim Defaults**, set **Dim H** `7` and **Arrow** `4` so the numbers suit a part this size.
2. Hub `REC 0,-20 → 50 Tab 40`, rim `REC 13,-60 → 24 Tab 120`, and the bore lines `L 0,-10 → 50,-10` and `L 0,10 → 50,10`.
3. Three `DVE` dimensions on the left of the part: bore `0,-10` to `0,10`, hub `0,-20` to `0,20`, rim `13,-60` to `13,60`. Place them anywhere to the left; uneven is fine.
4. Type `DIMPOST %%c<>`. Every value now starts with the diameter sign: ⌀20.00, ⌀40.00, ⌀120.00.
5. Type `DIMSPACE 20`. The three dimension lines line up 20 mm apart.



Pulley side view with an even stack of ⌀20, ⌀40 and ⌀120 dimensions

Challenge: Type `DIMOVERRIDE precision 0` to drop the decimals (⌀20, ⌀40, ⌀120), then `DIMPOST` with no value to remove the sign again.

See also

- DIMSTYLE
- Smart Dimension

DIMSTYLE and STYLE - Style Manager

Manage text styles and dimension styles - font, height, arrows, decimals, units and colour.

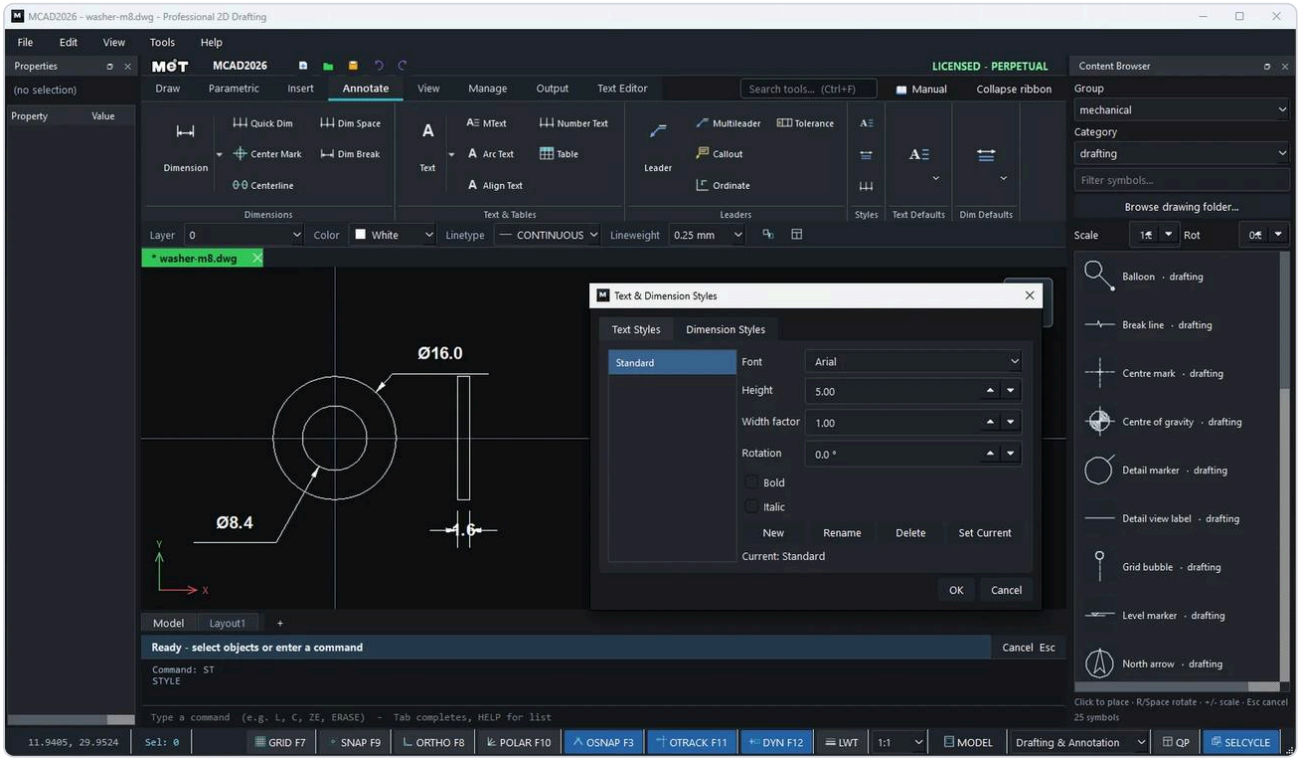
What it does

The **Text & Dimension Styles** dialog creates and edits the styles used by text and dimensions. When you edit a style, every object that uses it is updated.

Where to find it

- Ribbon: **Annotate › Styles › Text Style / Dim Style** or **Manage › CAD Standards**
- Command line: `DIMSTYLE` (aliases `D`, `DST`, `DDIM`)
- Command line: `STYLE` (alias `ST`) opens the Text Styles tab
- Quick defaults: **Annotate › Text Defaults / Dim Defaults**

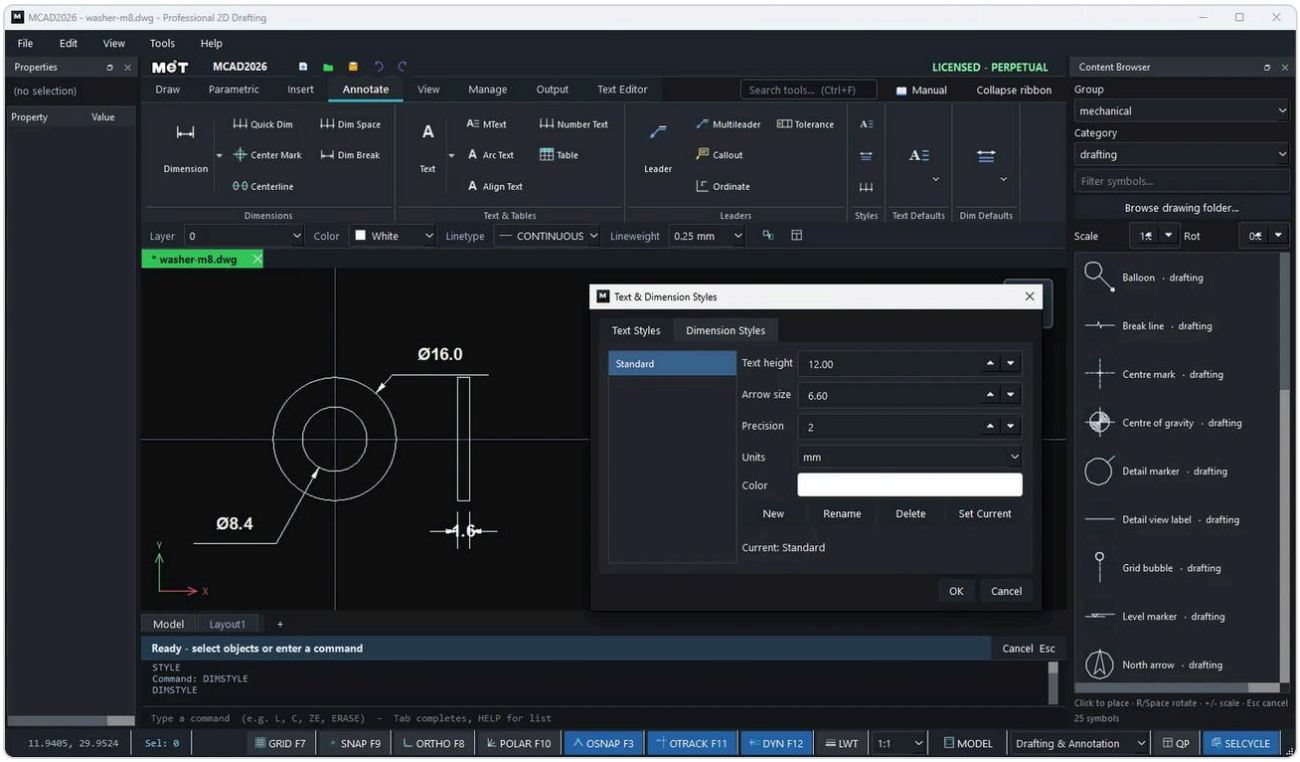
Text Styles tab



Text Styles tab

Option	Meaning
Font	Any Windows font.
Height	The text height.
Width factor	The text width (1 = normal).
Rotation	The text angle.
Bold / Italic	Bold / italic text.

Dimension Styles tab



Dimension Styles tab

Option	Meaning
Text height	The height of the dimension numbers.
Arrow size	The size of the arrowheads.
Precision	The number of decimals.
Units	mm, cm, m, in, ft
Color	The dimension colour.

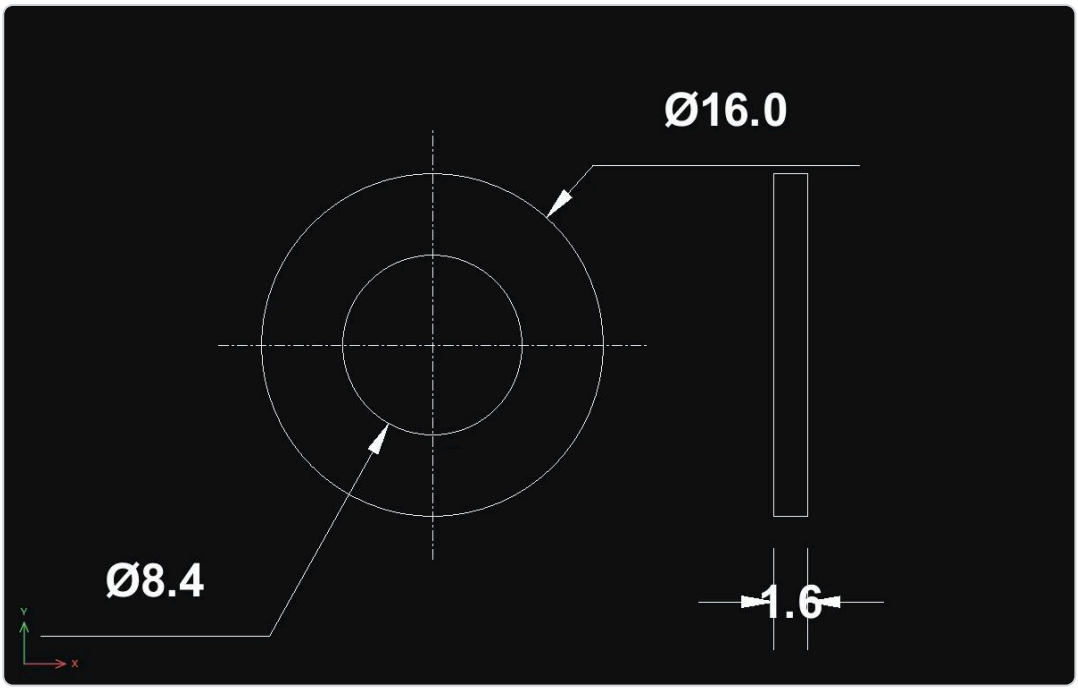
Common buttons

Button	Meaning
New / Rename / Delete	Creates, renames or deletes a style.
Set Current	Uses this style for new objects.
OK	Saves and updates the existing objects.

Try it: Dimension style for small parts

A DIN 125 M8 washer is only 16 mm across. With a style made for big plates its numbers would be larger than the part. A SMALL PARTS style with 2.5 mm text and one decimal fixes that once for every small part you draw.

1. Front view: `C` `0,0` radius `8` and `C` `0,0` radius `4.2`. Edge view: `REC` `16,-8` → `1.6` `Tab` `16`.
2. Select the outer circle and type `CENTERMARK 10`.
3. Type `DIMSTYLE`. On the **Dimension Styles** tab click **New** and name it `SMALL PARTS`. Set Text height `2.5`, Arrow size `1.5` and Precision `1`. Click **Set Current**, then **OK**.
4. `DDI` on the outer circle: `Ø16.0`. `DDI` on the hole: `Ø8.4`.
5. `DLI` across the edge view, from `16,-8` to `17.6,-8`, placed below: `1.6`.



M8 washer dimensioned with the SMALL PARTS style: Ø16.0, Ø8.4 and 1.6

Challenge: Open `DIMSTYLE` again, change SMALL PARTS to Text height `2` and click **OK**. All three dimensions update together.

See also

- Editing dimensions
- TEXT and MTEXT
- Annotation Scale

CENTERMARK and CENTERLINE

Draw a centre mark (CENTERMARK) on circles and arcs, and a centreline (CENTERLINE) between two lines.

What it does

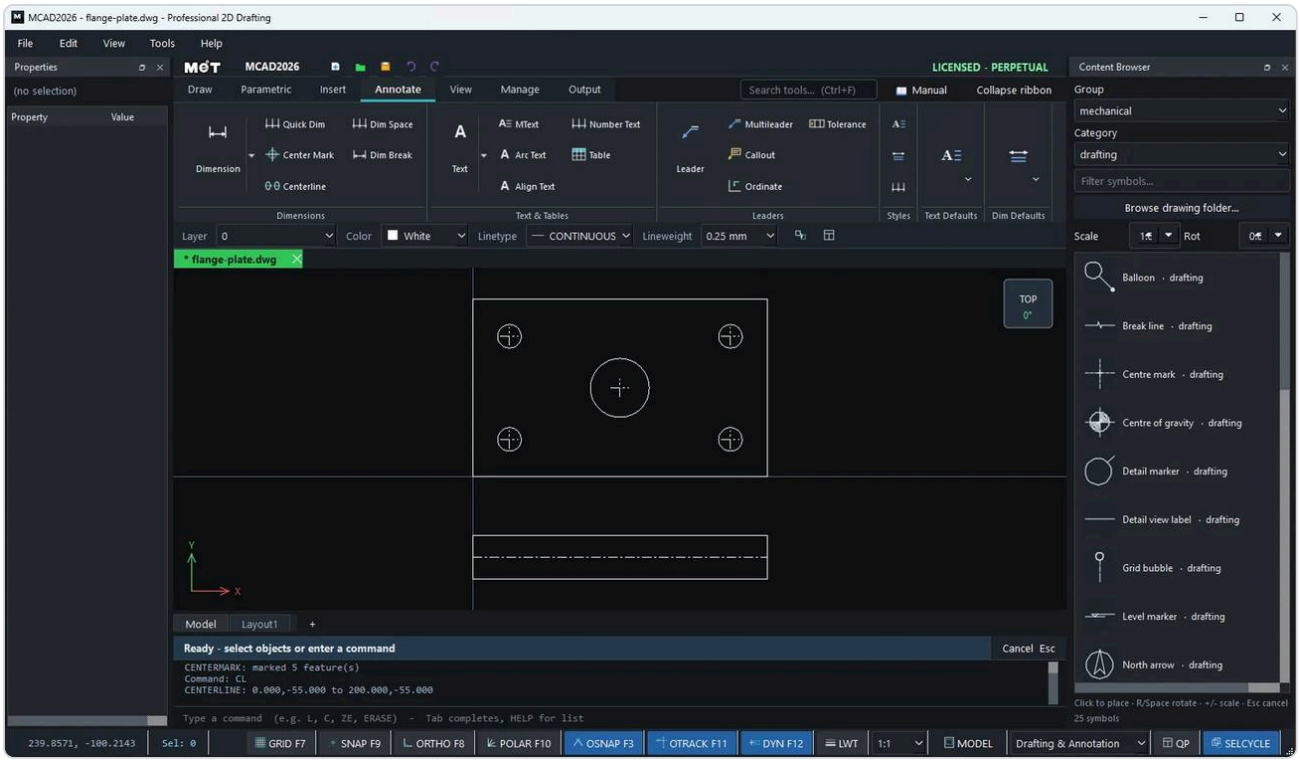
- **CENTERMARK** – draws a cross in the CENTER linetype at the centre of the selected circles or arcs.
- **CENTERLINE** – draws a CENTER line midway between two lines. For parallel lines you get the middle line. For lines that converge you get the angle bisector.

Where to find it

- Ribbon: **Annotate › Dimensions › Center Mark / Centerline** or **Draw** (Center Mark submenu)
- Command line: `CENTERMARK` (aliases `DIMCENTER`, `DCE`)
- Command line: `CENTERLINE` (alias `CL`)

Steps

1. Select the circles (for example with `QSELECT circle`).
2. Type `CENTERMARK` and press Enter (or `CENTERMARK 5` to set the arm length to 5).
3. For CENTERLINE: select two lines, then type `CL`.



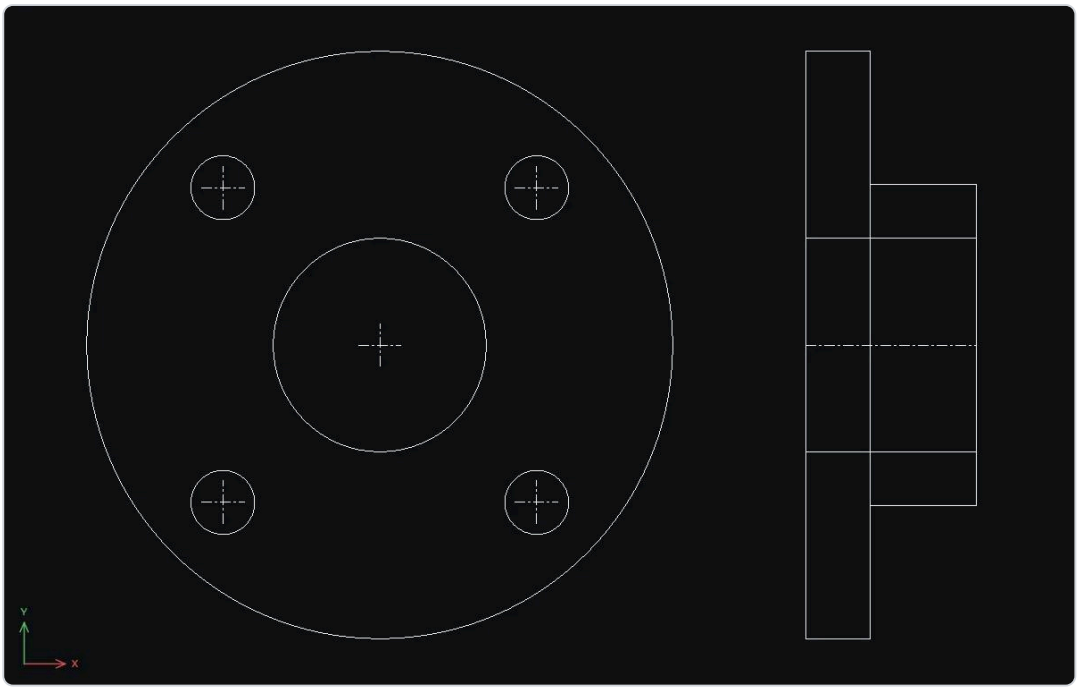
CENTERMARK on every hole and CENTERLINE between the two lower lines

Option	Meaning
`CENTERMARK [size]`	The arm length. If you omit it, 25% of the radius is used.

Try it: DN50 pipe flange

A standard DN50 pipe flange: Ø165 outside, Ø60 bore and four Ø18 bolt holes on a 125 mm pitch circle, in a front view and a side view. Two commands add every centre line.

1. Front view: `C` `0,0` radius `82.5`, then `C` `0,0` radius `30` for the bore.
2. One bolt hole: `C` `44.19,44.19` radius `9`. Select it, type `ARRAYPOLAR 4` and click `0,0` as the centre.
3. Side view: `REC` `120,-82.5` → `18` `Tab` `165` (flange), `REC` `138,-45` → `30` `Tab` `90` (hub), then the two bore lines `L` `120,-30` → `168,-30` and `L` `120,30` → `168,30`.
4. Select the six circles of the front view and type `CENTERMARK 6`. Every hole and the bore get a 6 mm centre cross.
5. Select the two bore lines and type `CL`. A CENTER line appears midway between them: the axis of the flange.



DN50 flange with centre marks on every hole and the axis in the side view

Challenge: Undo step 4 and run `CENTERMARK` with no number. The arms become 25% of each radius, so the bolt holes get 2.25 mm arms and the bore 7.5 mm.

See also

- Selecting objects
- DIMDIAMETER

QLEADER, LEADER, MLEADER and Callout

Arrows with notes: QLEADER/LEADER, Multileader (MLEADER), and Callout boxes.

What it does

Command	Alias	Meaning
QLEADER	`LE`	Multi-point leader, then you type the text
LEADER	`LEAD`	Same as QLEADER
MLEADER	`MLD`	`MLEADER <text>` Arrow, elbow line, and text in three clicks
Callout	–	Leader arrow with a text box (on the Ribbon)

Where to find it

- Ribbon: **Draw › Annotation › Leader** (submenu: Leader, Multileader, Callout, Ordinate) and **Annotate › Leaders**

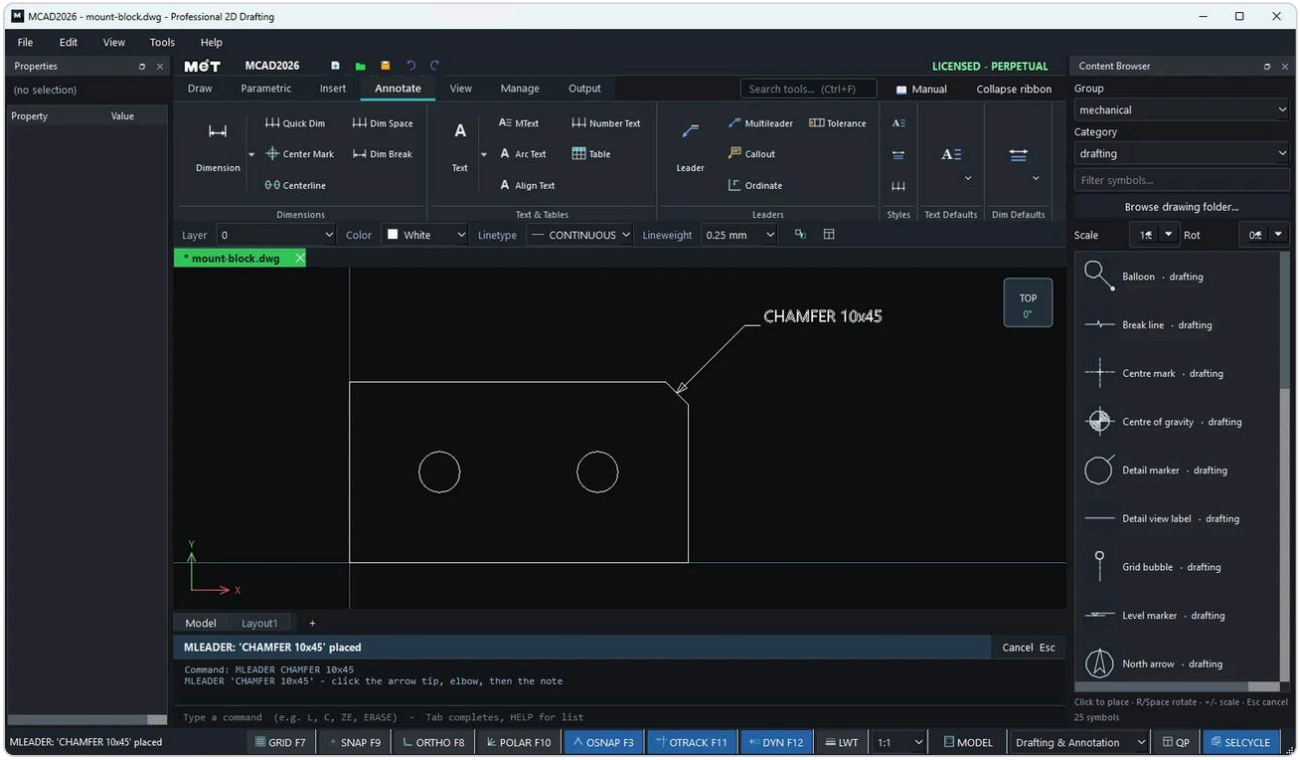
QLEADER steps

1. Type `LE` and press Enter.

2. Click the arrow tip, then click each following bend point (`Leader: click more points, Enter to finish.`).
3. Press Enter. The **Leader** window opens. Type the text and click OK.

MLEADER steps

1. Type `MLEADER CHAMFER 1x45` and press Enter (without text, a window asks for the Note).
2. `MLEADER: click the arrow tip` Click the arrow tip.
3. `MLEADER: click the elbow` Click the elbow point.
4. `MLEADER: click where the note sits` Click the text location.



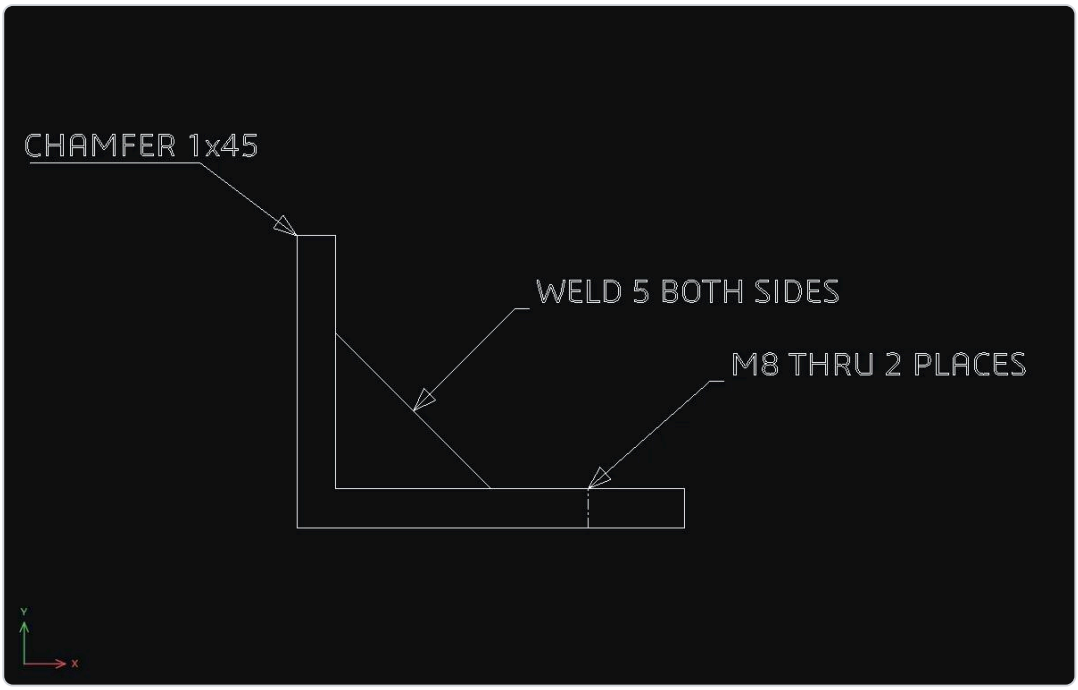
Multileader pointing at a corner of the part

> **Note:** A Multileader is exploded into lines and text when you export to DXF. You can also explode it yourself with `EXPLODE` .

Try it: Weld and machining notes on an angle bracket

The side view of a welded steel angle bracket with a gusset. Three multileaders tell the workshop where to weld, which edge to chamfer and which holes to drill.

1. Angle: `PL` `0,0` `80,0` `80,8` `8,8` `8,60` `0,60` `0,0` , then Enter.
2. Gusset edge: `L` `8,40` → `40,8` . Hole centre line: `L` `60,0` → `60,8` .
3. Type `MLEADER WELD 5 BOTH SIDES` . Click the arrow tip on the gusset at `24,24` , the elbow at `45,45` and the note at `48,45` .
4. Type `MLEADER CHAMFER 1x45` . Arrow tip on the top corner `0,60` , elbow `-20,75` , note `-55,75` .
5. Type `MLEADER M8 THRU 2 PLACES` . Arrow tip on the hole line at `60,8` , elbow `85,30` , note `88,30` .



Angle bracket with three multileader notes: weld, chamfer and holes

Challenge: Add a fourth note with `LE` (QLEADER) using three bend points, and type the note in the **Leader** window instead.

See also

- TEXT
- TOLERANCE

TOLERANCE - GD&T Feature Control Frame

Place a GD&T Feature Control Frame with symbol, tolerance value, and datums.

What it does

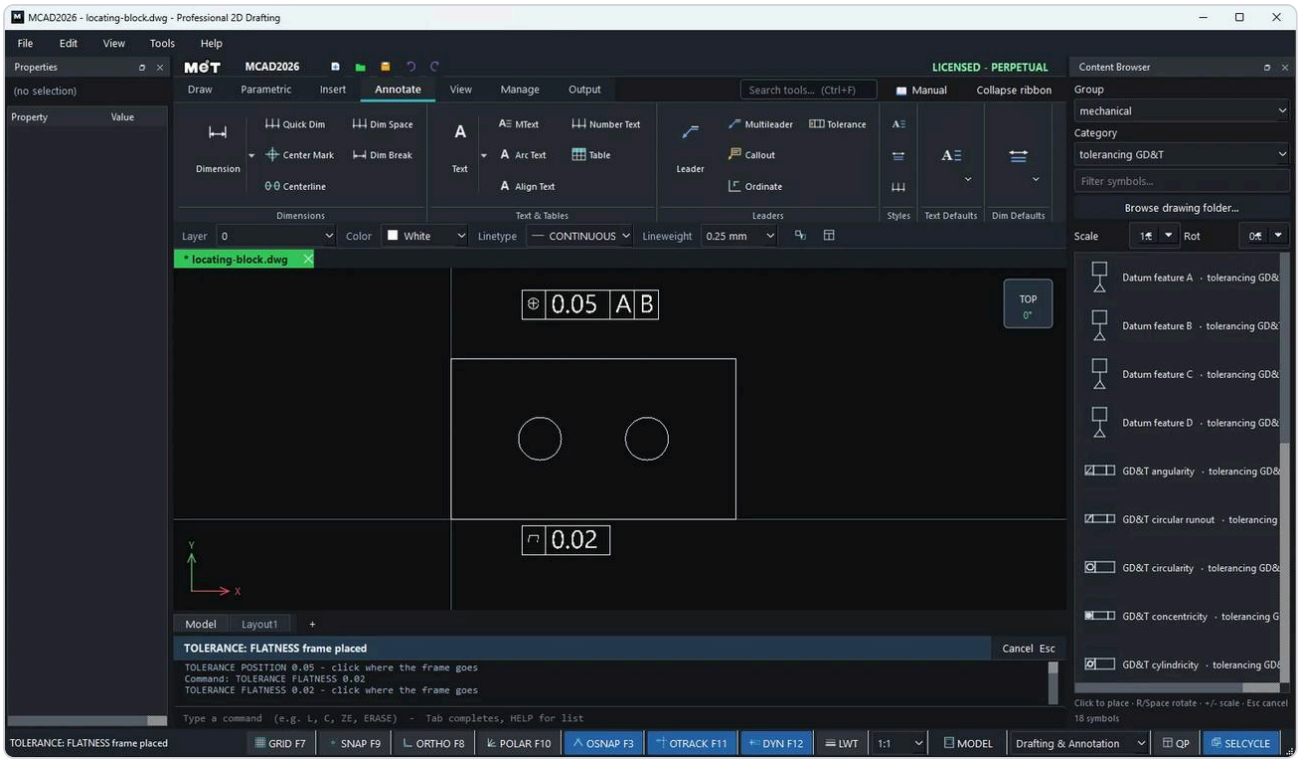
TOLERANCE creates a Geometric Tolerance (GD&T) frame from lines and text, so it exports to DXF and prints correctly in any program.

Where to find it

- Ribbon: **Annotate › Leaders › Tolerance** (shows the list of symbols)
- Command line: `TOLERANCE` (Alias `TOL`)

Steps

1. Type `TOLERANCE POSITION 0.05 A B` and press Enter (symbol, value, datums...).
2. `TOLERANCE: click where the frame goes` Click a location.



Position 0.05 A B and Flatness 0.02 frames

Supported symbols

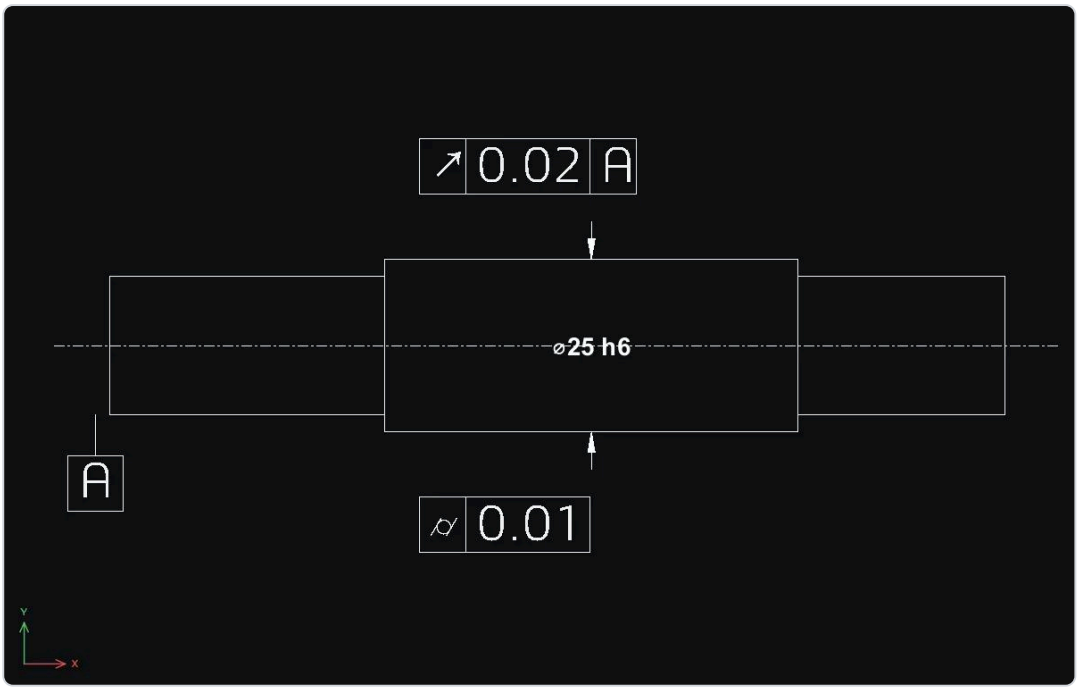
Keyword	Characteristic
STRAIGHTNESS	Straightness
FLATNESS	Flatness
CIRCULARITY	Circularity
CYLINDRICITY	Cylindricity
PROFILE / PROFILELINE	Profile of a surface / profile of a line
ANGULARITY	Angularity
PERPENDICULARITY	Perpendicularity
PARALLELISM	Parallelism
POSITION	Position
CONCENTRICITY	Concentricity
SYMMETRY	Symmetry
RUNOUT / TOTALRUNOUT	Runout / total runout

`TOLERANCE ?` lists the symbols on the Command line. The text height follows the current text height setting.

Try it: GD&T on a motor shaft

A stepped motor shaft whose middle Ø25 h6 section carries the bearing. Two feature control frames tell the inspector how round and how true that seat must be, measured against datum A.

1. Shaft, left to right: `REC 0,-10` → `40` `Tab 20`, `REC 40,-12.5` → `60` `Tab 25` (bearing seat), `REC 100,-10` → `30` `Tab 20`. Axis: `L -8,0` → `138,0`.
2. Bearing seat: `DVE` from `70,-12.5` to `70,12.5`, placed on the shaft. Then `DIMPOST %%c<> h6` and `DIMOVERRIDE precision 0`. It reads `⌀25 h6`.
3. Datum A: `REC -6,-24` → `8` `Tab 8`, `DT` with Text height `5` and type `A` inside the box, then `L -2,-16` → `-2,-10` up to the shaft.
4. `TOLERANCE RUNOUT 0.02 A`, then click above the seat at `45,22`.
5. `TOLERANCE CYLINDRICITY 0.01`, then click below the seat at `45,-30`. Both frames use the text height 5 from step 3.



Stepped shaft with ⌀25 h6, a runout 0.02 A frame and a cylindricity 0.01 frame

Challenge: Type `TOLERANCE ?` to list all symbols, then add `TOLERANCE PERPENDICULARITY 0.03 A` for the shoulder at X 40.

See also

- QLEADER and MLEADER

TEXT, MTEXT and TEXTEDIT

Type single-line text (TEXT) and multiline text (MTEXT) directly in the drawing area, and edit text (TEXTEDIT).

What it does

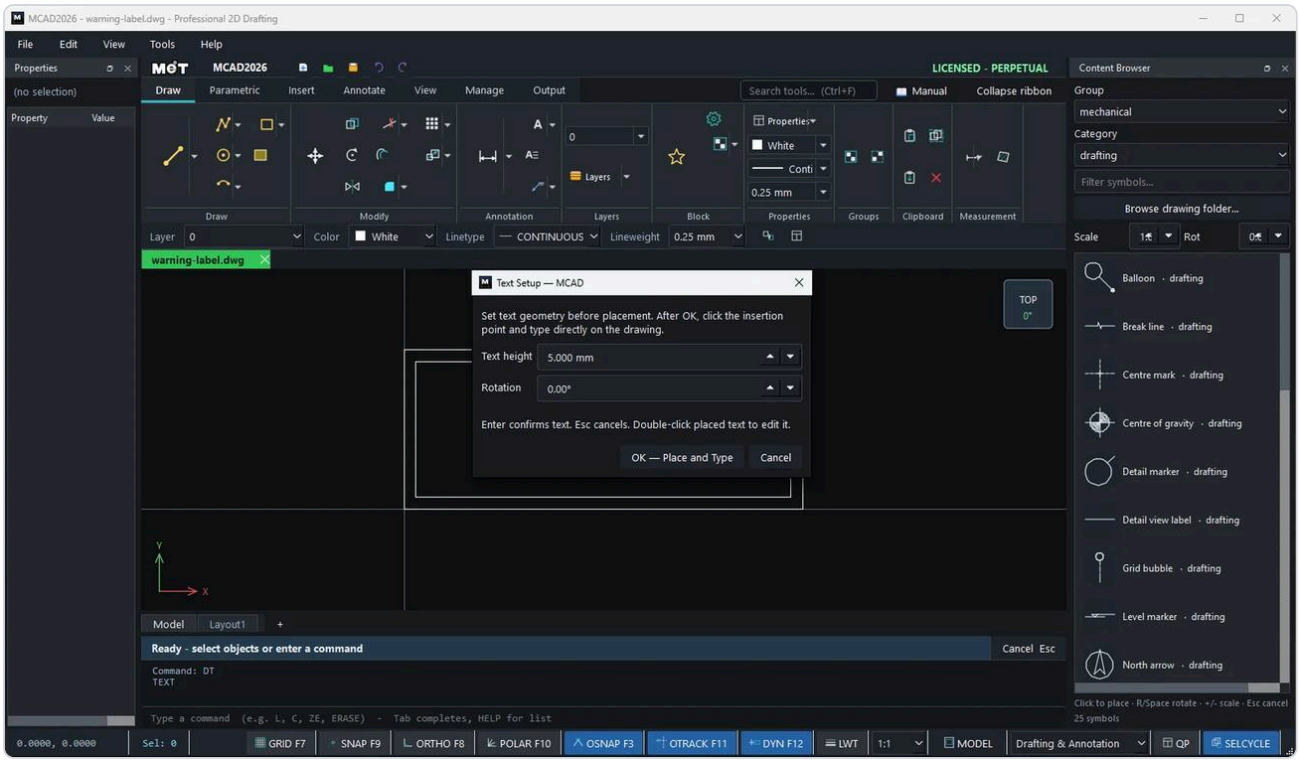
Command	Alias	Meaning
TEXT	`DT`, `DTEXT`	Single-line text
MTEXT	`T`, `MT`	Multiline text with formatting
TEXTEDIT	`ED`, `DDEDIT`, `MTEDIT`	`TEXTEDIT [new text]` Edits the selected text

Where to find it

- Ribbon: **Draw › Annotation › Text** (Single Line Text / Multiline Text), **MText**, and **Annotate › Text & Tables**

TEXT steps

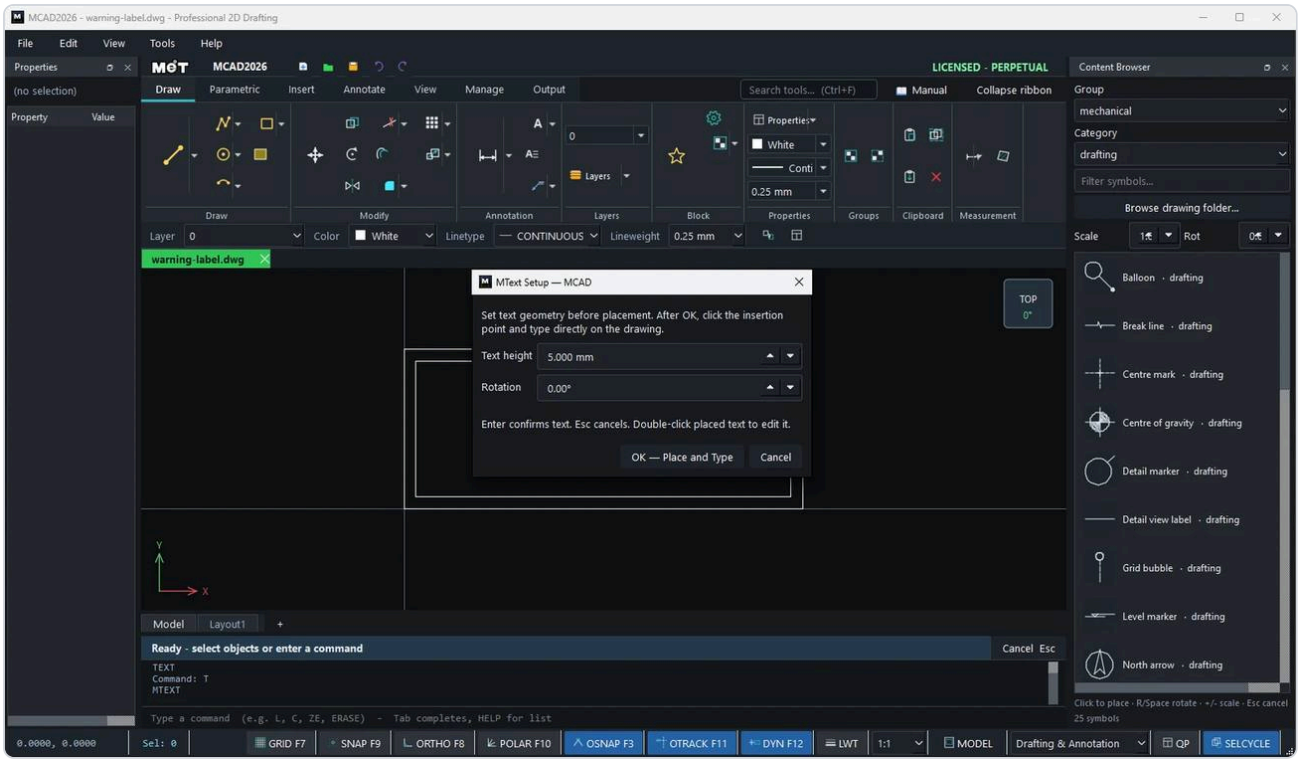
1. Type `DT` and press Enter. The **Text Setup** window opens.
2. Set the **Text height** and **Rotation**, then click **OK — Place and Type**.
3. Click the text location and type directly in the drawing area.
4. Press Enter to confirm (Esc = cancel).



