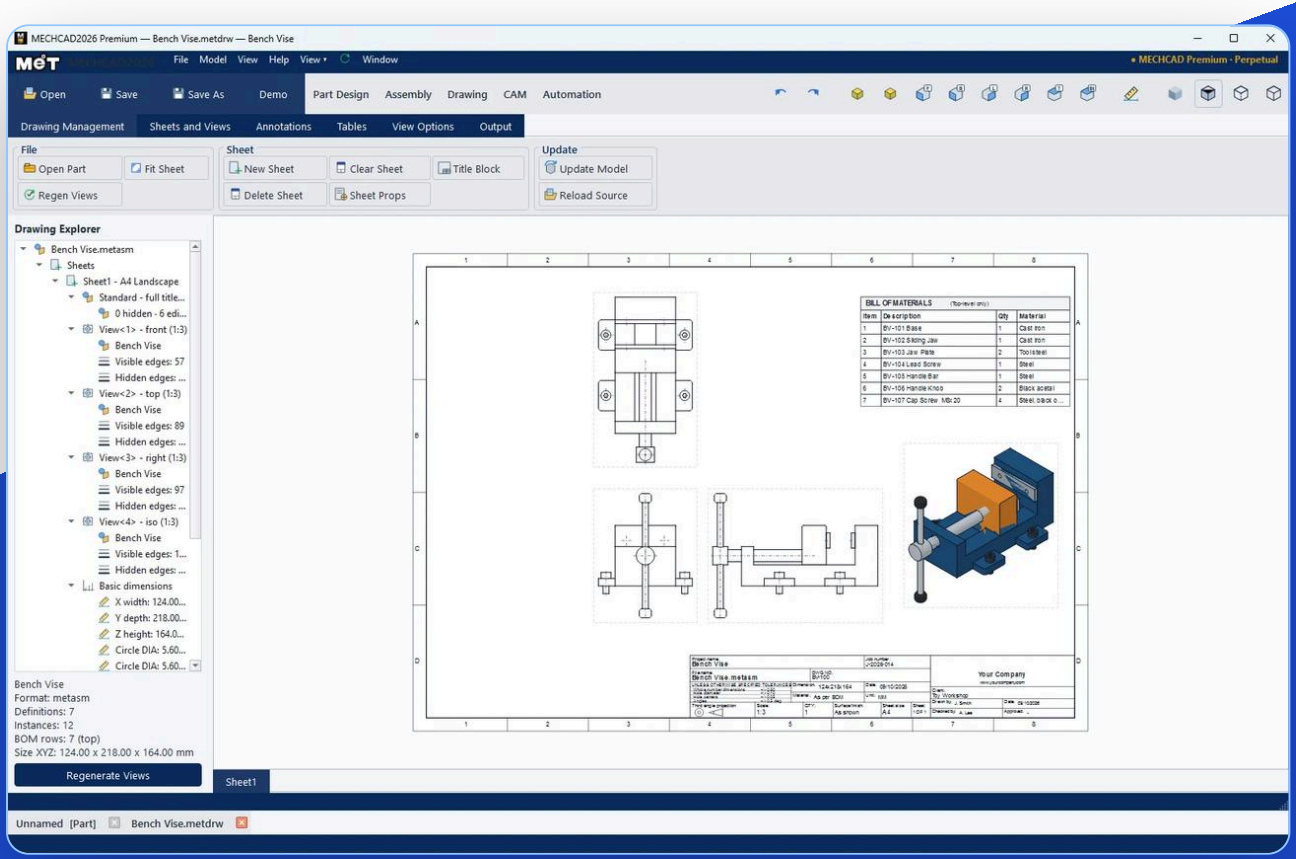
MECHATRONICS DESIGNED CO., LTD.

USER MANUAL

MECHCAD2026

3D part design, assembly, drawings and CAM



English edition · for version 2026.80

25 October 2026 · www.mechdesigned.com

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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/mechcad

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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 MECHCAD 2026 (Direct Download / Microsoft Store)
   How to install MECHCAD 2026 on Windows with the direct installer (free 30-day Premium trial), and about the Microsoft Store listing.
2. Welcome Window and 30-Day Premium Trial
   Start a free 30-day MECHCAD Premium trial from the Welcome window.
3. Activate a licence key
   Enter a licence key to activate MECHCAD on this computer.
4. MECHCAD Editions: Standard, Expert and Premium
   Compare what each MECHCAD Edition includes, and what you see when a feature is not in your Edition.
5. Start Center: Recent Documents (Start / Recent Documents)
   Open recent work with thumbnails, create a new part, or open a file from the Start window.
6. Create your first part in 5 minutes
   Practise the basic Sketch › Extrude workflow until you have your first 3D part.
7. Sample jobs (Demo) included with the program
   Open the sample Camera, PET Bottle, Modern Chair, Robot 6 Axis and CAM jobs.

Install MECHCAD 2026 (Direct Download / Microsoft Store) Standard

How to install MECHCAD 2026 on Windows with the direct installer (free 30-day Premium trial), and about the Microsoft Store listing.

What it does

MECHCAD 2026 is a 3D CAD/CAM program for Windows. The program is the same however you install it. The edition (Standard / Expert / Premium) depends on the activated licence, not on how you installed.

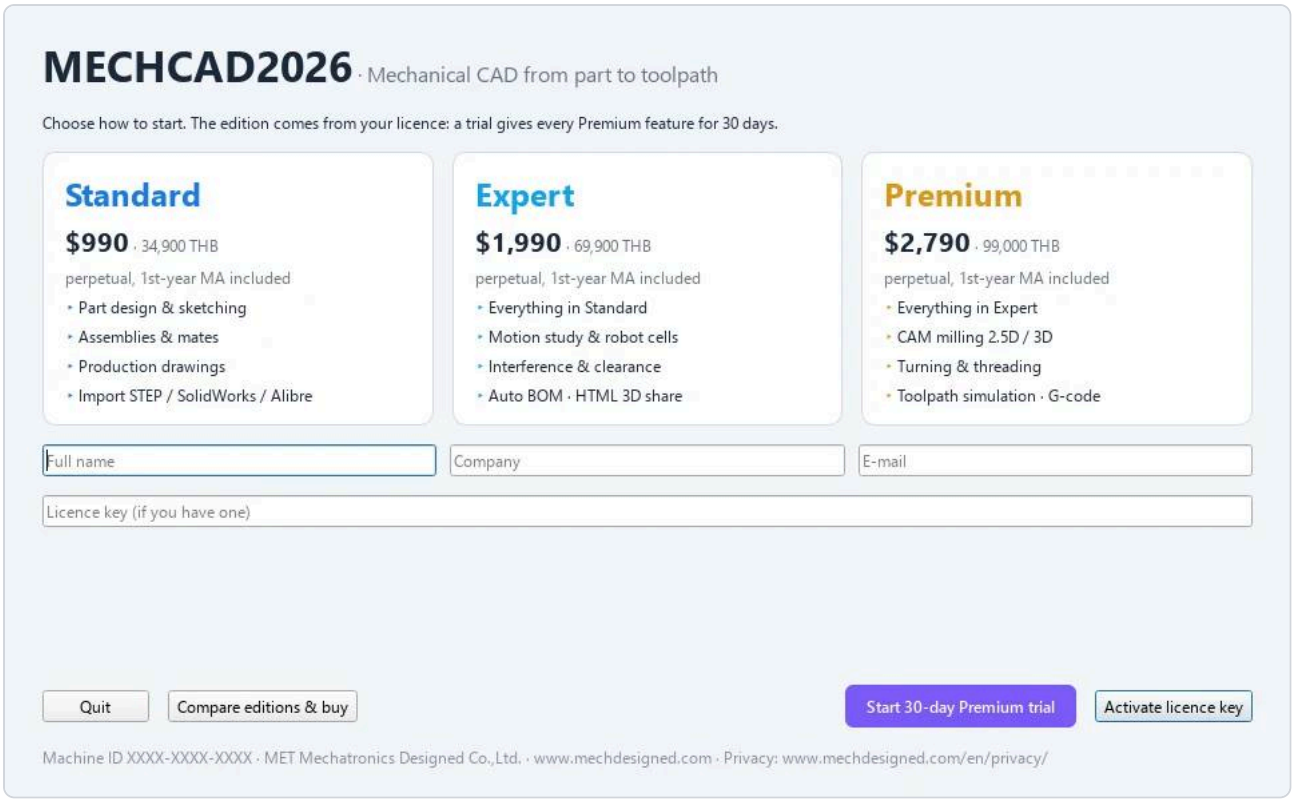
Channel	Best for	Updates
Direct download (recommended): https://license.mechdesigned.com/download/MECHCAD	Everyone: new users start a free 30-day Premium trial from the Welcome screen, MA customers activate their key	**Help › Check for Updates...** for MA customers, or download the latest installer from the same link
Microsoft Store	MECHCAD 2026 on the Microsoft Store	Updated through the Microsoft Store