Text Setup window

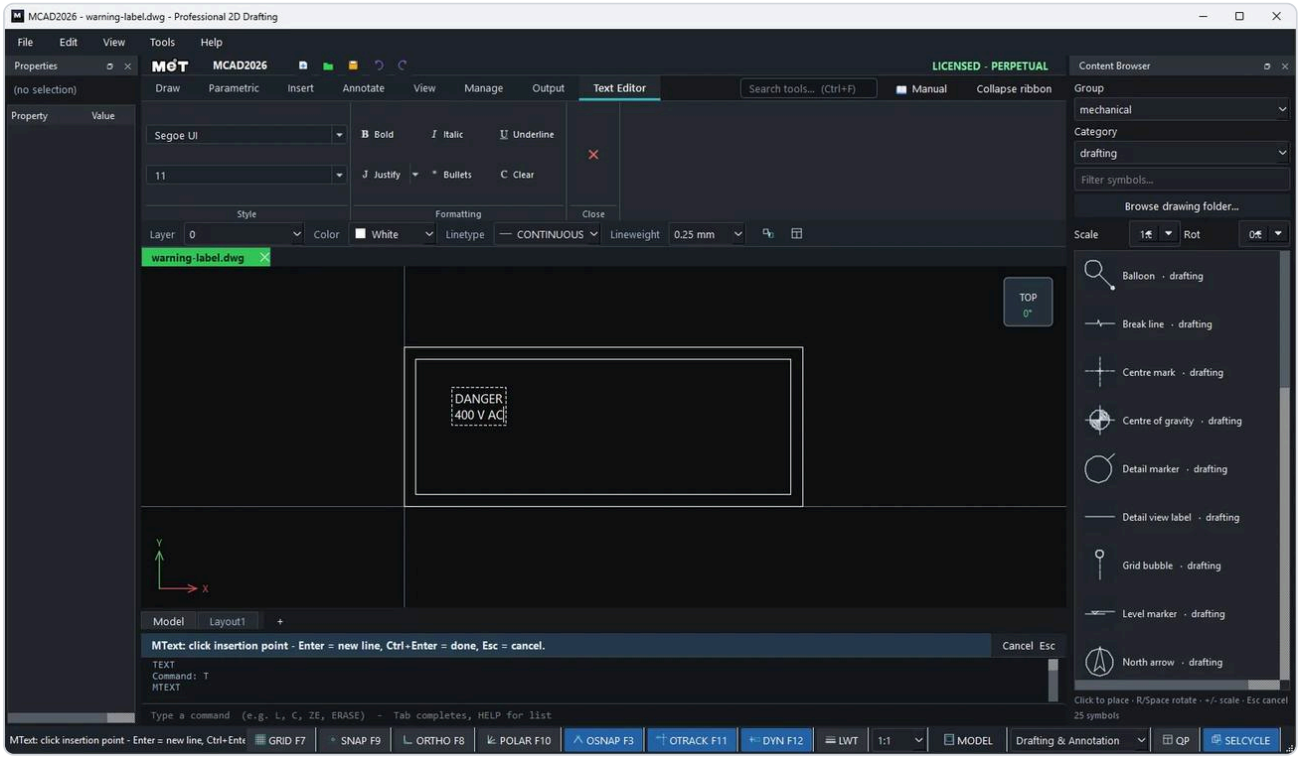
MTEXT steps

1. Type `T` and press Enter. Set the options in the MText Setup window and click OK.



MText Setup window

2. Click the location, then type (Enter = new line, `Ctrl+Enter` = finish, Esc = cancel).
3. While you type, the **Text Editor** tab appears on the Ribbon.



Typing MTEXT with the Text Editor tab on the Ribbon

Text Editor tab

Option	Meaning
Font / size	Font and character size

Option	Meaning
Bold / Italic / Underline	Bold, italic, underline
Justify	Left / Center / Right
Bullets	Add bullets
Clear	Clear formatting
Close Text Editor	Finish typing

Edit text

- Double-click the text to edit it in place.
- Or select the text and use `ED NEW TEXT` / `ED` (opens a window for editing).
- Right-click › Text Properties... to change the font, height, and color.

> **Note:** You can type Thai text as usual. Choose a font that supports Thai in the Text Style.

Try it: DANGER 400 V nameplate

An engraved warning plate, 100 x 40 with four Ø3.2 fixing holes, for the door of an electrical cabinet. Three single-line texts at three heights make it readable from across the room.

1. `REC` `0,0` → `100` `Tab` `40`. Fixing holes: `C` radius `1.6` at `5,5`, `95,5`, `5,35` and `95,35`.
2. `DT`. In **Text Setup** set Text height `12` and Rotation `0`, click **OK — Place and Type**, click at about `21,22`, type `DANGER` and press Enter.
3. `DT` again with height `5`: `400 V AC` at about `21,13`.
4. `DT` with height `4`: `ISOLATE BEFORE OPENING` at about `21,6`.



100 x 40 nameplate reading DANGER, 400 V AC, ISOLATE BEFORE OPENING

Challenge: Replace steps 3 and 4 with one `MTEXT` (press Enter for the second line, `Ctrl+Enter` to finish), then make `DANGER` bold with the **Text Editor** tab.

See also

- Text Tools
- DIMSTYLE and STYLE

Text Tools (ARCTEXT, TALIGN, TCOUNT, FIND ...)

Arc-shaped text, align text to a line, number text, justify, find/replace, and spell check.

What it does

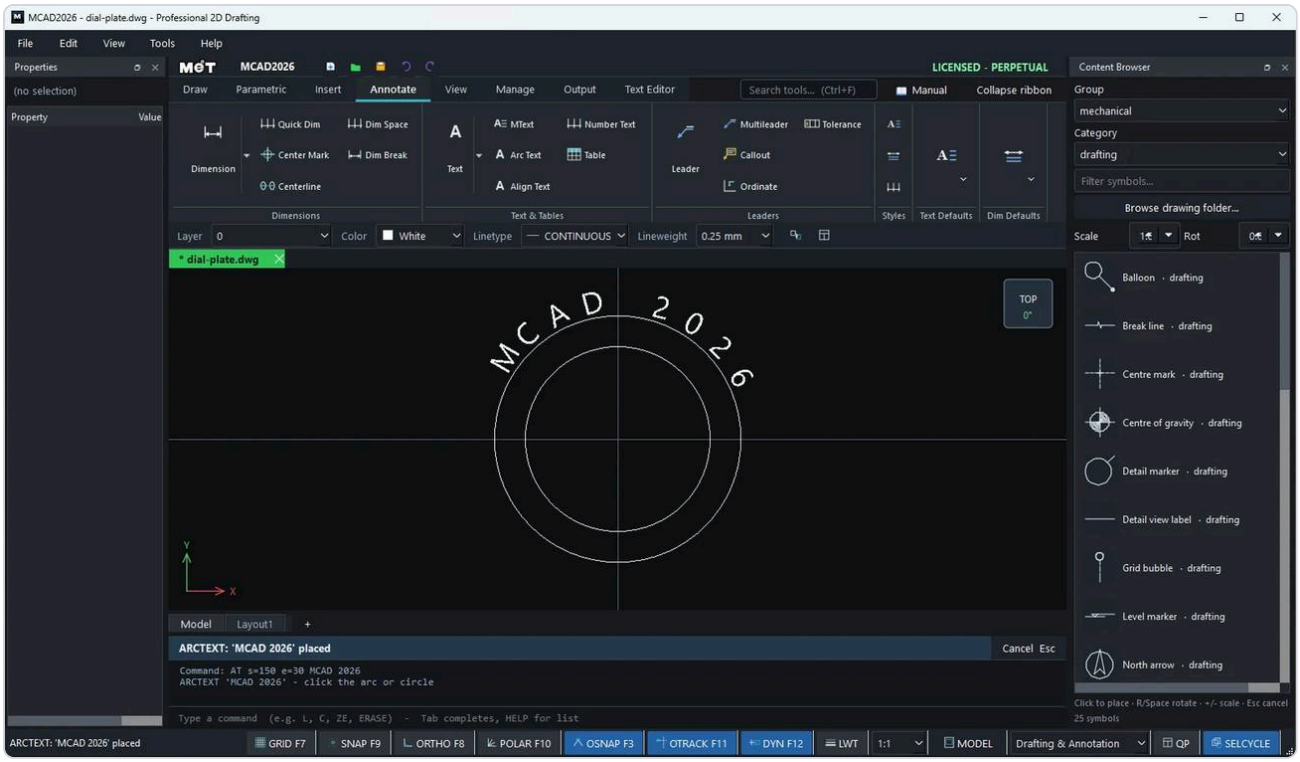
Command	Alias	Format	Meaning
ARCTEXT	`AT`	`ARCTEXT [s=degrees] [e=degrees] [p=degrees/character] [sp=multiplier] [h=height] <text>`	Places text curved along a circle/arc
TALIGN	`TA` , `ALIGNTEXT`	–	Rotates the selected text parallel to a clicked line and places it on the line
TEXTALIGN	–	–	The AutoCAD name for TALIGN
TCOUNT	`TC`	`TCOUNT [start] [step] [p=prefix] [s=suffix] [pick] [over]`	Numbers the selected text in reading order (top to bottom). `pick` = in the order you selected. `over` = replace the existing text
JUSTIFYTEXT	`JT`	`JUSTIFYTEXT <Left|Center|Right>`	Changes the justification point without moving the text
FIND	–	`FIND <word> [replacement]`	Finds text (selects what it finds) or replaces it
SPELL	–	–	Checks for words commonly misspelled in drawings, repeated words, double spaces, and unbalanced brackets
TEXTTOFRONT	`TTF`	`TEXTTOFRONT [Text|Dim|Both]`	Brings text/dimensions in front of Hatch

Where to find it

- Ribbon: **Annotate › Text & Tables › Arc Text / Align Text / Number Text**

ARCTEXT steps

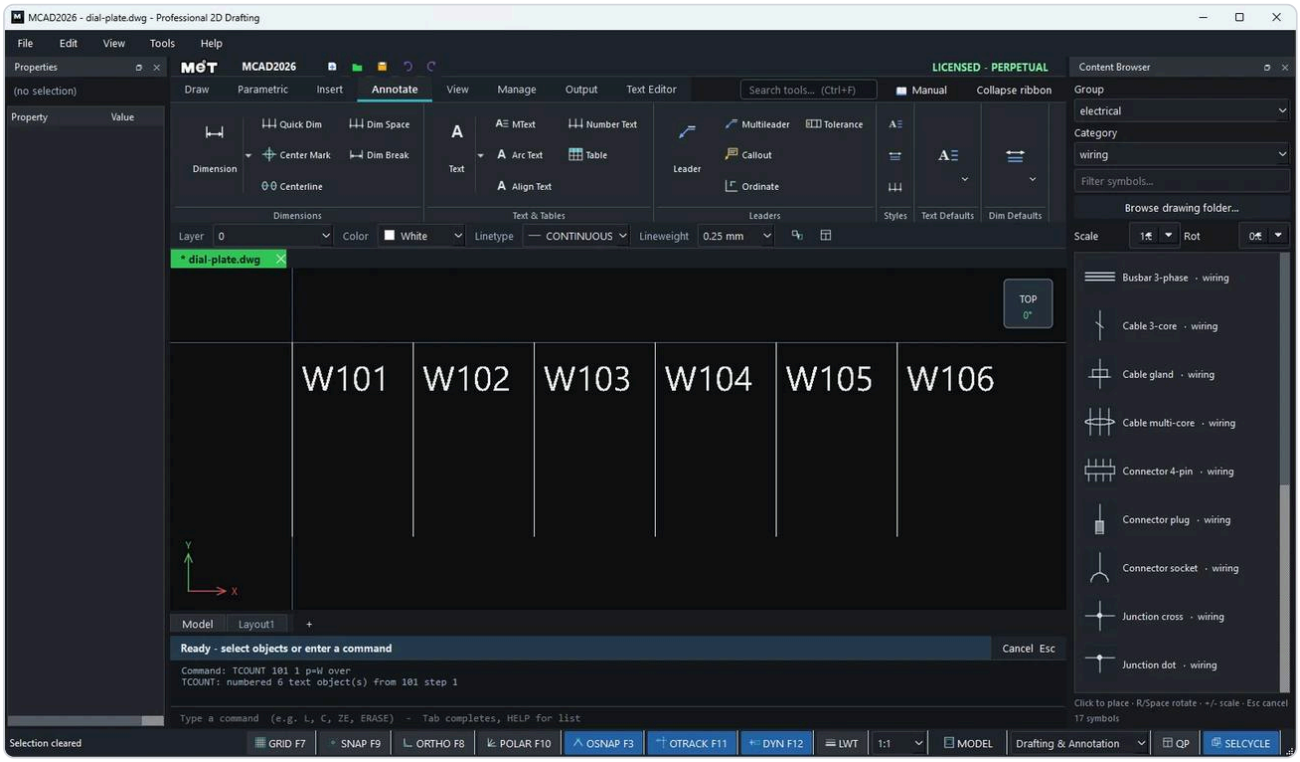
1. Type `AT s=150 e=30 MCAD 2026` and press Enter.
2. `ARCTEXT: click the arc/circle to wrap onto` Click the circle.



The text MCAD 2026 curved along a circle

TCOUNT steps

1. Select the text objects you want to number.
2. Type `TCOUNT 1 1 p=X1: over` and press Enter. The result is X1:1, X1:2, ...



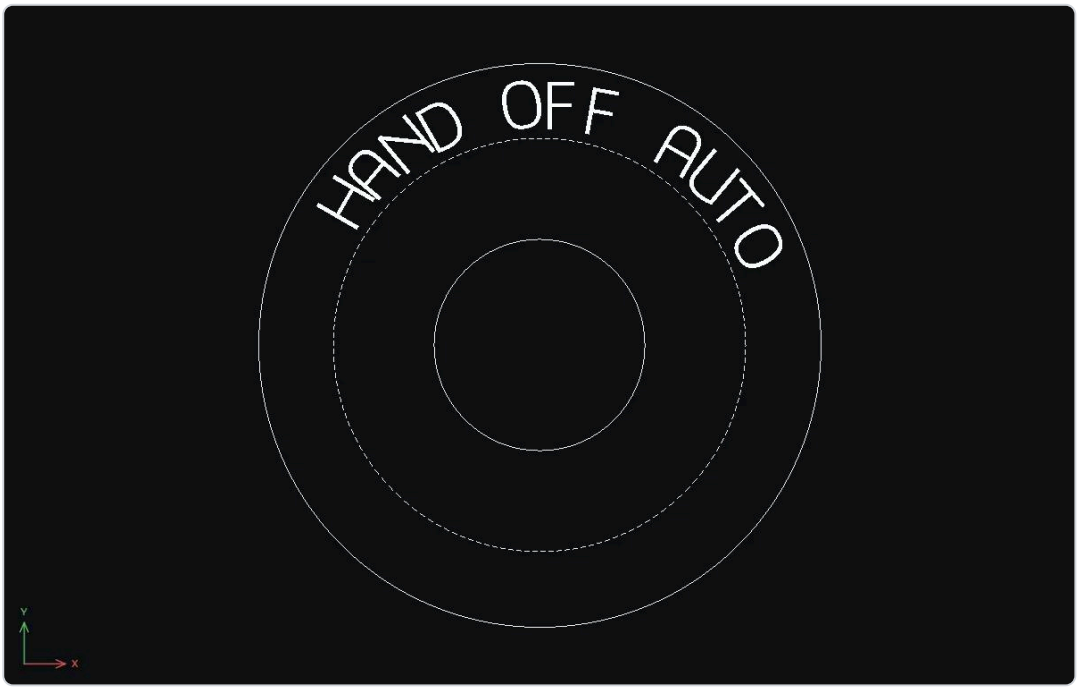
Text numbered with TCOUNT

> **Note:** SPELL does not include a full English dictionary. It checks only words that are commonly misspelled in drafting (for example TOLERENCE, DIAMTER) and errors it can prove.

Try it: Hand-Off-Auto selector legend

The round legend plate that sits behind a 22 mm selector switch on a pump panel, with HAND, OFF and AUTO curved over the top. ARCTEXT does the curve in one line.

1. Plate: `C 0,0` radius `30`. Hole for the switch: `C 0,0` radius `11.25`.
2. Guide circle for the text: `C 0,0` radius `22`.
3. Type `AT s=150 e=30 h=5 HAND OFF AUTO` and press Enter.
4. At `ARCTEXT: click the arc/circle to wrap onto`, click the guide circle (for example at `0,22`). The words run from 150° round to 30° across the top.
5. Erase the guide circle with `E` if you do not want it engraved (it is shown dashed in the picture).



Round selector legend with HAND OFF AUTO curved over the top

Challenge: Replace `s=150 e=30` with `p=12` to get a fixed 12° per character, handy when several plates must look identical.

See also

- TEXT, MTEXT
- Terminal and Wire Numbering

TABLE and DATALINK - Tables

Create tables (TABLE), fill in cells, export to CSV, and link to Excel files (DATALINK).

What it does

Command	Alias	Format	Meaning
TABLE	`TB2`	`TABLE <rows> <columns> [column width] [row height] [text height]` / `TABLE CSV <file>`	Places a table
TABLECELL	`TCELL`	`TABLECELL <row> <column> <text>`	Writes into a cell (counting starts at 0)
TABLEDIT	`TED`	`TABLEDIT [row column text]`	Edits a cell of the selected table
TABLEEXPORT	`TBEXP`	`TABLEEXPORT <file.csv>`	Exports the table to CSV
DATALINK	`DL`	`DATALINK <file.xlsx|csv> [sheet]` / `UPDATE` / `DETACH`	A table that pulls data from Excel/CSV. UPDATE = read again, DETACH = break the link (the existing values stay)

Where to find it

- Ribbon: **Annotate › Text & Tables › Table** (opens the settings window)

Steps from the Ribbon

1. Click **Table**. The settings window opens.
2. Set the Table style, Rows, Columns, Column width, Row height, and Text height.
3. Click **OK — Pick insertion point**, then click the top-left corner of the table.
4. Double-click a cell to type. Drag the blue Grips to resize rows/columns.

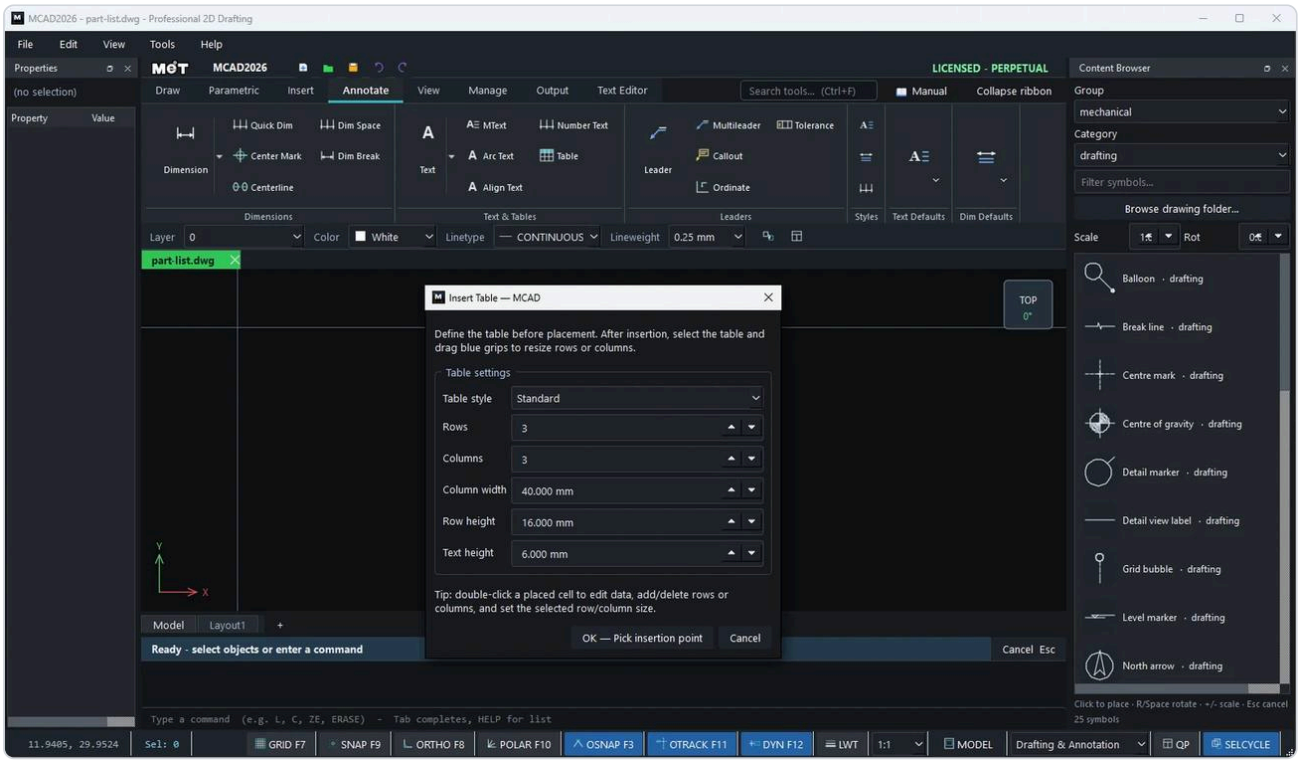
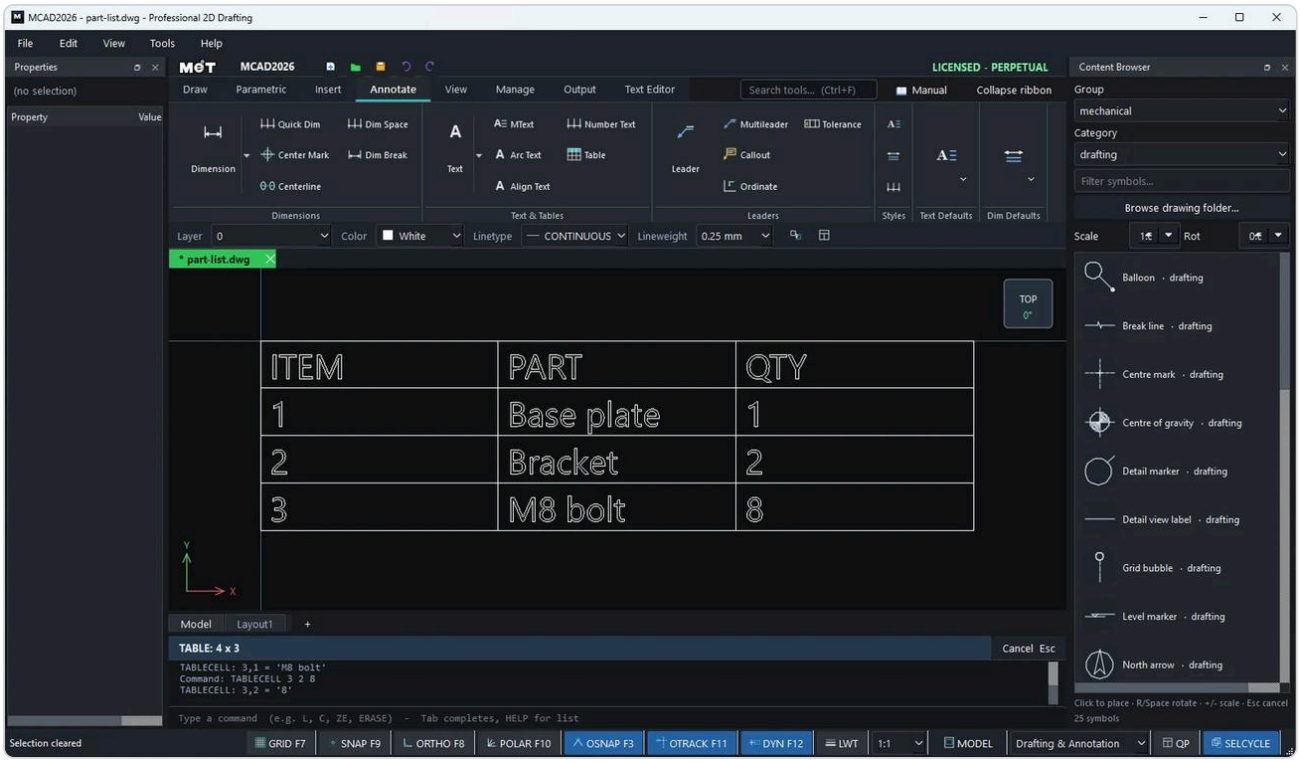


Table settings window

Steps from the Command line

1. Type `TABLE 4 3 40 8 4` (text height 4), then click the top-left corner (the table stays selected).
2. Fill in each cell: `TABLECELL 0 0 ITEM`, `TABLECELL 0 1 PART`, ...



Part list table with 4 rows and 3 columns

> **Note:** The table keeps its values inside the drawing, so anyone you send the drawing to can see the data even without the Excel file. When you export to DXF, the table is broken into lines and text.

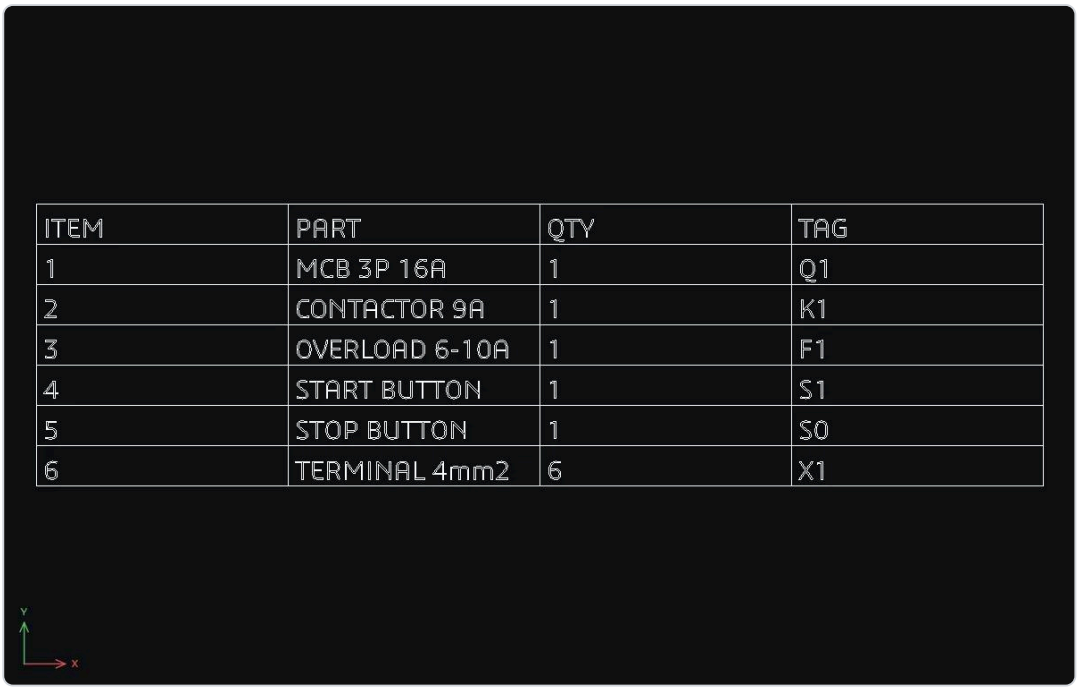
Try it: Parts list for a motor starter panel

The parts list that goes on the drawing of a small direct-on-line motor starter: breaker, contactor, overload, two push-buttons and the terminals. Built with typed commands, then exported for purchasing.

1. Type `TABLE 7 4 50 8 3.5` (7 rows, 4 columns, 50 wide, 8 high, text 3.5) and click the top-left corner. The new table stays selected.
2. Header row: `TABLECELL 0 0 ITEM`, `TABLECELL 0 1 PART`, `TABLECELL 0 2 QTY`, `TABLECELL 0 3 TAG`.
3. Fill rows 1 to 6 the same way (or double-click each cell):

Row	ITEM	PART	QTY	TAG
1	1	MCB 3P 16A	1	Q1
2	2	CONTACTOR 9A	1	K1
3	3	OVERLOAD 6-10A	1	F1
4	4	START BUTTON	1	S1
5	5	STOP BUTTON	1	S0
6	6	TERMINAL 4mm2	6	X1

4. With the table still selected, type `TABLEEXPORT starter_parts.csv` and send the CSV to purchasing.



Seven-row parts list for a motor starter panel

Challenge: Keep the list in Excel instead and pull it in with `DATALINK starter_parts.xlsx`; after a change in Excel, `DATALINK UPDATE` refreshes the drawing.

See also

- FIELD
- TITLEBLOCK

FIELD and UPDATEFIELD

Text that calculates its own value (FIELD), such as the date, file name or area, and how to update it (UPDATEFIELD).

What it does

FIELD places text that stores an expression and displays the calculated value, such as the date or the file name in a title block.

Where to find it

- Ribbon: type the command
- Command line: `FIELD`
- Command line: `UPDATEFIELD` (alias `UPDATEF`)

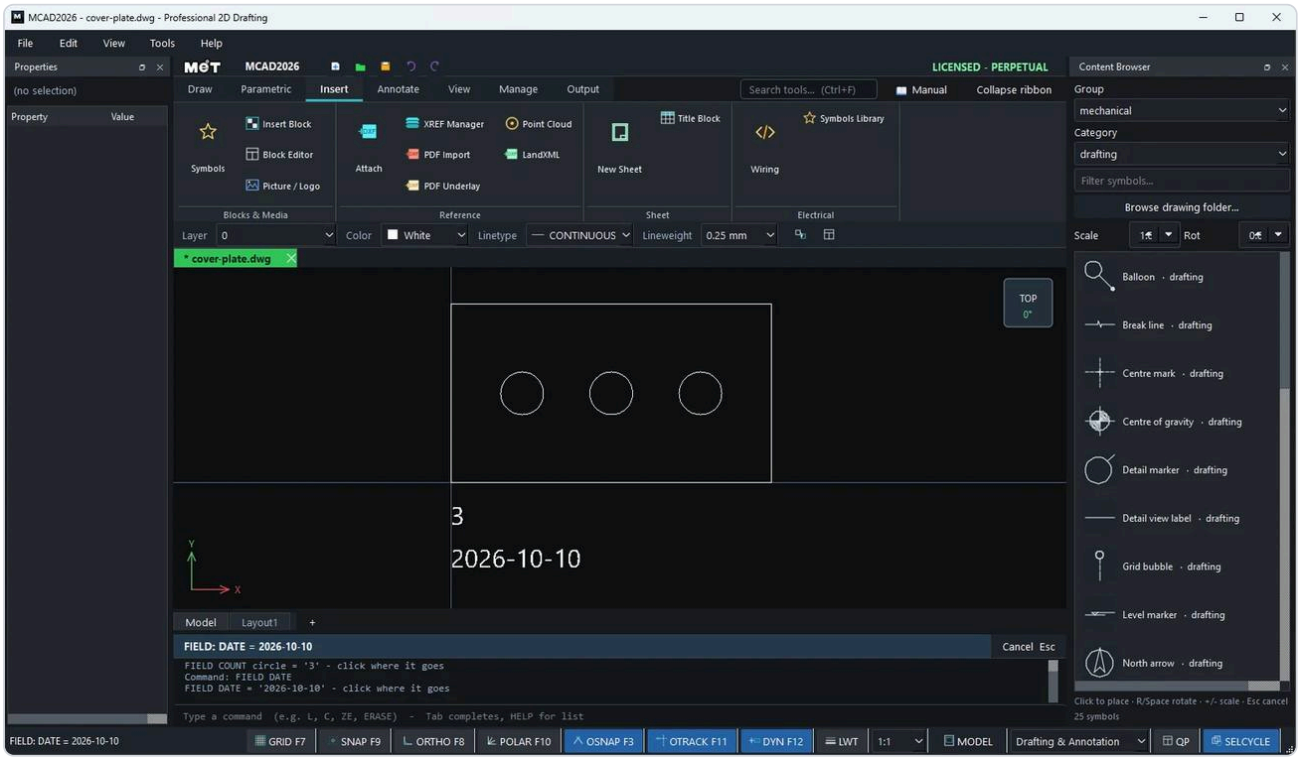
Available fields (`FIELD ?`)

Field	Value
AREA	The total closed area in the drawing.
COUNT	The number of objects (`COUNT circle` counts circles).
DATE / DATETIME / TIME	The date / the date and time / the time.

Field	Value
FILENAME / FILEPATH	The file name / the full path.
LAYER	The current layer.
LOGIN	The Windows user name.
SHEET	The current layout name.

Steps

1. Type `FIELD COUNT circle` and press Enter. The prompt bar shows a sample value.
2. Click the position to place it.
3. After you edit the drawing, type `UPDATEFIELD` to recalculate every field.



FIELD COUNT circle shows 3, and FIELD DATE shows the date

> **Note:** When you save as DXF, the latest calculated value is saved as text, so other CAD programs show the correct value.

Try it: Hole count that keeps itself up to date

A 300 x 200 back plate for an electrical cabinet with a grid of M6 tapping holes. The hole count in the note is a FIELD, so when the customer asks for two more holes, the note fixes itself.

1. `REC` `0,0` → `300` `Tab` `200`.
2. First hole: `C` `30,20` radius `3.25`. Select it, type `ARRAYRECT 4 80 0`, then select the four holes and type `ARRAYRECT 3 0 80`. You now have 12 holes.
3. `DT` with Text height `8`, type `HOLES:` at about `0,-18`. (FIELD uses this text height too.)
4. Type `FIELD COUNT circle`. The prompt bar shows the value `'12'`. Click at about `45,-18`.
5. Two extra holes: `C` `70,140` radius `3.25` and `C` `230,140` radius `3.25`.

6. Type `UPDATEFIELD`. The note now reads `HOLES: 14`.



Back plate with 14 holes and a FIELD note that counted them

Challenge: Put `FIELD DATE` and `FIELD FILENAME` under the note, so the printout always shows which file and which day it came from.

See also

- TEXT
- TABLE

Annotation Scale (OBJECTSCALE, ANNOALLVISIBLE)

Set the annotation scale so text and dimensions display at the right size for the drawing scale (OBJECTSCALE, ANNOALLVISIBLE).

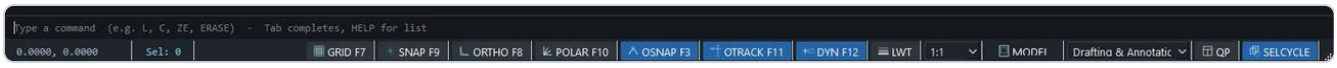
What it does

An annotative object is displayed only when the current annotation scale matches a scale assigned to it. Use this when you plot a drawing at several scales.

Command	Alias	Syntax	Meaning
OBJECTSCALE	`ANNOOBJECTSCALE`	`OBJECTSCALE <Add|Delete|List> [scale]`	Add, delete or list the scales of the selected annotative objects. If you give no scale, the current DIMSCALE is used.
ANNOALLVISIBLE	–	`ANNOALLVISIBLE <ON|OFF>`	ON = show annotative objects at all scales.

Where to find it

- Status bar: the Annotation scale box (1:1, 1:2, 1:5, 1:10, 1:20, 1:50, 1:100, 1:200, 1:500, 1:1000, 2:1, 5:1)
- Ribbon: **Annotate › Styles › Add Object Scale**



Status bar with the Annotation scale box set to 1:1

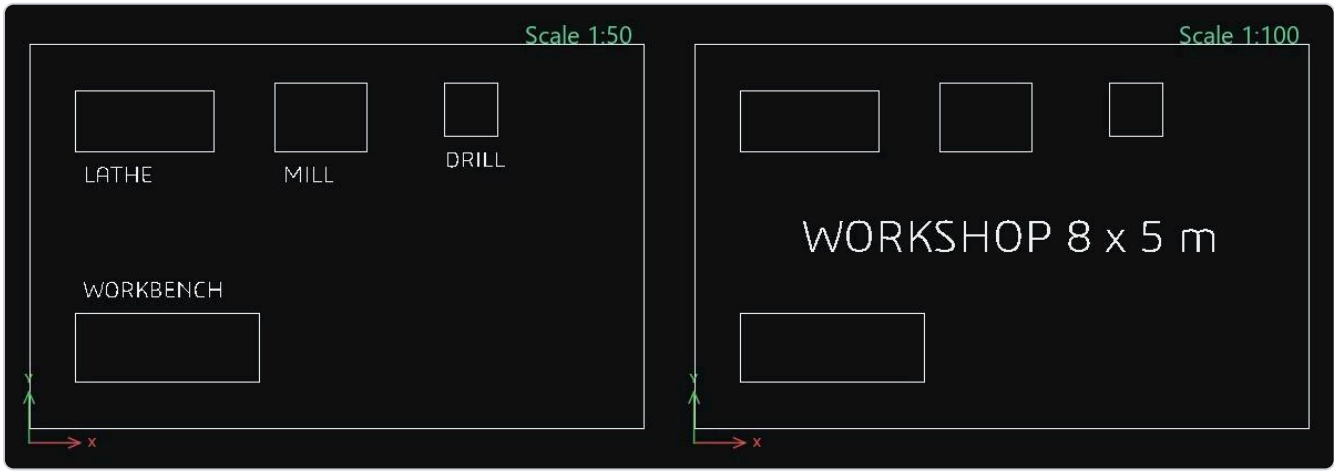
Steps

1. Choose an annotation scale on the Status bar, for example `1:10`.
2. Select the text or dimensions.
3. Click **Add Object Scale** (or type `OBJECTSCALE ADD`).
4. Check the result with `OBJECTSCALE LIST`.

Try it: Workshop floor plan at two scales

One 8 x 5 m workshop plan, two printouts. At 1:50 you want every machine named; at 1:100 only the room title fits. Annotative text shows the right labels at each scale on its own.

1. Walls: `REC` `0,0`, then `8000` `Tab` `5000` Enter. Machine footprints, also with `REC`: lathe `600,3600` → `1800` `Tab` `800`, mill `3200,3600` → `1200` `Tab` `900`, drill `5400,3800` → `700` `Tab` `700`, workbench `600,600` → `2400` `Tab` `900`.
2. Machine labels: `DT` with Text height `200`. Type `LATHE` at about `700,3200`, `MILL` at `3300,3200`, `DRILL` at `5400,3400` and `WORKBENCH` at `700,1700`.
3. Room title: `DT` with Text height `400`, `WORKSHOP 8 x 5 m` at about `1400,2300`.
4. Set the Annotation scale box on the Status bar to `1:50`. Select the four machine labels and type `OBJECTSCALE ADD`. With no number it uses the current scale, so they now belong to 1:50.
5. Select the room title and type `OBJECTSCALE ADD 100`.
6. Switch the Status bar between `1:50` and `1:100`. The machine labels appear only at 1:50 and the title only at 1:100. `OBJECTSCALE LIST` confirms the scales of the selected text.



The same workshop plan at 1:50 (machine labels) and at 1:100 (room title)

Challenge: Type `ANNOALLVISIBLE ON` to see every label at once while you edit, then `ANNOALLVISIBLE OFF` to go back.

See also

- DIMSTYLE
- Units and system variables

Chapter 5 quiz: Dimension & Annotation

Choose the one best answer. Answers are in Appendix A

1. **With Smart Dimension (DIM), what do you get when you click one edge and then a parallel edge?**

   A. The angle between them
   B. The length of the first edge only
   C. The distance between the edges
   D. A diameter dimension

2. **While a Smart Dimension is attached to the mouse, what happens if you type a number?**

   A. It replaces the displayed value (override)
   B. It changes the text height
   C. It moves the dimension by that distance
   D. It sets the number of decimals

3. **Which command (alias `D`) opens the dialog for changing the dimension text height, arrow size and decimals?**

   A. DIM
   B. DIMSTYLE
   C. DIMLINEAR
   D. DIMSCALE

4. **While typing multiline text (MTEXT), which key finishes the text?**

   A. Enter
   B. Esc
   C. Tab
   D. Ctrl+Enter

5. **Which command measures the true distance along the line that joins two points?**

   A. DIMLINEAR (DLI)
   B. DIMVERTICAL (DVE)
   C. DIMALIGNED (DAL)
   D. DIMORDINATE

6. **With `MLEADER <text>`, what are the three clicks, in order?**

   A. Text location, elbow, arrow tip
   B. Arrow tip, elbow, text location
   C. Elbow, arrow tip, text location
   D. Arrow tip, text location, elbow

CHAPTER 6

Layer & Properties

1. LAYER - Layer Manager

   Turn layers on and off, freeze and lock them with the Layer Manager, set the current layer, rename layers and save layer states.

2. Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)

   Quick layer commands that work from selected objects - isolate (LAYISO), turn off (LAYOFF), freeze, lock and set current.

3. PROPERTIES, MATCHPROP and CHPROP

   View and edit object properties in the Properties palette, copy properties (MATCHPROP), and change Layer/color with CHPROP.

4. COLOR, LINETYPE and LWEIGHT

   Set the colour (COLOR), linetype (LINETYPE) and lineweight (LWEIGHT) for new objects or the selected objects.

LAYER - Layer Manager

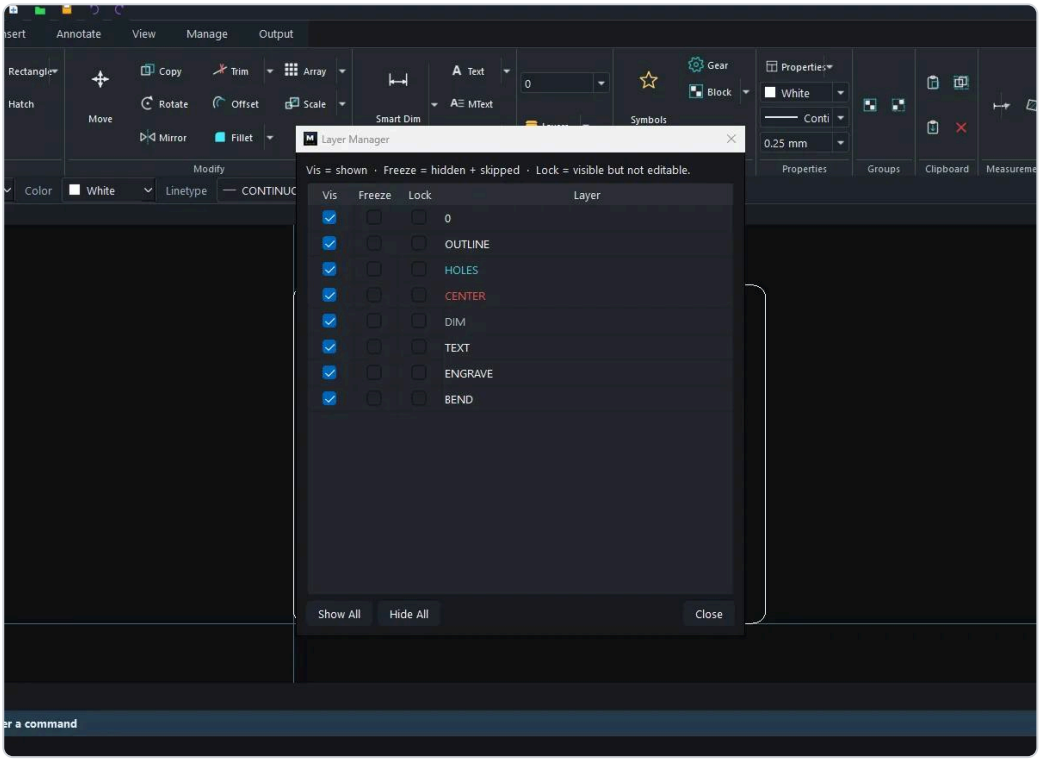
Turn layers on and off, freeze and lock them with the Layer Manager, set the current layer, rename layers and save layer states.