Install with the direct installer

1. Download the installer from https://license.mechdesigned.com/download/MECHCAD (MA customers can also use the `MECHCAD2026-Setup-V<version>.exe` file they received from MET).
2. Double-click the file and answer **Yes** when Windows asks for administrator permission. The installer needs admin rights to register file types and Explorer thumbnails.
3. Choose the additional options (table below), then click **Install**.
4. When it finishes, tick **Launch MECHCAD** and click **Finish**.
5. The first time, the **Welcome** window appears. Click **Start 30-day Premium trial**, or enter your licence key. See Welcome window and 30-day trial.

Installer option	Meaning
Create a desktop shortcut	Creates an icon on the desktop.
Prefer high-performance graphics for MECHCAD	Makes Windows use the discrete GPU for MECHCAD on laptops with two GPUs. We recommend turning it on.

The program is installed in `%LOCALAPPDATA%\Programs\MECHCAD2026`. It registers these file types so you can open them by double-clicking: `.metprt` (Part), `.metasm` (Assembly), `.metdrw` (Drawing), `.metcam` (CAM), `.metauto` (Automation) and `.metpkg` (Pack and Go). It also registers file thumbnails in Windows Explorer.



Welcome window shown the first time after installation

Microsoft Store

You can also install MECHCAD 2026 from the Microsoft Store. It is the same program with the same licence, and “Start 30-day Premium trial” works the same. The Store copy updates through the Store and may trail the direct installer slightly.

System requirements

Item	Recommended
Operating system	Windows 10 / 11, 64-bit
Graphics card	OpenGL 3.2 or later (a discrete GPU makes large parts rotate more smoothly)
Memory	16 GB or more for large assemblies
Internet	Needed to activate the licence and for periodic checks (works offline within a 7-day grace period)

> **Note:** DWG open and save work right after installation: the DWG converter ships with the program, so you do not need to install the ODA File Converter (if ODA is installed, it is only used as a fallback). PDF and DXF also work immediately.

Try it: Set up a home-office laptop from scratch

Scenario: you want MECHCAD on your personal laptop tonight and to design a vertical laptop dock at the weekend.

1. Open https://license.mechdesigned.com/download/MECHCAD and save the installer.

2. Run it, answer **Yes** to the admin prompt, and tick **Prefer high-performance graphics for MECHCAD** (your laptop has two GPUs).
3. Click **Install**, then **Launch MECHCAD** › **Finish**.
4. In the Welcome window, enter your name, company and e-mail and click **Start 30-day Premium trial**.
5. Click **Demo** and open **MET Camera** to check that rotation is smooth. If it is not, see Display troubleshooting.

Challenge: Double-click any `.metprt` file in Explorer: it opens straight in MECHCAD with its thumbnail.

See also

- Welcome window and 30-day trial
- Activate with a license key
- MECHCAD editions
- Updates (Check for Updates)

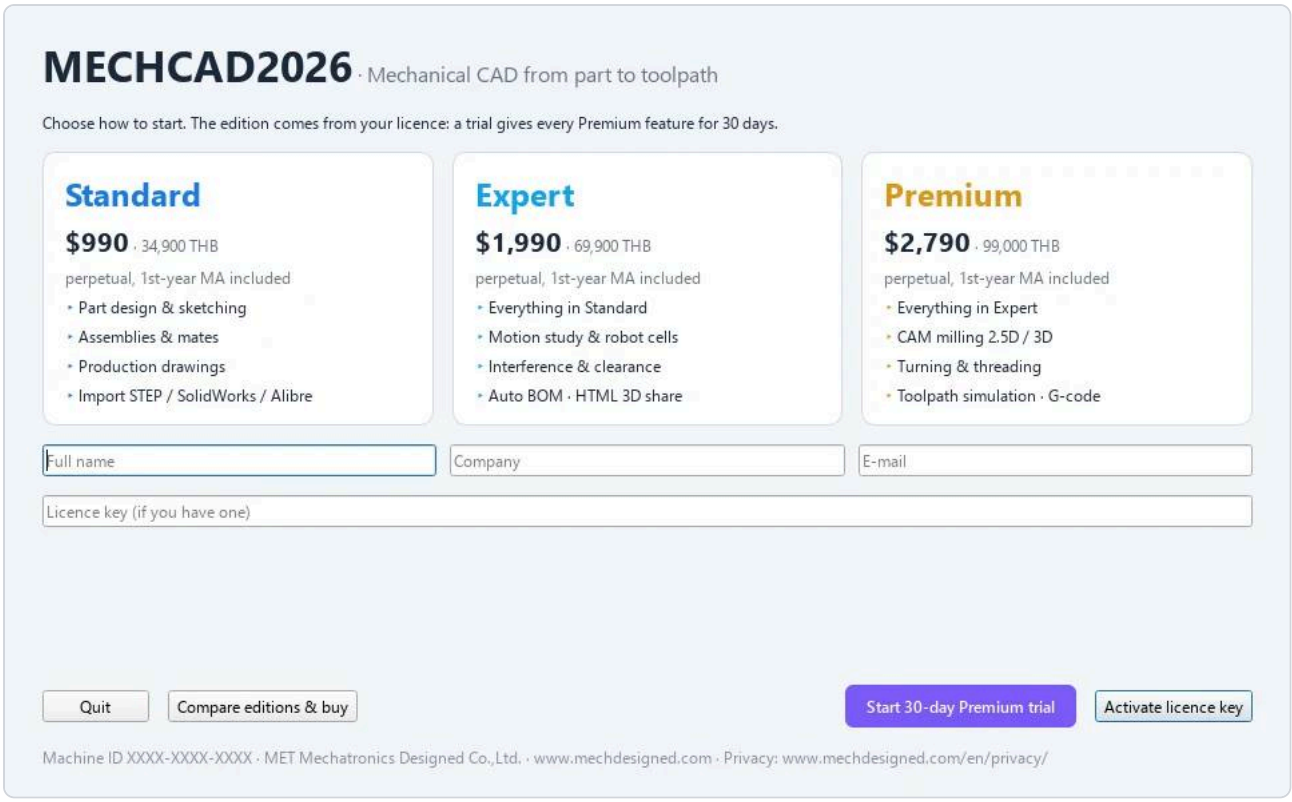
Welcome Window and 30-Day Premium Trial Standard

Start a free 30-day MECHCAD Premium trial from the Welcome window.

What it does

The **MECHCAD2026 — Welcome** window opens by itself when the computer has no licence (a new install, or an expired trial). You can open it at any time from the Edition badge at the top right, or from **Help › Activate License (online)...**. It has three ways in:

- **Start 30-day Premium trial** — try every Premium feature for 30 days
- **Activate licence key** — enter a license key you have bought. See Activate with a license key.
- **Compare editions & buy** — opens mechdesigned.com to compare and order



Welcome window: edition cards, input fields and the trial button

Start the 30-day trial

1. Open MECHCAD for the first time. The Welcome window appears.
2. Enter your **Full name**, **Company** and **E-mail**.
3. Click **Start 30-day Premium trial** (the purple button).
4. Wait for the message *Requesting your 30-day trial from the MET licence server…*. When it succeeds, the program opens with the **MECHCAD Trial · N days left** badge at the top right.

Field / button	Meaning
Full name	User name (required for the trial)
Company	Company name
E-mail	The e-mail used to request the trial (required)
Licence key (if you have one)	Enter it only if you already have a license key
Quit	Exits the program (on first start) / Close when opened from the menu
Compare editions & buy	Opens the website for prices and ordering
Start 30-day Premium trial	Requests a 30-day trial licence from the MET server
Activate licence key	Activates with a license key

> **Note:** The trial is available **once per computer and per e-mail**. If you have used it already, you see *A free trial was already used on this computer or with this e-mail.* Contact MET to buy a licence or ask for an extension.

> **Note:** During the trial, the badge at the top right counts the days left and turns orange when 7 days or fewer remain. When the trial expires, the next time you start the program it returns to the Welcome window so you can enter a license key.

Try it: A weekend trial with a real project

Scenario: you want to know if MECHCAD suits you, and you have a bookend design in mind.

1. Start MECHCAD for the first time. The Welcome window opens.
2. Enter **Full name**, **Company** (your own name is fine for a freelancer) and **E-mail**.
3. Click **Start 30-day Premium trial** and wait for the message from the licence server.
4. The badge now reads **MECHCAD Trial · N days left**. Every feature is open, including Simulation and CAM.
5. Try the bookend: an L shape **120 × 100** mm, **4** mm thick, **Extrude 100** mm wide. Then try **Drawing** and **CAM** on the same part to see what each edition gives.

Challenge: Click the badge at any time to open the Welcome window again and compare the editions.

See also

- Activate with a license key
- MECHCAD editions
- License management

Activate a licence key Standard

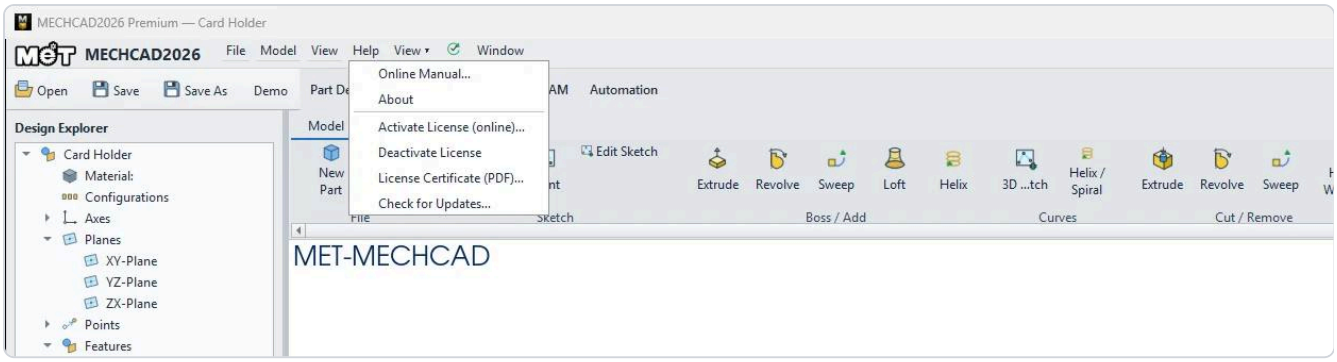
Enter a licence key to activate MECHCAD on this computer.

What it does

Activates this computer with a licence key issued by MET (for example `MECHCAD-XXXXX-XXXXX-XXXXX-XXXXX`). The licence determines which Edition you can use. You can change Edition later without changing the key.

Where to find it

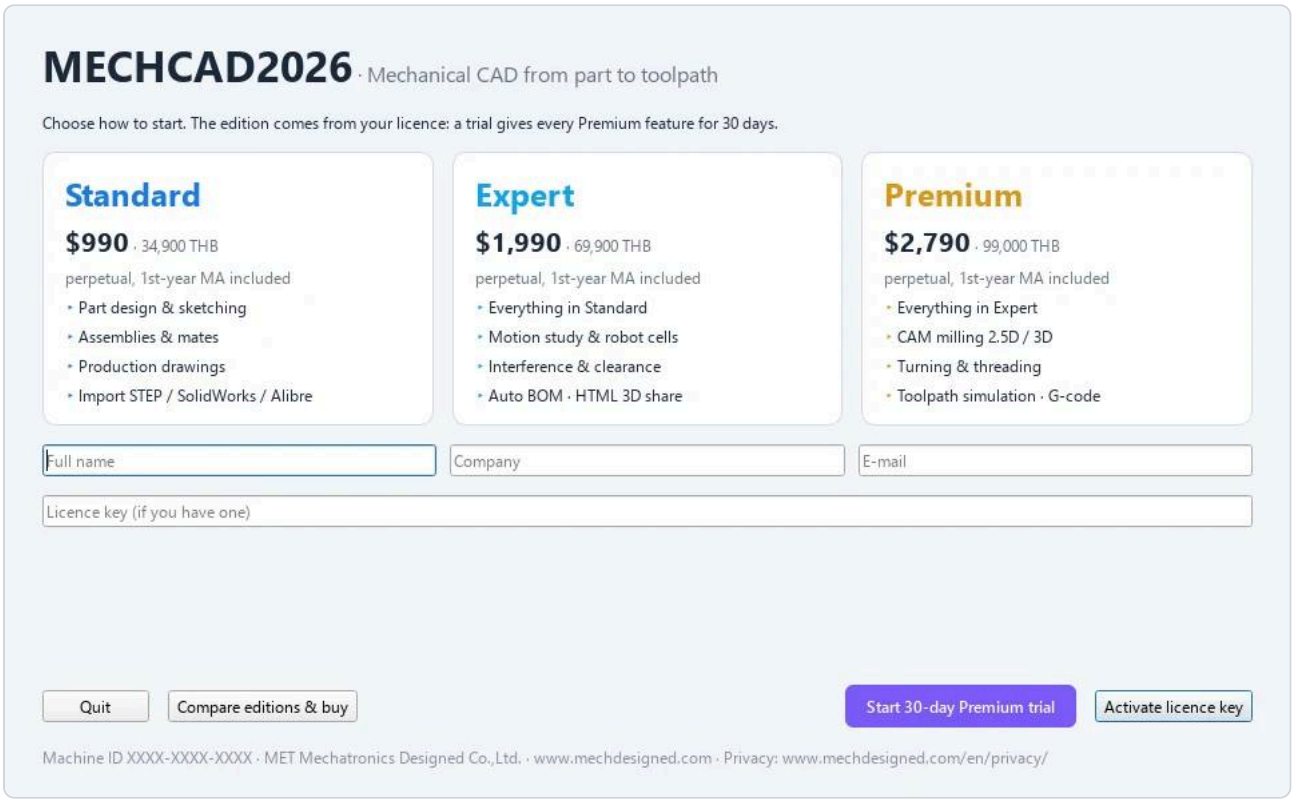
- The **Welcome** window (opens automatically when there is no licence yet)
- **Help › Activate License (online)...**
- Click the Edition badge at the top right of the window (for example ● **MECHCAD Premium · Perpetual**)