What it does

The Layer Manager lists every layer in the drawing and controls how each layer is displayed and edited.

Where to find it

- Ribbon: **Draw › Layers › Layers** (the submenu has the full set of LAY* commands) or **Manage › CAD Standards › Layers**
- Command line: `LAYER` (alias `LA`)
- The Layer box on the Ribbon (Layers group) and the Properties bar below the Ribbon set the current layer.
- Menu: **View › Layers Palette**



Layer Manager window

Options

Option	Meaning
Vis	Show (ticked) / hide.
Freeze	Hides the layer and skips it in processing. LAYON does not turn it back on. You must thaw it.
Lock	The layer is visible, but you cannot select or edit its objects.
Layer	The layer name.
Show All / Hide All	Shows / hides every layer.
Close	Closes the window.

Set the current layer

- Choose from the Layer box on the Ribbon. New objects go on that layer.
- Or click an object and type `LAYMCUR` (makes the object's layer current).

Create a new layer

A new layer is created when you give an object a layer name:

1. Select the objects.
2. Type `CHPROP layer=HOLES` and press Enter.
3. Click that object and type `LAYMCUR` to make HOLES the current layer.

> **Note:** All layers in an opened DWG/DXF file appear automatically in the Layer Manager and in the Layer box.

Layer-related commands

Command	Alias	Syntax	Meaning
RENAME	`REN`	`RENAME <old name> <new name>`	Renames a layer and moves all its objects.
LAYERSTATE	`LAS`, `LMAN`	`LAYERSTATE <Save|Restore|List|Delete> [name]`	Saves or restores the Off/Freeze/Lock state of every layer.

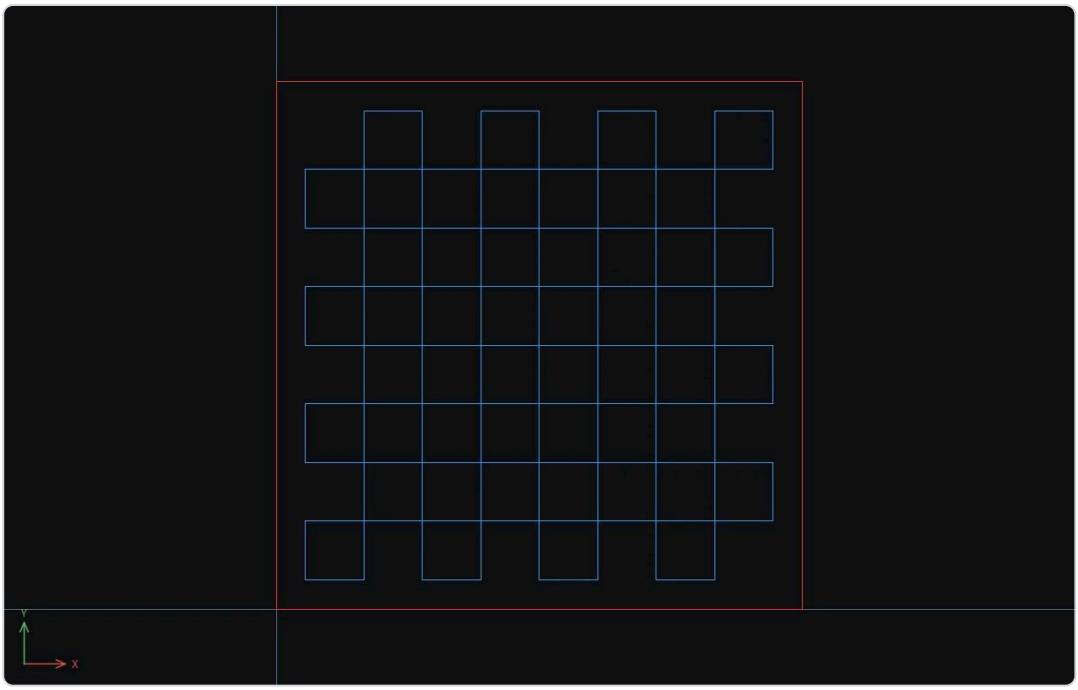
Steps: save a layer state for drawing review

1. Turn off or freeze the layers you do not want to see.
2. Type `LAYERSTATE SAVE review`.
3. Later, type `LAYERSTATE RESTORE review` to bring back the state of every layer.

Try it: Pocket checkers board with CUT and ENGRAVE layers

A 180 mm plywood checkers board. The laser cuts the outline and engraves the 32 dark squares. Put each job on its own layer, then use the Layer Manager to check the cut path on its own.

1. Board: `REC`, `0,0`, `@180,180`, Esc. Click it and type `CHPROP layer=CUT color=#FF3030`. The new CUT layer appears in the Layer box.
2. Dark squares: `REC 10,10 @20,20`, Esc, and `REC 30,30 @20,20`, Esc.
3. Drag a window around both squares and type `ARRAYRECT 4 40 0`. Drag a window around the 8 squares (from 5,5 to 175,55) and type `ARRAYRECT 4 0 40`. That makes 32 squares.
4. Type `QSELECT rect 0`. It selects the 32 squares that are still on layer 0. Then type `CHPROP layer=ENGRAVE color=#3399FF`.
5. Type `LA` to open the Layer Manager and untick **Vis** for ENGRAVE. Only the cut path is left. Type `LAYERSTATE SAVE cutcheck`. The result reads `LAYERSTATE: saved 'cutcheck' - 1 off, 0 frozen, 0 locked`.
6. Tick ENGRAVE back on. From now on, `LAYERSTATE RESTORE cutcheck` gives you the cut-only view in one command.



Checkers board with a red CUT outline and 32 blue ENGRAVE squares

Challenge: Add a layer called SCORE for a light line around the playing area with `REC 8,8 @164,164` and `CHPROP layer=SCORE color=#00C000`.

See also

- Layer tools
- PROPERTIES and CHPROP
- PURGE

Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...)

Quick layer commands that work from selected objects - isolate (LAYISO), turn off (LAYOFF), freeze, lock and set current.

What it does

These commands control layers by clicking objects in the drawing, so you do not need to remember layer names. If nothing is selected, the command asks `select object on layer`. Click an object, then press Enter.

Command	Alias	Meaning
LAYISO	`LI2`	Hides every layer except the layer of the selected objects.
LAYUNISO	–	Cancels LAYISO and shows every layer.
LAYOFF	–	Hides the layer of the selected objects.
LAYON	–	Shows every layer.
LAYFRZ	–	Freezes the layer of the selected objects, or `LAYFRZ <name>`.
LAYTHW	–	Thaws every layer, or `LAYTHW <name>`.

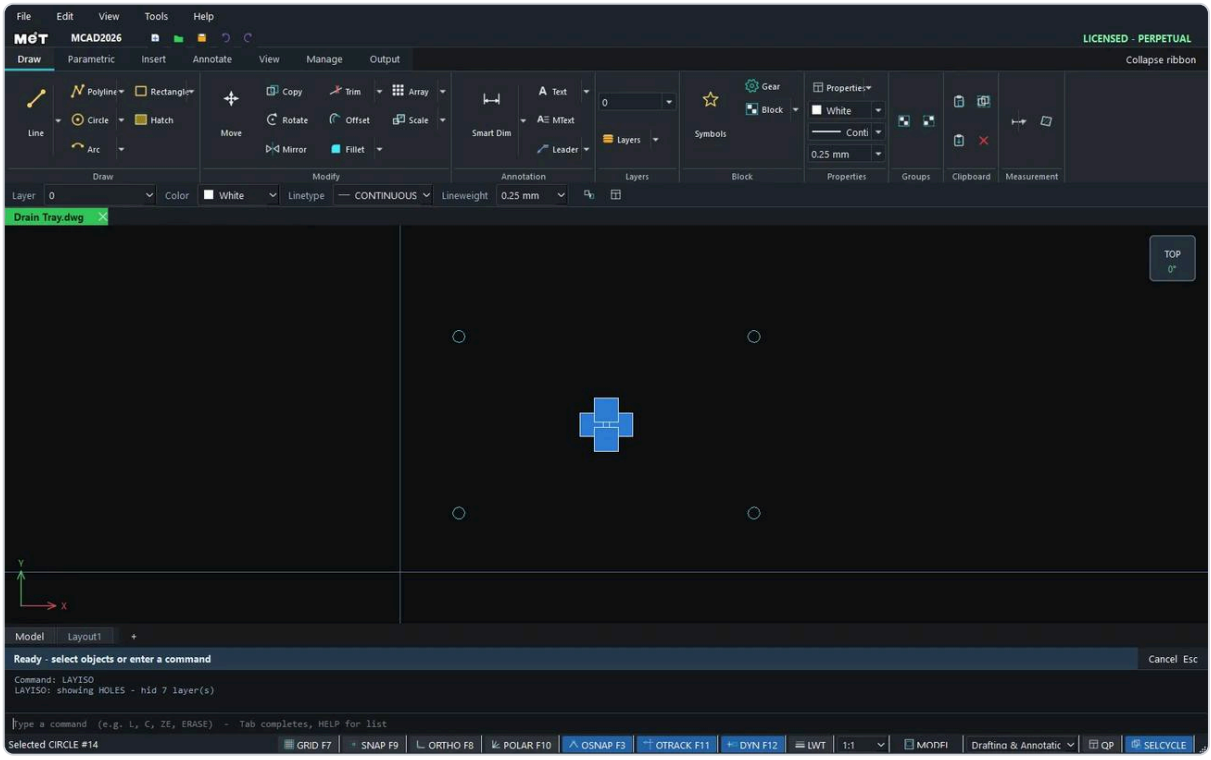
Command	Alias	Meaning
LAYLCK	–	Locks the layer of the selected objects, or `LAYLCK <name>`.
LAYULK	–	Unlocks every layer, or `LAYULK <name>`.
LAYMCUR	`LAYCUR`	Makes the layer of the selected object the current layer.

Where to find it

- Ribbon: **Draw › Layers › Layers** ▾ (Isolate, Un-isolate, Turn Off, Turn Every Layer On, Freeze, Thaw Every Layer, Lock, Unlock Every Layer, Make Current, Saved Layer States)

Steps: view only the hole layer

1. Click one hole.
2. Type `LAYISO` and press Enter. The result reads `LAYISO: showing HOLES - hid 2 layer(s)`.
3. When you finish editing, type `LAYUNISO`.



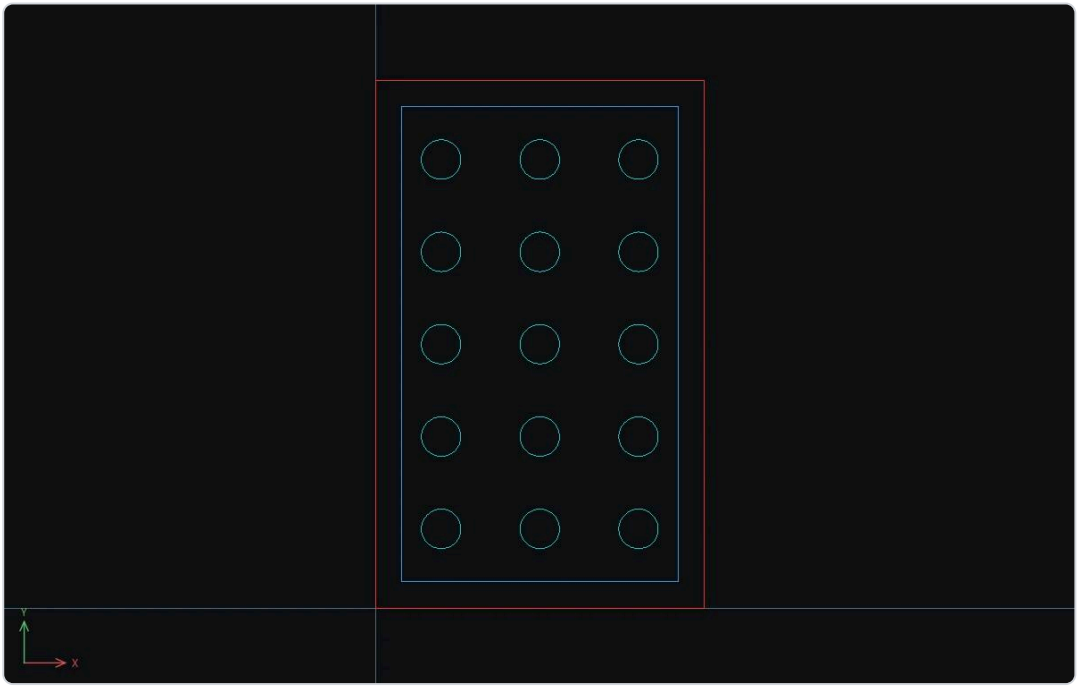
After LAYISO, only the HOLES layer remains

Try it: Lantern side panel

One side of a 100 x 160 mm plywood lantern: the outline is cut, a border is engraved, and 15 light holes are cut on their own layer. Layer tools let you check each job just by clicking an object.

1. Panel: `REC`, `0,0`, `@100,160`, Esc. Click it and type `CHPROP layer=CUT color=#FF3030`.
2. Border: `REC`, `8,8`, `@84,144`, Esc. Click it and type `CHPROP layer=ENGRAVE color=#3399FF`.
3. Light hole: `C`, `20,24`, `6`, Esc. Click it and type `CHPROP layer=HOLES color=#00D0D0`.
4. Click the hole and type `ARRAYRECT 3 30 0`. Drag a window around the 3 holes and type `ARRAYRECT 5 0 28`. You now have 15 holes, all on HOLES.
5. Click one hole and type `LAYISO`. The result reads `LAYISO: showing HOLES - hid 2 layer(s)`. Check the hole spacing, then type `LAYUNISO`.

6. Click the border and type `LAYOFF` (`LAYOFF: hid ENGRAVE`). This is what the cut-only job looks like. Type `LAYON` to bring everything back.



Lantern panel with CUT, ENGRAVE and HOLES layers

Challenge: Click the border and type `LAYLCK`. Now drag a window over the whole panel: the engraved border can no longer be picked by mistake. `LAYULK` unlocks it.

See also

- Layer Manager
- ISOLATEOBJECTS

PROPERTIES, MATCHPROP and CHPROP

View and edit object properties in the Properties palette, copy properties (MATCHPROP), and change Layer/color with CHPROP.

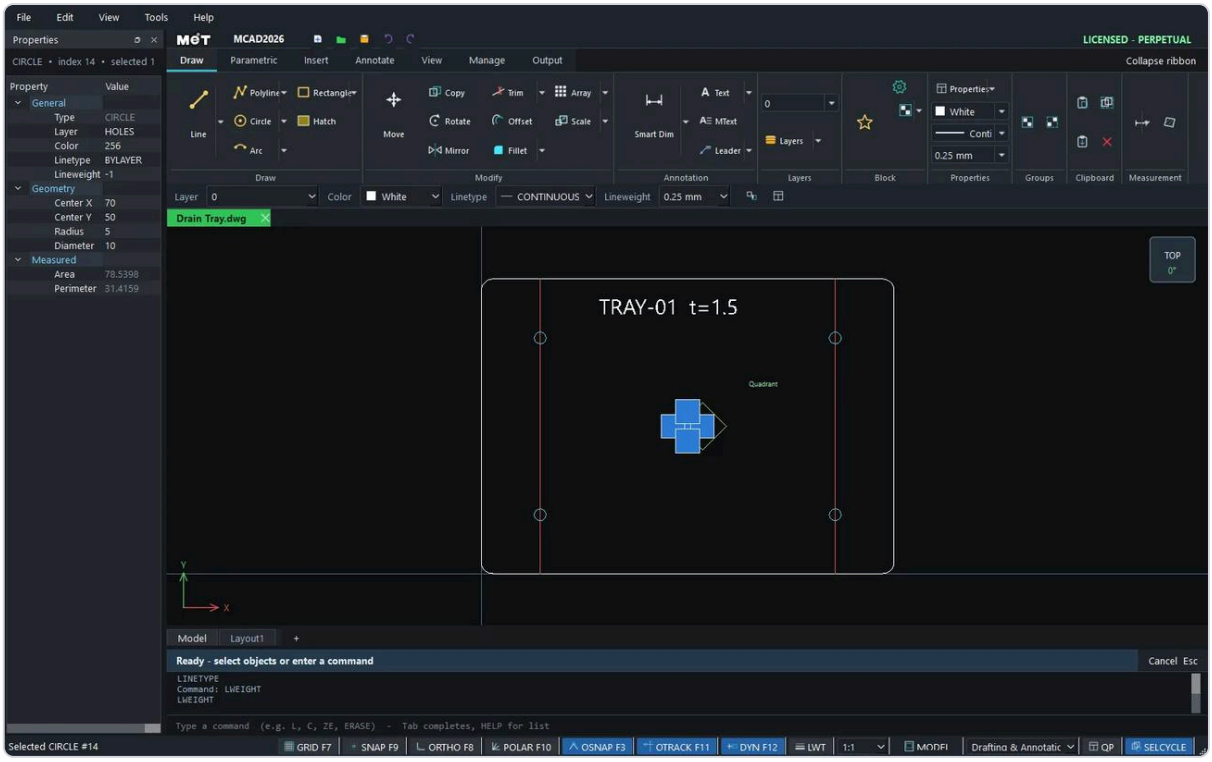
What it does

Command	Alias	Shortcut	Meaning
PROPERTIES	`CH`, `MO`, `PR`, `DDMODIFY`	`Ctrl+1`	Opens the Properties palette and summarizes the selection
MATCHPROP	`MA`	–	Click a source object, then click target objects to give them the same properties (Layer, color, Linetype, Lineweight)
CHPROP	`CHANGE`	–	`CHPROP layer=<name> color=<#RRGGBB or ACI number>`

Where to find it

- Ribbon: **Draw › Properties › Properties** ▾ (Properties Palette, Match Properties)

- Properties bar under the Ribbon: Layer, Color, Linetype, and Lineweight of the selected objects (or of new objects if nothing is selected)
- Right-click an object › Properties



Properties palette for the selected circle (left)

Properties palette

Section	Information
General	Type, Layer, Color, Linetype, Lineweight
Geometry	Coordinates and size, such as Center X/Y, Radius, Diameter (editable)
Measured	Area and Perimeter of closed objects

MATCHPROP steps

1. Type `MA` and press Enter.
2. Click the source object.
3. Click each object you want to change. Press Esc to finish.

CHPROP steps

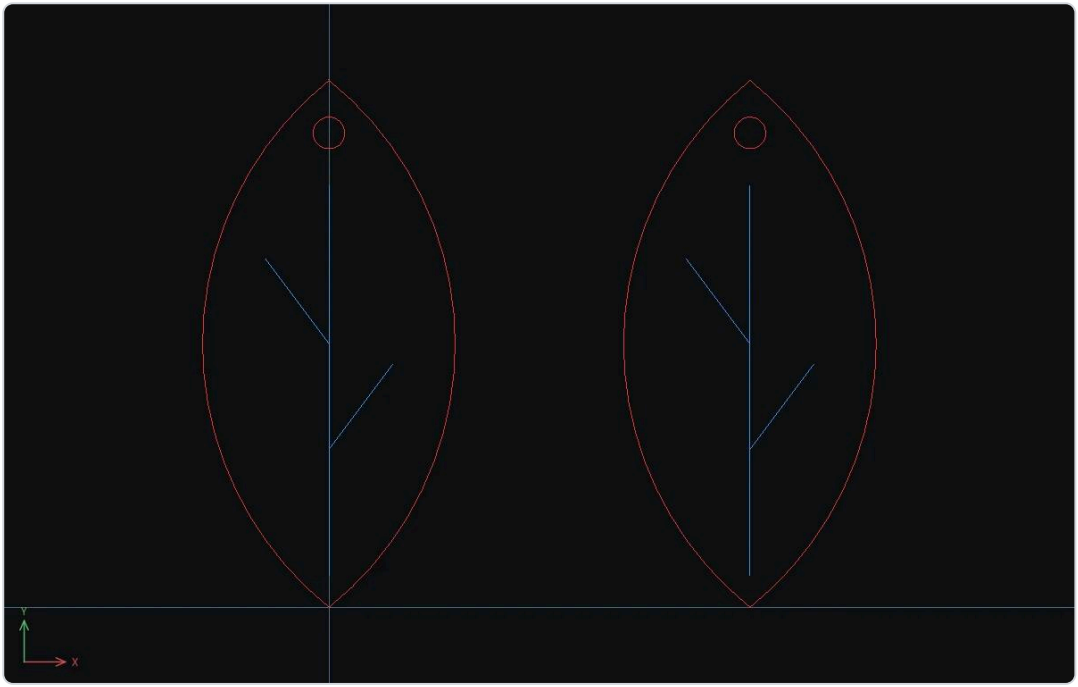
1. Select the objects.
2. Type `CHPROP layer=CENTER color=#FF0000` and press Enter.

Try it: Leaf earrings, cut and engraved

A pair of 50 mm leaf earrings in thin plywood: the outline and hook hole are cut, the veins are engraved. Style one earring, then MATCHPROP copies the look onto the other.

1. Leaf outline: `ARC`, then `0,0` `12,25` `0,50`, Esc. Then `ARC`, `0,0` `-12,25` `0,50`, Esc.
2. Veins: `L` `0,3` `0,40`, Esc. `L` `0,15` `@6,8`, Esc. `L` `0,25` `@-6,8`, Esc. Hook hole: `C`, `0,45`, `1.5`, Esc.

3. Drag a window around the leaf, type `CO` and press Enter. Base point `0,0`, destination `@40,0`, then press Enter. That is the second earring.
4. Style the first earring: click one arc, `Ctrl`+click the other arc and the hole, and type `CHPROP layer=CUT color=#FF3030`. Then select its 3 veins the same way and type `CHPROP layer=ENGRAVE color=#3399FF`.
5. Type `MA` and press Enter. Click an outline arc of the first leaf, then click both arcs and the hole of the second leaf. Press Esc.
6. Type `MA` again. Click a vein of the first leaf, then the 3 veins of the second leaf. Press Esc.
7. Select a vein of the second leaf and press `Ctrl+1`. **General** shows layer ENGRAVE.



Two leaf earrings with red cut outlines and blue engraved veins

Challenge: Select everything and type `CHPROP` with nothing after it. It reports which layers the selection is on, which is a quick check before you send the job to the laser.

See also

- COLOR, LINETYPE, LWEIGHT
- Editing with Grips and Quick Properties

COLOR, LINETYPE and LWEIGHT

Set the colour (COLOR), linetype (LINETYPE) and lineweight (LWEIGHT) for new objects or the selected objects.

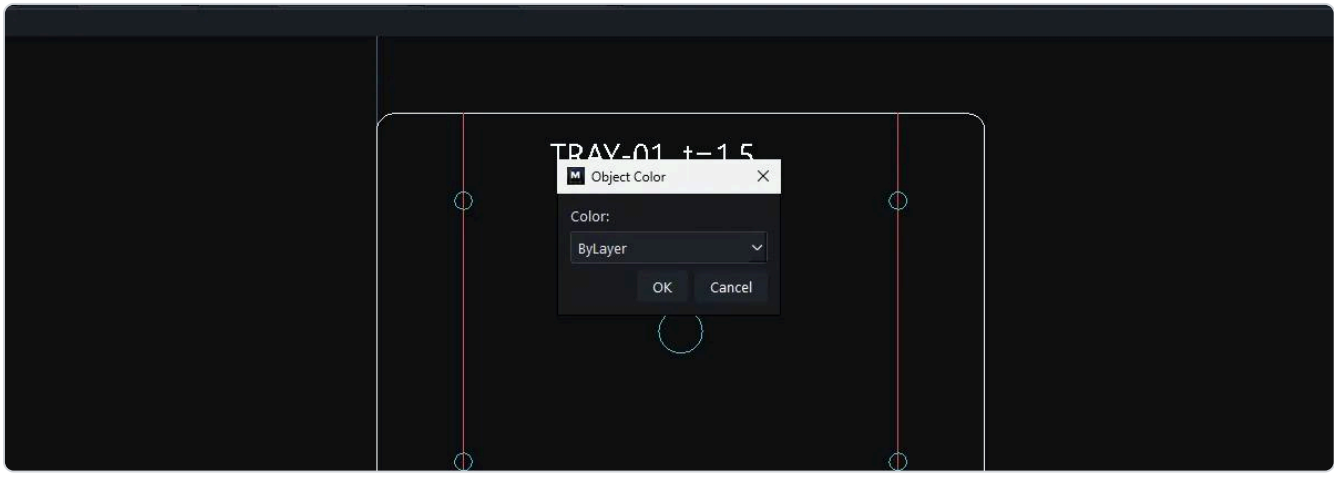
What it does

Command	Alias	Options
COLOR	`COL`, `DDCOLOR`	ByLayer, Red, Yellow, Green, Cyan, Blue, Magenta, White
LINETYPE	`LT`, `DDLTYPE`	BYLAYER, CONTINUOUS, DASHED, HIDDEN, CENTER
LWEIGHT	`LINEWEIGHT`	ByLayer, 0.13 – 0.70 mm

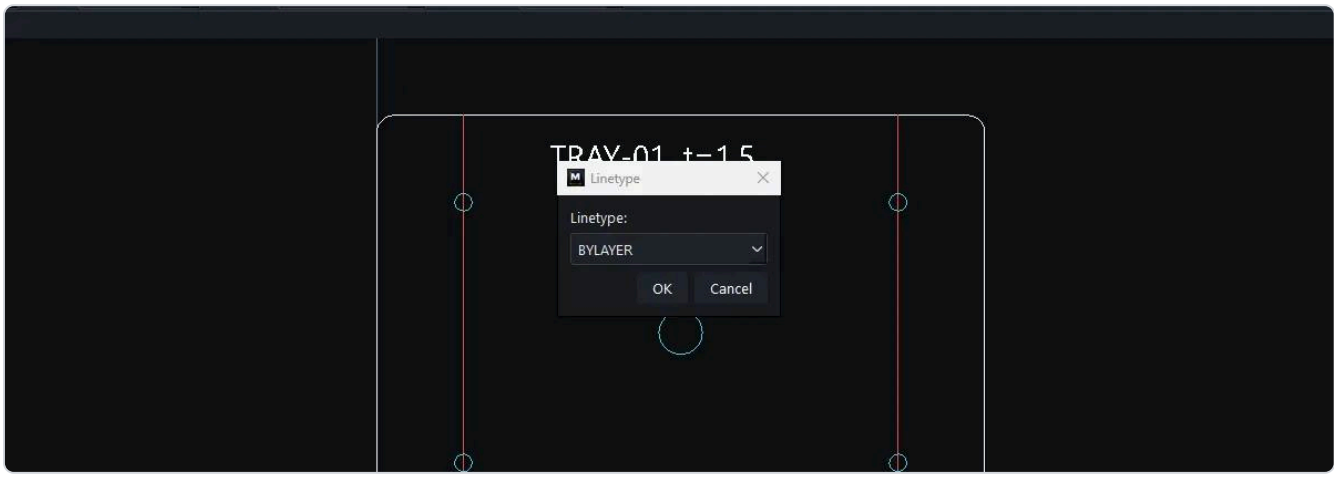
The value you choose applies to the new objects you draw from then on.

Where to find it

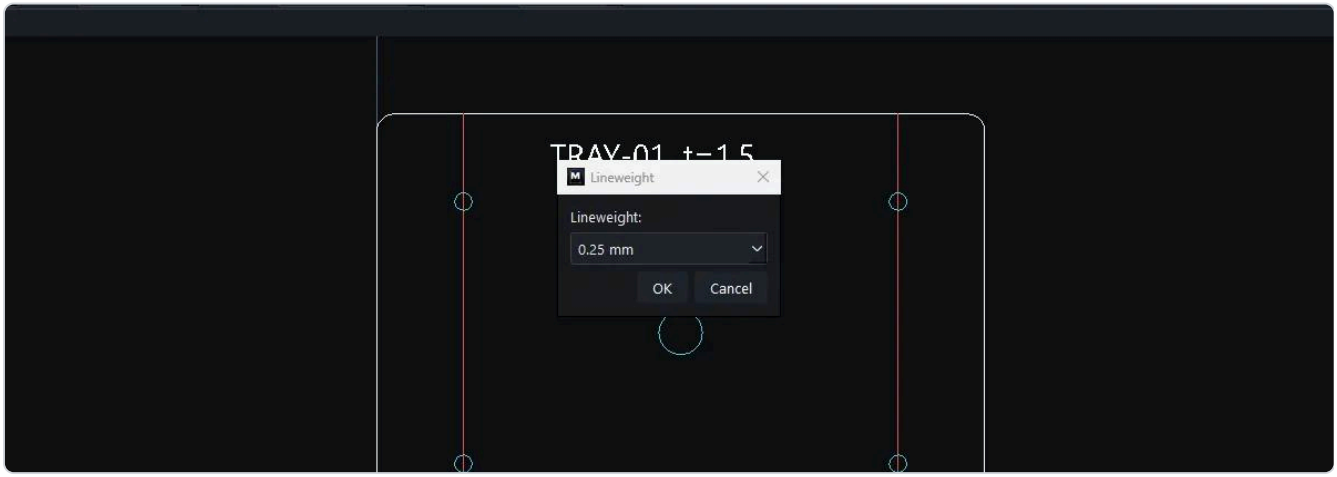
- The Properties bar below the Ribbon: the **Color**, **Linetype** and **Lineweight** boxes. If objects are selected, they change immediately.
- Ribbon: **Draw › Properties**
- Command line: see the table above



COLOR dialog



LINETYPE dialog



LWEIGHT dialog

Steps

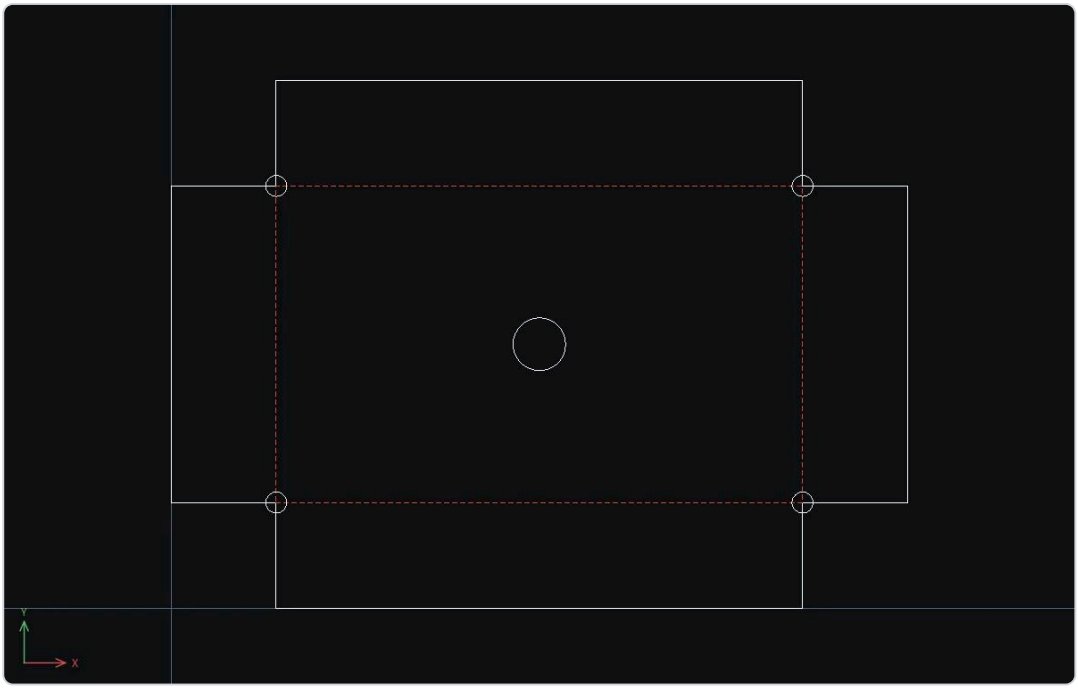
1. Type `LT` and press Enter.
2. Select `CENTER`, then click OK.
3. Draw a new line. It is a center line.

> **Note:** If the dashes look too dense or too sparse, adjust them with `LTSCALE`. See Units and system variables. Click LWT on the Status bar to show lineweights on screen.

Try it: Sheet-metal tray flat pattern

The flat pattern of a 100 x 60 mm tray with 20 mm walls. Cut lines stay continuous, and bend lines are red and dashed, so whoever runs the press brake sees at once where to fold.

1. Outline: `PL`, then `20,0` `@100,0` `@0,20` `@20,0` `@0,60` `@-20,0` `@0,20` `@-100,0` `@0,-20` `@-20,0` `@0,-60` `@20,0` `20,0`, and press Enter.
2. Type `LT`, press Enter, choose `DASHED` and click OK. Type `COL`, press Enter, choose `Red` and click OK.
3. Bend lines: `REC`, `20,20`, `@100,60`, Esc. They are drawn red and dashed.
4. Back to normal: `LT` → `CONTINUOUS`, then `COL` → `ByLayer`.
5. Corner relief holes: `C` `20,20` `2`, `C` `120,20` `2`, `C` `20,80` `2`, `C` `120,80` `2`. Drain hole: `C` `70,50` `5`. Press Esc after each one.



Tray flat pattern with red dashed bend lines

Challenge: Select the outline and use `LWEIGHT` to give it 0.50 mm, then click LWT on the status bar to see the cut line stand out.

See also

- PROPERTIES
- Layer Manager

Chapter 6 quiz: Layer & Properties

Choose the one best answer. Answers are in Appendix A

1. **What does locking a layer (Lock) do?**

   A. Hides the layer
   B. The layer stays visible, but you cannot select or edit its objects
   C. Deletes all objects on the layer
   D. Only stops the layer from printing

2. **A layer is frozen. How do you make it visible again?**

   A. Run LAYON
   B. Run LAYULK
   C. Run LAYMCUR
   D. Thaw it (for example with LAYTHW)

3. **You used LAYISO to show only one layer. Which command shows every layer again?**

   A. LAYUNISO
   B. LAYOFF
   C. LAYFRZ
   D. LAYLCK

4. **Which shortcut opens the Properties palette?**

   A. Ctrl+9
   B. Ctrl+0
   C. Ctrl+1
   D. Ctrl+Shift+P

5. **Which command copies the Layer, colour, linetype and lineweight from a source object to other objects?**

   A. CHPROP
   B. MATCHPROP (MA)
   C. LAYMCUR
   D. COPY

6. **A CENTER or DASHED line looks too dense or too sparse. What should you adjust?**

   A. LWEIGHT
   B. COLOR
   C. DIMSCALE
   D. LTSCALE

CHAPTER 7

Block & Symbol

1. BLOCK, INSERT and WBLOCK

   Create a block from selected objects (BLOCK), insert it (INSERT), list blocks (BLOCKLIST) and write one to a file (WBLOCK).

2. Block Editor and Dynamic Blocks (BEDIT, BPARAM)

   Edit a block definition in the Block Editor (BEDIT/BSAVE/BCLOSE) and add dynamic parameters (BPARAM/BSET).

3. Attributes (ATTDEF, ATTEDIT, ATTDISP)

   Define attribute text in a block (ATTDEF), edit its values (ATTEDIT) and show or hide it (ATTDISP).

4. Symbol Library (Content Browser)

   Browse and place Blocks and Symbols from the built-in library using the Content Browser.

5. External Reference (XREF, XATTACH, XCLIP, XBIND)

   Attach other DWG/DXF files as External References, clip their boundary (XCLIP), and merge them into the drawing (XBIND).

6. Images and PDF Underlays (IMAGE, PDFATTACH)

   Insert pictures and logos (IMAGE, INSERTOBJ) and a PDF as a background (PDFATTACH), with clipping and fading.

BLOCK, INSERT and WBLOCK

Create a block from selected objects (BLOCK), insert it (INSERT), list blocks (BLOCKLIST) and write one to a file (WBLOCK).

What it does

A block is a group of objects with a single definition that you can insert many times. When you edit the definition once, every copy changes.

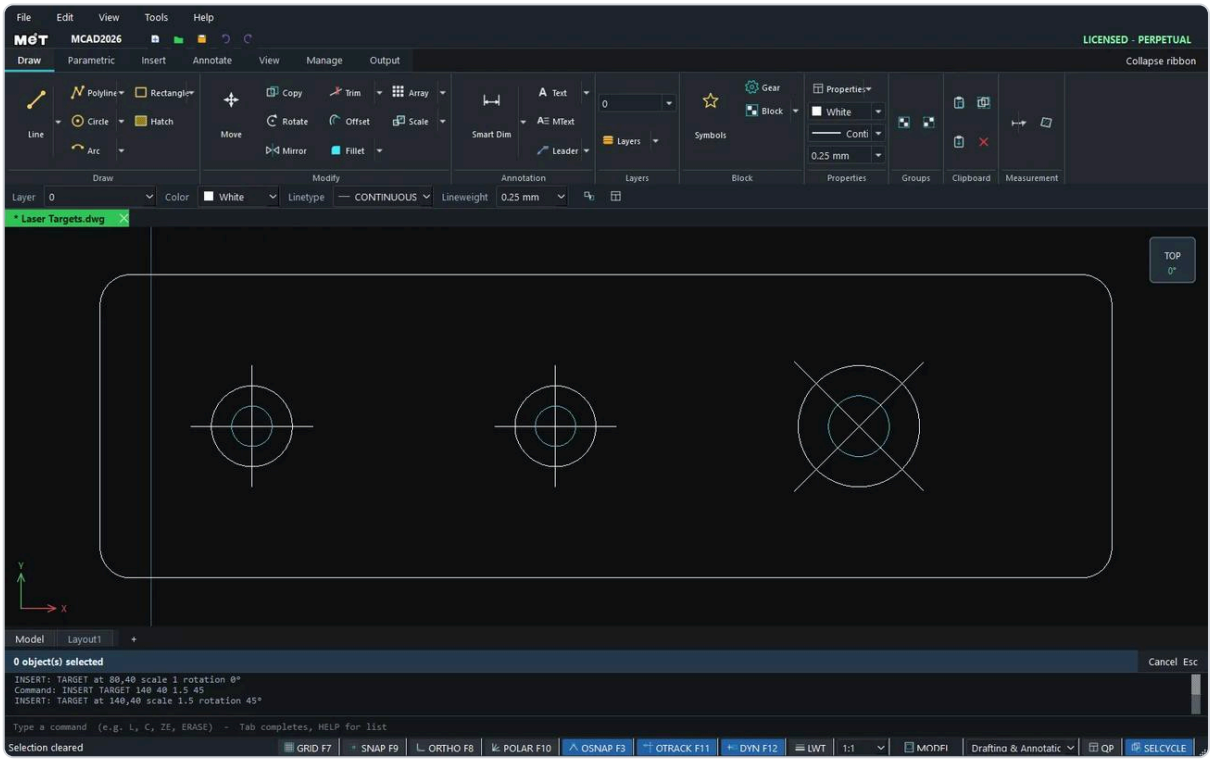
Command	Alias	Syntax	Meaning
BLOCK	`B`, `BMAKE`	`BLOCK <name> [baseX baseY]`	Creates a block definition from the selected objects and replaces them with one block reference. The default base point is the lower-left corner.
INSERT	`I`	`INSERT <name> [x y] [scale] [angle]`	Inserts a block that exists in the drawing. If you give no name, the Content Browser opens.
BLOCKLIST	`BL`	–	Lists the blocks in the drawing, their dynamic parameters and how many times each is used.
WBLOCK	`W`	–	Writes the selected objects to a new DXF file.

Where to find it

- Ribbon: **Draw › Block › Block** ▾ (Insert Block, Picture / Logo, Block Editor) and **Insert › Blocks & Media**

Steps

1. Select the objects to turn into a block.
2. Type `BLOCK TARGET 0 0` and press Enter (name TARGET, base point 0,0).
3. Insert more copies: `INSERT TARGET 50 0` and `INSERT TARGET 100 0 1.5 45` (scale 1.5, rotated 45°).
4. Check with `BLOCKLIST`. The result is `TARGET: static, no parameters [3 inserted]`.



Block TARGET inserted 3 times, the right one at scale 1.5 and rotated 45°

> **Note:** GROUP only keeps objects together, while BLOCK creates a reusable definition. In a block from an opened DWG/DXF file, one click selects the whole block, and you can break it apart with EXPLODE.

Try it: Wall sockets around a workshop

A 6 x 4 m workshop plan with power outlets on every wall. Draw the socket symbol once, make it a block, then INSERT places each socket with the right position and rotation.

1. Socket symbol at the origin: `A` (3-point arc) through `-150,150`, `0,300`, `150,150`; then `L` `-150,150` → `150,150` and `L` `0,0` → `0,150`.
2. Select the three objects and type `BLOCK SOCKET 0 0`. The point `0,0`, where the symbol touches the wall, becomes the insertion point.
3. Erase this first copy with `E`. The definition stays in the drawing.
4. Walls: `REC` `0,0` → `6000` `Tab` `4000`, then `REC` `-150,-150` → `6300` `Tab` `4300`.
5. Sockets: `INSERT SOCKET 1500 0`, `INSERT SOCKET 4500 0`, `INSERT SOCKET 3000 4000 1 180`, `INSERT SOCKET 0 2000 1 -90`, `INSERT SOCKET 6000 2000 1 90`.
6. A bigger outlet for the welder: `INSERT SOCKET 6000 3200 1.5 90`. `BLOCKLIST` reports `SOCKET: static, no parameters [6 inserted]`.



Workshop plan with six SOCKET blocks on the walls, one at scale 1.5 for the welder

Challenge: Open the definition with BEDIT, add two short earth-pin lines, `BSAVE`, and all six sockets change together.

See also

- Block Editor and dynamic blocks
- Attributes
- Symbol Library

Block Editor and Dynamic Blocks (BEDIT, BPARAM)

Edit a block definition in the Block Editor (BEDIT/BSAVE/BCLOSE) and add dynamic parameters (BPARAM/BSET).