Help menu: Activate / Deactivate License, License Certificate and Check for Updates

Steps

1. Open the Welcome window.
2. Enter **Full name**, **Company** and **E-mail**. The e-mail must match the one MET used to issue the key.
3. Enter the key in the **Licence key (if you have one)** field.
4. Click **Activate licence key** (or press Enter in the key field).
5. When activation succeeds, the window closes and the badge at the top right shows your Edition.



Welcome window for entering a licence key

Messages you may see

Message	Cause / fix
Enter full name, company and the e-mail the key was issued to.	Required fields are incomplete.
This license key has expired.	The key has expired. Contact MET.

Message	Cause / fix
All N seats are in use. Release one from another PC first.	All seats are in use. Use **Deactivate License** on a previous computer first.
Offline grace elapsed — reconnect to the license server.	The computer has been offline for more than 7 days. Connect to the internet and restart the program.

> **Note:** The program checks the licence with the server every 5 minutes in the background. If there is no internet connection, you can keep working during the 7-day grace period.

Try it: Activate a new office laptop in two minutes

Scenario: you received the key `MECHCAD-XXXXX-XXXXX-XXXXX-XXXXX` by e-mail at `you@company.com` and want to start designing a desk nameplate today.

1. Start MECHCAD. The Welcome window opens because the laptop has no licence yet.
2. Type your **Full name**, **Company**, and **you@company.com**, the e-mail the key was sent to. Another address does not work.
3. Paste the key into **Licence key (if you have one)** and press **Enter**.
4. The window closes. The badge at the top right now shows your edition, for example ● **MECHCAD Standard · Perpetual**.
5. To check it works, sketch a **150 × 40** rectangle, **Extrude** it **8** mm and save it as `nameplate.metprt`.

Challenge: Choose **Help › License Certificate (PDF)…** and keep the certificate with your purchase records.

See also

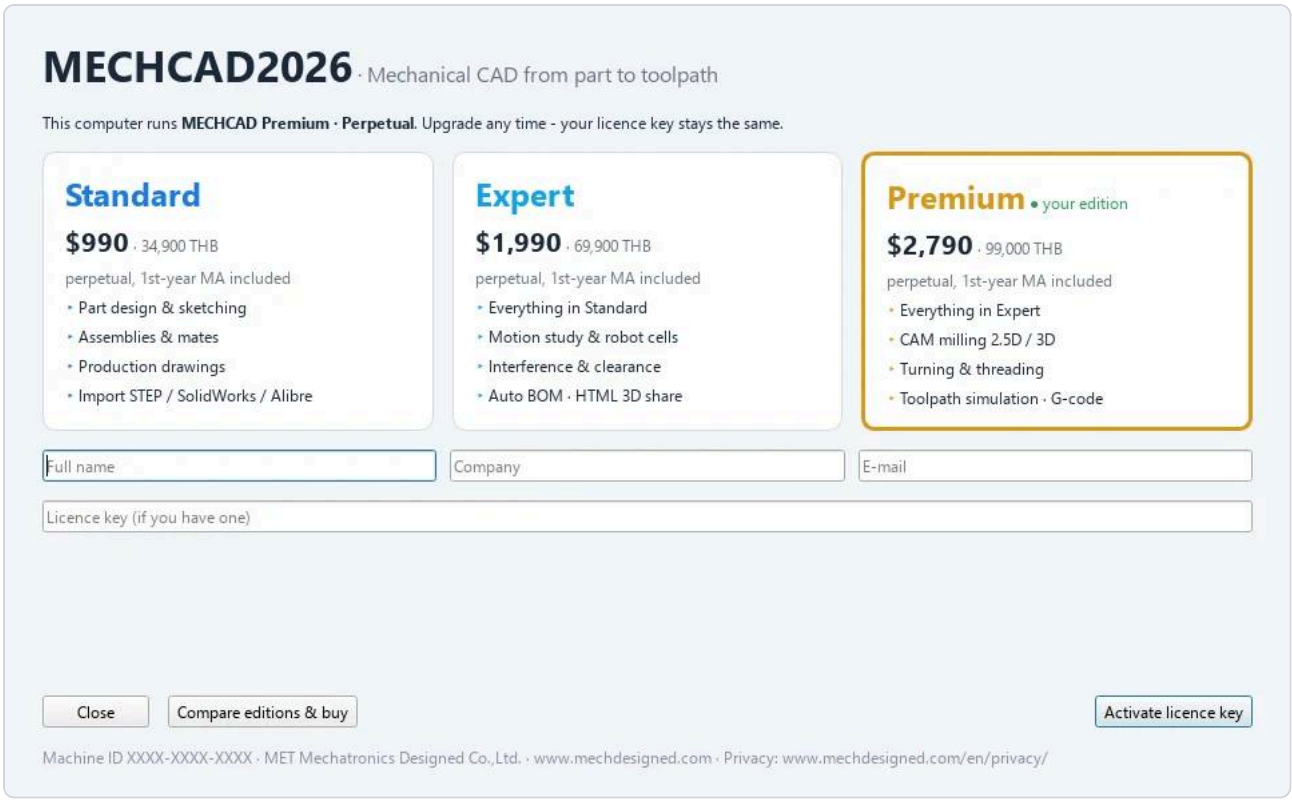
- Welcome window and 30-day trial
- Manage licences (move computer / certificate)
- Licence troubleshooting

MECHCAD Editions: Standard, Expert and Premium Standard

Compare what each MECHCAD Edition includes, and what you see when a feature is not in your Edition.

What it does

MECHCAD is a single program. Your licence determines which Edition this computer uses. You do not choose the Edition yourself, and you can upgrade with the same licence key.



Edition comparison cards in the Welcome window (● your edition is the current Edition)

Compare Editions

Capability	Standard	Expert	Premium
Part Design and Sketch (parameters and equations)	✓	✓	✓
Import STEP / IGES / SolidWorks / Alibre / Siemens NX	✓	✓	✓
Assembly (mates, sub-assemblies, explode)	✓	✓	✓
Drawing (views, dimensions, BOM, export PDF/DXF/DWG)	✓	✓	✓
Surface modeling	–	✓	✓
Sheet metal (flat pattern, DXF for laser cutting)	–	✓	✓
Simulation: Motion Study, Motion Studio, Automation workbench, video/GIF recording	–	✓	✓
Assembly analysis: Interference, Clearance, Auto BOM, Export HTML	–	✓	✓
CAM: 2.5D / 3D milling, turning and threading, machining simulation, G-code	–	–	✓
30-day trial	Every feature, same as Premium		

Prices

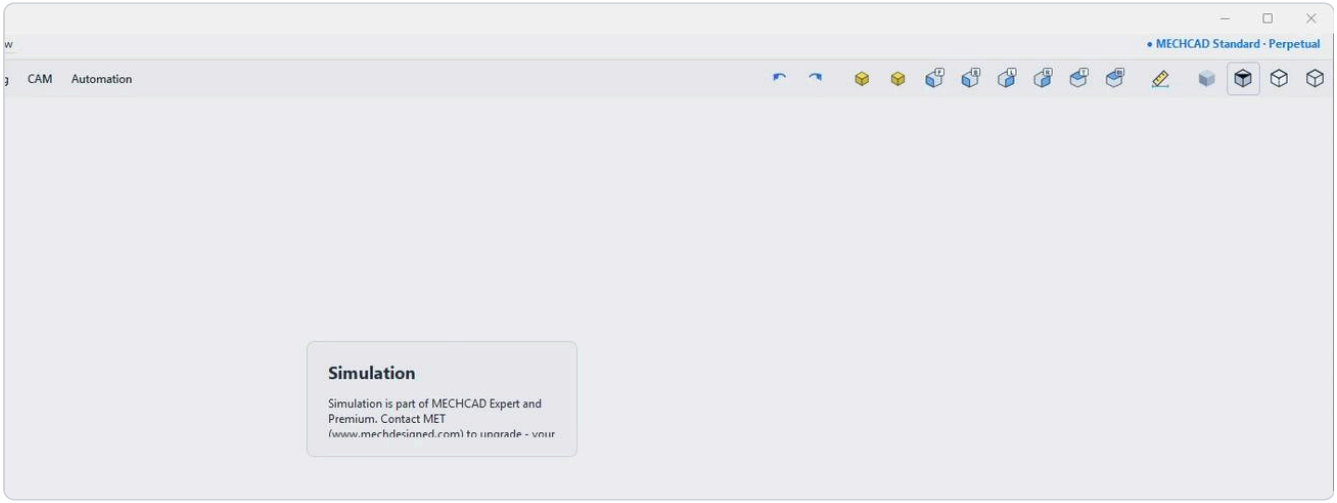
Prices are per seat (one computer at a time). Every THB price **includes 7% VAT**.

Item	Standard	Expert	Premium
Perpetual, first year of MA included	$990 / ฿34,900	$1,990 / ฿69,900	$2,790 / ฿99,000
MA renewal per year (from year 2)	$290 / ฿9,900	$450 / ฿15,900	$690 / ฿23,900
Annual subscription	$390 / ฿13,900	$790 / ฿27,900	$1,190 / ฿39,900
Network (floating) licence	+25%	+25%	+25%

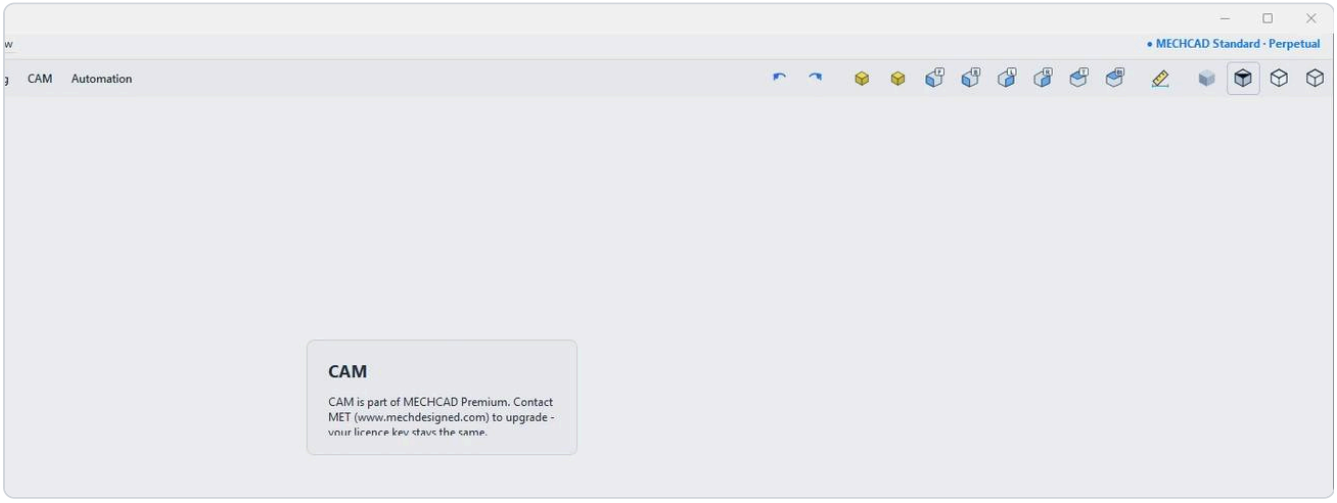
Upgrading pays only the price difference: Standard → Expert $1,000 (฿35,000 incl. VAT) and Expert → Premium $800 (฿29,100 incl. VAT), with the same licence key. Current prices are on the MECHCAD pricing page.

When you use a feature that is not in your Edition

- The **Automation** and **CAM** tabs are still visible, but show a card saying which Edition includes the feature.
- Locked buttons (for example Interference, Auto BOM, Motion Studio) show a message in the status bar and in the tooltip, such as *Simulation is part of MECHCAD Expert and Premium. Contact MET (www.mechdesigned.com) to upgrade - your licence key stays the same.*



Automation tab on Standard: upgrade card



CAM tab on Standard

> **Note:** The badge at the top right always shows your Edition and licence term, for example *MECHCAD Premium · Perpetual*, *MECHCAD Trial · 12 days left* or *MECHCAD Expert · until 2027-01-31*.

Try it: Pick the right edition for a laptop-stand project

Scenario: you design aluminium laptop stands. Use this page to decide which edition you need, step by step.

1. Drawing the stand, its parts and the drawings for the workshop: **Standard** ($990 / 34,900 THB) is enough.
2. You want to animate the folding hinge with a Motion Study and check the parts for interference: you need **Expert** ($1,990 / 69,900 THB).
3. You want to mill the aluminium plates yourself and make G-code: you need **Premium** ($2,790 / 99,000 THB).
4. Not sure yet? Start the 30-day trial: it includes every Premium feature, so you can try all three steps first.
5. On a Standard licence, click the **CAM** tab: you see the card that tells you which edition includes it, and nothing is lost if you upgrade later, because the key stays the same.

Challenge: Write down which of the three steps your next project really needs before you contact MET.