What it does

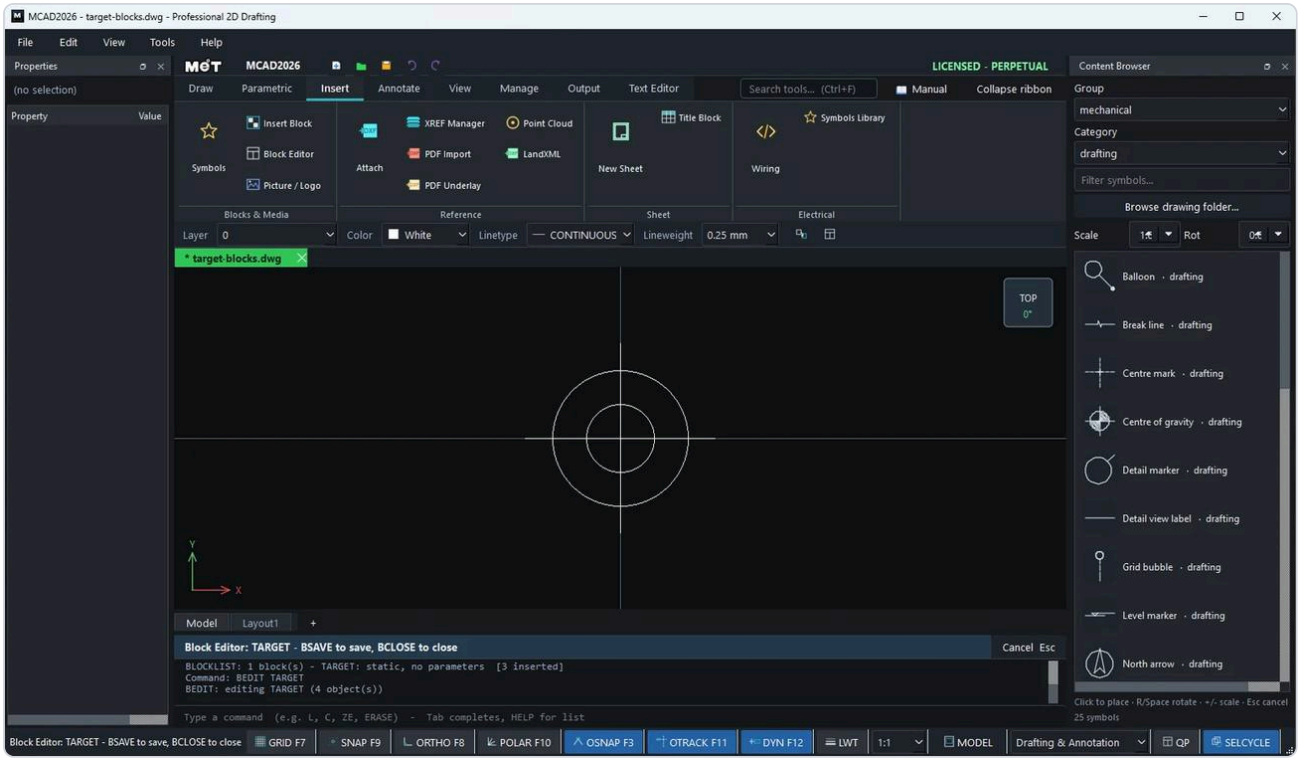
Command	Alias	Meaning
BEDIT	`BE`	`BEDIT [name]` opens the block definition (the selected block, or the one you name) separately in the drawing area for editing.
BSAVE	–	Saves the definition and updates every block reference.
BCLOSE	–	`BCLOSE SAVE` or `BCLOSE DISCARD` closes the Block Editor.
BPARAM	`BPAR`	Adds a dynamic parameter to the selected block. Type `BPARAM` alone to list all syntaxes.
BSET	`BPROP`	`BSET <parameter> <value>` sets a parameter of the selected block. If you give no value, it shows the current value.

Where to find it

- Ribbon: **Insert › Blocks & Media › Block Editor** or **Draw › Block ▾ › Block Editor (BEDIT)**

Steps to edit a block

1. Type `BEDIT TARGET` (or click the block, then type `BE`).
2. The prompt bar shows `Block Editor: TARGET - BSAVE to save, BCLOSE to close`. Other objects are hidden. Edit the definition with the normal commands.
3. Type `BSAVE` to update every TARGET in the drawing, or `BCLOSE DISCARD` to discard your edits.



Block Editor with the TARGET definition open

BPARAM syntax

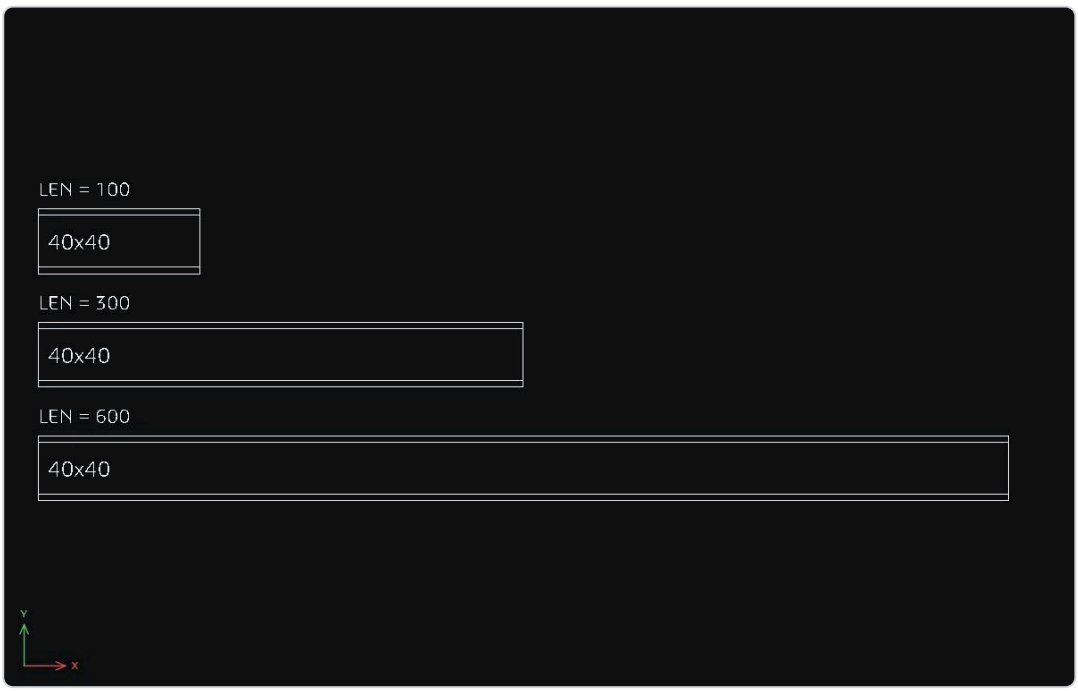
Type	Syntax
Linear (stretch)	`BPARAM <name> LINEAR <x|y> <base> <default> [min] [max]`
Flip	`BPARAM <name> FLIP <x|y> [at]`
Rotate	`BPARAM <name> ROTATE [aboutX] [aboutY]`
Array (repeat)	`BPARAM <name> ARRAY <x|y> <spacing>`
Visibility (display state)	`BPARAM <name> VISIBILITY <state>=<tag,tag> ...`
Lookup (value table)	`BPARAM <name> LOOKUP <row>=<PARAM:value,...> ...`

Example: select a block and type `BPARAM LEN LINEAR x 0 100 50 300`, then type `BSET LEN 150` to stretch it to 150.

Try it: A wiring duct that stretches

Panel builders use slotted wiring duct in every length. Make one 40 x 40 duct block, add a stretch parameter, and the same block becomes a 100, 300 or 600 mm piece.

1. Duct: `REC 0,0 → 100 Tab 40`, then the cover lines `L 0,4 → 100,4` and `L 0,36 → 100,36`.
2. Select all three and type `BLOCK DUCT40 0 0`.
3. Type `BEDIT DUCT40`. Add the size label: `DT` with Text height `9`, type `40x40` at about `6,15`. Type `BSAVE`.
4. Click the duct and type `BPARAM LEN LINEAR x 50 100 50 2000`. Everything right of X 50 now stretches; the default length is 100, the limits 50 to 2000.
5. `INSERT DUCT40 0 -70`, then `BSET LEN 300`.
6. `INSERT DUCT40 0 -140`, then `BSET LEN 600`. Check with `BLOCKLIST`.



One DUCT40 block at lengths 100, 300 and 600

Challenge: `BEDIT DUCT40` again, change the label to `40x60`, `BSAVE`, and all three ducts update at once.

See also

- BLOCK, INSERT
- EXPLODE

Attributes (ATTDEF, ATTEDIT, ATTDISP)

Define attribute text in a block (ATTDEF), edit its values (ATTEDIT) and show or hide it (ATTDISP).

What it does

An attribute is text with a tag that is attached to a block, for example a device number. When you insert the block, the program asks for the value using the prompt you defined.

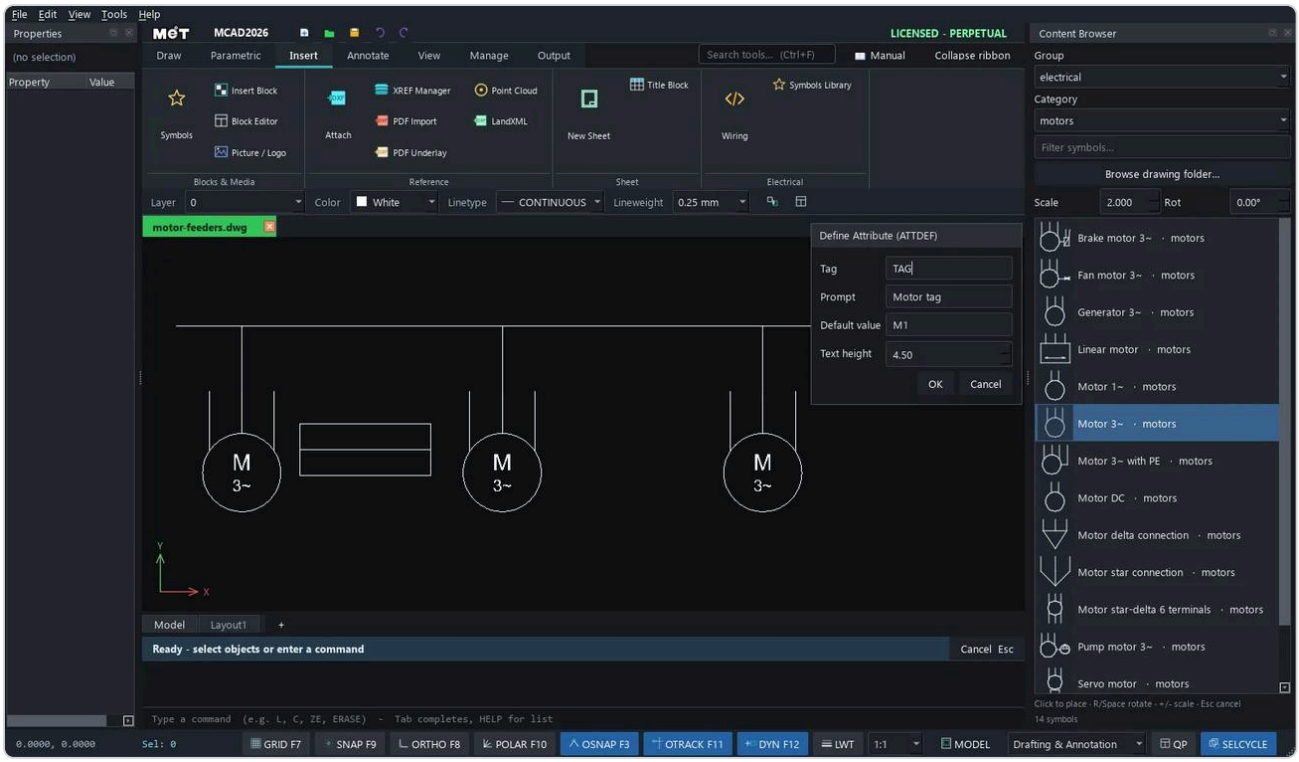
Command	Alias	Meaning
ATTDEF	`ATT`	Creates an attribute definition (tag, prompt, default value, height).
ATTEDIT	`ATE` , `EATTEDIT`	Click a block that has attributes, then edit their values.
ATTDISP	–	`ATTDISP <ON|OFF|Normal>` shows or hides all attributes.

Where to find it

- Ribbon: type the command
- Command line: `ATTDEF` (alias `ATT`)

Steps

1. Type `ATT` and press Enter. The **Define Attribute (ATTDEF)** dialog opens.
2. Enter a Tag (for example `TAG`), a Prompt (for example `Device tag`), a Default value and a Text height. Click OK, then click the position.
3. Select the shapes and the attribute, then create the block with `BLOCK <name>` .
4. When you place that block from the Content Browser (Group `Drawing`), the program asks for each attribute value. A block you just made shows up in that group after you save and reopen the drawing. Typing `INSERT <name>` places the block with its default values and does not ask.



Define Attribute dialog

Option	Meaning
Tag	The attribute name (used as an identifier).
Prompt	The text shown when you insert the block.
Default value	The initial value.
Text height	The text height.

> **Note:** Use `ATTDISP OFF` to hide all part numbers before you plot a general arrangement drawing.

Try it: Motor tags with power ratings

Three conveyor motors on one supply, each with a small tag box that shows its name and power. The tag text is attributes, so you change the values in a form instead of hunting for the right text.

1. Motors: open the Content Browser, Group `electrical`, Category `motors`, set **Scale** `2`, click **Motor 3~** and click at `0,0`, `80,0` and `160,0`. The motor hangs below the point you click, with its middle lead at X+10. Press Esc. Supply line: `L` `-10,20` → `240,20`, and a drop to each motor's middle lead: `L` `10,0` → `10,20`, `L` `90,0` → `90,20` and `L` `170,0` → `170,20`.
2. Tag box: `REC` `28,-26` → `40` `Tab` `16`, then `L` `28,-18` → `68,-18` across the middle.
3. `ATT`. Tag `TAG`, Prompt `Motor tag`, Default value `M1`, Text height `4.5`. Click **OK** and click at `31,-16`.
4. `ATT` again. Tag `POWER`, Prompt `Rating`, Default value `0.75 kW`, Text height `4`. Click at `31,-24`.
5. Drag a window around the tag box and its two attributes, type `CO`, click `28,-26` as the base point and `108,-26`, `188,-26` as the destinations.
6. Drag a window around the middle tag and type `ATE`. The form asks **Motor tag:** and **Rating:**. Type `M2` and `1.5 kW`, then OK. Do the same on the right tag: `M3`, `2.2 kW`.



Three motors on one supply, tagged M1 0.75 kW, M2 1.5 kW and M3 2.2 kW

Challenge: Type `ATTDISP OFF` to hide every tag for a clean layout print, then `ATTDISP ON` to bring them back.

See also

- BLOCK
- Electrical Symbols

Symbol Library (Content Browser)

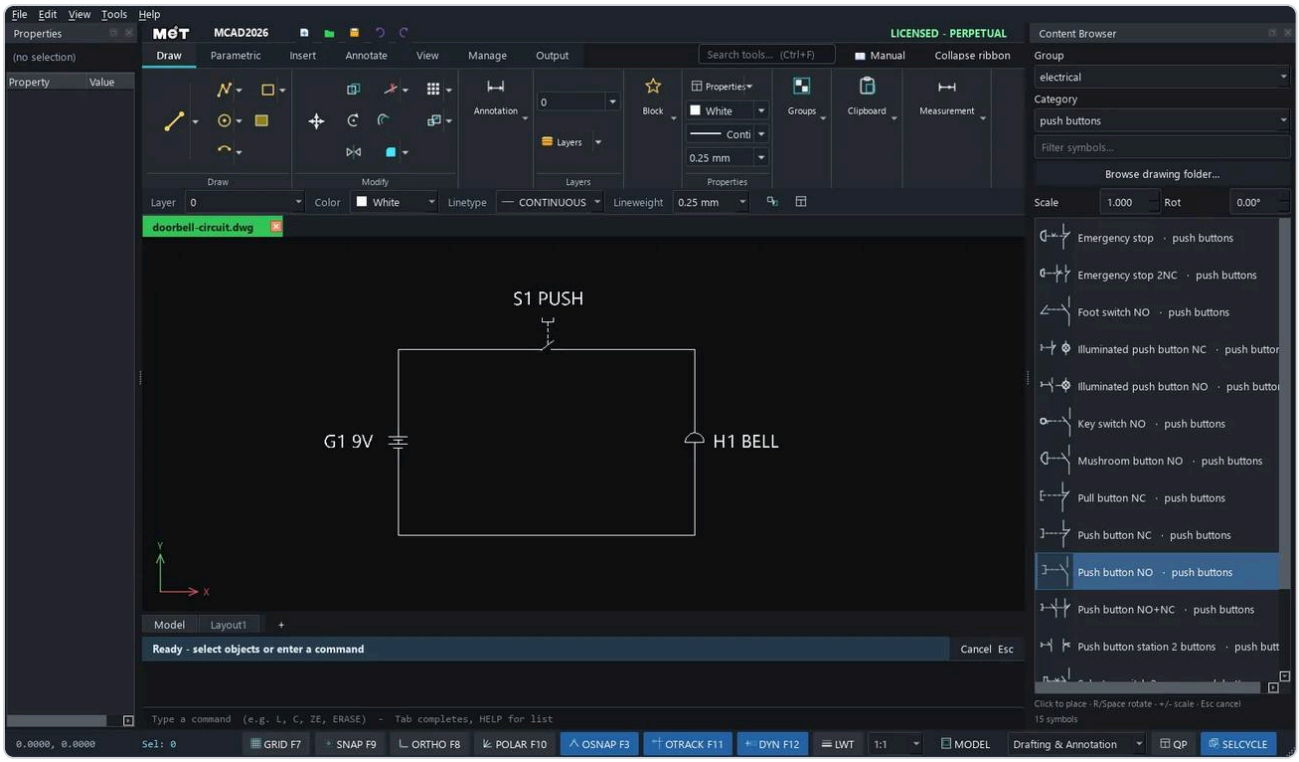
Browse and place Blocks and Symbols from the built-in library using the Content Browser.

What it does

The Content Browser (Symbols palette) holds MET's own library of 547 symbols, drawn by MET to IEC 60617 (electrical), ISO 1219 (pneumatic and hydraulic) and ISO 128 (mechanical drafting). They are sorted into four groups, and each one has a thumbnail. Click a symbol and place it on the drawing right away.

Where to find it

- **Content Browser** palette on the right side of the screen
- Ribbon: **Draw › Block › Symbols**, **Insert › Blocks & Media › Symbols**, **Insert › Electrical › Symbols Library**
- Menu **View › Symbols Palette**
- Command line: `INSERT` (without a name)



Content Browser, electrical group, push buttons category, with a doorbell circuit placed on the drawing

Options

Option	Meaning
Group	`electrical` (240 symbols), `pneumatic` (90), `mechanical` (156) and `architecture` (61). Also `Drawing` (the Blocks in the current drawing) and `Folder` (your own DXF folder)
Category	A subcategory of the group, for example `push buttons`, `protection` or `motors` in electrical, `valves` or `cylinders` in pneumatic, `fasteners` or `welding` in mechanical. Choose (all) to see the whole group
Filter symbols...	Search by symbol name or category name
Browse drawing folder...	Use the DXF files in your own folder as a library (Folder group)
Scale / Rot	Scale and angle when placing

Steps

1. Choose a **Group** and **Category**.
2. Click a Symbol in the list.
3. Click a location on the drawing. While placing: `R` or `Space` = rotate, `+` / `-` = scale, `Esc` = cancel.
4. You can place it repeatedly until you press Esc.

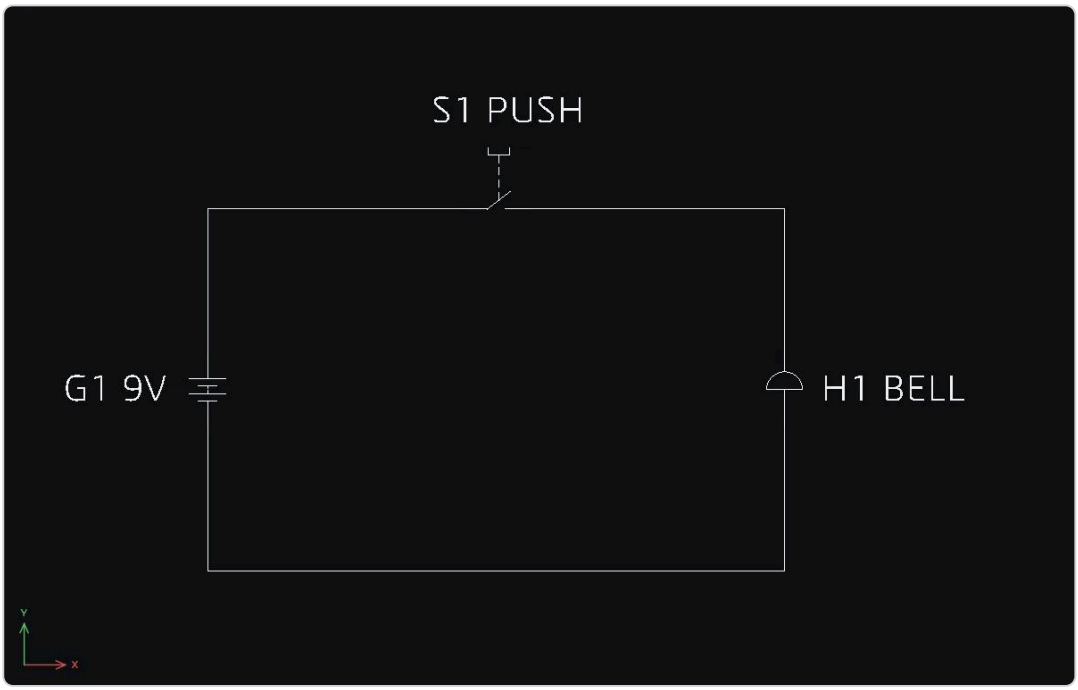
Every symbol is drawn in millimetres on a 2.5 mm grid. The insertion point is pin 1 (the top connection of a vertical symbol), so the symbol lands with its pin exactly on the point you click. Electrical and pneumatic symbols are drawn vertically; set **Rot** `90` or `-90` to lay one on a horizontal wire.

> **Note:** The symbols are drawn by MET and licensed with MCAD. You may use them in any drawing you make with MCAD. See `ATTRIBUTION.md` in the program's `library` folder for the standards each group follows.

Try it: Doorbell circuit

The simplest circuit in the house: a 9 V battery, a push button and a bell. Three symbols from the library and a few wires, done in two minutes.

1. Content Browser: Group `electrical`, Category `power supply`, click **Battery** and click at `0,30`. The battery stands upright with its pins at `0,30` and `0,20`. Press Esc.
2. Category `push buttons`, set **Rot** `-90`, click **Push button NO** (a normally open push button) and click at `45,50`. The button lies on the top wire between `35,50` and `45,50`. Press Esc and set **Rot** back to `0`.
3. Category `indication`, click **Bell** and click at `80,30`.
4. Wires with `L`, snapping to the symbol pins: `0,30` → `0,50` → `35,50`, then `45,50` → `80,50` → `80,30`, then `80,20` → `80,0` → `0,0` → `0,20`.
5. Labels with `DT`, Text height `3.5`: `G1 9V` at `-20,23.5`, `S1 PUSH` at `31,62`, `H1 BELL` at `85,23.5`.



Doorbell circuit: battery G1, push-button S1 and bell H1

Challenge: Swap the bell for a **Buzzer** from the same category. Type `buzz` in the Filter box to find it quickly.

See also

- Electrical Symbols
- BLOCK

External Reference (XREF, XATTACH, XCLIP, XBIND)

Attach other DWG/DXF files as External References, clip their boundary (XCLIP), and merge them into the drawing (XBIND).

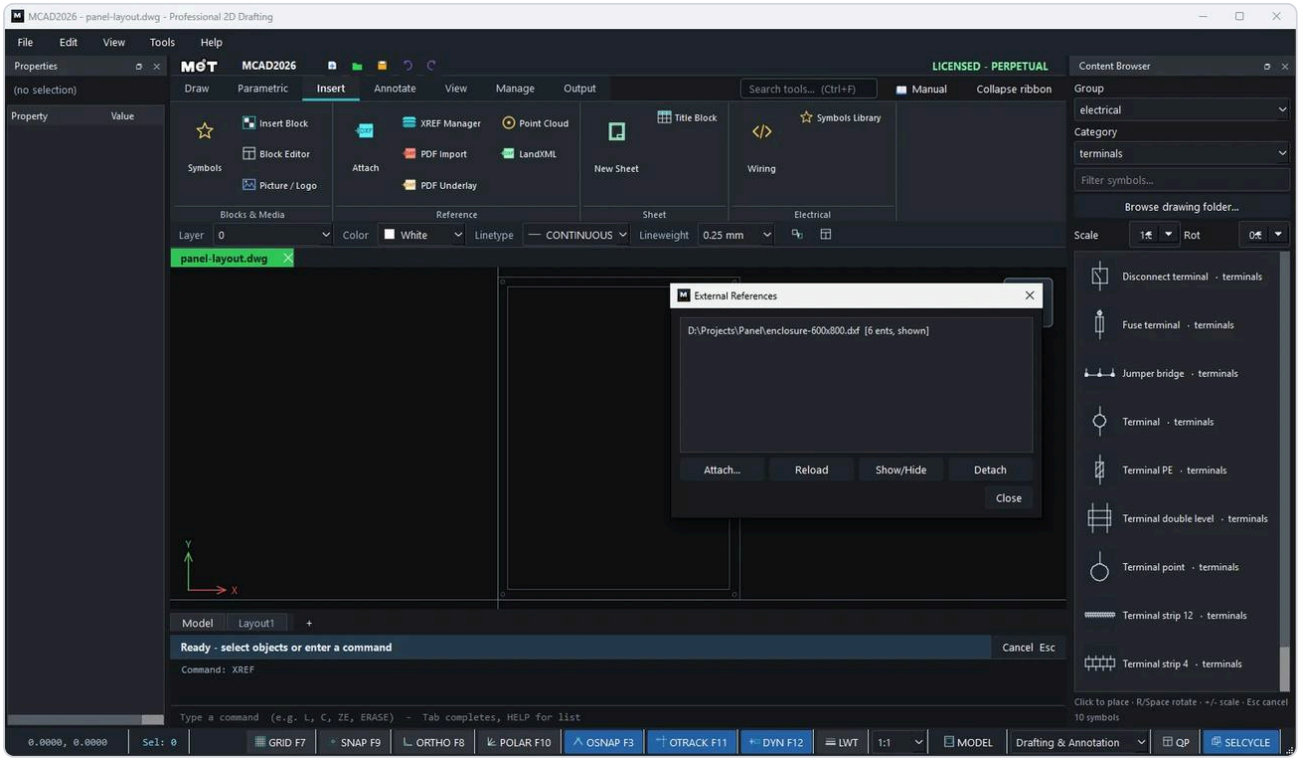
What it does

An External Reference (Xref) shows a drawing from another file as a reference background without copying it into your drawing.

Command	Alias	Format	Meaning
XREF	`XR`	–	Opens the External References window (Attach, Reload, Show/Hide, Detach)
XATTACH	`XA`	–	Choose a DWG/DXF file to attach
XCLIP	`XC`	`XCLIP [Delete] [index]`	Shows only the part of the Xref inside the selected closed shape
XBIND	`XB`	`XBIND [index]`	Copies the Xref objects into the drawing (onto Layer `XREF-<file name>`) and breaks the link

Where to find it

- Ribbon: **Insert › Reference › Attach / XREF Manager**



External References window

Steps

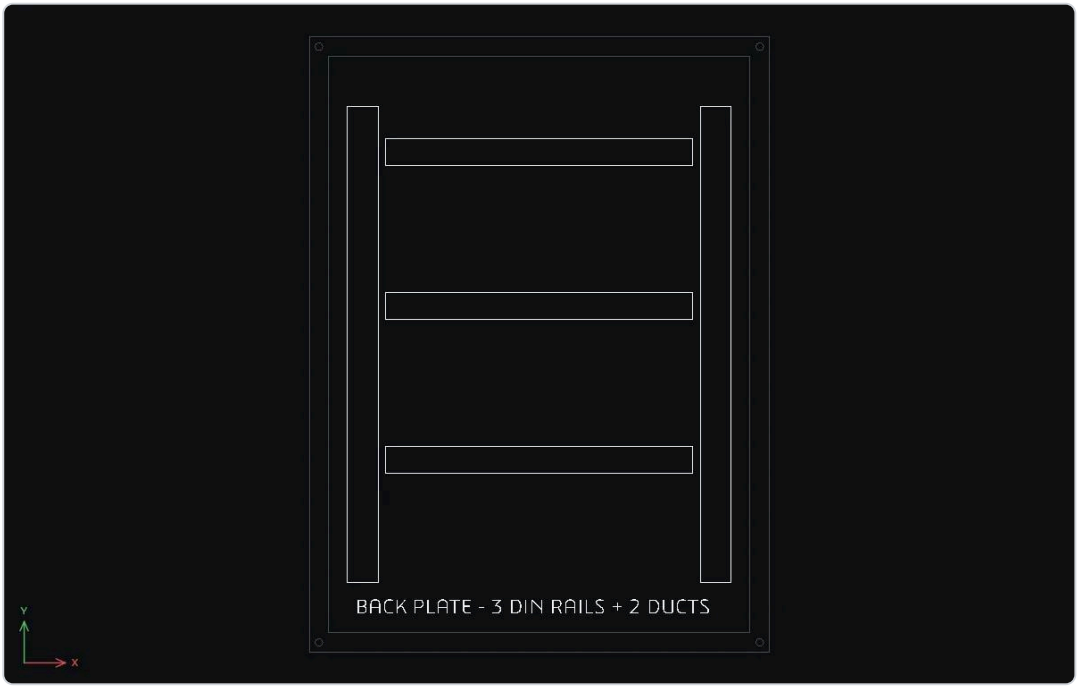
1. Choose **Insert › Reference › Attach** (or type `XA`) and select a file.
2. Open `XREF` to see the list. Use Reload when the source file changes, Show/Hide, or Detach.
3. To clip: draw a closed shape, select it, then run `XCLIP` (delete the clip with `XCLIP D`).
4. Before you send the drawing to someone else, use `XBIND` or `ETRANSMIT`.

Try it: Back-plate layout over the enclosure drawing

The enclosure supplier's drawing stays in its own file. You attach it as an Xref and lay out DIN rails and wiring ducts on top, so a new revision from the supplier just needs Reload.

1. New drawing for the enclosure: `REC 0,0 → 600 Tab 800`, `REC 25,25 → 550 Tab 750`, and four Ø10 fixing holes `C` radius `5` at `12.5,12.5`, `587.5,12.5`, `12.5,787.5` and `587.5,787.5`. Save it as `enclosure_600x800.dwg`.

2. Start another new drawing and choose **Insert › Reference › Attach** (`XA`). Pick `enclosure_600x800.dwg` . It appears faded at `0,0` .
3. Three 35 mm DIN rails: `REC` `100,632.5` → `400` `Tab` `35` , then the same at `100,432.5` and `100,232.5` .
4. Two vertical wiring ducts: `REC` `50,90` → `40` `Tab` `620` and `REC` `510,90` → `40` `Tab` `620` .
5. Note: `DT` with Text height `18` , `BACK PLATE - 3 DIN RAILS + 2 DUCTS` at about `60,50` .
6. Open `XREF` to see the attached file. Use Reload whenever the enclosure file changes.



Faded enclosure Xref with three DIN rails and two ducts drawn on top

Challenge: Before you send the job out, run `XBIND` so the enclosure becomes part of your drawing on its own layer.

See also

- ETRANSMIT
- Images and PDF Underlay

Images and PDF Underlays (IMAGE, PDFATTACH)

Insert pictures and logos (IMAGE, INSERTOBJ) and a PDF as a background (PDFATTACH), with clipping and fading.

What it does

Command	Alias	Syntax	Meaning
IMAGE	`IM` , `IMAGEATTACH`	`IMAGE <Attach [file]|List|Detach>`	Attaches a PNG/JPG/BMP picture. The picture is stored inside the drawing, so it is not lost when you send the file elsewhere.
INSERTOBJ	`IMAGEATT`	–	Inserts a picture or logo from a file (same as Picture / Logo on the Ribbon).

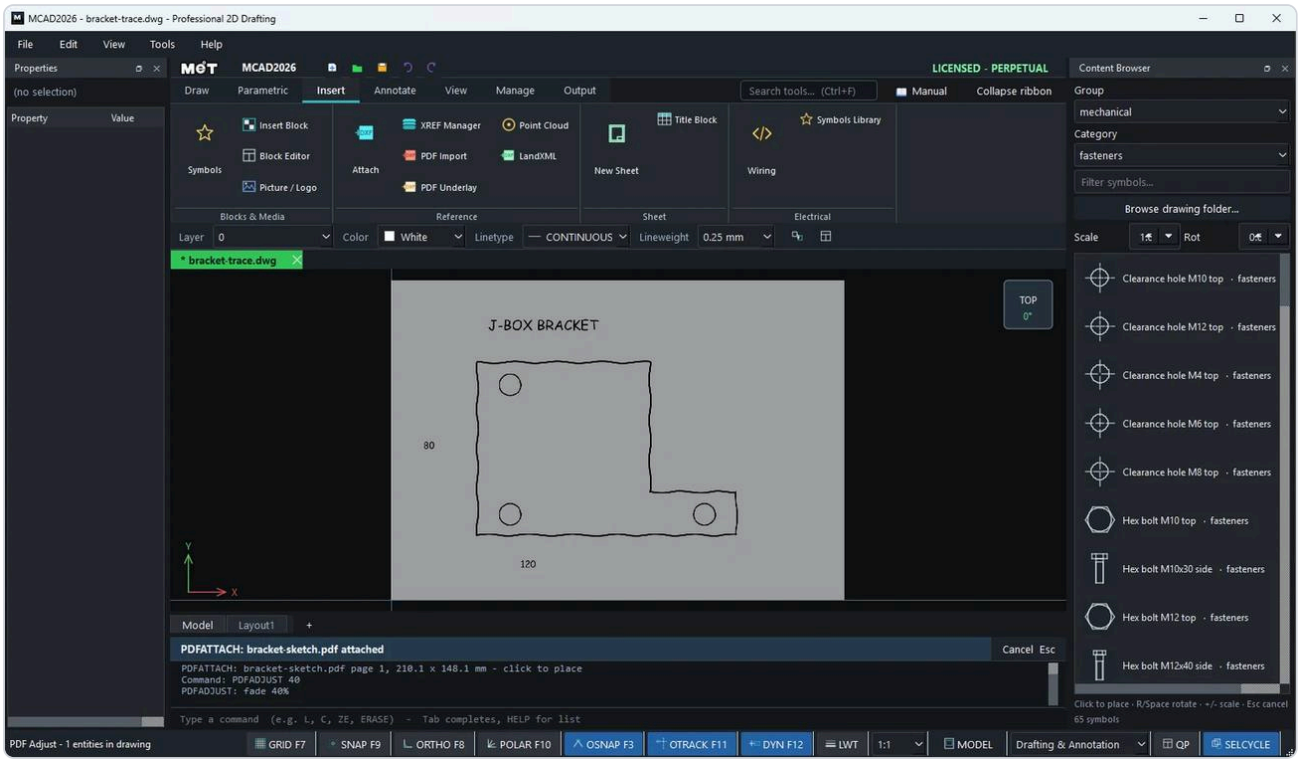
Command	Alias	Syntax	Meaning
PDFATTACH	`PDFUNDERLAY`	`PDFATTACH [file] [page]`	Places the chosen PDF page as a background image. Then click the position.
PDFCLIP	–	`PDFCLIP [Delete]`	Select a PDF underlay and a closed shape to clip it so it shows only inside the boundary.
PDFADJUST	–	`PDFADJUST <0..100>`	The fade of the PDF underlay (%).

Where to find it

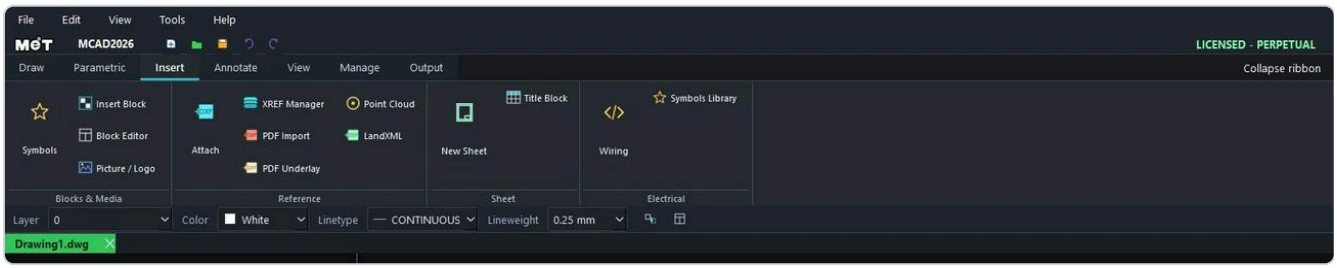
- Ribbon: **Insert › Blocks & Media › Picture / Logo**, **Insert › Reference › PDF Underlay**, **Draw › Block ▾ › Picture / Logo**
- Right-click a picture › Picture Properties...

Steps for a PDF underlay

1. Click **Insert › Reference › PDF Underlay** (or type `PDFATTACH`) and choose the PDF file.
2. The prompt bar shows the page size (mm). Click the position to place it.
3. `PDFADJUST 60` fades it so you can draw over it.
4. Check a distance with `DIST`. If the scale is wrong, use `SCALEREF`.



PDF underlay faded to 40% (PDFADJUST 40)



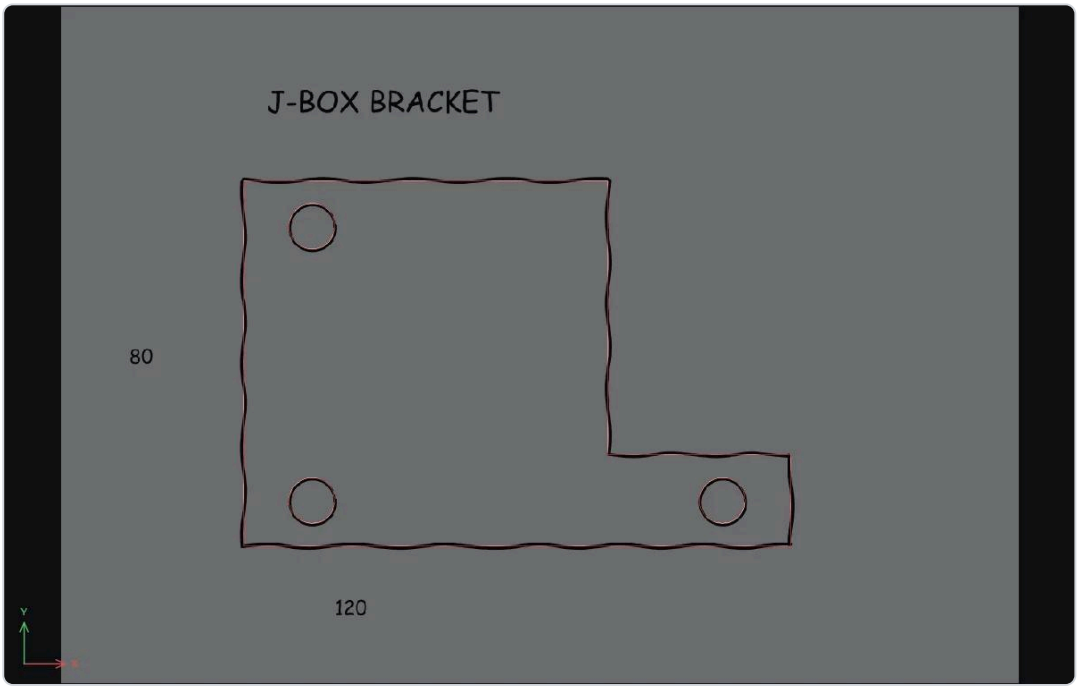
Insert tab, Reference group, with PDF Import and PDF Underlay

> **Note:** If you want editable lines from a vector PDF, use `PDFIMPORT` instead. See Import.

Try it: Trace a junction-box bracket from a PDF sketch

An electrician on site sends a scanned A5 sketch of an L-shaped junction-box bracket, 120 x 80 with three holes. Put the PDF under your drawing and trace it into real geometry. Any PDF sketch of your own works the same way.

1. **Insert › Reference › PDF Underlay** (or `PDFATTACH`), choose the PDF and click `0,0`. The prompt bar shows the page size, here `210.1 x 148.1 mm`.
2. `PDFADJUST 60` fades the sketch so your lines stand out.
3. Check the scale: `DIST` along the bottom edge of the bracket should read about 120, the size written on the sketch. If it does not, fix it with `SCALEREF`.
4. Trace the outline with **Polyline - Closed Loop**: `40,30` `160,30` `160,50` `120,50` `120,110` `40,110`, Enter.
5. Holes: `C` radius `5` at `55,40`, `145,40` and `55,100`.
6. Give the traced objects a strong colour such as red (right-click › Properties) so you can see them over the sketch.



Faded PDF sketch of the bracket with the traced outline and holes in red

Challenge: When the tracing is done, draw a rectangle round the bracket, select it with the underlay and run `PDFCLIP` so only that part of the page stays visible.

See also

- Import DXF, PDF, Point Cloud, LandXML
- SCALEREF

Chapter 7 quiz: Block & Symbol

Choose the one best answer. Answers are in Appendix A

1. **If you create a BLOCK without giving a base point, which point is used as the base point?**

   A. The lower-left corner
   B. The centre of the objects
   C. The drawing origin 0,0
   D. The upper-right corner

2. **What happens when you type `INSERT` without a block name?**

   A. The last block is inserted again
   B. An error message appears
   C. The Content Browser opens
   D. A new block is created

3. **Before plotting a general arrangement drawing, which command hides all attribute values (such as part numbers)?**

   A. ATTDEF
   B. ATTDISP OFF
   C. ATTEDIT
   D. LAYOFF

4. **What does XBIND do to an External Reference?**

   A. Reloads it from the source file
   B. Hides it
   C. Clips it to a closed shape
   D. Copies its objects into the drawing and breaks the link

5. **Which command shows only the part of an Xref that lies inside a selected closed shape?**

   A. XCLIP
   B. XATTACH
   C. XBIND
   D. VPCLIP

6. **While placing a symbol from the Content Browser, which key rotates it?**

   A. Tab
   B. + or -
   C. R or Space
   D. Ctrl+R

CHAPTER 8

Electrical & Wiring

1. Electrical Symbols - Placing electrical symbols
   Place electrical symbols (switches, relays, fuses, motors and more) from the Electrical library into a circuit drawing.
2. Wiring Wizard (WIRING) - PLC IO as A4 Pages
   Automatically generate A4 electrical circuit drawings from a PLC tag list (TIA Portal) or from MET templates.
3. Terminal and Wire Numbering (TCOUNT / Re-tag)
   Number terminals and wires with TCOUNT, and fix addresses/wire numbers on copied drawing pages.
4. SCRIPT - Python Macro API
   Write and run Python macros with the SCRIPT command through the mcad object for automation.