See also

- Welcome window and 30-day trial
- Activate a licence key
- Manage licences

Start Center: Recent Documents (Start / Recent Documents) Standard

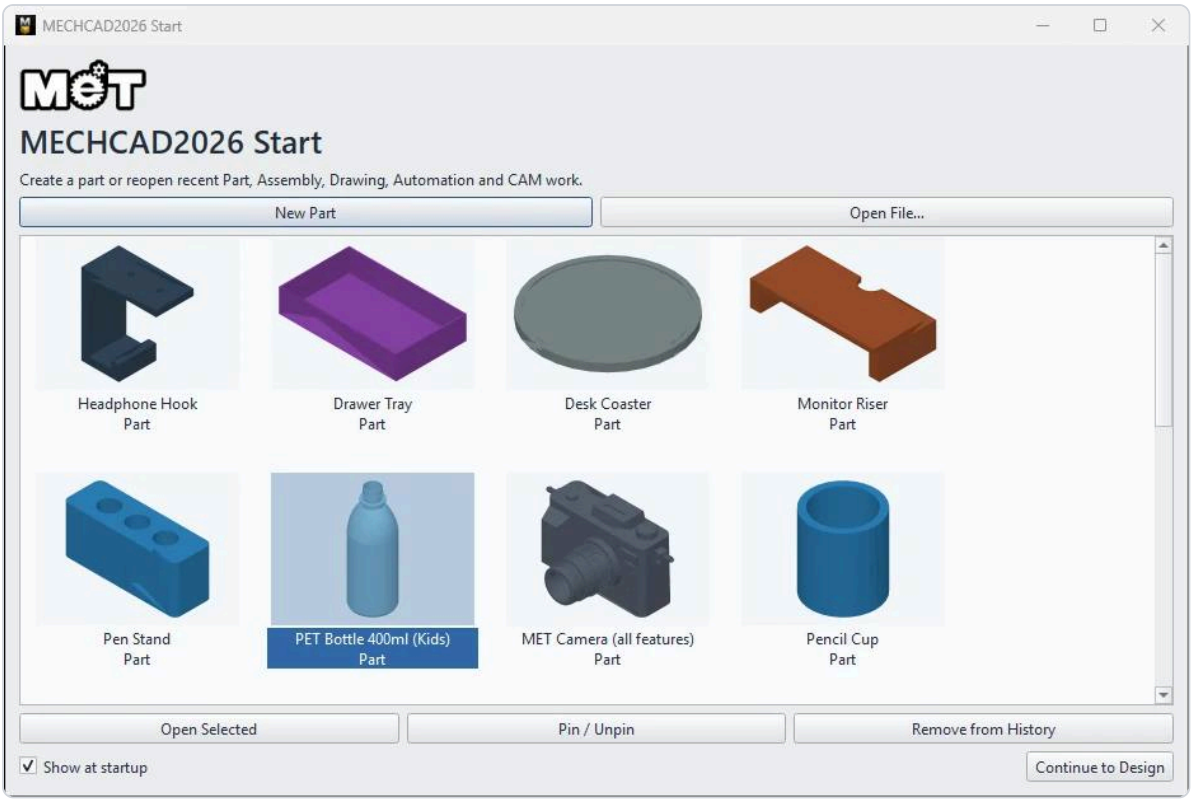
Open recent work with thumbnails, create a new part, or open a file from the Start window.

What it does

The **MECHCAD2026 Start** window shows the documents you opened recently (Part, Assembly, Drawing, Automation and CAM) with thumbnails, so you can get back to work quickly. Opening a file from the list does not overwrite the original file.

Where to find it

- Opens by itself when the program starts (turn it off with **Show at startup**)
- **File › Start / Recent Documents…**



MECHCAD2026 Start window with the recent documents list

Steps

1. Double-click the document thumbnail you want, or select it and click **Open Selected**.
2. To start new work, click **New Part**, or click **Open File...** to choose another file.
3. Click **Continue to Design** to minimize this window and go back to the main window.

Button / option	Meaning
New Part	Starts a new part
Open File...	Opens any MECHCAD file type (`.metprt .metasm .metdrw .metcam .metauto .metpkg`)
Open Selected	Opens the selected document
Pin / Unpin	Pins a document (marked with ★ and always at the top)
Remove from History	Removes it from the list (the actual file is not deleted)
Show at startup	Shows this window every time the program starts
Continue to Design	Returns to the main window

> **Note:** The list keeps the 40 most recent documents plus any pinned ones. If a file has been moved, it shows *File is unavailable at its saved location*.

Try it: Keep your desk-calendar stand one click away

Scenario: you work on a desk-calendar stand all week and want it at the top of the list every morning.

1. Make a quick part: sketch a rectangle **120 × 60**, **Extrude 10** mm and save it as `desk-calendar-stand.metprt`. Close it.
2. Choose **File › Start / Recent Documents…**. The stand is in the list with its thumbnail.
3. Select it and click **Pin**. It gets a ★ and stays at the top.
4. Select an old test file you no longer need and click **Remove from History** (the file itself is not deleted).
5. Make sure **Show at startup** is ticked. Close MECHCAD and start it again: the pinned stand is the first thing you see. Double-click it.

Challenge: Move the `.metprt` to another folder and open the Start window again: you see *File is unavailable at its saved location*.

See also

- MECHCAD file types
- The MECHCAD main screen

Create your first part in 5 minutes Standard

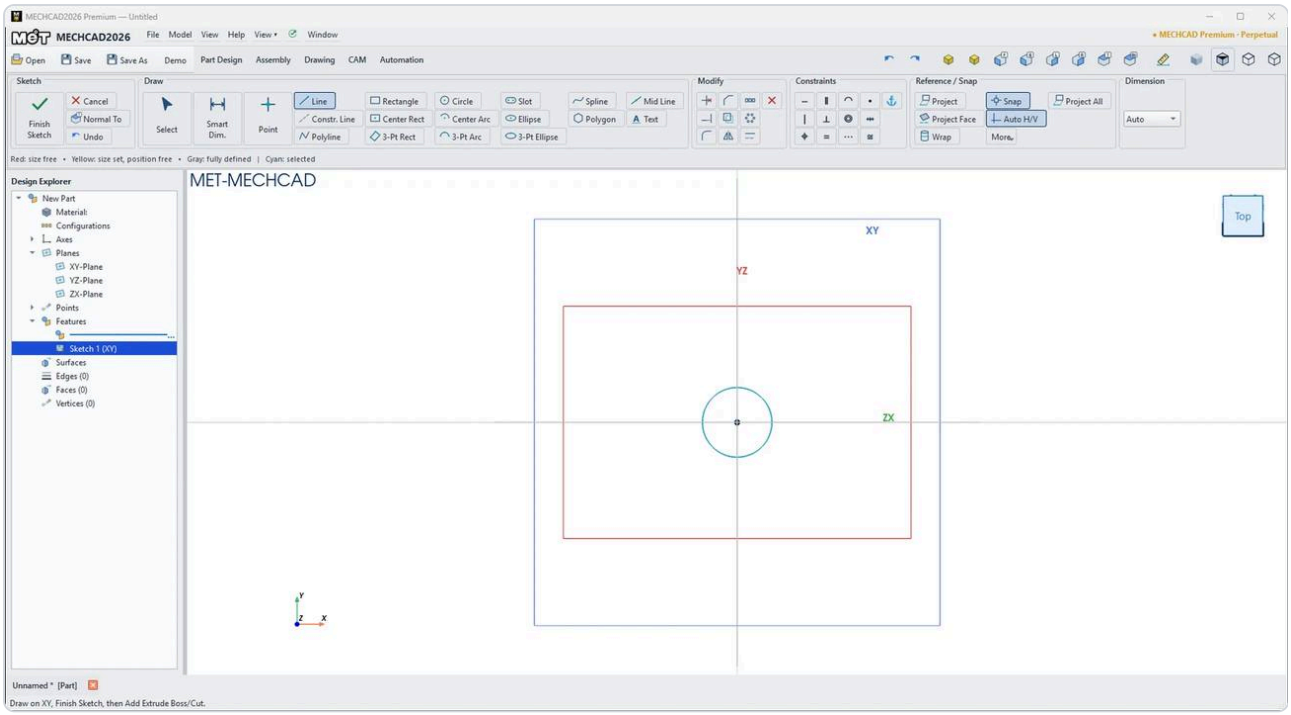
Practise the basic Sketch › Extrude workflow until you have your first 3D part.