Electrical Symbols - Placing electrical symbols

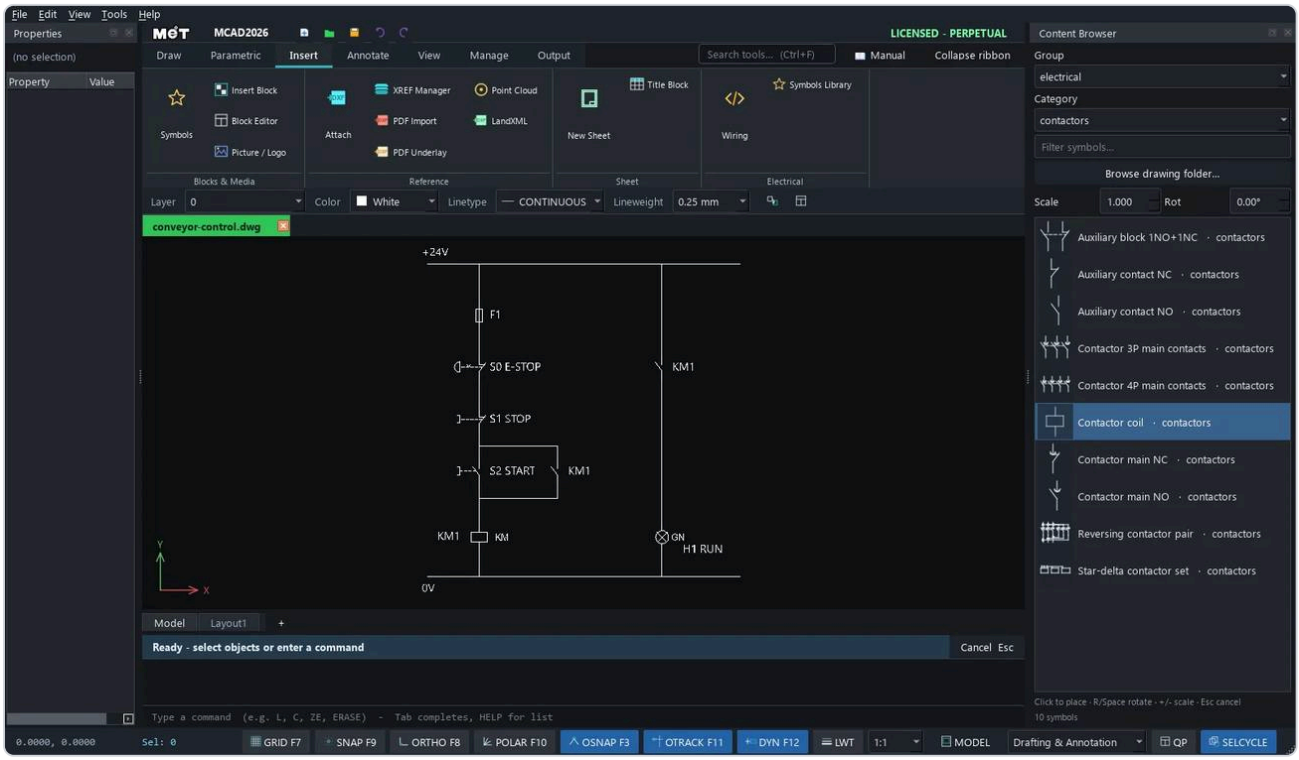
Place electrical symbols (switches, relays, fuses, motors and more) from the Electrical library into a circuit drawing.

What it does

MCAD 2026 includes 240 electrical symbols drawn by MET to IEC 60617: contacts, push buttons, breakers, contactors, relays, motors, drives, sensors, PLC modules, terminals and more. Use them to draw control circuits, power circuits and panel layouts. Each symbol is drawn vertically on a 2.5 mm grid, and its insertion point is pin 1 (the top pin).

Where to find it

- Ribbon: **Insert › Electrical › Symbols Library**
- **Content Browser** palette › Group = `electrical`



Content Browser, electrical group, contactors category, next to a conveyor control circuit

Categories in the electrical group

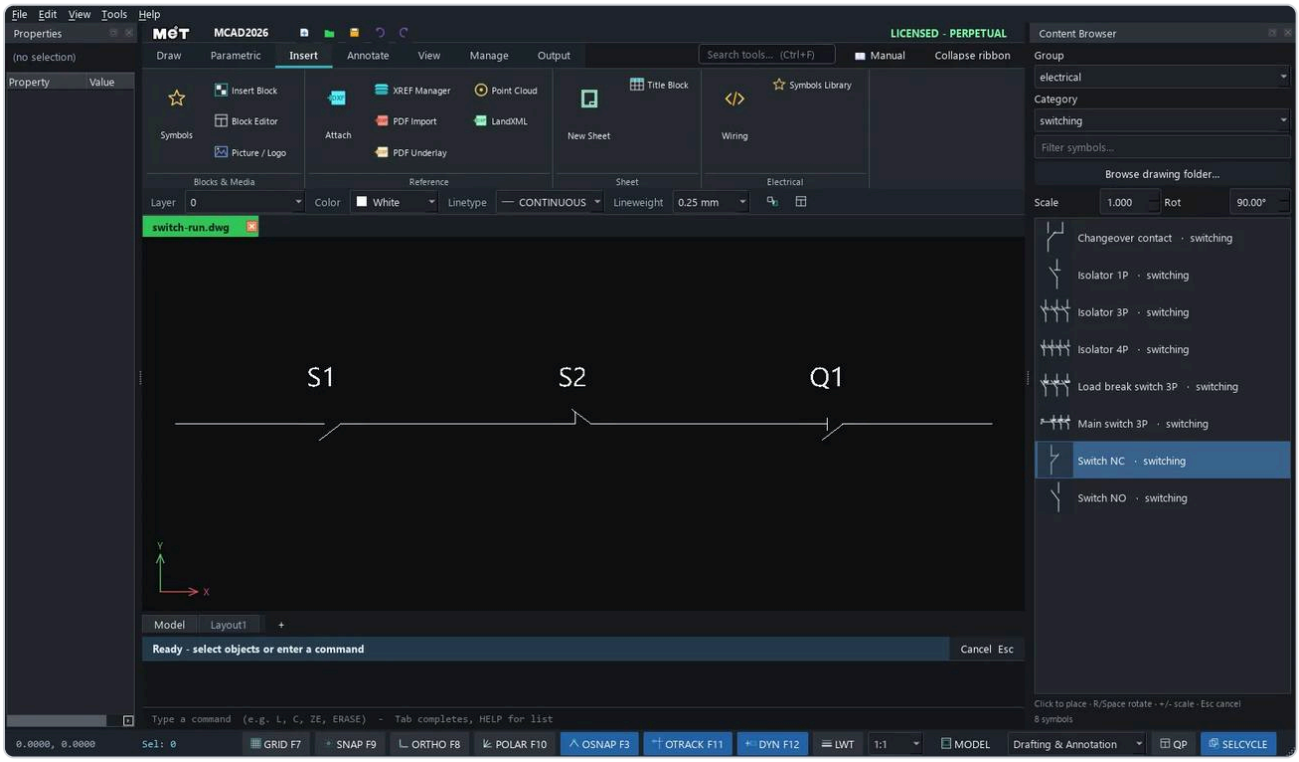
actuators, contactors, drives, earthing, indication, motors, outlets, passive, plc, power supply, protection, push buttons, relays & timers, sensors, switching, terminals, wiring (17 categories)

The same Content Browser also has the `pneumatic` group (ISO 1219 valves, cylinders, air treatment and vacuum) for the pneumatic side of a machine.

Steps

1. Open **Insert › Electrical › Symbols Library**.
2. Set Group = `electrical` and Category = `switching` (or type a name in the Filter box, for example `push button`).
3. Click a symbol, then click a position on the drawing. Press `R` to rotate it 90° before you place it.
4. Turn on grid snap (`SNAP 5`) to keep the terminals in line with the wires.

5. Draw the wires with `LINE` or `PLINE` (turn on Ortho with `F8`).



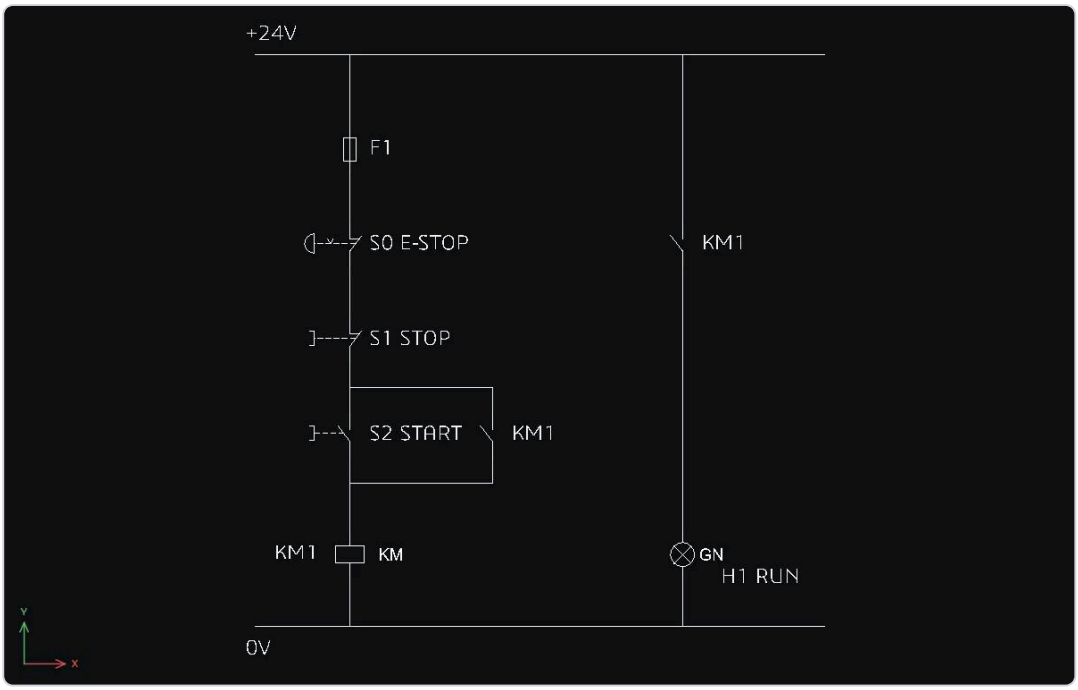
Switch NO, Switch NC and Isolator 1P placed on a horizontal wire with Rot 90

> **Note:** To make your own symbol, draw it and run `BLOCK <name>`. Add a tag with `ATTDEF`, or keep the DXF file in a folder and use **Browse drawing folder...**.

Try it: Conveyor start/stop circuit with E-stop

The classic control circuit of a belt conveyor: fuse, E-stop, stop and start buttons, a contactor coil that holds itself in, and a green RUN lamp. It is drawn the IEC way, with vertical current paths between a +24V rail at the top and a 0V rail at the bottom, so every symbol goes in upright with **Rot** `0`.

1. Type `SNAP 5` and press `F8` for Ortho.
2. Rails: `L` `0,120` → `120,120` (+24V) and `L` `0,0` → `120,0` (0V).
3. Current path 1 at X 20, from the top: Category `protection` › **Fuse** at `20,105`; Category `push buttons` › **Emergency stop** at `20,85`, **Push button NC** at `20,65` (Stop) and **Push button NO** at `20,45` (Start); Category `contactors` › **Contactor coil** at `20,20`. Each symbol is 10 long, so its bottom pin is 10 below the point you click.
4. Hold-in contact: Category `contactors` › **Auxiliary contact NO** at `50,45`. Wire it in parallel with the Start button: `L` `20,50` → `50,50` → `50,45` and `L` `50,35` → `50,30` → `20,30`.
5. Current path 2 at X 90: **Auxiliary contact NO** at `90,85` (the KM1 contact) and Category `indication` › **Pilot lamp GN** at `90,20`.
6. Join the symbols with `L`, snapping to the pins: `20,120` → `20,105`, `20,95` → `20,85`, `20,75` → `20,65`, `20,55` → `20,45`, `20,35` → `20,20`, `20,10` → `20,0`, and `90,120` → `90,85`, `90,75` → `90,20`, `90,10` → `90,0`.
7. Labels with `DT`, Text height `3`: `F1`, `S0 E-STOP`, `S1 STOP` and `S2 START` to the right of each symbol, `KM1` at the coil and at both contacts, `H1 RUN` under the lamp, and `+24V` and `0V` on the rails. The coil and the lamp already show `KM` and `GN` inside the symbol.



Conveyor control circuit: F1, S0 E-stop, S1 stop, S2 start, KM1 coil with hold-in contact, and H1 run lamp

Challenge: Lengthen the rails and add a third current path at X 120 with Category `protection` › **Overload NO 97-98** and Category `indication` › **Pilot lamp RD** as a red fault lamp. Number the wires afterwards with TCOUNT.

See also

- Symbol Library
- Wiring Wizard
- Terminal and wire numbering

Wiring Wizard (WIRING) - PLC IO as A4 Pages

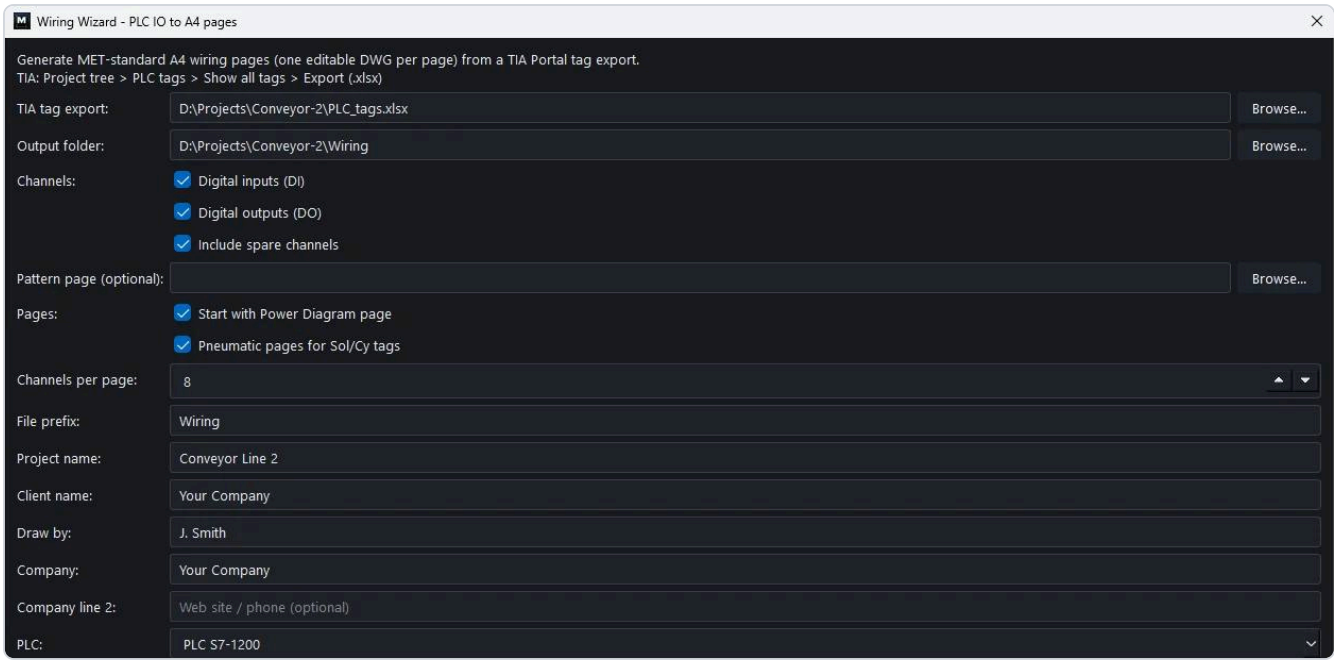
Automatically generate A4 electrical circuit drawings from a PLC tag list (TIA Portal) or from MET templates.

What it does

Wiring Wizard reads a tag file exported from Siemens TIA Portal (`.xlsx`) and creates IO circuit drawings on A4 pages to the MET standard. Each page is an editable DWG file, and a `.mprj` project file opens all the pages at once.

Where to find it

- Ribbon: **Insert › Electrical › Wiring**
- Command line: `WIRING` (Alias `WW`)



Wiring Wizard window

Prepare the file from TIA Portal

In TIA Portal: **Project tree › PLC tags › Show all tags › Export**. Save it as `.xlsx`.

Steps

1. Click **Insert › Electrical › Wiring**.
2. **TIA tag export:** Click Browse... and choose the `.xlsx` file. The summary line below shows the number of tags and the number of DI/DO channels.
3. **Output folder:** Choose the folder where the DWG files are saved.
4. Choose the channel options and extra pages.
5. Fill in File prefix, Project name, Client name, Draw by, Company (and Company line 2 if you want), and PLC (this data goes into the title block).
6. (Optional) Tick the MET template pages in the list below, in the order you want. Click a name to preview it.
7. Click **Generate**. Every page opens as a tab. The Status bar shows `Wiring Wizard: N page(s) -> <prefix>.mprj`.

Options

Option	Meaning
TIA tag export	The tag `.xlsx` file from TIA Portal
Output folder	Where the output files are saved
Digital inputs (DI) / Digital outputs (DO)	The channel types to generate
Include spare channels	Include Spare channels
Pattern page (optional)	A finished drawing page used as the pattern (the program changes the %I/%Q addresses and wire numbers)

Option	Meaning
Start with Power Diagram page	Start with a Power Diagram page
Pneumatic pages for Sol/Cy tags	Generate Pneumatic pages for solenoid/cylinder tags
Channels per page	Number of channels per page (4–12, default 8)
File prefix	Prefix of the file names (default `Wiring`)
Project name / Client name / Draw by	Title block information
Company / Company line 2	Your company name and a second line (web site or phone) for the title block. The program remembers them for next time
PLC	PLC S7-1200, S7-1500, S200, or type your own
MET template pages	Template pages (Blower, Ground Wiring, HMI and Network diagram, INPUT & OUTPUT ...). Tick them to use these pages instead of the automatic ones

> **Note:** If you do not choose a tag file, you must tick at least one template page. Otherwise you see the warning `Select a TIA tag export .xlsx file, or tick template pages to clone.`

Try it: IO pages for a box conveyor

Ten PLC tags for a small box conveyor (buttons, E-stop relay, two photo-eyes, overload, motor, lamps and horn) turn into two ready-to-edit A4 IO pages in one click. You do not need TIA Portal for this: a hand-made Excel sheet with the same columns works.

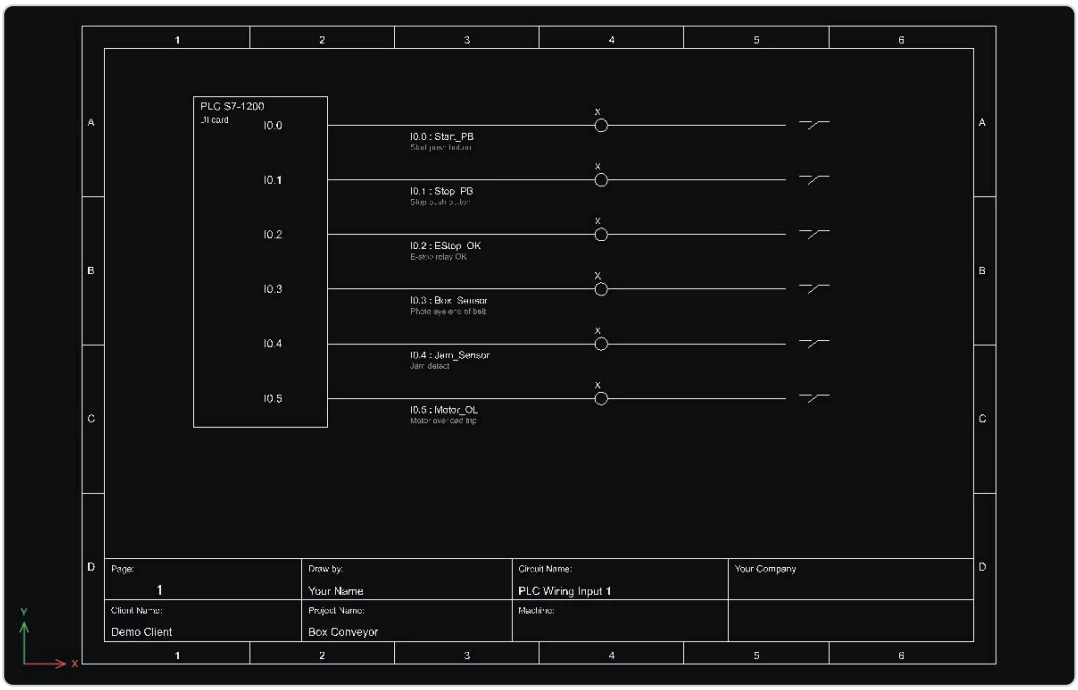
1. In Excel, make a sheet with the columns `Name`, `Path`, `Data Type`, `Logical Address`, `Comment` and these rows (Path `Default tag table`, Data Type `Bool`):

Name	Logical Address	Comment
Start_PB	%I0.0	Start push button
Stop_PB	%I0.1	Stop push button
EStop_OK	%I0.2	E-stop relay OK
Box_Sensor	%I0.3	Photo eye end of belt
Jam_Sensor	%I0.4	Jam detect
Motor_OL	%I0.5	Motor overload trip
Motor_Run	%Q0.0	Conveyor motor contactor
Lamp_Run	%Q0.1	Green lamp running
Lamp_Fault	%Q0.2	Red lamp fault
Horn	%Q0.3	Warning horn

Save it as `conveyor_tags.xlsx`.

2. **Insert › Electrical › Wiring**, then **Browse...** for `conveyor_tags.xlsx`.

3. Keep **Digital inputs** and **Digital outputs** ticked. Untick **Start with Power Diagram page** and **Pneumatic pages**. Channels per page `8`.
4. File prefix `Conveyor`, Project name `Box Conveyor`, Client name `Demo Client`, Draw by your name, Company `Your Company`, PLC `PLC S7-1200`. Leave the template pages unticked.
5. Click **Generate**. Two pages open as tabs, Conveyor_P01 with the six inputs and Conveyor_P02 with the four outputs, plus `Conveyor.mprj` that opens both.



Generated A4 input page: I0.0 to I0.5 with tag names and comments, title block filled in

Challenge: Add four more inputs (%I0.6 to %I1.1) to the sheet and generate again: the inputs now fill page 1 and roll on to a second input page.

See also

- SAVEPROJECT - Project (.mprj)
- Terminal and Wire Numbering
- PUBLISH

Terminal and Wire Numbering (TCOUNT / Re-tag)

Number terminals and wires with TCOUNT, and fix addresses/wire numbers on copied drawing pages.

What it does

Electrical drawings contain many terminal and wire numbers in sequence. `TCOUNT` adds sequence numbers to all the selected text in one step.

Where to find it

- Ribbon: **Annotate › Text & Tables › Number Text**
- Command line: `TCOUNT` (Alias `TC`)

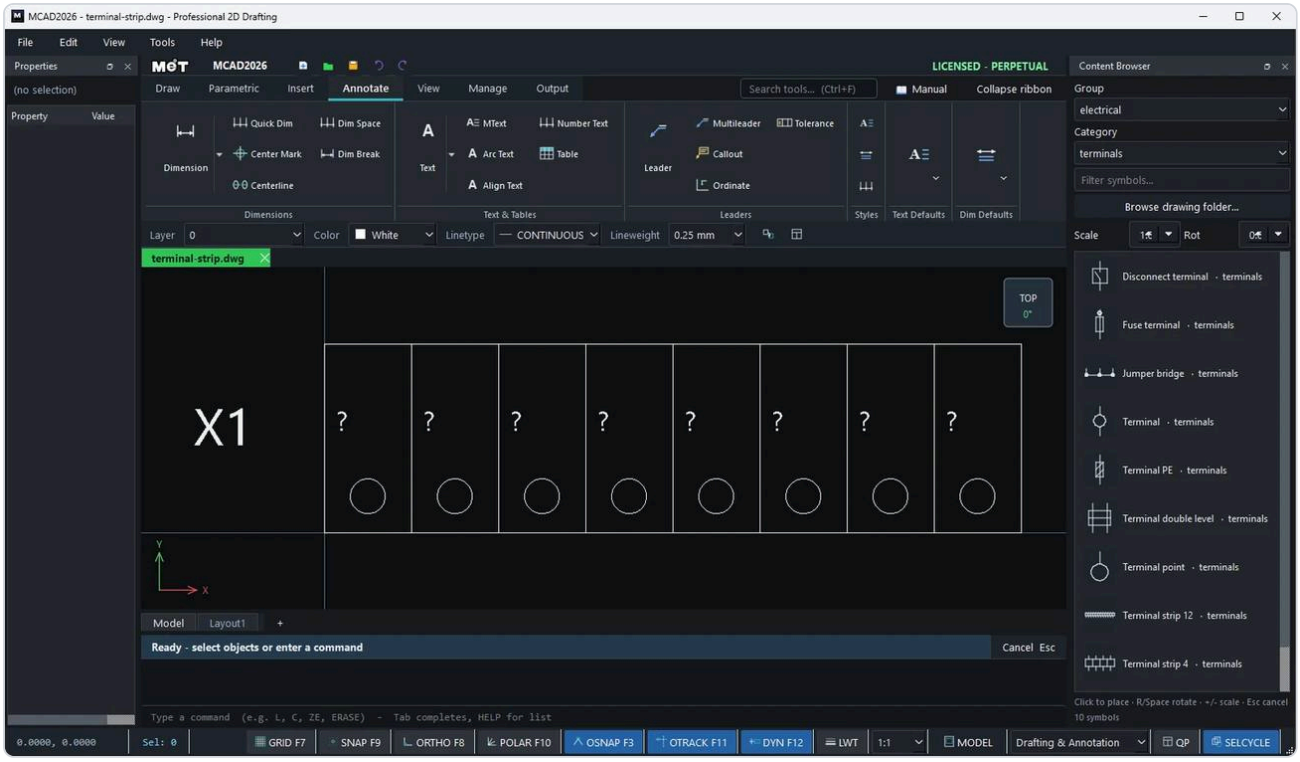
Format

`TCOUNT [start] [step] [p=prefix] [s=suffix] [pick] [over]`

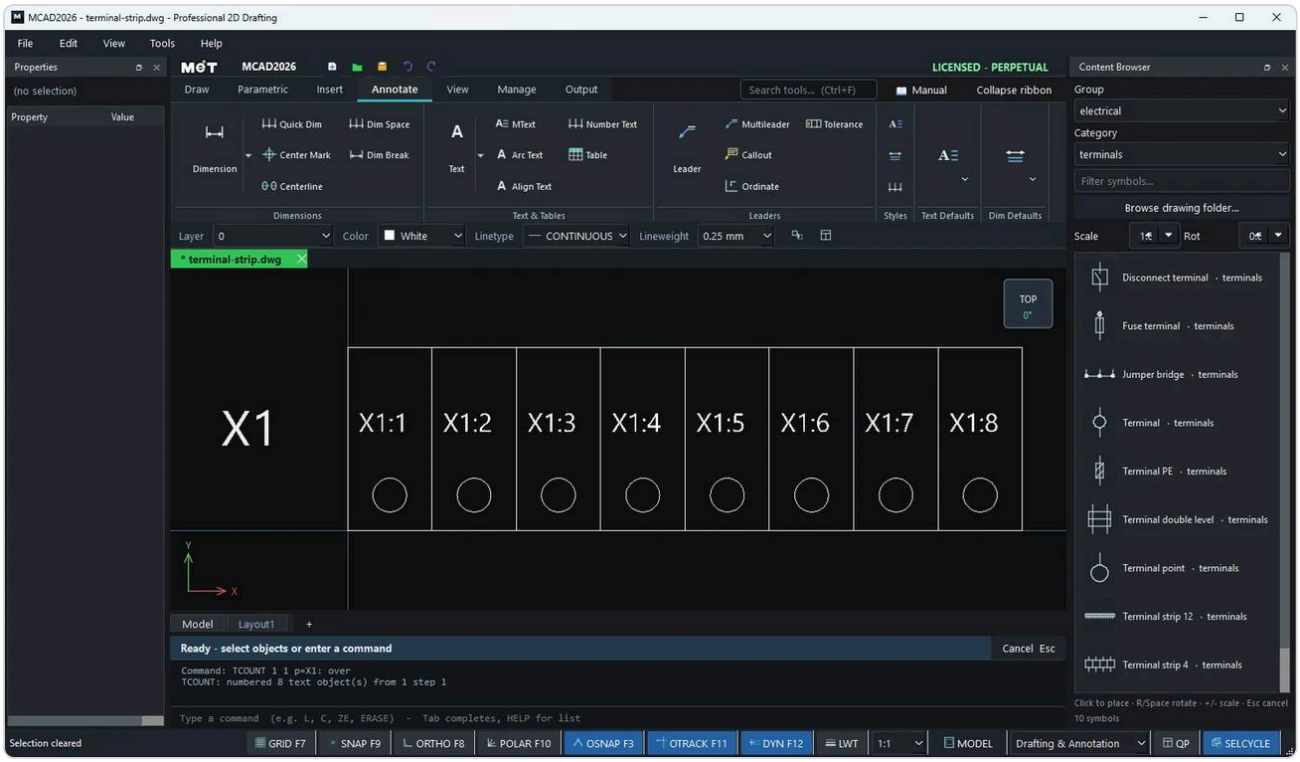
Option	Meaning
start	First number (default 1)
step	Increment (default 1)
`p=`	Prefix, for example `p=X1:`
`s=`	Suffix
`pick`	Number in the order you click (the default is by position, top to bottom, left to right)
`over`	Replace all of the existing text (without it, the number is added in front of the existing text)

Steps: number terminals X1:1 to X1:8

1. Draw one terminal with the text `?`, then ARRAY it to make the full set.
2. Select all of the text (for example `QSELECT text` and then click to remove the ones you do not want, or `FIND ?`).
3. Type `TCOUNT 1 1 p=X1: over` and press Enter.



Terminal strip before numbering



After TCOUNT: X1:1 to X1:8

Wire numbers by row

- Wire numbers 101, 102, ...: `TCOUNT 101 1 over`
- Terminal numbers in steps of 2: `TCOUNT 1 2 p=X2:`

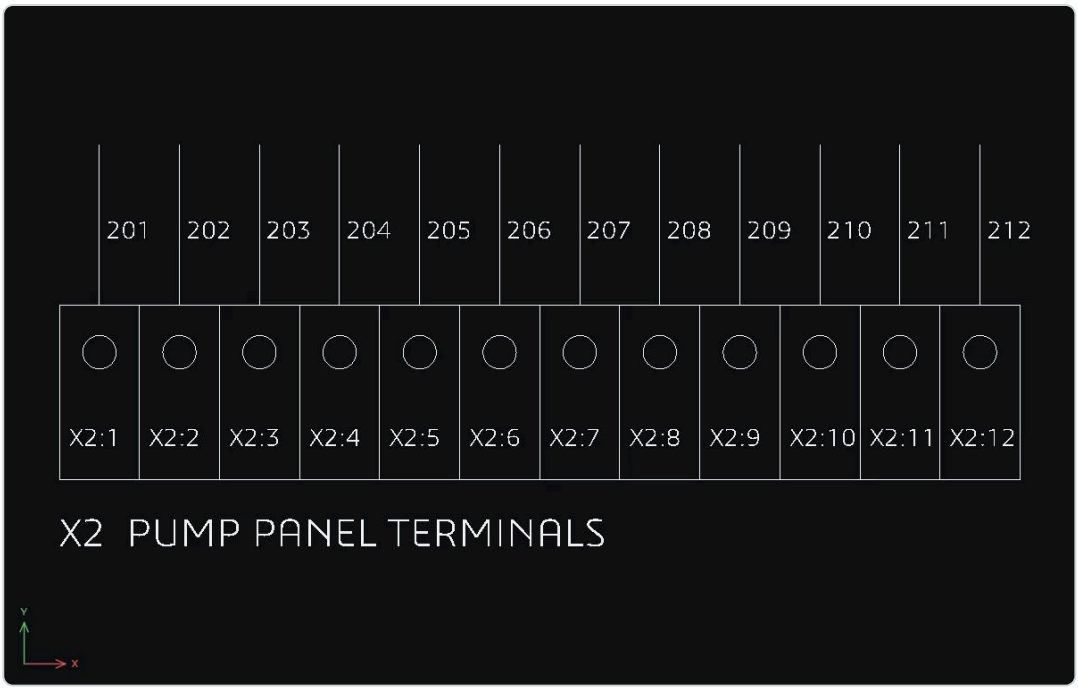
Fix addresses and wire numbers on copied pages

When you copy a finished IO drawing page to make the next group, Wiring Wizard (the **Pattern page** field) automatically changes the addresses `%I0.0` → `%I8.0` and the 3-digit wire numbers (for example 200 → 300) in the text. For special cases, use FIND to replace word by word, for example `FIND %I0. %I8.`

Try it: X2 terminal strip for a pump panel

Twelve terminals on a DIN rail with the wires coming in from above. Draw one terminal, array it, then two TCOUNT runs write every terminal number and wire number in order.

1. One terminal: `REC 0,0` → `12 Tab 26`, screw `C 6,19` radius `2.5`, wire `L 6,26` → `6,50`.
2. Placeholders with `DT`, Text height `2.5`: a `?` at about `1.5,5` (terminal number) and a `?` at about `7,36` (wire number).
3. Select the five objects and type `ARRAYRECT 12 12 0`. You now have 12 terminals.
4. Drag a window from left to right around the bottom row of `?` only (about `-2,2` to `146,10`) and type `TCOUNT 1 1 p=X2: over`. They become X2:1 to X2:12.
5. Window the upper row of `?` (about `-2,33` to `146,41`) and type `TCOUNT 201 1 over`. The wires are now 201 to 212.
6. Title: `DT` with height `4`, `X2 PUMP PANEL TERMINALS` at about `0,-10`.



Terminal strip X2:1 to X2:12 with wire numbers 201 to 212

Challenge: Make it a double-level strip: copy the row up by 60 and number it with `TCOUNT 13 1 p=X2: over` and `TCOUNT 213 1 over`.

See also

- Text Tools
- Wiring Wizard
- SCRIPT - Python Macro

SCRIPT - Python Macro API

Write and run Python macros with the SCRIPT command through the mcad object for automation.

What it does

SCRIPT runs a Python (`.py`) file inside MCAD. It is comparable to AutoLISP/VBA. Use it to build repeated drawings, number items, or edit many drawings automatically. Every change a macro makes can be undone with `Ctrl+Z`.

Where to find it

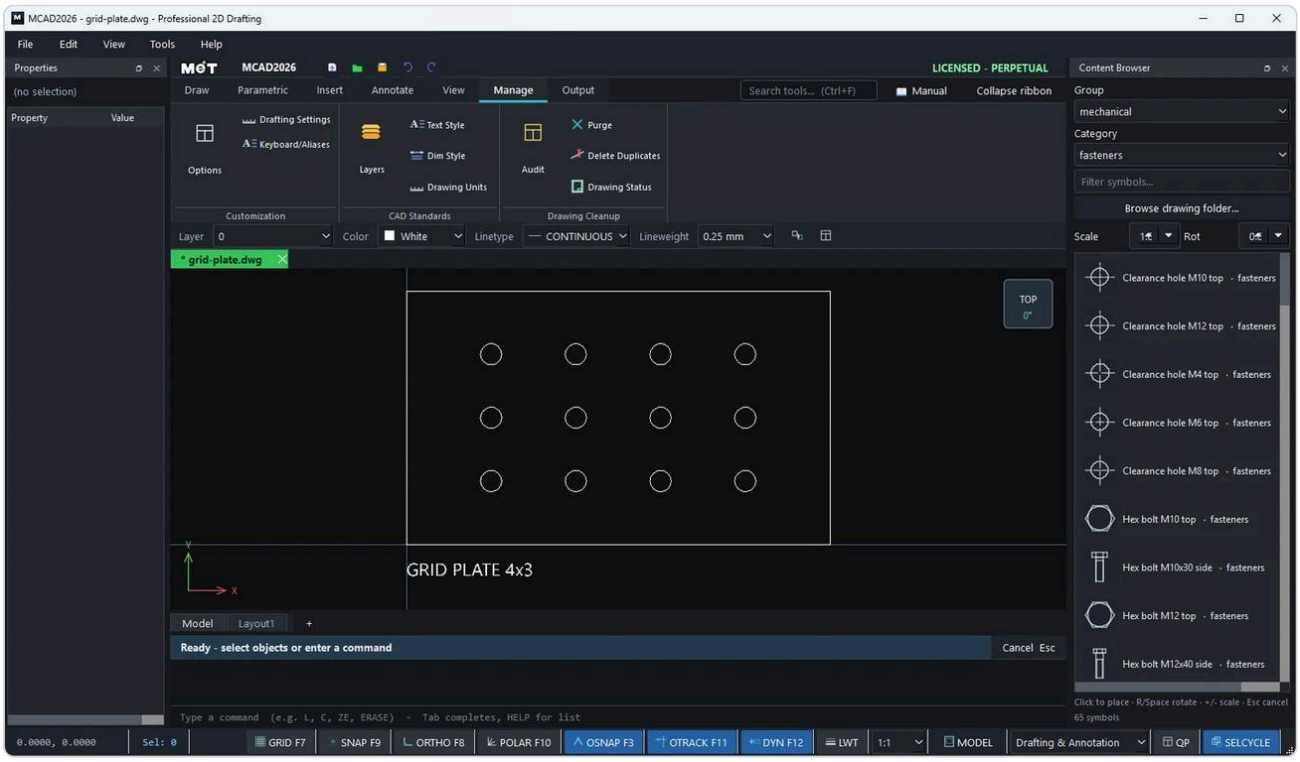
- Ribbon: type the command
- Command line: `SCRIPT` (Alias `SCR`, `MACRO`)

Steps

1. Create a file `grid_holes.py` with Notepad:

```
# MCAD macro demo: a plate with a 4 x 3 hole grid
mcad.add_rect(0, 0, 200, 120, layer="OUTLINE")
for i in range(4):
    for j in range(3):
        mcad.add_circle(40 + i * 40, 30 + j * 30, 5, layer="HOLES")
mcad.add_text(0, -15, "GRID PLATE 4x3", height=6)
mcad.zoom_fit()
mcad.log(f"{mcad.entity_count} entities")
```

2. Type `SCRIPT` and press Enter, then choose the file (or `SCRIPT C:\Macros\grid_holes.py`).
3. The result appears in the drawing, and the Status bar shows `SCRIPT done: grid_holes.py`.



Macro result: a plate with a 4 x 3 hole grid

API reference

Document

Command	Meaning
`mcad.entity_count`	Number of objects
`mcad.entities()`	List of objects (dicts)
`mcad.entity(i)`	Object number i
`mcad.open(path)`	Open a DWG/DXF file
`mcad.save()`	Save the current drawing

Create objects (returns the index of the new object)

Command	Meaning
`mcad.add_line(x1, y1, x2, y2, **style)`	Line

Command	Meaning
`mcad.add_circle(cx, cy, r, **style)`	Circle
`mcad.add_arc(cx, cy, r, start_deg, end_deg, **style)`	Arc
`mcad.add_rect(x, y, w, h, **style)`	Rectangle
`mcad.add_polyline([(x, y), ...], closed=False, **style)`	Polyline
`mcad.add_text(x, y, "text", height=2.5, **style)`	Text
`mcad.add_entities([dict, ...])`	Add several objects in one Undo step (fast for many objects)

`**style`: `layer="..."`, `color="#RRGGBB"` (or an ACI number), `linetype="DASHED"`, `lineweight=25` (= 0.25 mm)

Edit and other commands

Command	Meaning
`mcad.move(indices, dx, dy)`	Move
`mcad.erase(indices)`	Erase
`mcad.select(indices)` / `mcad.selection()` / `mcad.clear_selection()`	Manage the selection
`mcad.zoom_fit()` / `mcad.redraw()`	Zoom to fit / redraw
`mcad.command("ZOOM E")`	Run any Command line command
`mcad.log("msg")`	Show a message on the Status bar
`mcad.math`	Python's `math` module

> **Note:** If a macro fails, a window shows the Traceback and the program does not close. Editing an object's dict directly bypasses the Undo system, so use the `mcad` commands instead.

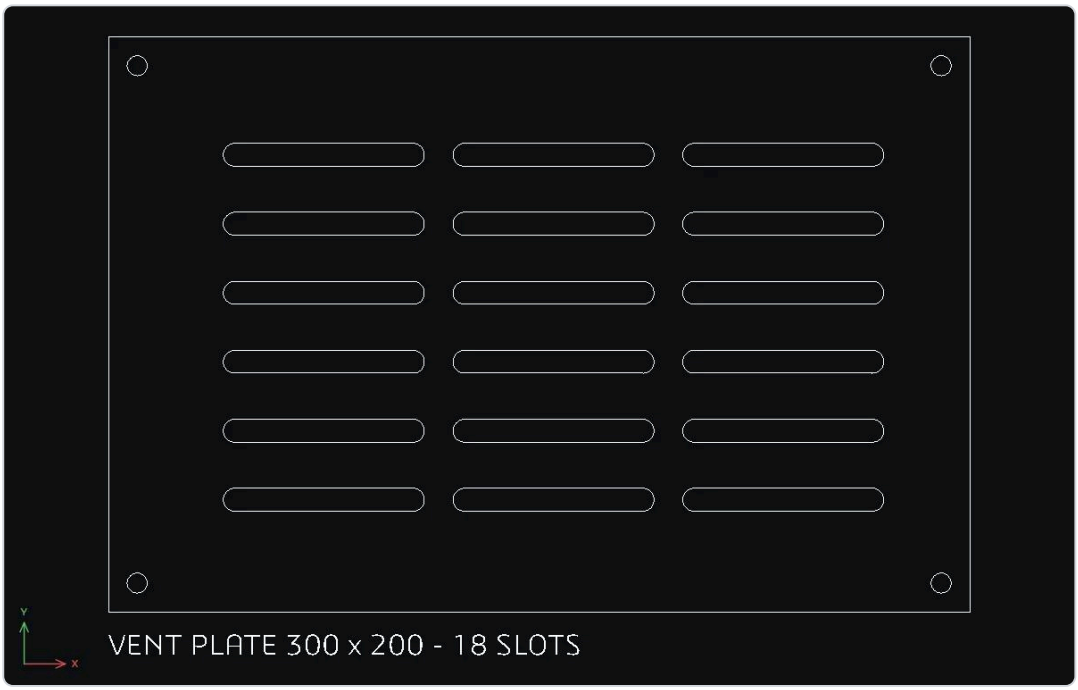
Try it: Vent plate for an enclosure door

A 300 x 200 louvre plate for a hot control cabinet: 18 rounded slots in three columns plus four fixing holes. Drawing 72 lines and arcs by hand is no fun; a 20-line macro does it in a blink, and changing one number redraws the whole pattern.

1. Save this as `vent_plate.py`:

```
# MCAD macro: enclosure door vent plate with 3 x 6 louvre slots
W, H = 300, 200
mcad.add_rect(0, 0, W, H, layer="OUTLINE")
for (x, y) in ((10, 10), (W - 10, 10), (10, H - 10), (W - 10, H - 10)):
    mcad.add_circle(x, y, 3.5, layer="HOLES")
slot_w, slot_h, r = 70, 8, 4
for col in range(3):
    for row in range(6):
        x0 = 40 + col * 80
        y0 = 35 + row * 24
        mcad.add_line(x0 + r, y0, x0 + slot_w - r, y0, layer="SLOTS")
        mcad.add_line(x0 + r, y0 + slot_h, x0 + slot_w - r, y0 + slot_h, layer="SLOTS")
        mcad.add_arc(x0 + r, y0 + r, r, 90, 270, layer="SLOTS")
        mcad.add_arc(x0 + slot_w - r, y0 + r, r, -90, 90, layer="SLOTS")
mcad.add_text(0, -15, "VENT PLATE 300 x 200 - 18 SLOTS", height=7)
mcad.zoom_fit()
mcad.log(f"{mcad.entity_count} entities")
```

2. Type `SCRIPT` and choose `vent_plate.py`.
3. The plate appears and the Status bar reports the run. The drawing now holds 78 objects: the plate, 4 holes, 72 slot lines and arcs and the title.



Vent plate with 18 louvre slots generated by the macro

Challenge: Change `range(6)` to `range(5)` and `slot_w` to `90`, erase the old plate and run the script again.

See also

- Command Index
- Terminal Numbering

Chapter 8 quiz: Electrical & Wiring

Choose the one best answer. Answers are in Appendix A

1. **Which input file does Wiring Wizard read to generate the IO circuit pages?**

   A. A DXF file of the panel layout
   B. A CSV of wire numbers
   C. A tag export (.xlsx) from Siemens TIA Portal
   D. A .mprj project file

2. **What is the default value of Channels per page in Wiring Wizard?**

   A. 4
   B. 8
   C. 12
   D. 16

3. **In TCOUNT, what does the `over` option do?**

   A. Replaces all of the existing text with the number
   B. Numbers in the order you click
   C. Adds a suffix
   D. Starts counting from 0

4. **In `TCOUNT 1 1 p=X1: over`, what does `p=X1:` set?**

   A. The step
   B. The start number
   C. The suffix
   D. The prefix

5. **What kind of file does the SCRIPT command run?**

   A. An AutoLISP (.lsp) file
   B. A Python (.py) file
   C. A VBA macro
   D. A batch (.bat) file

6. **In the electrical symbol library, how can you recognise IEC-style categories?**

   A. Their names start with `iec_`
   B. They are in a separate Group called `iec`
   C. Their names end in `-iso`
   D. They are shown in blue

CHAPTER 9

Print & Export

1. PLOT and PREVIEW - Print and PDF
   Print to a printer or save as PDF. Choose the area, paper size, scale, color, and Plot style (CTB).
2. PUBLISH - Combine All Layouts into One PDF
   Combine Model and all Layouts into a single multi-page PDF file.
3. Layouts and viewports (LAYOUT, MVIEW, PAGESETUP)
   Set up the plot sheet with layouts, add viewports, switch between model and paper space, set the paper size and per-viewport layers.
4. TITLEBLOCK and MET Drawing Sheet
   Insert the standard MET title block into a Layout and create new drawings from MET A4–A0 sheets.
5. SHEETSET - Sheet Set Manager
   Combine layouts from several files into a Sheet Set and print them as one PDF.
6. EXPORT - Exporting files
   Export the drawing to a DXF file from the Output tab, and the limits of the EXPORT command.
7. GCODE - Generate G-code for CNC and laser
   Generate G-code for CNC, laser, plasma or plotter from the lines in a drawing, and simulate the cut before you save.
8. NEST - Arrange Parts on a Sheet
   Automatically arrange parts on raw material sheets (nesting) for laser, plasma, and wire-cut jobs.

PLOT and PREVIEW - Print and PDF

Print to a printer or save as PDF. Choose the area, paper size, scale, color, and Plot style (CTB).

What it does

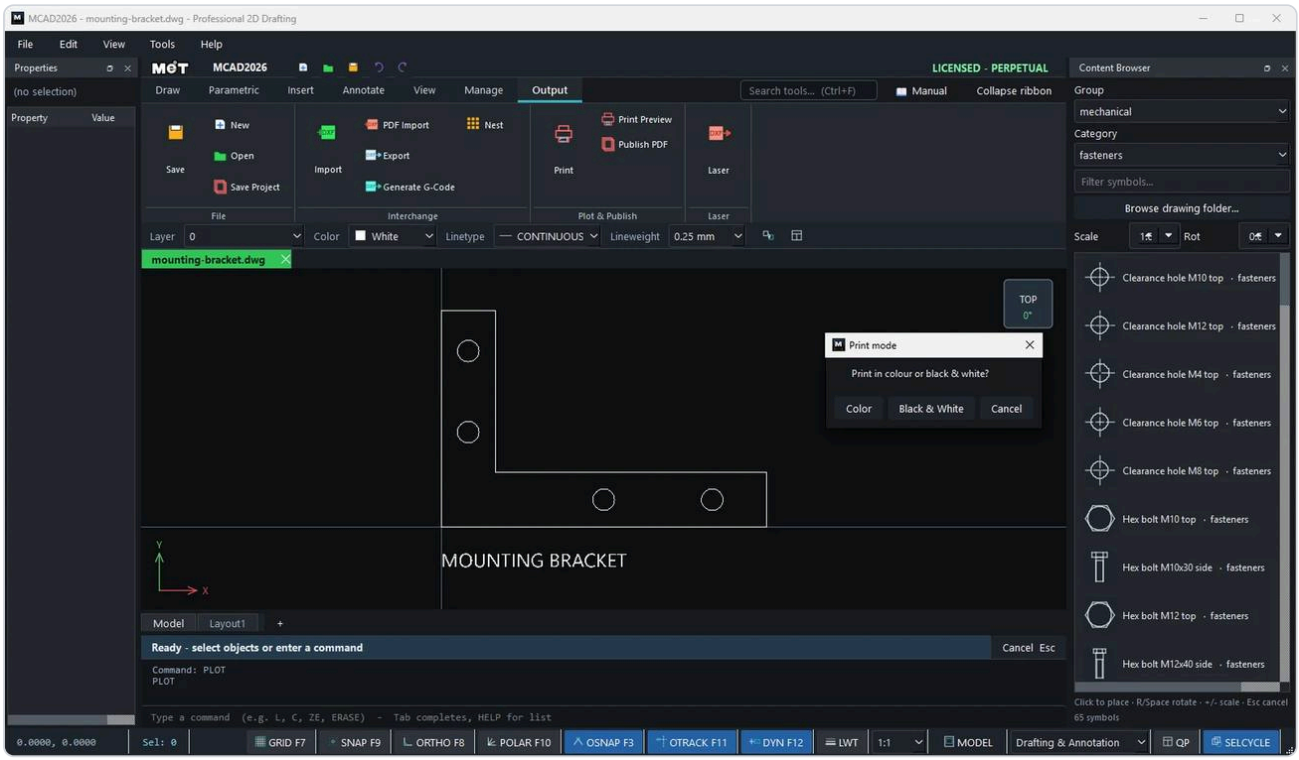
- **PLOT** – prints the selected area to a Windows printer.
- **PREVIEW** – shows a preview before printing.
- **Print Window (PDF)...** (right-click menu › View) – the Plot / PDF window with all options. It saves a vector PDF.

Where to find it

- Ribbon: **Output › Plot & Publish › Print / Print Preview**
- Command line: `PLOT` (Alias `PRINT`)
- Command line: `PREVIEW` (Alias `PRE`)
- Right-click in the drawing area › View › Print Window (PDF)...

PLOT steps

1. Type `PLOT` and press Enter. The **Print mode** window asks `Print in colour or black & white?`
2. Choose **Color** or **Black & White**.
3. `Print area: drag a window, or click 2 corners (snap on)` Drag a window around the area to print, or click 2 corners.
4. The Windows printer window opens. Choose a printer and click Print.

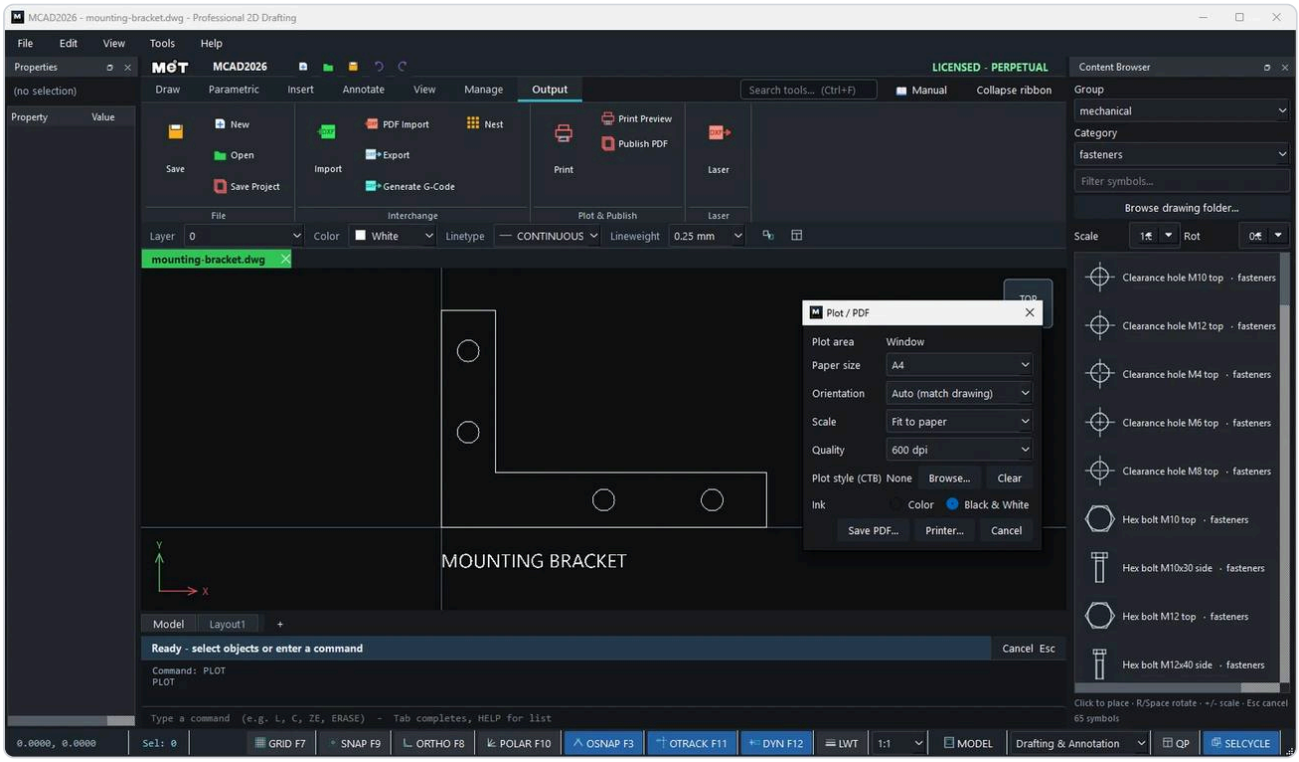


Print mode window

Print Window (PDF) steps

1. Right-click in the drawing area › **View › Print Window (PDF)...**
2. Drag a window around the area.

3. Set the options in the **Plot / PDF** window, then click **Save PDF...** (or **Printer...**).



Plot / PDF window

Option	Meaning
Plot area	The area to print (Window, Current view, Extents)
Paper size	A4, A3, A2, A1, A0, Letter
Orientation	Auto (match drawing), Landscape, Portrait
Scale	Fit to paper, 1:1, 1:2, 1:5, 1:10, 1:20, 1:50, 1:100, 1:200
Quality	300 / 600 / 1200 dpi
Plot style (CTB)	A `.ctb` / `.stb` file that sets colors/line weights when plotting (Browse... / Clear)
Ink	Color or Black & White
Save PDF... / Printer...	Save a vector PDF / send to a printer

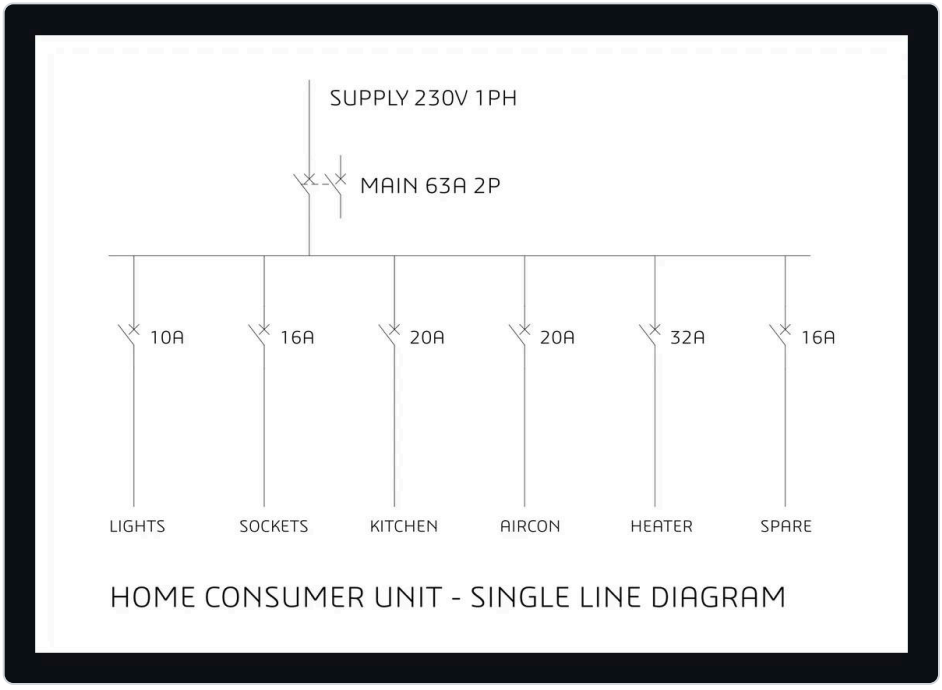
> **Note:** If a Layout tab is open, the plot comes out at the Layout's actual paper size, 1:1. The last settings you used are remembered.

Try it: Home consumer-unit diagram to PDF

The single-line diagram of a home consumer unit: a 63 A main switch feeding six circuit breakers. You plot it to an A4 PDF in black and white, ready to stick inside the cabinet door.

1. Main switch: Content Browser, Group `electrical`, Category `protection`, set **Scale** `2.5`, click **MCB 2P** and click at `100,260`. Its pins are at `100,260` (top) and `100,235` (bottom). Supply line `L` `100,290` → `100,260`, busbar `L` `20,220` → `300,220`, and a link `L` `100,235` → `100,220`.
2. First breaker: **MCB 1P** (still **Scale** `2.5`) at `30,200`, plus `L` `30,220` → `30,200` and `L` `30,175` → `30,120`. Select the breaker and its two lines and type `ARRAYRECT 6 52 0`.

3. Labels with `DT`, Text height `5`: `10A 16A 20A 20A 32A 16A` next to the breakers, and `LIGHTS SOCKETS KITCHEN AIRCON HEATER SPARE` under them (height `4.5`). Add `MAIN 63A 2P`, `SUPPLY 230V 1PH` and a title.
4. Right-click in the drawing area › **View › Print Window (PDF)...** and drag a window around the diagram.
5. In **Plot / PDF**: Paper size `A4`, Orientation `Landscape`, Scale `Fit to paper`, Quality `300 dpi`, Ink `Black & White`. Click **Save PDF...** and name it `consumer_unit.pdf`.



Consumer-unit single-line diagram: main switch, busbar and six labelled breakers

Challenge: Plot the same window at Scale `1:2` on A3 and compare the text size with the fit-to-paper version.

See also

- PUBLISH
- Layouts and Viewports

PUBLISH - Combine All Layouts into One PDF

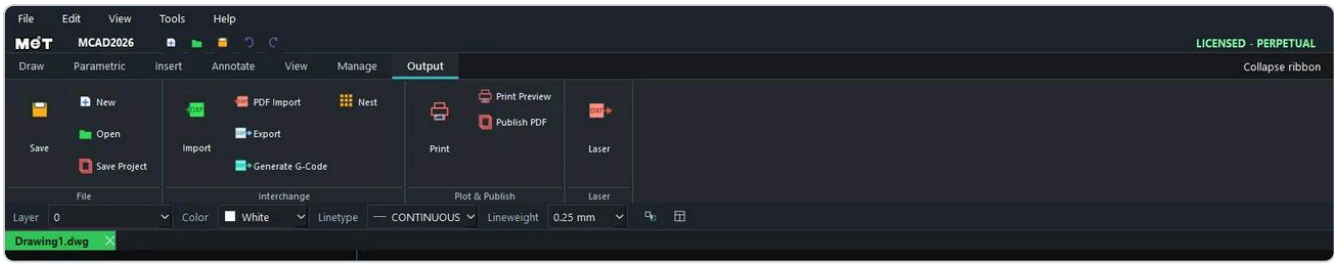
Combine Model and all Layouts into a single multi-page PDF file.

What it does

PUBLISH plots the Model (scaled to fit A4 paper) and every Layout in the drawing (actual paper size, 1:1, black and white) into one PDF file at 600 dpi.

Where to find it

- Ribbon: **Output › Plot & Publish › Publish PDF**
- Command line: `PUBLISH` (Alias `PUB`)



Output tab, Plot & Publish group

Steps

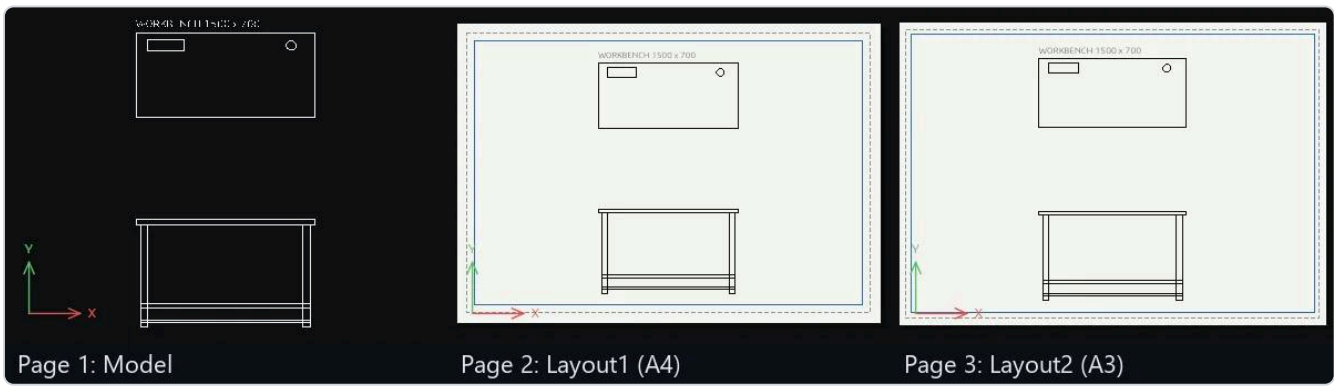
1. Set up all your Layouts (paper size, title block, viewports).
2. Type `PUB` and press Enter.
3. Choose a file name (default `<drawing name>-published.pdf`) and click Save.

> **Note:** To combine several drawing files into one PDF set, use Sheet Set Manager.

Try it: Workbench drawing set in one PDF

A 1500 x 700 workbench with a top view and a front view. Model space plus two layouts go out as one three-page PDF that the workshop can print in one go.

1. Top view: `REC 0,900 → 1500 Tab 700`, vice pad `REC 100,1450 → 300 Tab 100`, grommet hole `C 1300,1500` radius `40`.
2. Front view: worktop `REC 0,0 → 1500 Tab 40`, legs `REC 40,-860 → 60 Tab 860` and `REC 1400,-860 → 60 Tab 860`, rails `REC 40,-700 → 1420 Tab 40` and `REC 40,-820 → 1420 Tab 20`. Title: `DT` height `60`, `WORKBENCH 1500 x 700` at about `0,1650`.
3. Click the **Layout1** tab (A4). A viewport fitted to the bench appears.
4. Type `LAYOUT New Layout2`, click the **Layout2** tab, type `PAGESETUP A3 Landscape` and drag the viewport's corner so it fills the larger sheet.
5. Type `PUB` and save `workbench-published.pdf`. The PDF has three pages: Model fitted to A4, Layout1 on A4 and Layout2 on A3.



The three sheets that go into the PDF: Model, Layout1 (A4) and Layout2 (A3)

Challenge: Put a title block on both layouts before you publish, and fill in a different drawing number on each.

See also

- PLOT

- Layouts and Viewports

Layouts and viewports (LAYOUT, MVIEW, PAGESETUP)

Set up the plot sheet with layouts, add viewports, switch between model and paper space, set the paper size and per-viewport layers.

What it does

A layout is the sheet of paper you plot. It contains viewports that look into model space.

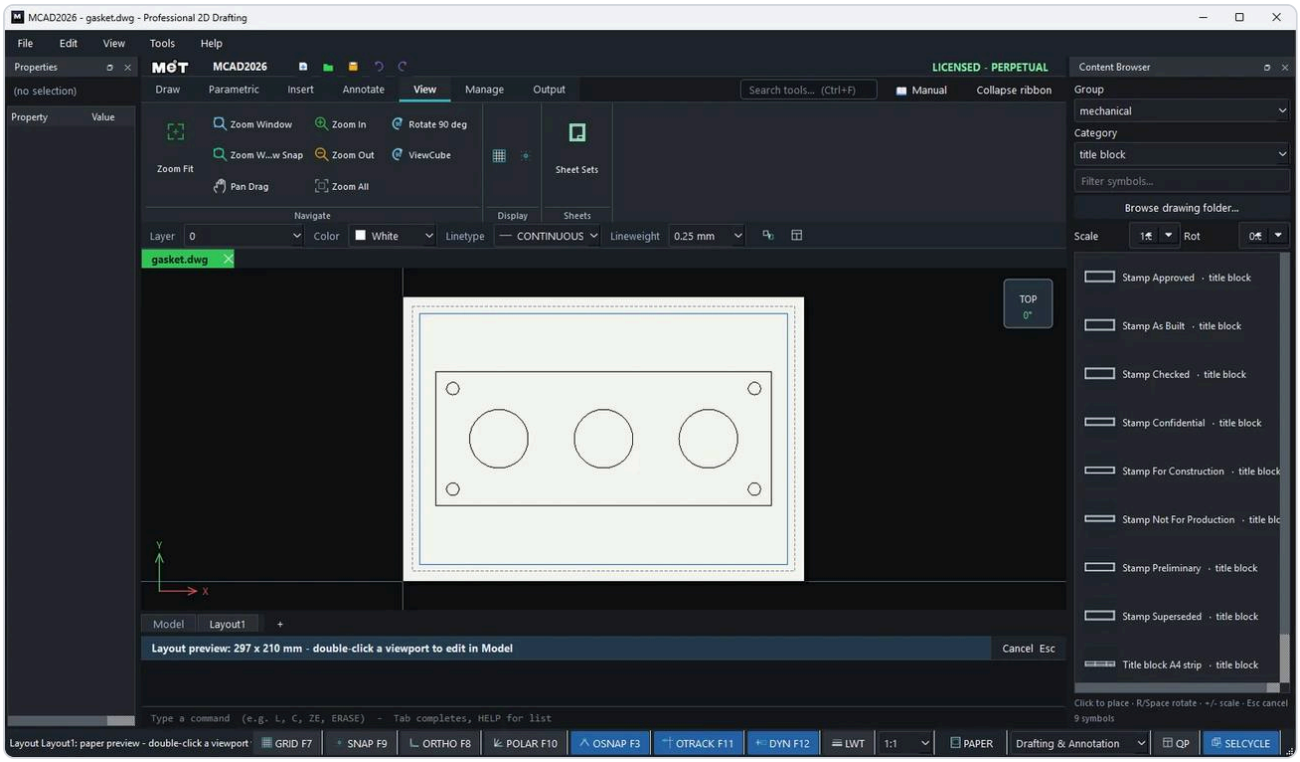
Command	Alias	Syntax	Meaning
LAYOUT	`LO`	`LAYOUT <New|Set|Rename|Delete|List> [name]`	Creates, opens, renames, deletes or lists layouts.
MVIEW	`MV`	`MVIEW [width] [height]`	Adds a viewport (in mm on the paper; the default is 40% of the paper).
PSPACE	`PS`	–	Goes to paper space (the layout).
MSPACE	`MS`	–	Returns to model space.
PAGESETUP	`PAGE`	`PAGESETUP <A0–A4|LETTER|TABLOID|widthxheight> [Portrait|Landscape] [margin]`	The paper size of the layout (`PAGESETUP ?` lists the sizes).
VPLAYER	–	`VPLAYER <Freeze|Thaw|List> <layer> [viewport]`	Hides a layer in one viewport only.
VPCLIP	–	`VPCLIP [Delete] [viewport]`	Clips a viewport to the selected closed shape.

Where to find it

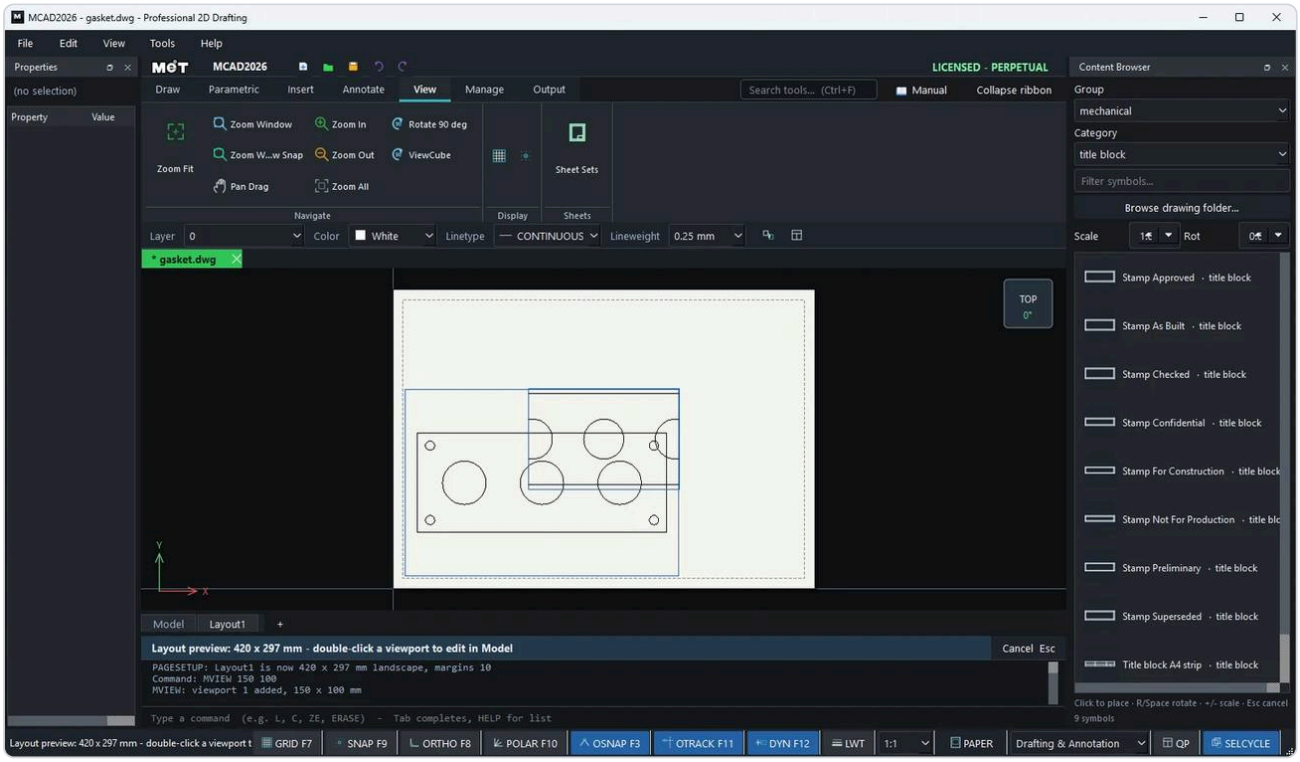
- The **Model / Layout1 / +** tabs below the drawing area (`+` = add a layout)
- `Ctrl+PgDown` / `Ctrl+PgUp` switch between layout tabs

Steps

1. Click the **Layout1** tab. The prompt bar reads `Layout preview: 297 x 210 mm - double-click a viewport to edit in Model` .
2. Set the paper: `PAGESETUP A3 Landscape 10`
3. Add a viewport: `MVIEW 150 100`
4. Double-click the viewport to edit the drawing in model space.
5. Add a title block. See TITLEBLOCK.



A4 layout with a viewport



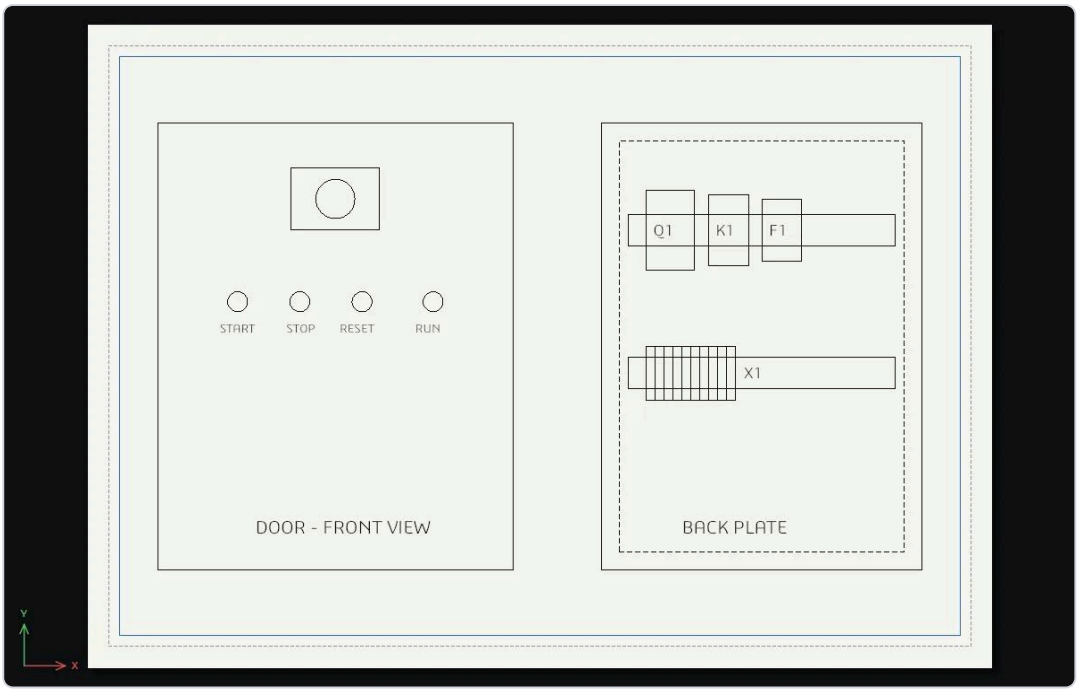
A3 layout after PAGESETUP and MVIEW

Try it: A3 sheet for a motor starter panel

The door and the back plate of a small motor starter enclosure, side by side on one A3 sheet. Setting the paper first means the viewport Layout1 creates already fits it.

1. Door: `REC 0,0` → `400 Tab 500`. Button holes `C 90,300` radius `11.25`, select it and `ARRAYRECT 3 70 0`; lamp hole `C 310,300` radius `11.25`; isolator `REC 150,380` → `100 Tab 70` with `C 200,415` radius `22`.

2. Back plate: `REC 500,0 → 360 Tab 500`, two DIN rails `REC 530,362.5 → 300 Tab 35` and `REC 530,202.5 → 300 Tab 35`, and a box for each device (Q1, K1, F1 and a row of X1 terminals). Label both views with `DT`.
3. While you are still in Model, type `PAGESETUP A3 Landscape 10`.
4. Click the **Layout1** tab. One viewport fills the A3 sheet and shows the door and the back plate together.
5. Double-click inside the viewport to pan or zoom the drawing, then double-click outside it to go back to the sheet.



A3 Layout1 with one viewport showing the starter panel door and back plate

Challenge: Add a close-up of the back plate with `MVIEW 120 80`, then put the title block on with TITLEBLOCK.

See also

- PLOT
- PUBLISH
- Sheet Set

TITLEBLOCK and MET Drawing Sheet

Insert the standard MET title block into a Layout and create new drawings from MET A4–A0 sheets.

What it does

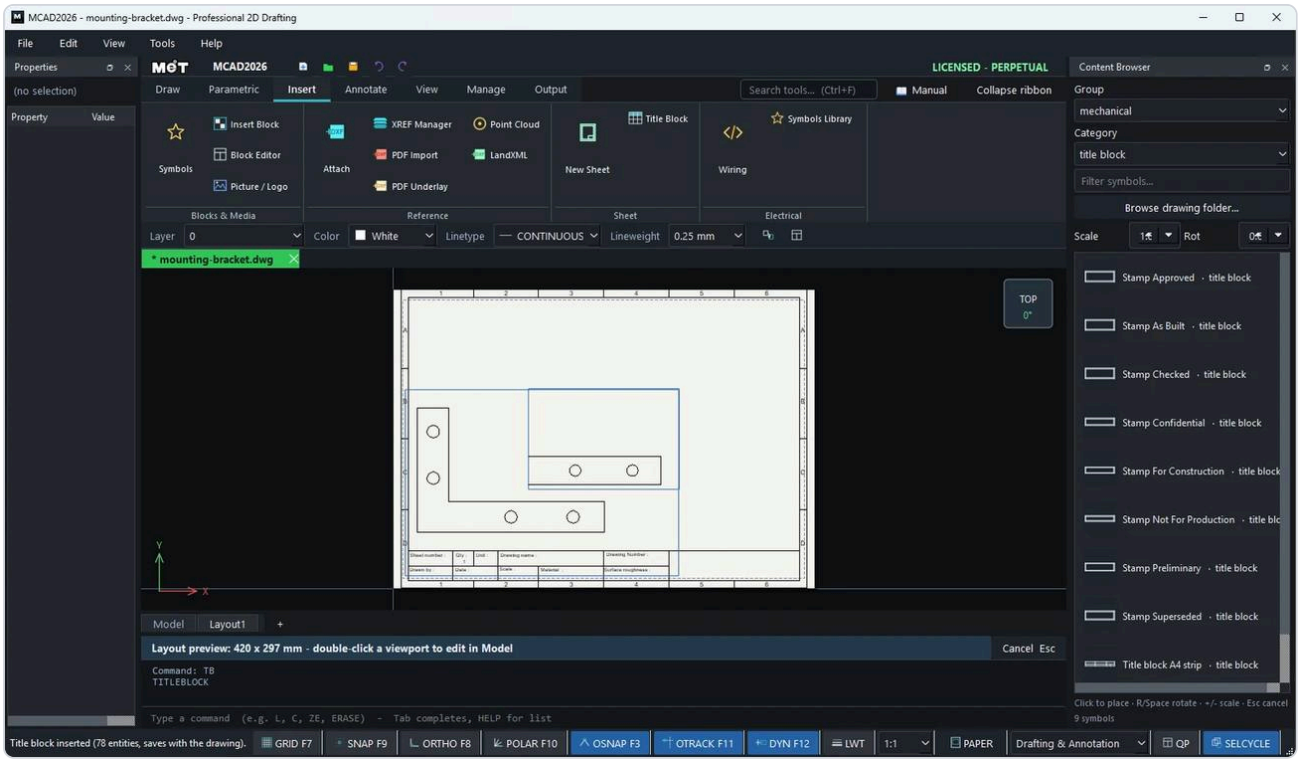
- **TITLEBLOCK** – places the standard MET border and title block on the current Layout. It adjusts to the paper size and is saved with the drawing.
- **New Sheet / MET Drawing Sheet** – creates a new drawing from a MET sheet in A4, A3, A2, A1, or A0 (landscape).

Where to find it

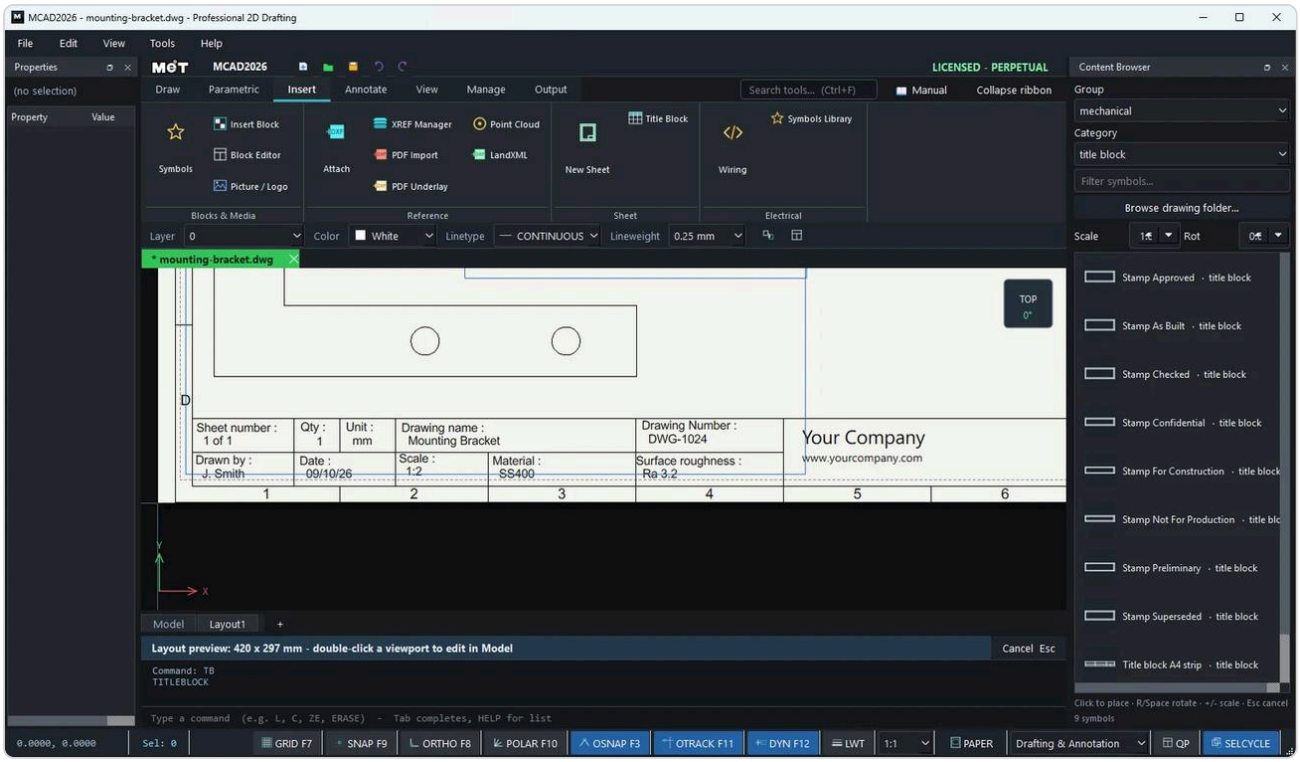
- Ribbon: **Insert › Sheet › Title Block / New Sheet** or **Draw › Block** (submenu Table ▾ › Title Block / New Sheet)
- Command line: `TITLEBLOCK` (Alias `TB`)
- Start Center › **MET Drawing Sheet...**

TITLEBLOCK steps

1. Click the **Layout1** tab. You must be in a Layout, otherwise you see `Title Block: open a Layout tab first.`
2. Type `TB` and press Enter. The Status bar shows `Title block inserted (... entities, saves with the drawing).`
3. Double-click the text in the title block fields to fill in Drawing name, Drawing Number, Material, Drawn by, Date, and Scale.



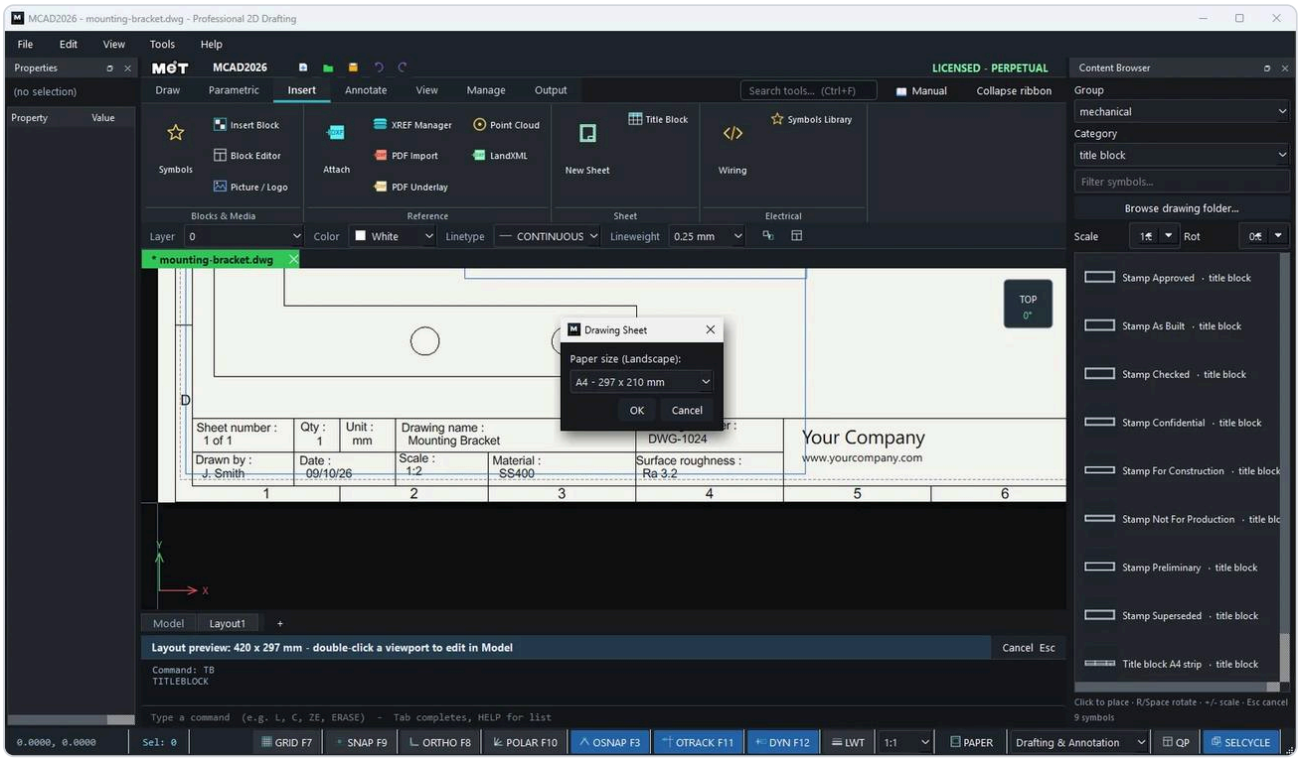
Layout with the title block



Title block with the fields filled in

New Sheet steps

1. Choose **Insert › Sheet › New Sheet**.
2. Select the Paper size (Landscape) and click OK.
3. The new drawing opens in a new tab with the border and title block.