What it does

A short tutorial that builds a rectangular plate with a hole in the middle. It shows the main MECHCAD workflow: choose a plane › draw a sketch › Finish Sketch › Extrude.

Steps

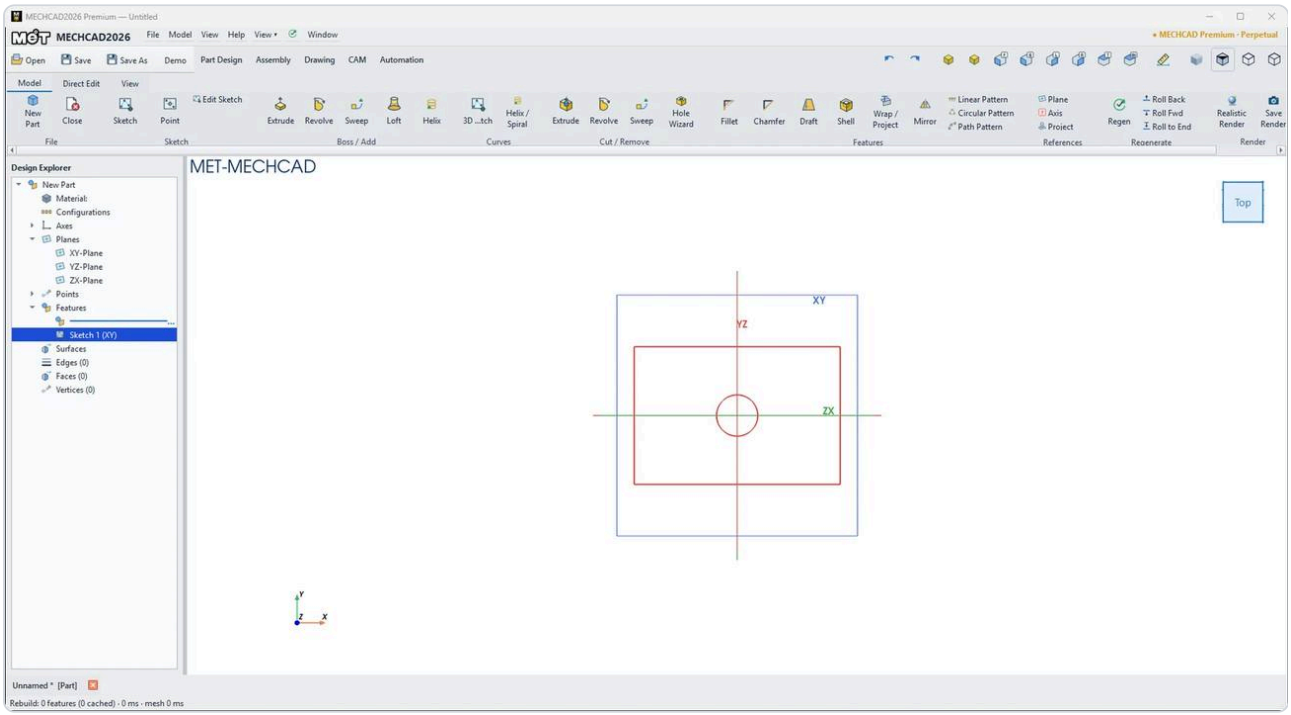
1. On the **Part Design** tab, click **Model › Sketch › Sketch** (or double-click **XY-Plane** in the Design Explorer). Choose the **XY** plane. You can press Enter straight away if you want XY.
2. Click **Rectangle**. Click the first corner, then the opposite corner.
3. Click **Circle**. Click the centre point inside the rectangle, then click again to set the radius.



Rectangle and circle drawn on the XY plane

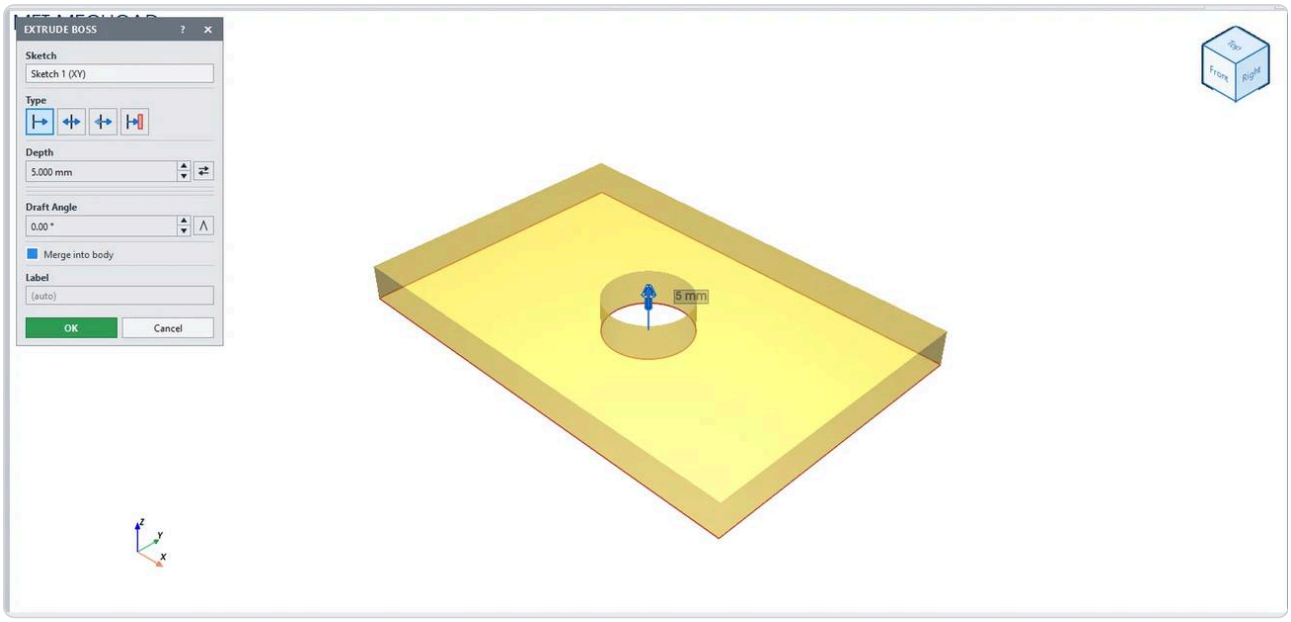
4. (Optional) Use **Smart Dim.** to set real sizes, for example width 60, height 40 and Ø12. See Smart Dimension.