Paper size window

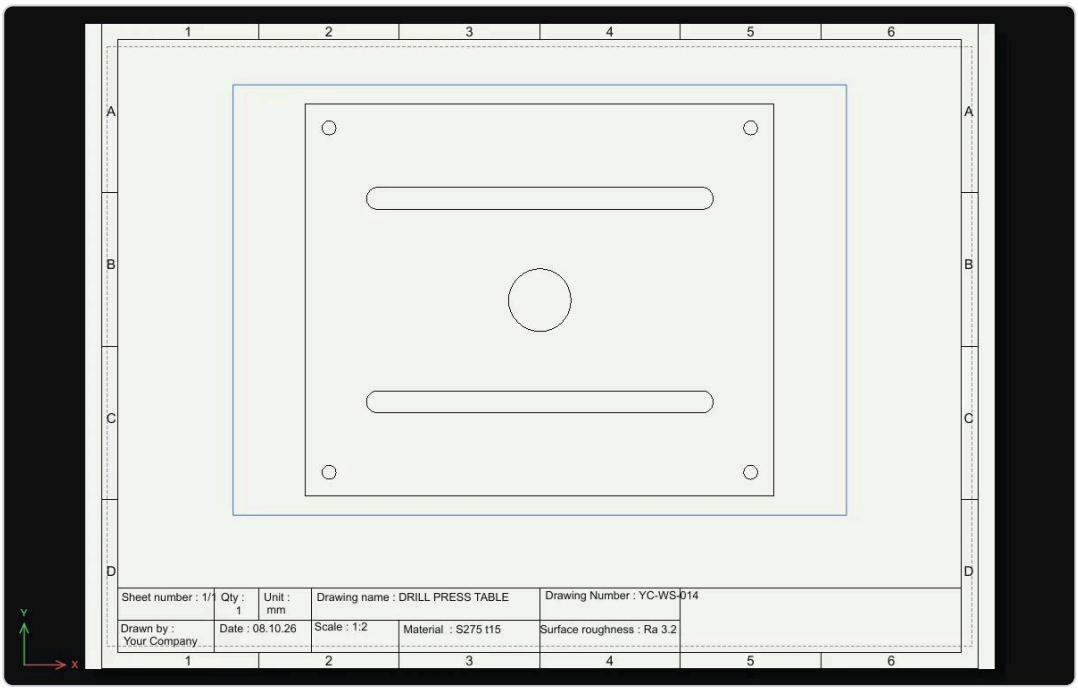
Title block fields

Sheet number, Qty, Unit, Drawing name, Drawing Number, Drawn by, Date, Scale, Material, Surface roughness

Try it: Drill press table drawing, ready to issue

A 300 x 250 steel table for a drill press, with two clamping slots and a Ø40 through hole. Put it on an A4 sheet with the title block and fill in every field, so it is ready to hand to the workshop.

1. Table: `REC 0,0 → 300 Tab 250`, through hole `C 150,125` radius `20`, corner holes `C` radius `4.5` at `15,15`, `285,15`, `15,235` and `285,235`.
2. Clamping slots: `SLOT 46,60 @208,0 @0,7`, then `SLOT 46,190 @208,0 @0,7`.
3. Click the **Layout1** tab (A4 landscape) and type `TB`. The Status bar reports `Title block inserted (... entities, saves with the drawing).`
4. Drag the bottom edge of the viewport up until the table clears the title block.
5. Double-click each field and fill it in: Drawing name `DRILL PRESS TABLE`, Drawing Number `YC-WS-014`, Material `S275 t15`, Drawn by `Your Company`, Date `08.10.26`, Scale `1:2`, Sheet number `1/1`, Unit `mm`, Surface roughness `Ra 3.2`.



A4 layout with the drill press table and a filled-in title block

Challenge: Use FIELD `DATE` and `FILENAME` in the Date and Drawing Number boxes so they update themselves.

See also

- Layouts and Viewports
- FIELD

SHEETSET - Sheet Set Manager

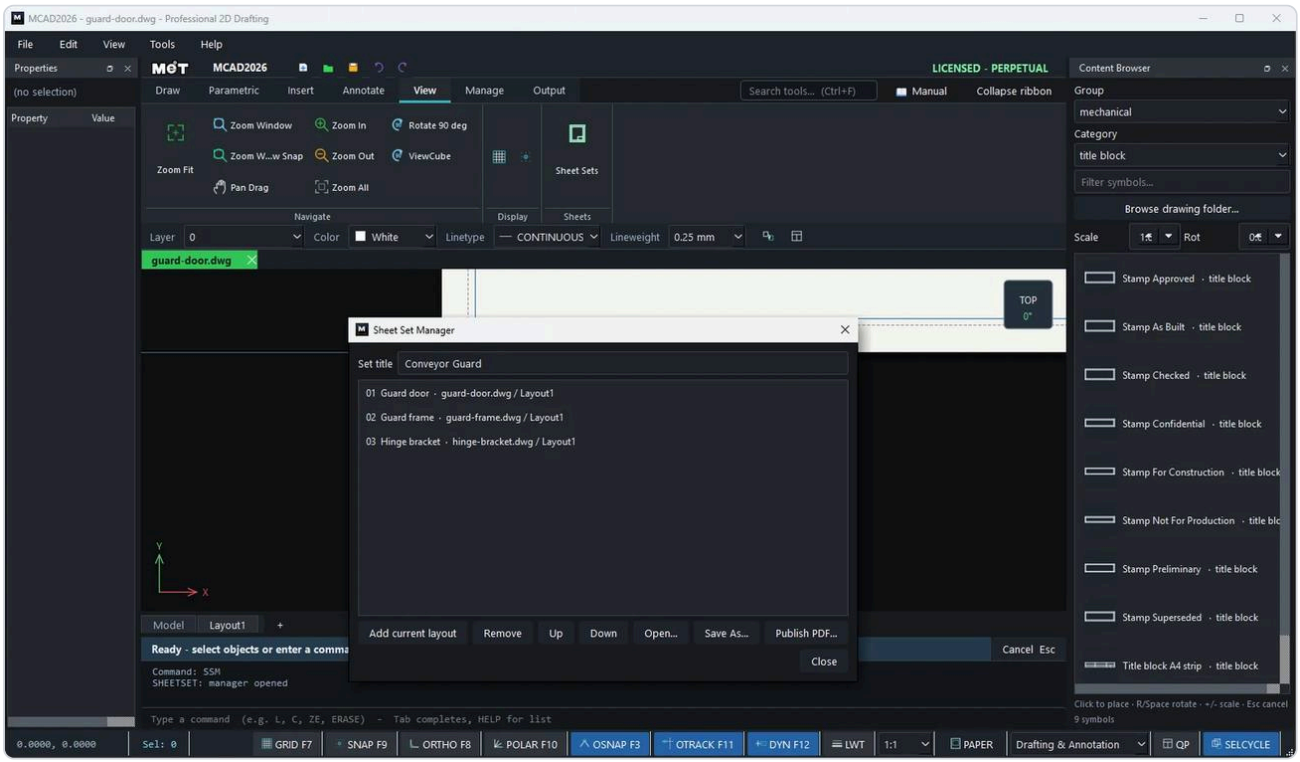
Combine layouts from several files into a Sheet Set and print them as one PDF.

What it does

Sheet Set Manager groups Layouts from multiple drawing files into a set. You can reorder the sheets, save the set as a `.mset` file, and Publish it as a single PDF.

Where to find it

- Ribbon: **View › Sheets › Sheet Sets**
- Command line: `SHEETSET` (Alias `SSM`, `SHEETSETMANAGER`)



Sheet Set Manager

Steps

1. Open each drawing file (it must be saved), then open the Layout you want.
2. In **Sheet Sets**, set the **Set title**, then click **Add current layout**.
3. Repeat with the other files. Reorder the sheets with **Up / Down**.
4. Click **Save As...** to store the set as a `.mset` file.
5. Click **Publish PDF...** to get one PDF containing all the sheets.

Options

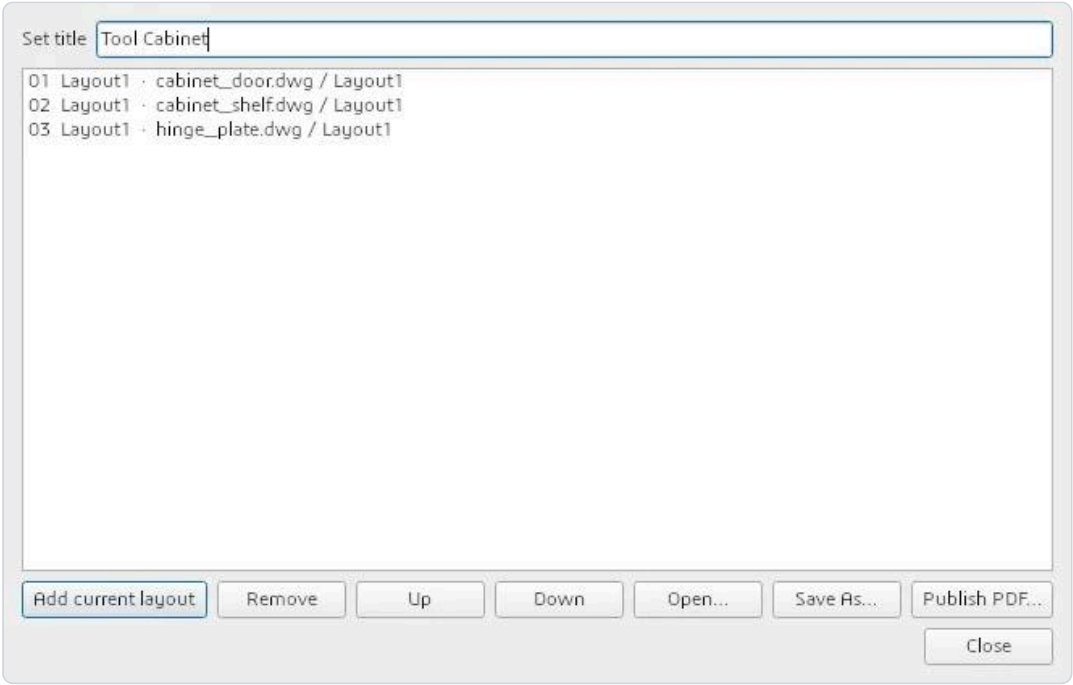
Button / command form	Meaning
Add current layout / `SHEETSET Add [layout]`	Add the current Layout
Remove / `SHEETSET Remove <n>`	Remove a sheet
Up / Down	Change the order
Open... / `SHEETSET Open <file.mset>`	Open a set
Save As... / `SHEETSET Save <file.mset>`	Save the set
Publish PDF... / `SHEETSET Publish [file.pdf]`	Plot the whole set

Button / command form	Meaning
`SHEETSET New [name]` / `SHEETSET List`	Create a new set / list the sheets

Try it: Tool cabinet drawing set

A tool cabinet drawn in three files: the door, a shelf and a hinge plate. A sheet set collects their layouts in the right order and publishes them as one PDF for the workshop.

1. Draw and save the three parts as `cabinet_door.dwg`, `cabinet_shelf.dwg` and `hinge_plate.dwg`. In each file click the **Layout1** tab once so it has a viewport.
2. Type `SHEETSET New Tool Cabinet`.
3. In `cabinet_door.dwg`, type `SHEETSET Add Layout1`. Switch to `cabinet_shelf.dwg` and then `hinge_plate.dwg` and type the same command in each.
4. Type `SHEETSET` on its own to open **Sheet Set Manager**. The three sheets are listed in order; use **Up / Down** if you want the hinge plate first.
5. **Save As…** `tool_cabinet.mset`, then **Publish PDF…** for one PDF with all three sheets.



Sheet Set Manager with the Tool Cabinet set: door, shelf and hinge plate

Challenge: Next week, `SHEETSET Open tool_cabinet.mset` and publish again. Any changes you saved in the three files are in the new PDF.

See also

- PUBLISH
- Layouts

EXPORT - Exporting files

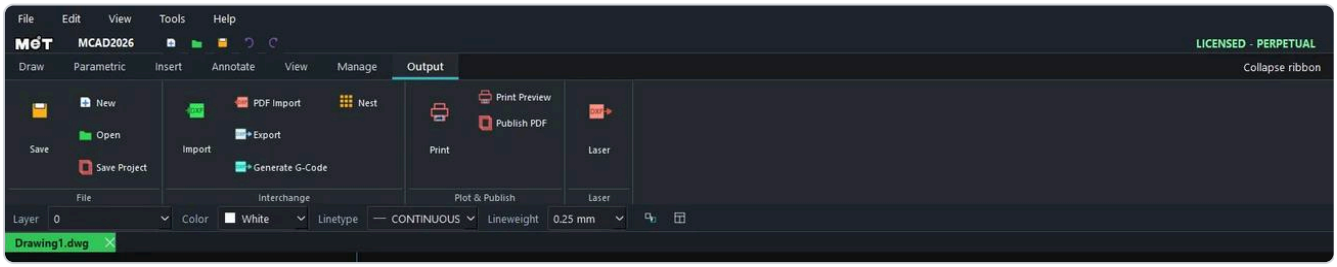
Export the drawing to a DXF file from the Output tab, and the limits of the EXPORT command.

What it does

Exports a 2D drawing as a DXF file (R2018) for other CAD/CAM programs.

Where to find it

- Ribbon: **Output › Interchange › Export** (Export DXF)
- Menu: **File › Export DXF...**
- Command line: `EXPORT` (aliases `EXP`, `DXFOUT`)



Output tab, Interchange group

Steps

1. Click **Output › Interchange › Export**.
2. Name the `.dxf` file and click Save.

> **Note:** In this 2D edition, the typed commands `EXPORT` and `DXFOUT` are for exporting 3D (STEP/IGES/STL) and report `No 3D shape to export.` Use the **Export** button on the Ribbon, or `SAVEAS` with the DXF type, instead. Special objects (Table, Multileader, Tolerance) are exploded into lines and text in the DXF.

Exporting other formats

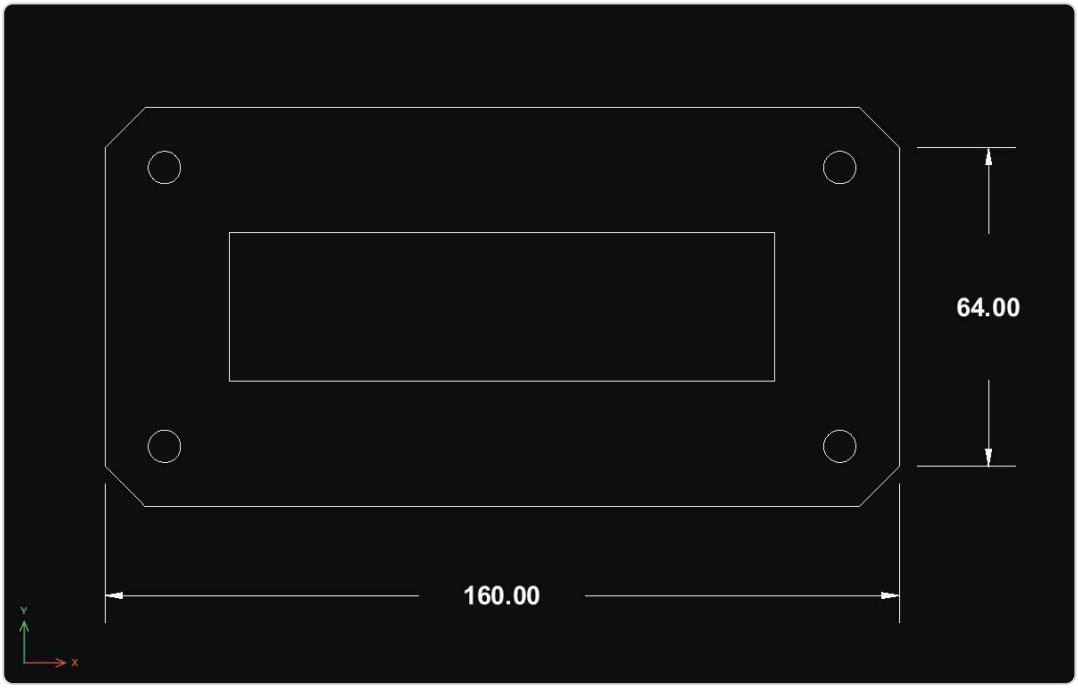
To get	Use
DWG	SAVEAS
PDF	PLOT, PUBLISH
G-code	GCODE
Separate DXF sheets after nesting	NEST
CSV of a table	`TABLEEXPORT`, see TABLE
ZIP for sending	ETRANSMIT

Try it: Cable-entry gasket for the waterjet shop

A 160 x 80 rubber gasket that seals a cable-entry cut-out in an enclosure wall. The waterjet shop asks for a DXF, so you draw it and export it in under three minutes.

1. Outline with chamfered corners: **Polyline - Closed Loop** through `8,0` `152,0` `160,8` `160,72` `152,80` `8,80` `0,72` `0,8`, then Enter.
2. Cable opening: `REC` `25,25` → `110` `Tab` `30`. Bolt holes: `C` radius `3.25` at `12,12`, `148,12`, `12,68` and `148,68`.
3. **Output › Interchange › Export**, name the file `cable_entry_gasket.dxf` and click Save.

4. Open the DXF to check it: the same six closed shapes come back in millimetres, ready for the cutter. Add a `DLI` of 160 and a `DVE` of 64 between the corner flats if the shop wants a check size.



The exported gasket DXF opened again, with check dimensions 160 and 64

Challenge: Send the same gasket to an AutoCAD user as DWG with SAVEAS.

See also

- NEW, SAVE and SAVEAS

GCODE - Generate G-code for CNC and laser

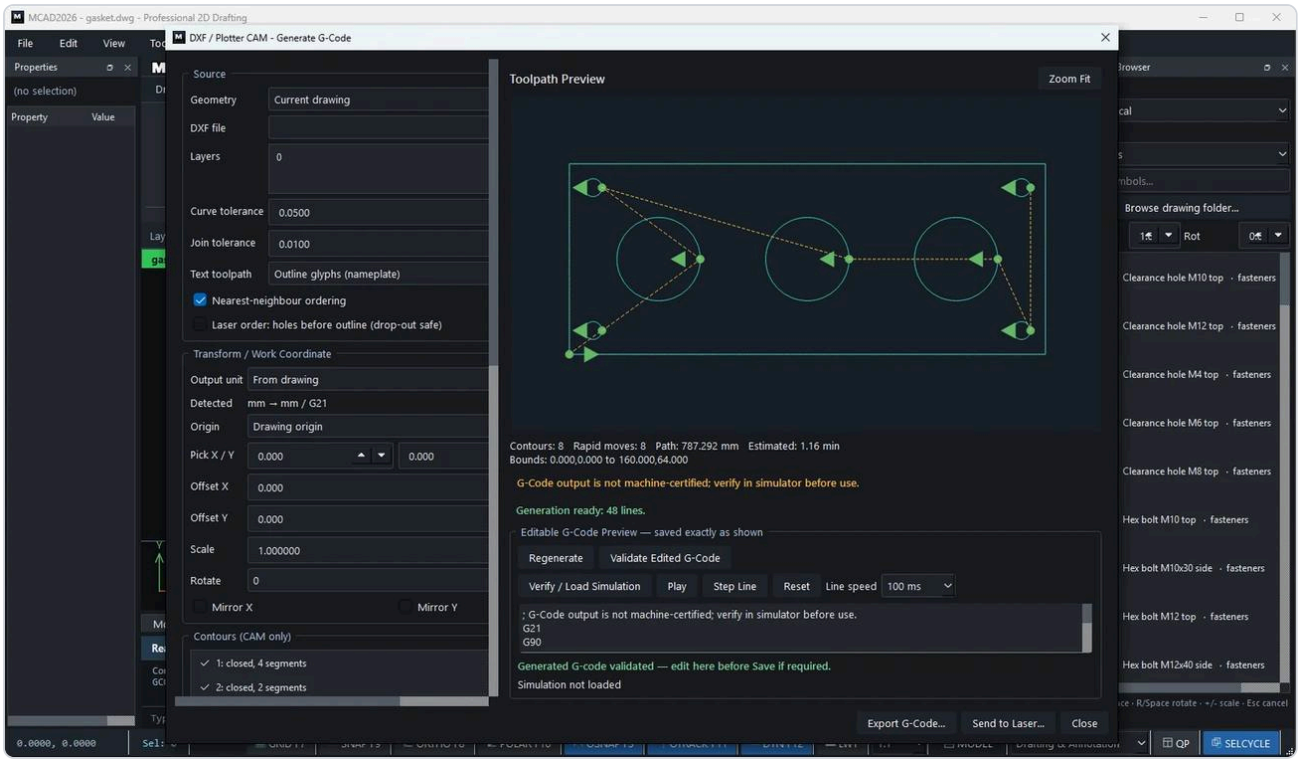
Generate G-code for CNC, laser, plasma or plotter from the lines in a drawing, and simulate the cut before you save.

What it does

GCODE converts the lines in a drawing (or an external DXF file) into toolpaths and G-code. It shows a preview, lets you edit the G-code, and simulates the run before you export.

Where to find it

- Ribbon: **Output › Interchange › Generate G-Code**
- Command line: `GCODE` (aliases `CAM`, `NCOUT`)
- Menu: **File › Export › Generate G-Code...**



DXF / Plotter CAM - Generate G-Code dialog

Steps

1. Draw or open the drawing (closed shapes for cutting).
2. Click **Generate G-Code**. Choose **Source › Geometry** (Current drawing, Selected entities, Visible entities, Selected layers, External DXF file).
3. Set **Transform / Work Coordinate** (Origin, Offset, Scale, Rotate, Mirror).
4. Choose **Machine / Post-processor › Profile**, then enter Feed, Plunge, Rapid, Safe Z, Cut Z and Passes.
5. Check the **Toolpath Preview** image and the summary of Contours / Path / Estimated time.
6. Click **Verify / Load Simulation**, then **Play** or **Step Line** to simulate.
7. Click **Export G-Code...** and save the file.

Main options

Group	Options
Source	Geometry, DXF file, Layers, Curve tolerance, Join tolerance, Text toolpath (Outline glyphs / SHX strokes), Nearest-neighbour ordering, Laser order: holes before outline
Transform / Work Coordinate	Output unit (From drawing / mm / inch), Origin (Drawing origin / Lower-left / Center / Pick point), Offset X/Y, Scale, Rotate (0/90/180/270), Mirror X/Y
Contours (CAM only)	Reverse cut direction, Open / Close, Start Next
Machine / Post-processor	Profile (Generic CNC, GRBL Laser/Router, Laser / Plasma cutter and others), Feed rate, Plunge rate, Rapid rate, Safe Z / Pen up, Cut Z / Pen down, Passes, Pass depth, Enable spindle / laser, Power / RPM, Precision, Extension (.nc .gcode .tap .txt), G90 absolute, Preserve arcs as G2/G3, Line numbers, Work width/height, Header, Footer

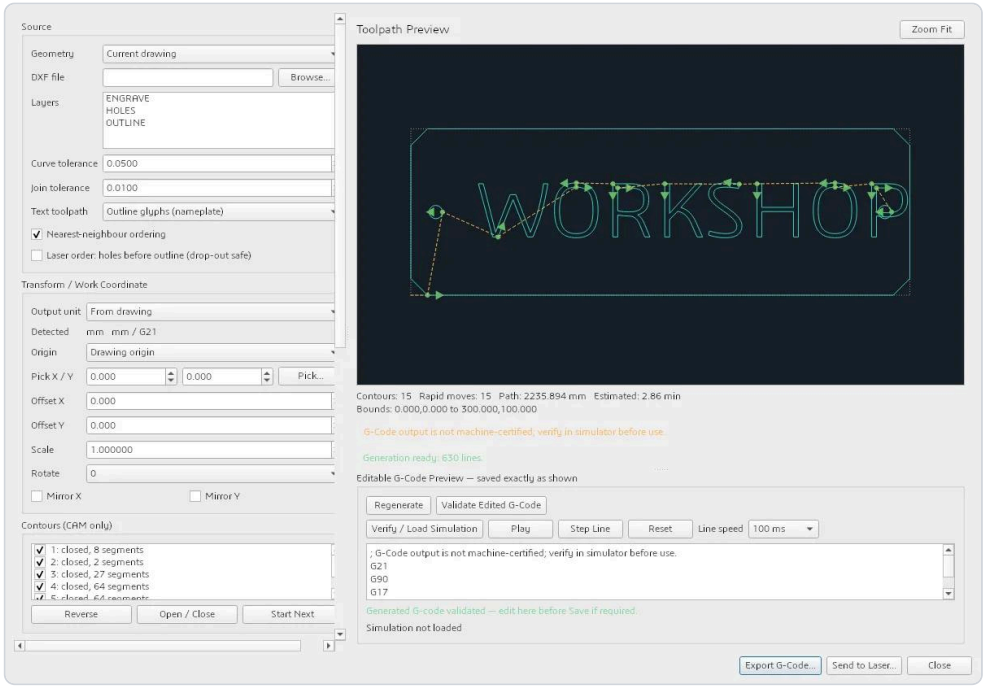
Group	Options
Editable G-Code Preview	Edit the G-code directly. Regenerate / Validate Edited G-Code.

> **Note:** The G-code is not certified for your machine. Always check it with the simulator and make a test cut before real production. An orange warning message tells you which objects were skipped.

Try it: CNC-routed WORKSHOP sign

A 300 x 100 sign for the workshop door: chamfered corners, two fixing holes and the word WORKSHOP cut as letter outlines. GCODE turns all of it into one toolpath you can watch in the simulator before the router touches the board.

1. Sign blank: **Polyline - Closed Loop** through `10,0` `290,0` `300,10` `300,90` `290,100` `10,100` `0,90` `0,10`, then Enter.
2. Fixing holes: `C` `15,50` radius `4` and `C` `285,50` radius `4`.
3. `DT` with Text height `32`, type `WORKSHOP` at about `40,35`.
4. **Output › Interchange › Generate G-Code**. Source: Geometry `Current drawing`, Text toolpath `Outline glyphs`.
5. Machine / Post-processor: Profile `GRBL Laser/Router`. The Toolpath Preview shows 15 contours: the outline, 2 holes and 12 letter contours (O, R and P each have an inside).
6. Click **Verify / Load Simulation**, then **Play**. When it looks right, **Export G-Code...** as `workshop_sign.gcode`.



G-Code dialog with the WORKSHOP sign toolpath in the preview

Challenge: Tick **Laser order: holes before outline** so the sign cannot drop out of the sheet before its holes and letters are cut.

See also

- NEST

- Gear
- REVERSE

NEST - Arrange Parts on a Sheet

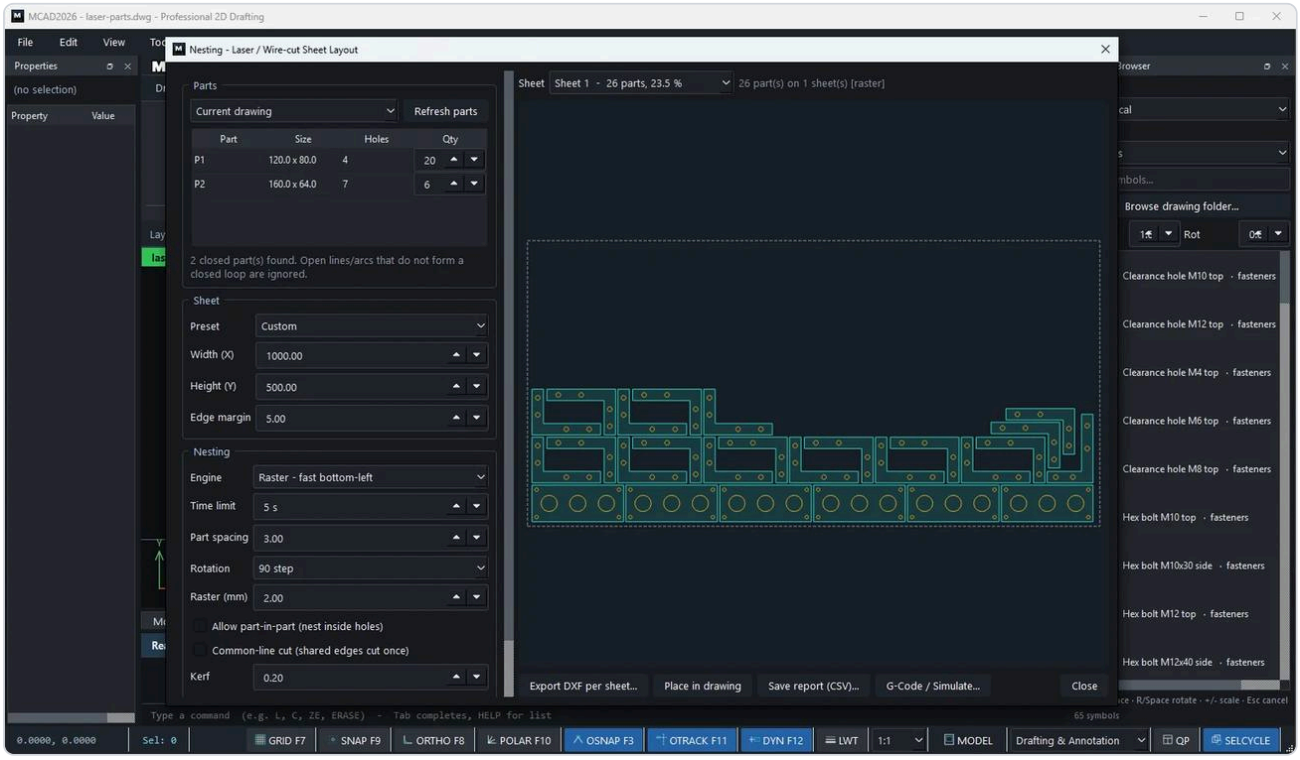
Automatically arrange parts on raw material sheets (nesting) for laser, plasma, and wire-cut jobs.

What it does

NEST places the closed shapes in your drawing onto raw material sheets so that as little material as possible is wasted. It then exports one DXF per sheet, a CSV report, or passes the result on to G-Code.

Where to find it

- Ribbon: **Output › Interchange › Nest**
- Command line: `NEST` (Alias `NESTING`, `SHEETNEST`)



Nesting - Laser / Wire-cut Sheet Layout window

Steps

1. Draw the parts as closed shapes (open shapes are not counted).
2. Click **Output › Nest** and check the **Parts** list (you can edit the quantity of each part).
3. Choose a **Sheet › Preset**, or set Width/Height/Edge margin.
4. Set the **Nesting** options (Engine, Time limit, Part spacing, Rotation, Kerf).
5. Click **Nest** and review each sheet.
6. Export with **Export DXF per sheet...**, **Place in drawing**, **Save report (CSV)...**, or **G-Code / Simulate...**

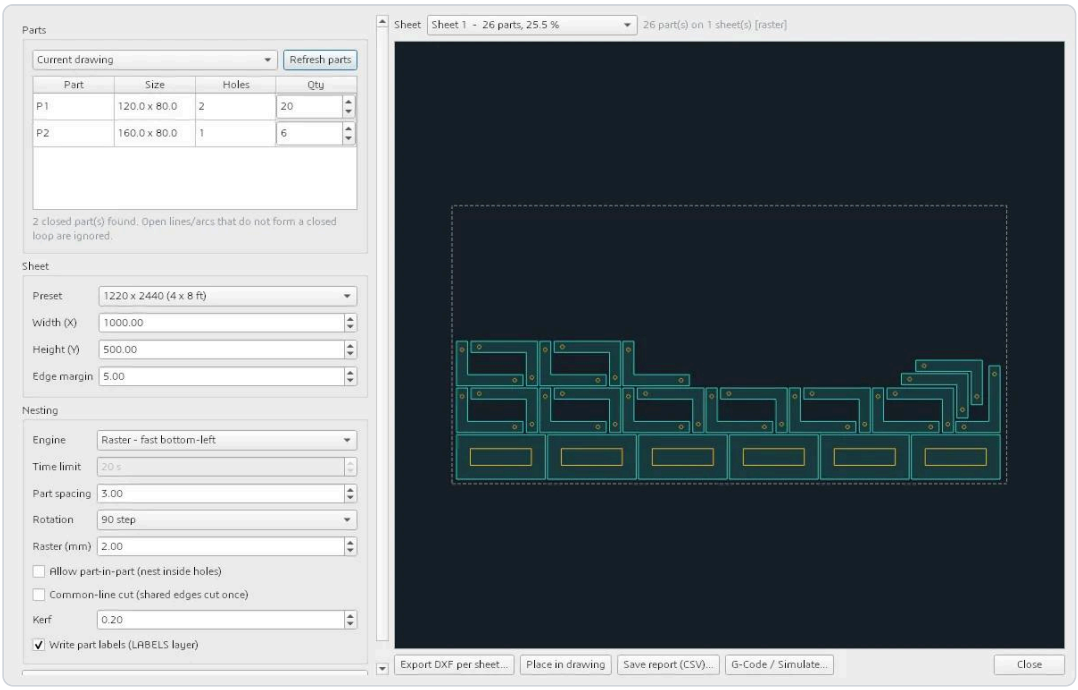
Options

Option	Meaning
Parts	Current drawing / Selected entities / Visible entities, Refresh parts
Preset	1220 x 2440 (4 x 8 ft), 1250 x 2500, 1500 x 3000, 1000 x 2000, Custom
Edge margin	Margin around the sheet edge
Engine	Auto, Sparrow (true-shape optimiser), Raster (fast, packs toward the lower left)
Time limit	Maximum calculation time
Part spacing	Distance between parts
Rotation	Fixed (0), 0/180, 90 step, 45 step, 30 step, 15 step
Allow part-in-part	Place small parts inside the holes of larger parts
Common-line cut	Cut shared edges once
Kerf	Width of the cut
Write part labels	Write part names on Layer LABELS

Try it: L-brackets and gland plates on one sheet

A batch for an enclosure job: 20 L-shaped mounting brackets and 6 gland plates, laser-cut from a 1000 x 500 offcut. NEST works out the layout and tells you how much of the sheet you use.

1. Bracket: **Polyline - Closed Loop** through `0,0` `120,0` `120,20` `20,20` `20,80` `0,80`, then Enter. Holes `C` `10,65` and `C` `105,10`, both radius `3.25`.
2. Gland plate: `REC` `200,0` → `160` `Tab` `80` with the cut-out `REC` `225,25` → `110` `Tab` `30`.
3. **Output › Interchange › Nest**. The Parts list shows P1 `120.0 x 80.0` with 2 holes and P2 `160.0 x 80.0` with 1 hole. Set Qty `20` for P1 and `6` for P2.
4. Sheet: Width `1000`, Height `500`, Edge margin `5`.
5. Nesting: Engine `Raster - fast bottom-left`, Rotation `90 step`, Part spacing `3`. Click **Nest**.
6. The result line reads `26 part(s) on 1 sheet(s)`. Use **Export DXF per sheet...** for the laser, or **G-Code / Simulate...** to go straight on.



Nesting window with 20 brackets and 6 gland plates on a 1000 x 500 sheet

Challenge: Tick **Allow part-in-part** and add a small part that fits inside the gland plate cut-out, then nest again and compare the utilisation.

See also

- GCODE
- EXPORT

Chapter 9 quiz: Print & Export

Choose the one best answer. Answers are in Appendix A

1. **If a Layout tab is open when you plot, at what size and scale does the plot come out?**

   A. Always A4, Fit to paper
   B. The size of the current view
   C. A3 at 1:2
   D. The Layout's actual paper size at 1:1

2. **What does PUBLISH (PUB) create?**

   A. One PDF that contains the Model and every Layout of the drawing
   B. A separate PDF for each Layout
   C. A ZIP of the drawing and its Xrefs
   D. A DXF file for each sheet

3. **In this 2D edition, how should you export the drawing to DXF?**

   A. Type `EXPORT`
   B. Type `DXFOUT`
   C. Use the Export button on the Ribbon (Output › Interchange › Export) or SAVEAS with the DXF type
   D. Use PUBLISH

4. **Which command adds a viewport to a Layout?**

   A. PSPACE
   B. MVIEW
   C. VPCLIP
   D. LAYOUT

5. **You type `TB` (TITLEBLOCK) and get `Title Block: open a Layout tab first.` What must you do?**

   A. Save the drawing first
   B. Turn on grid snap
   C. Run PAGESETUP A4
   D. Switch to a Layout tab, then run TITLEBLOCK

6. **Which command sets the paper size of a Layout, for example `A3 Landscape`?**

   A. MVIEW
   B. LAYOUT
   C. PAGESETUP
   D. UNITS

CHAPTER 10

DWG/DXF files

1. NEW, SAVE and SAVEAS - Save DWG/DXF

   Create a new drawing (NEW), save (SAVE), and save under a new name (SAVEAS) as DWG or DXF.

2. Import DXF, PDF, Point Cloud and LandXML

   Import DXF, vector shapes from a PDF (PDFIMPORT), point clouds and LandXML into the current drawing.

3. SAVEPROJECT - Save Multiple Tabs as a Project (.mprj)

   Save all open drawing tabs as one Project (.mprj) file so you can reopen the whole set at once.

4. ETRANSMIT - Package a drawing for sending

   Package the drawing and all its referenced files into a ZIP with a MANIFEST list, ready to send to someone else.

5. AUDIT, RECOVER and PURGE

   Check for bad objects (AUDIT), open a damaged file (RECOVER) and remove unused layers and styles (PURGE).

NEW, SAVE and SAVEAS - Save DWG/DXF

Create a new drawing (NEW), save (SAVE), and save under a new name (SAVEAS) as DWG or DXF.

What it does

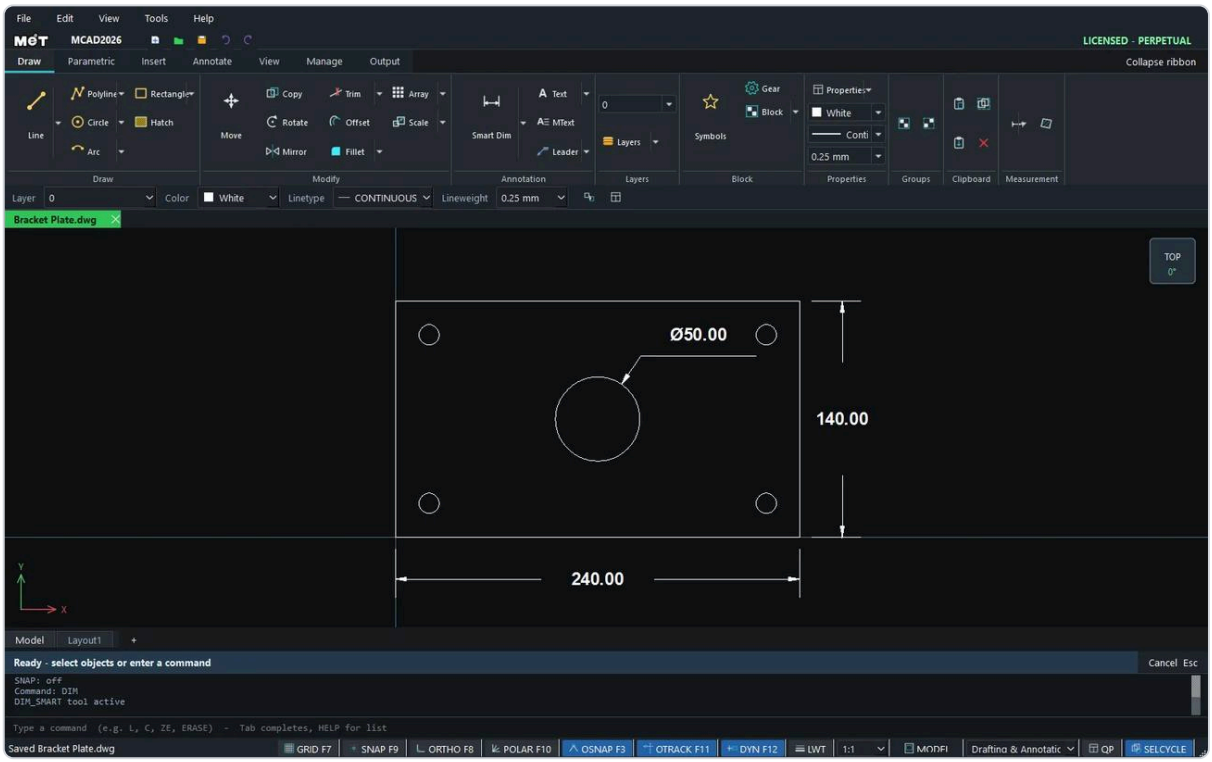
Command	Alias	Shortcut	Meaning
NEW	`QNEW`	`Ctrl+N`	New 2D drawing in a new tab (the existing tabs stay open)
SAVE	`QSAVE`	`Ctrl+S`	Saves the existing file (a new drawing opens Save As)
SAVEAS	–	`Ctrl+Shift+S`	Saves under a new name/type (menu **File › Save As...**)

Where to find it

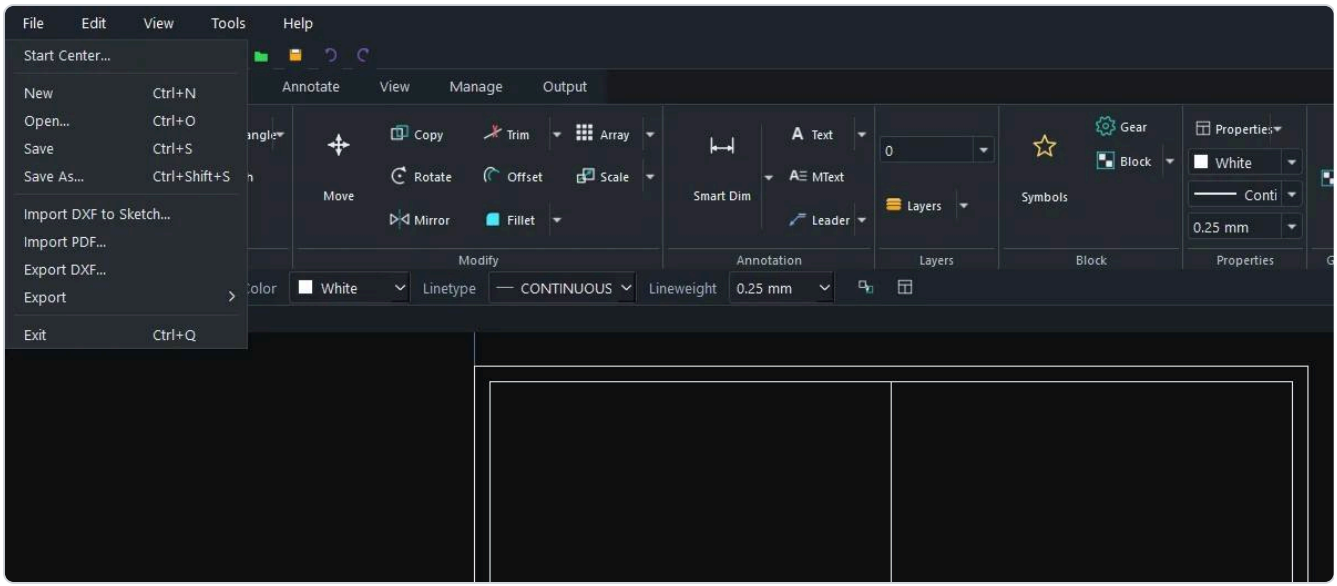
- Ribbon: **Output › File › Save / New / Open**
- Quick Access Toolbar, **File** menu

Steps

1. Press `Ctrl+Shift+S`, or type `SAVEAS` and press Enter.
2. Choose the file type `AutoCAD Drawing (*.dwg)` or `DXF (*.dxf)`.
3. Enter a name and click Save. The tab shows the file name, and the Status bar shows `Saved <file name>`.



A saved drawing: the tab name and Status bar show the file name



File menu

Save details

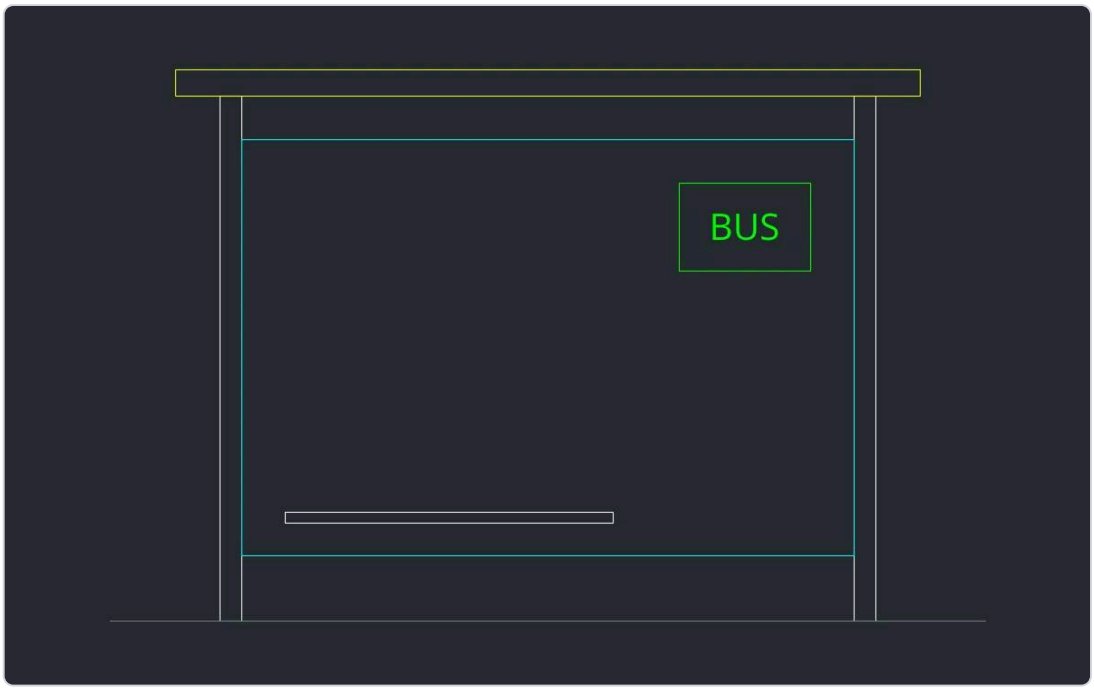
Topic	Details
File version	DWG and DXF in AutoCAD 2018 format. Opens in AutoCAD 2018 and later and in common CAD programs
Backup file	When you save a DWG over an existing file, the previous file is kept as `<name>.dwg.bak`
Background save	Large files are saved in the background, so you can keep working
No changes	If you have not changed anything, SAVE shows `No changes to save`
MCAD-specific data	Smart Dimension and extra annotation data are stored in the user's cache, so no extra files appear next to the DWG file
Thumbnail	Windows Explorer shows thumbnails of files saved by MCAD

> **Note:** When you close a tab that is not saved (the tab name has a `*`), the program asks before it discards your changes.

Try it: A bus shelter elevation, saved two ways

A 3 m bus shelter, saved as DWG for the client and as DXF for the steel cutter, from the same drawing.

1. Press `Ctrl+N` to start a new drawing in a new tab.
2. Posts: `REC` `0,0`, then `100` `Tab` `2400`, and `REC` `2900,0`, then `100` `Tab` `2400`.
3. Roof: `REC` `-200,2400`, then `3400` `Tab` `120`. Bench: `REC` `300,450`, then `1500` `Tab` `50`. Timetable sign: `REC` `2100,1600`, then `600` `Tab` `400`.
4. Press `Ctrl+Shift+S`, choose `AutoCAD Drawing (*.dwg)`, name it `bus-shelter` and click Save.
5. Press `Ctrl+Shift+S` again, choose `DXF (*.dxf)` with the same name and click Save. Both files are now in the same folder.



3 m bus shelter elevation

Challenge: Change the roof height and press `Ctrl+S`. The previous DWG is kept automatically as `bus-shelter.dwg.bak`.

See also

- Open DWG/DXF files
- SAVEPROJECT
- EXPORT

Import DXF, PDF, Point Cloud and LandXML

Import DXF, vector shapes from a PDF (PDFIMPORT), point clouds and LandXML into the current drawing.

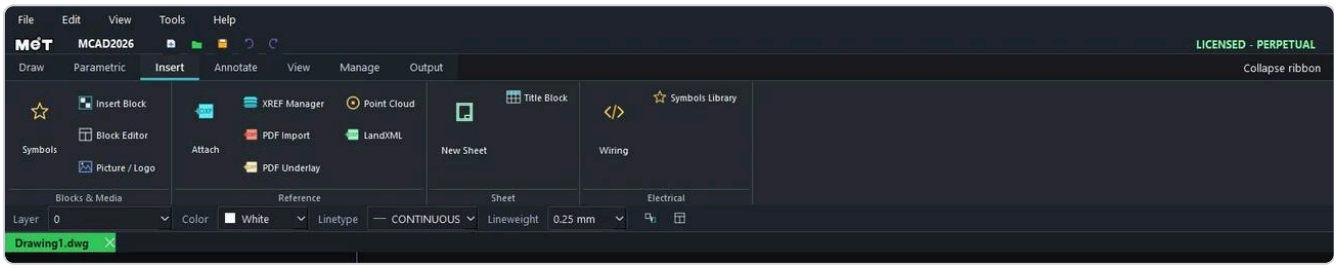
What it does

Command	Alias	Syntax	Meaning
Import (DXF)	–	Ribbon/menu	Imports a DXF as a sketch.
PDFIMPORT	`PDFIN`	`PDFIMPORT [file] [page]`	Converts the vector lines and text in a PDF into editable objects. (For a PDF that is a scanned picture, use IMAGE instead.)
POINTCLOUDATTACH	`POINTCLOUD`, `PCATTACH`	`POINTCLOUDATTACH [file] [max points]`	Imports points from XYZ/PTS/CSV/TXT (the default maximum is 250,000 points).
LANDXMLIMPORT	`LANDXML`	`LANDXMLIMPORT [file]`	Imports points, surfaces and alignments from LandXML.

Where to find it

- Ribbon: **Output › Interchange › Import / PDF Import**

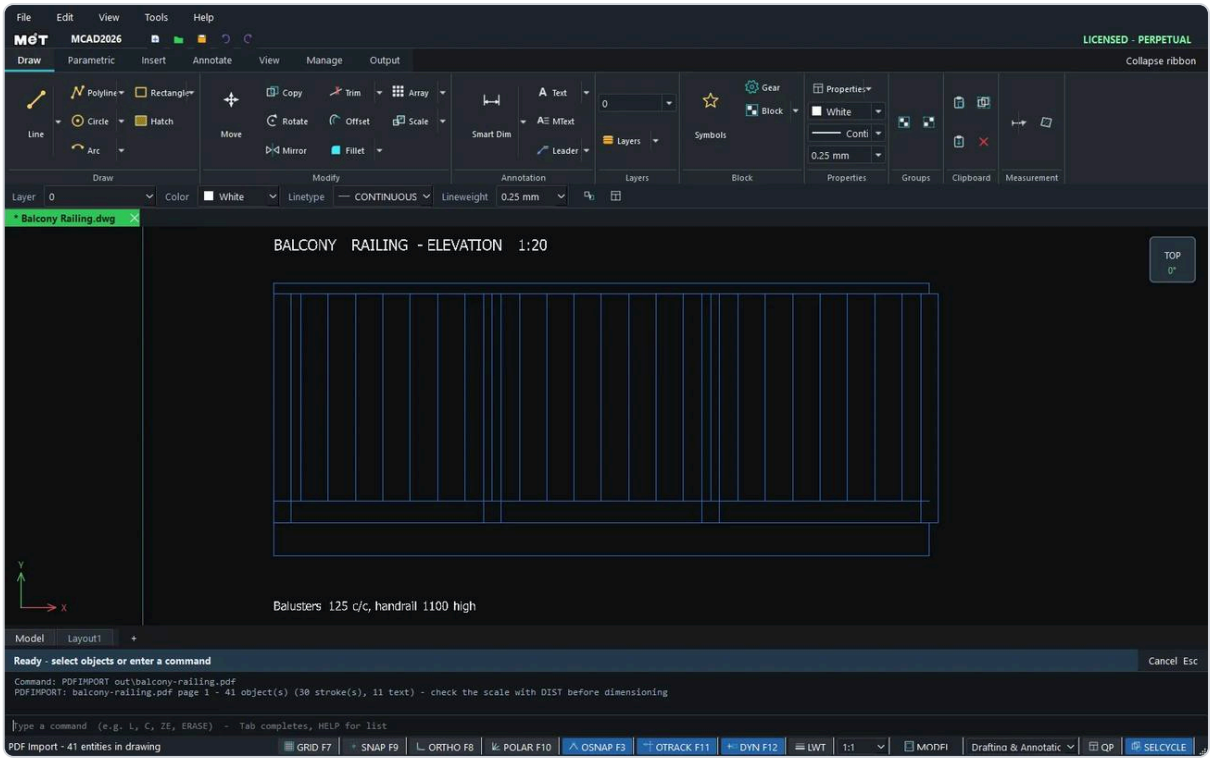
- Ribbon: **Insert › Reference › PDF Import / Point Cloud / LandXML**
- Menu: **File › Import DXF to Sketch... / Import PDF...**



Insert tab, Reference group

Steps for PDFIMPORT

1. Type `PDFIN` and press Enter, then choose the PDF file (or `PDFIMPORT C:\Drawings\part.pdf 2` for page 2).
2. The result reads `PDFIMPORT: part.pdf page 1 - N object(s) ... - check the scale with DIST before dimensioning`.
3. Measure a known distance with `DIST`. If it does not match, adjust with `SCALEREF`.



Vector shapes imported from a PDF, every line editable

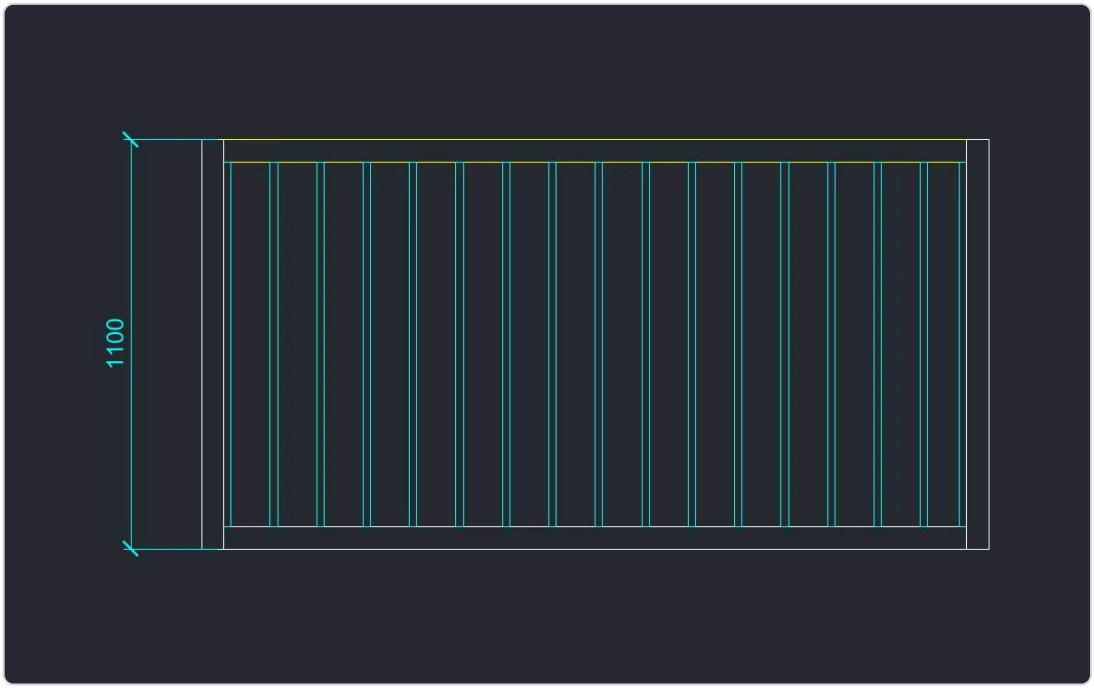
> **Note:** If the PDF has no vectors, the program reports `has no vector geometry - a scanned drawing is a picture, use IMAGE ATTACH`.

Try it: Bring a balcony railing from a supplier's PDF into your drawing

Scenario: a supplier sends a balcony railing drawing as a PDF, but you need to change the number of balusters to fit a 2 m balcony. Import it as editable lines.

1. Type `PDFIN`, press Enter and choose `balcony-railing.pdf`.
2. Read the Status bar: it says how many objects were imported and reminds you to check the scale with DIST.

3. Type `DI` and click from the floor to the top of the handrail. A standard railing is 1100 mm high. If you get another value, fix the scale with `SCALEREF`. See SCALE.
4. With the scale right, delete or add balusters just like lines you drew yourself. Balusters at 125 mm centres leave gaps under 105 mm.



2 m balcony railing, 1100 high, imported from a PDF

Challenge: If the PDF is a scan, the program tells you to use IMAGE ATTACH instead. Place the picture underneath and trace over it. See Images and PDF Underlays.

See also

- Images and PDF Underlays
- SCALEREF

SAVEPROJECT - Save Multiple Tabs as a Project (.mprj)

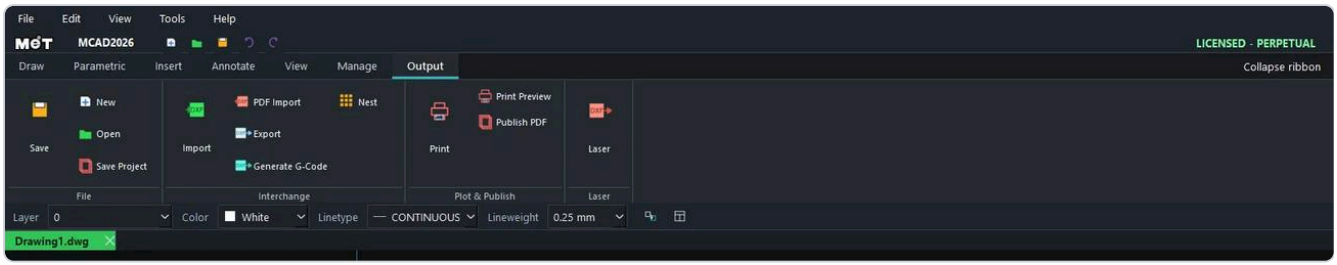
Save all open drawing tabs as one Project (.mprj) file so you can reopen the whole set at once.

What it does

SAVEPROJECT stores the list of every saved drawing tab in a `.mprj` file. When you open a `.mprj` with `OPEN`, all the sheets open at once as tabs. Wiring Wizard also creates a `.mprj` automatically.

Where to find it

- Ribbon: **Output › File › Save Project**
- Command line: `SAVEPROJECT` (Alias `SP`, `PROJ`)



Output tab, File group with Save Project

Steps

1. Save every tab as DWG/DXF first (unsaved Untitled tabs are skipped).
2. Type `SP` and press Enter. Name the `.mprj` file (the default is the folder name).
3. To open it later, press `Ctrl+O` and select the `.mprj` file.

> **Note:** Keep all the drawing files in the same folder as the `.mprj`. If any file is in another folder, the program warns you. If there are no saved tabs at all, it shows `No saved drawings open - save each tab as DWG first`.

Try it: A three-sheet bakery fit-out set

Scenario: a bakery fit-out has 3 sheets: the floor plan, the counter elevation and the shop sign. Every morning you need all 3 tabs open.

1. Save all 3 tabs as DWG in the same folder, for example `D:\Bakery\floor-plan.dwg`, `counter-elevation.dwg` and `shop-sign.dwg`.
2. Type `SP`, press Enter and name the file `bakery-fitout.mprj` in that folder.
3. Close the program. Tomorrow, press `Ctrl+O` and choose `bakery-fitout.mprj`. All 3 sheets open as tabs at once.
4. An unsaved Untitled tab is not included in the project. Save it first, then run `SP` again.

See also

- Wiring Wizard
- Sheet Set

ETRANSMIT - Package a drawing for sending

Package the drawing and all its referenced files into a ZIP with a MANIFEST list, ready to send to someone else.

What it does

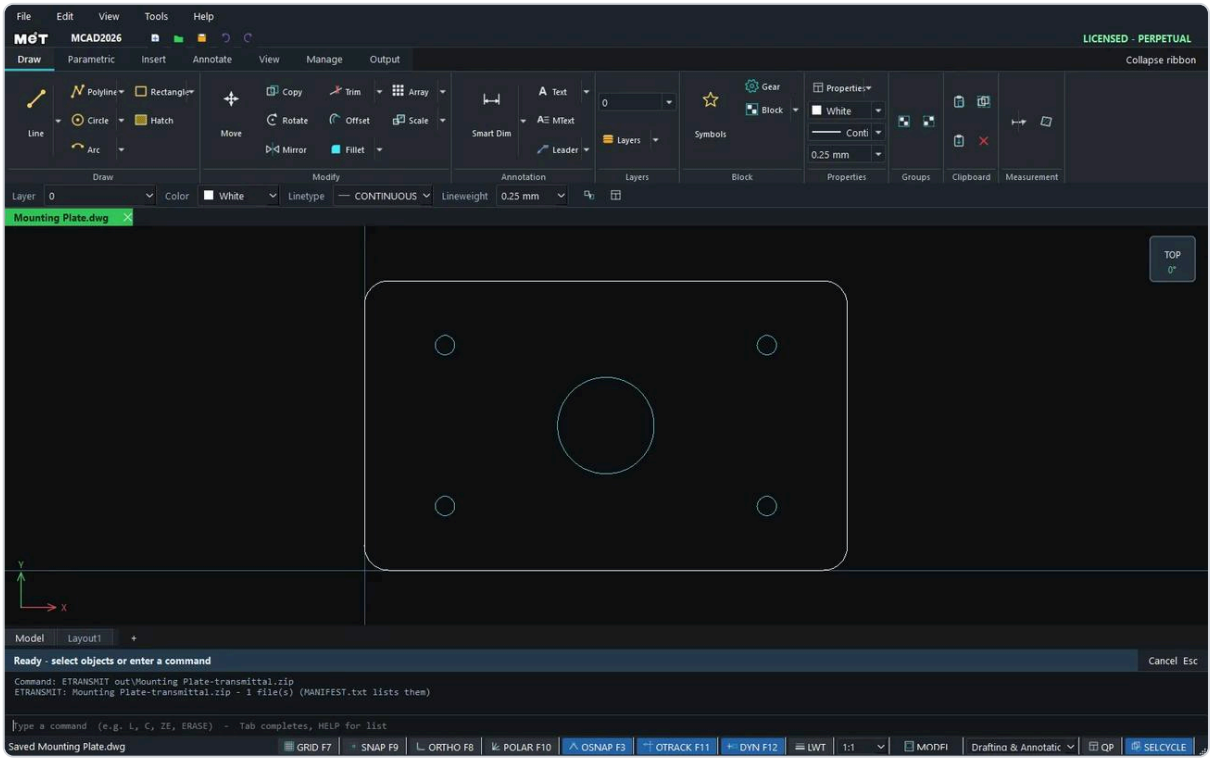
ETRANSMIT creates a ZIP file that contains the current drawing (as DXF), every Xref file, and a `MANIFEST.txt`. The manifest lists what is included and which referenced files could not be found.

Where to find it

- Ribbon: type the command
- Command line: `ETRANSMIT` (alias `ZIP`)

Steps

1. Type `ETRANSMIT` and press Enter.
2. Name the file (the default is `<drawing name>-transmittal.zip`) and click Save, or type `ETRANSMIT D:\Send\bracket.zip`.
3. The result reads `ETRANSMIT: bracket.zip - N file(s) (MANIFEST.txt lists them)`. If any referenced files are missing, it reports the `MISSING` count.



A drawing packaged with ETRANSMIT

In the ZIP file	Meaning
`<name>.dxf`	The current drawing.
`xrefs/...`	External Reference files.
`MANIFEST.txt`	The object count, the files included, the files missing and the objects that could not be exported.

Try it: Package a hotel room drawing for the contractor

Scenario: the fit-out drawing for hotel room 301 references the furniture layout as an Xref. Send only one file and the contractor sees an empty room. ETRANSMIT packs everything in one go.

1. Open `hotel-room-301.dwg` and type `ETRANSMIT D:\Send\hotel-room-301.zip`.
2. Read the result on the Status bar, for example `ETRANSMIT: hotel-room-301.zip - 3 file(s) (MANIFEST.txt lists them)`.
3. If you see `MISSING`, open `MANIFEST.txt` in the ZIP to see which Xref could not be found, fix its path in XREF and package it again.
4. Send the single ZIP file. When the contractor extracts it, they get the drawing as DXF plus an `xrefs/` folder.

See also

- XREF

- EXPORT

AUDIT, RECOVER and PURGE

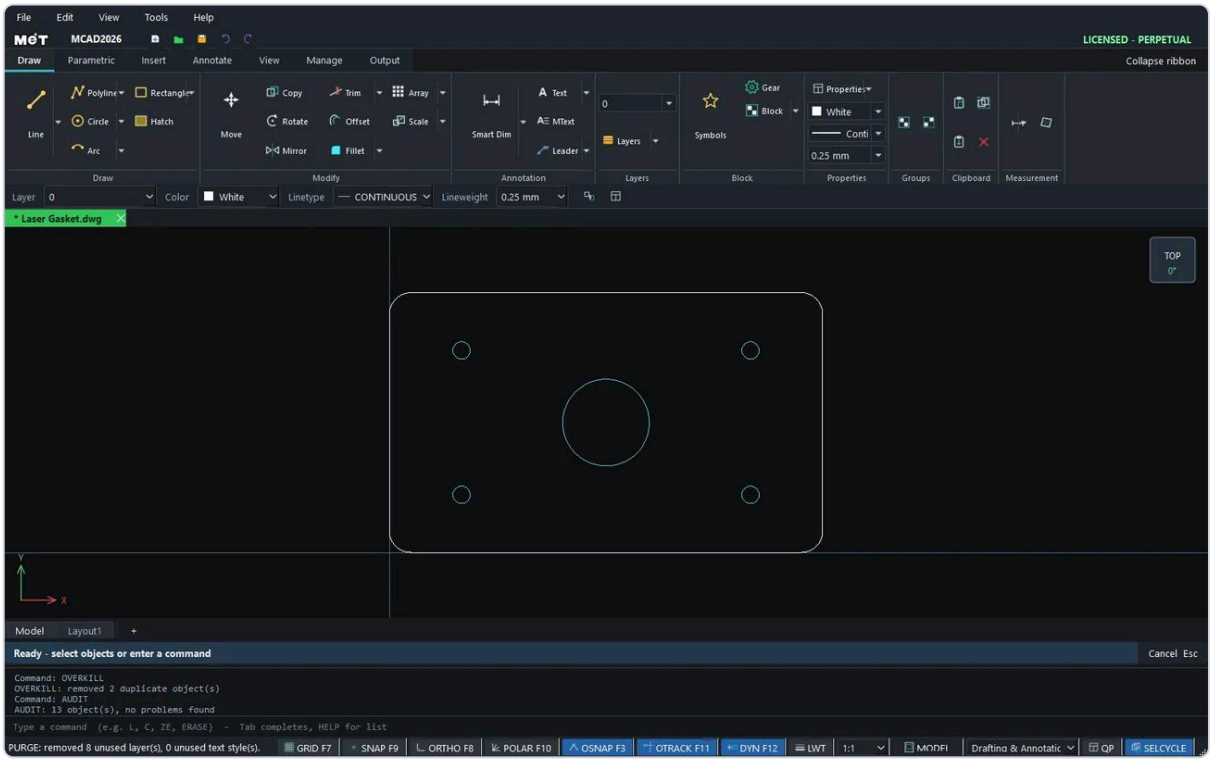
Check for bad objects (AUDIT), open a damaged file (RECOVER) and remove unused layers and styles (PURGE).

What it does

Command	Alias	Syntax	Meaning
AUDIT	–	`AUDIT [FIX]`	Reports objects that cannot be drawn (for example a zero radius or missing points). `AUDIT FIX` deletes the unusable objects.
RECOVER	–	`RECOVER [file]`	Opens a damaged DXF file with a repairing reader. It reports how many objects were repaired and how many could not be recovered.
PURGE	`PU`	–	Removes layers that contain no objects and text styles that are not used.

Where to find it

- Ribbon: **Manage › Drawing Cleanup › Audit / Purge / Delete Duplicates / Drawing Status**



Results of OVERKILL, AUDIT and PURGE on the command line

Steps to clean a drawing before you send it

1. Run `AUDIT`. If it finds problems, run `AUDIT FIX`.
2. Run `OVERKILL` to remove duplicate lines.
3. Run `PURGE` to remove unused layers and styles. The Status bar shows `PURGE: removed N unused layer(s), N unused text style(s).`
4. Run `SAVE`.

Steps for RECOVER

1. Type `RECOVER` and press Enter, then select the DXF file.
2. The result reads `RECOVER: <file> - N object(s), N repair(s), N unrecoverable (run AUDIT next)`.
3. Run `AUDIT` next, then save the drawing under a new name.

> **Note:** The program keeps backup files. If the program closes unexpectedly, open Start Center › **Drawing Recovery** to open the backup file.

Try it: Clean a mall wayfinding sign drawing before it goes to the printer

Scenario: a mall wayfinding sign drawing has been revised many times. It has duplicate lines, empty layers and some odd objects. Clean it in this order before you send it to the sign maker.

1. Type `AUDIT` and read how many objects cannot be drawn (for example a zero-radius circle from a bad copy).
2. If there are any, type `AUDIT FIX` to delete the unusable objects.
3. Type `OVERKILL` to remove duplicate lines, or the cutter will cut the same line twice.
4. Type `PURGE`. The Status bar shows how many unused layers and text styles were removed.
5. `SAVEAS` under a new name such as `mall-signage-clean.dwg`, and keep the original in case you need to go back.

See also

- OVERKILL, FLATTEN
- Common problems

Chapter 10 quiz: DWG/DXF files

Choose the one best answer. Answers are in Appendix A

1. **Which AutoCAD version format does MCAD use when it saves DWG and DXF?**

   A. AutoCAD R12
   B. AutoCAD 2018
   C. AutoCAD 2010
   D. AutoCAD 2004

2. **When you save a DWG over an existing file, where is the previous version kept?**

   A. In the Recycle Bin
   B. In a ZIP file next to the drawing
   C. It is not kept
   D. As `<name>.dwg.bak`

3. **Which command removes layers that contain no objects and unused text styles?**

   A. PURGE
   B. AUDIT
   C. OVERKILL
   D. RECOVER

4. **A DXF file is damaged and will not open normally. Which command opens it with a repairing reader?**

   A. AUDIT FIX
   B. PURGE
   C. RECOVER
   D. OPEN

5. **What does ETRANSMIT create?**

   A. A PDF of every Layout
   B. A ZIP with the drawing (as DXF), its Xref files and a MANIFEST.txt
   C. A .mprj project file
   D. A .dwg.bak backup

6. **SAVEPROJECT saves all open drawing tabs as one file that reopens them together. What is its file extension?**

   A. .dwt
   B. .zip
   C. .dxf
   D. .mprj

CHAPTER 11

Troubleshooting

1. Common Problems and Solutions

   Fixes for common MCAD 2026 problems with licences, opening files, snapping, display, printing, and Thai text.

Common Problems and Solutions

Fixes for common MCAD 2026 problems with licences, opening files, snapping, display, printing, and Thai text.

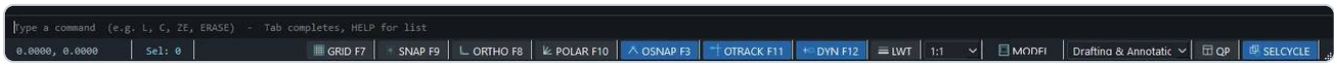
Licence

Symptom	Fix
The Activate License window appears every time you start the program	Check your internet connection, and enter the e-mail that is registered with the key. See Activate Your Licence
After moving to a new computer, you are told all seats are in use	On the old computer, use **Help › Release License** first, or contact MET with your Machine ID
The computer has no internet access	Ask MET for an offline licence file, then use **Help › Import Offline License File...**
Plot/Publish/Export commands do not work	The licence has expired or is a limited trial. Check the status at the top right of the Ribbon

Opening files

Symptom	Fix
A DWG will not open / an error message appears	Save the drawing as DXF from the source program and open that, or use `RECOVER`. See AUDIT, RECOVER
A large file opens slowly	The file loads in the background, and you can work in other tabs while you wait. Use `QTEXT ON` to pan faster
The screen is blank after opening	Type `ZE` (Zoom Extents), or `LAYON` and `LAYTHW` if the Layers are turned off
A drawing with Z levels makes Snap/Area wrong	`FLATTEN`
A file on OneDrive/a network drive will not open	Copy it to a local drive first

Drawing and snapping



Snap, Ortho, Polar, OSnap buttons on the Status bar

Symptom	Fix
Clicks do not land exactly on a point	Turn on OSNAP (`F3`). Turn off Grid Snap (`F9`) if you do not want it
Lines are forced horizontal/vertical on their own	Turn off ORTHO (`F8`) or POLAR (`F10`)
You cannot select objects	The Layer is locked. Use `LAYULK`
Typing numbers produces Thai characters	Switch the keyboard to English. The Smart Input number fields accept numbers from the numeric keypad even when the keyboard is set to Thai
A command is stuck	Press `Esc` , or click the **Cancel Esc** button on the Prompt bar

Symptom	Fix
Hatch reports `produced no lines`	Reduce the Pattern Scale
Boolean/Hatch reports no Closed profile found	Use `PEDIT` to join the lines into a closed Polyline, or use `BOUNDARY`

Display

Symptom	Fix
Dashed lines look solid	Adjust `LTSCALE`
Dimensions are too small/large	Change the Text height in `DIMSTYLE`, or use `DIMSCALE`
Hatch covers the text	`TEXTTOFRONT` or `HATCHTOBACK`
The Ribbon is gone	Double-click a Ribbon tab name, or press `Ctrl+0` if Clean Screen is on
The Command line is gone	`Ctrl+9`
A palette is gone	Menu **View** › Properties Palette / Layers Palette / Symbols Palette

Printing

Symptom	Fix
The PDF does not fill the page	Set Scale = Fit to paper, or use a Layout with the correct size set in `PAGESETUP`
Light-colored lines print too faintly	Set Ink = Black & White, or use a CTB file

Send information to support

Give us the Release version from **Help › About**, your Machine ID (Activate License window › Copy ID), and the steps that cause the problem. Keep **Options › General › Diagnostics** on so that reports are sent automatically when the program hangs or closes unexpectedly (no drawing data is sent).

Try it: A hotel lobby plan opens to a blank screen

Scenario: a contractor sends a hotel lobby plan. It opens blank, and once you can see it, the dashed lines look solid. Work through it step by step.

1. Type `ZE`. If the drawing sits far from the current view, it zooms to show everything at once.
2. Still blank? Type `LAYON`, then `LAYTHW`, to turn on and thaw every layer.
3. Column grid lines that should be dashed look solid: the drawing is in millimetres, so try `LTSCALE 20` and adjust until the dashes are clear.
4. You cannot select anything: the layer is locked. Use `LAYULK`.
5. Before you send it back, run `AUDIT` to check for bad objects. See AUDIT, RECOVER and PURGE.

See also

- Command Index
- Status Bar

Chapter 11 quiz: Troubleshooting

Choose the one best answer. Answers are in Appendix A

1. **A drawing opens to a blank screen. What should you type first?**

   A. REGEN
   B. AUDIT
   C. ZE (Zoom Extents)
   D. PURGE

2. **You cannot select any objects on a layer. What is the likely cause and fix?**

   A. The layer is locked; use LAYULK
   B. The layer is frozen; use LAYON
   C. OSNAP is off; press F3
   D. The drawing is read-only; use SAVEAS

3. **The Command line has disappeared. Which shortcut brings it back?**

   A. Ctrl+0
   B. Ctrl+9
   C. F2
   D. Ctrl+1

4. **The Ribbon has disappeared. How do you bring it back?**

   A. Press F2
   B. Press Ctrl+1
   C. Run REGEN
   D. Double-click a Ribbon tab name, or press Ctrl+0 if Clean Screen is on

5. **HATCH reports `produced no lines`. What is the fix?**

   A. Increase the Pattern Scale
   B. Reduce the Pattern Scale
   C. Change the hatch angle to 90°
   D. Use HATCHTOBACK

6. **Lines are forced horizontal or vertical on their own. What should you turn off?**

   A. OSNAP (F3) or Grid (F7)
   B. DYN (F12) or OTRACK (F11)
   C. ORTHO (F8) or POLAR (F10)
   D. LWT or QP

Appendix A · Quiz answers

Chapter 1 · Getting started

No.	Answer	Review topic
1	**B** Help › Check for Updates...	Install MCAD 2026 (p. 8)
2	**A** Help › About MCAD2026	Install MCAD 2026 (p. 8)
3	**D** Run Help › Release License (free this seat)	Activate Your Licence (p. 9)
4	**C** Import a .json licence file signed by MET with Help › Import Offline License File...	Activate Your Licence (p. 9)
5	**B** Ctrl+Tab	Open DWG/DXF Files and the Start Center (p. 12)
6	**A** FILLET works on LINE objects, not on an outline drawn with REC or PLINE	Draw your first part (Bracket Plate) (p. 14)

Chapter 2 · Interface & settings

No.	Answer	Review topic
1	**C** The last command is repeated	Command Line, Aliases and Smart Input (p. 44)
2	**D** ' (apostrophe)	Command Line, Aliases and Smart Input (p. 44)
3	**A** F3	Status Bar: Snap, Ortho, Polar, OSnap (p. 47)
4	**C** 45°	Status Bar: Snap, Ortho, Polar, OSnap (p. 47)
5	**B** ZE	Zoom, Pan and View Controls (p. 50)
6	**D** They are not scaled; use SCALE if you need to resize them	Units (UNITS) and System Variables (p. 59)

Chapter 3 · Draw

No.	Answer	Review topic
1	**A** You click the midpoint first, then one end, and the line extends equally on both sides	LINE (p. 68)
2	**C** Always 6 sides (a hexagon)	POLYGON - Regular Polygon (p. 83)
3	**B** Tab	RECTANG - Rectangle (p. 81)
4	**D** Pattern, scale, angle	HATCH and GRADIENT (p. 93)
5	**A** HATCHTOBACK	HATCH and GRADIENT (p. 93)
6	**C** XLINE extends in both directions; RAY extends from a start point in one direction	XLINE and RAY - Construction Lines (p. 70)

Chapter 4 · Modify

No.	Answer	Review topic
1	**B** Creates copies 10 units away on both sides without clicking a side	OFFSET - Parallel Copies (p. 122)
2	**D** The lines are joined at a sharp corner	FILLET and CHAMFER - Corner rounding and bevelling (p. 127)
3	**A** Only objects that are completely inside the box	Selecting Objects (SELECT, QSELECT, FILTER) (p. 109)
4	**C** OOPS	ERASE, OOPS, UNDO and REDO (p. 111)
5	**B** EXTRIM Outside (EXTRIM O)	TRIM, EXTEND and EXTRIM (p. 124)
6	**D** 360°	ARRAY - Rectangular / Polar / Path (p. 135)

Chapter 5 · Dimension & Annotation

No.	Answer	Review topic
1	**C** The distance between the edges	Smart Dimension (DIM) (p. 159)
2	**A** It replaces the displayed value (override)	Smart Dimension (DIM) (p. 159)
3	**B** DIMSTYLE	DIMSTYLE and STYLE - Style Manager (p. 174)
4	**D** Ctrl+Enter	TEXT, MTEXT and TEXTEDIT (p. 183)
5	**C** DIMALIGNED (DAL)	DIMLINEAR, DIMALIGNED and DIMVERTICAL (p. 162)
6	**B** Arrow tip, elbow, text location	QLEADER, LEADER, MLEADER and Callout (p. 179)

Chapter 6 · Layer & Properties

No.	Answer	Review topic
1	**B** The layer stays visible, but you cannot select or edit its objects	LAYER - Layer Manager (p. 198)
2	**D** Thaw it (for example with LAYTHW)	LAYER - Layer Manager (p. 198)
3	**A** LAYUNISO	Layer tools (LAYISO, LAYOFF, LAYFRZ, LAYLCK ...) (p. 200)
4	**C** Ctrl+1	PROPERTIES, MATCHPROP and CHPROP (p. 202)
5	**B** MATCHPROP (MA)	PROPERTIES, MATCHPROP and CHPROP (p. 202)
6	**D** LTSCALE	COLOR, LINETYPE and LWEIGHT (p. 204)

Chapter 7 · Block & Symbol

No.	Answer	Review topic
1	**A** The lower-left corner	BLOCK, INSERT and WBLOCK (p. 209)
2	**C** The Content Browser opens	BLOCK, INSERT and WBLOCK (p. 209)
3	**B** ATTDISP OFF	Attributes (ATTDEF, ATTEDIT, ATTDISP) (p. 213)
4	**D** Copies its objects into the drawing and breaks the link	External Reference (XREF, XATTACH, XCLIP, XBIND) (p. 217)
5	**A** XCLIP	External Reference (XREF, XATTACH, XCLIP, XBIND) (p. 217)
6	**C** R or Space	Symbol Library (Content Browser) (p. 215)

Chapter 8 · Electrical & Wiring

No.	Answer	Review topic
1	**C** A tag export (.xlsx) from Siemens TIA Portal	Wiring Wizard (WIRING) - PLC IO as A4 Pages (p. 226)
2	**B** 8	Wiring Wizard (WIRING) - PLC IO as A4 Pages (p. 226)
3	**A** Replaces all of the existing text with the number	Terminal and Wire Numbering (TCOUNT / Re-tag) (p. 229)
4	**D** The prefix	Terminal and Wire Numbering (TCOUNT / Re-tag) (p. 229)
5	**B** A Python (.py) file	SCRIPT - Python Macro API (p. 232)
6	**C** Their names end in `-iso`	Electrical Symbols - Placing electrical symbols (p. 224)

Chapter 9 · Print & Export

No.	Answer	Review topic
1	**D** The Layout's actual paper size at 1:1	PLOT and PREVIEW - Print and PDF (p. 238)
2	**A** One PDF that contains the Model and every Layout of the drawing	PUBLISH - Combine All Layouts into One PDF (p. 240)
3	**C** Use the Export button on the Ribbon (Output › Interchange › Export) or SAVEAS with the DXF type	EXPORT - Exporting files (p. 249)
4	**B** MVIEW	Layouts and viewports (LAYOUT, MVIEW, PAGESETUP) (p. 242)
5	**D** Switch to a Layout tab, then run TITLEBLOCK	TITLEBLOCK and MET Drawing Sheet (p. 244)
6	**C** PAGESETUP	Layouts and viewports (LAYOUT, MVIEW, PAGESETUP) (p. 242)

Chapter 10 · DWG/DXF files

No.	Answer	Review topic
1	**B** AutoCAD 2018	NEW, SAVE and SAVEAS - Save DWG/DXF (p. 259)
2	**D** As `<name>.dwg.bak`	NEW, SAVE and SAVEAS - Save DWG/DXF (p. 259)
3	**A** PURGE	AUDIT, RECOVER and PURGE (p. 266)
4	**C** RECOVER	AUDIT, RECOVER and PURGE (p. 266)
5	**B** A ZIP with the drawing (as DXF), its Xref files and a MANIFEST.txt	ETRANSMIT - Package a drawing for sending (p. 264)
6	**D** .mprj	SAVEPROJECT - Save Multiple Tabs as a Project (.mprj) (p. 263)

Chapter 11 · Troubleshooting

No.	Answer	Review topic
1	**C** ZE (Zoom Extents)	Common Problems and Solutions (p. 270)
2	**A** The layer is locked; use LAYULK	Common Problems and Solutions (p. 270)
3	**B** Ctrl+9	Common Problems and Solutions (p. 270)
4	**D** Double-click a Ribbon tab name, or press Ctrl+0 if Clean Screen is on	Common Problems and Solutions (p. 270)
5	**B** Reduce the Pattern Scale	Common Problems and Solutions (p. 270)
6	**C** ORTHO (F8) or POLAR (F10)	Common Problems and Solutions (p. 270)

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Appendix C · System requirements and support

System requirements

Operating system	Windows 10 / 11 64-bit
Processor	x64, 2 cores or more
Memory	4 GB (8 GB recommended for large DWG files)
Disk space	1 GB
Display	1366×768 or larger, 100-200% scaling

Installing and updating

- Download the installer from www.mechdesigned.com/en/software/mcad or the Microsoft Store (search "MCAD")
- The program tells you when a new version is out; users with active Maintenance (MA) update free for the MA term.

Support

E-mail	sales@mechdesigned.com
Phone	+66 2 520 4575 · +66 83 788 0003
LINE	@mechdesigned
Website	www.mechdesigned.com

When reporting a problem, include the version (Help › About), a screenshot and, if you can share it, the file you were working on.

About the author and MET

Suppakit ("P'Kit")

Mechatronics engineer · Engineering lead at MET · Creator of MCAD2026

P'Kit has spent his career building automated machines for factories in Thailand: mechanical design, Siemens TIA Portal and Codesys PLC programs, WinCC Unified HMIs, robots and cobots, and machine vision, delivered as complete systems that run on the shop floor.

MCAD and MECHCAD grew out of the daily work of the MET team: drafting, designing parts, assembling machines and programming CNC. The goal is affordable CAD/CAM that works fully in Thai and English, works with DWG, STEP, SOLIDWORKS and Alibre files, and keeps improving from what users ask for.

Mechatronics Designed Co., Ltd. (MET)

Founded in 2008 in Pathumthani, Thailand, MET designs and builds automated machinery for the food, cosmetics, pharmaceutical and packaging industries, from design and manufacture through installation and after-sales service.

Packaging machines	Filling, capping and labelling machines and complete packing lines
MT-COBOT	Cobot cells for palletizing, pick-and-place and packing
MCAD · MECHCAD	The CAD/CAM software the MET team uses to design its own machines

www.mechdesigned.com · sales@mechdesigned.com · +66 2 520 4575 · LINE @mechdesigned



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