5. Click **Finish Sketch** (the green tick). The sketch appears in the Design Explorer.



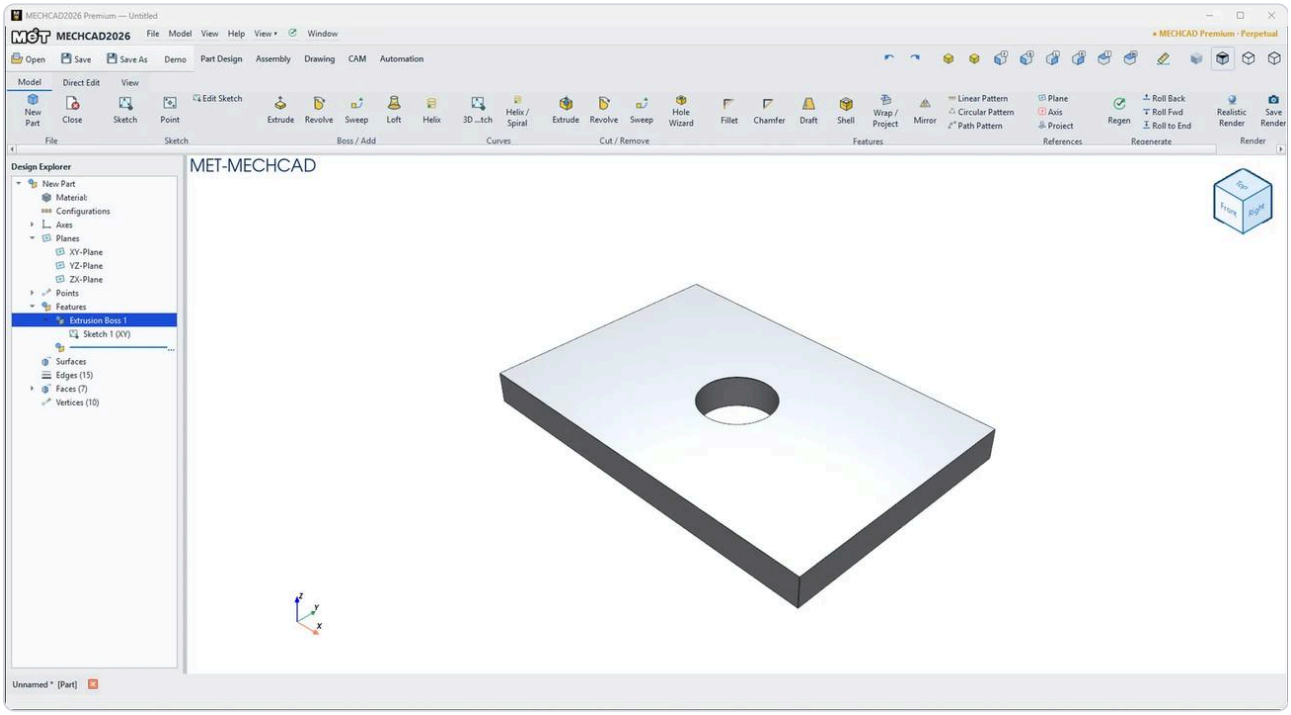
After Finish Sketch you have Sketch 1 (XY)

6. Select the sketch, then click **Model › Boss / Add › Extrude**. The **EXTRUDE BOSS** box opens with a yellow preview. Enter a **Depth** (for example 5 mm), or drag the arrow in the view.



EXTRUDE BOSS box with preview

7. Click **OK** to create **Extrusion Boss 1** in the Design Explorer. The inner circle becomes a hole automatically.



The first part in the Isometric view

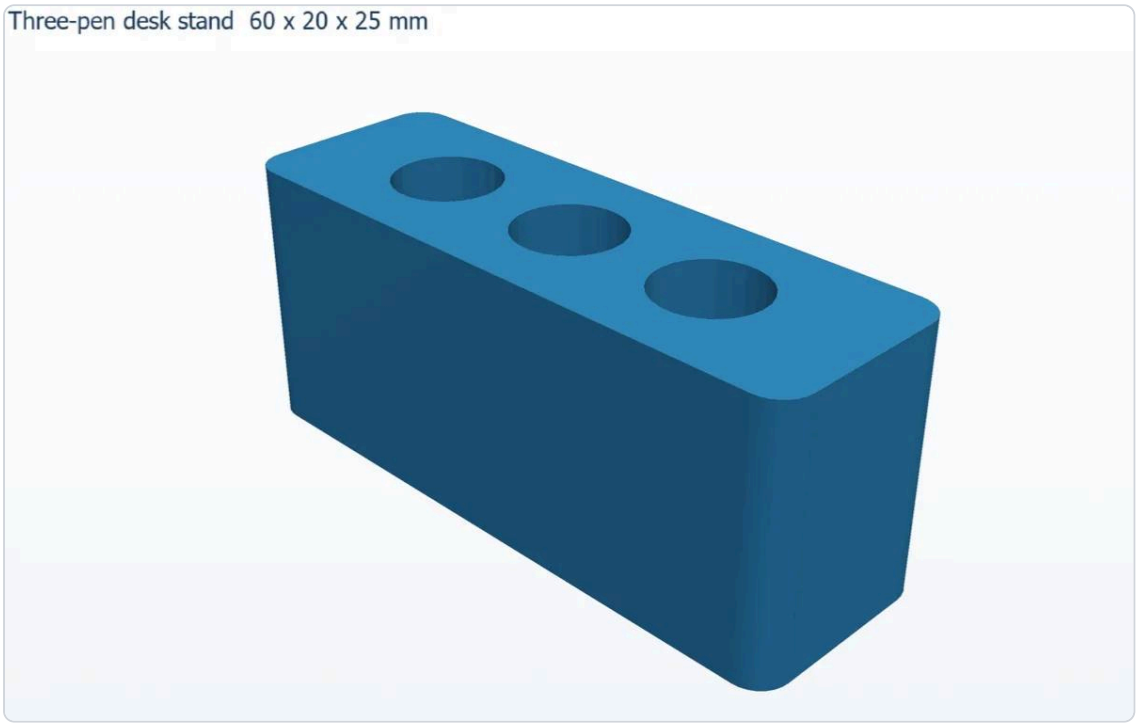
8. Click **Save** (Ctrl+S) to save as `.metprt`.

> **Note:** Every step can be undone with **Undo** (Ctrl+Z). You can edit it later by double-clicking the feature in the Design Explorer.

Try it: Three-pen desk stand

Turn the plate-with-a-hole idea into something you will actually use: a small block that keeps three pens standing upright next to your keyboard.

1. On the **Part Design** tab, start a sketch on the **XY** plane.
2. Press **R** (Rectangle), click the origin, type **60**, press **Tab**, type **20**, then press **Enter**. Press **Esc** twice.
3. Press **C** (Circle), click inside the rectangle, type **10** (diameter) and press **Enter**. Make three circles in a row.
4. Press **D** (Smart Dimension) and place the hole centres: **15**, **30** and **45** mm from the left edge, and each one **10** mm from the bottom edge.
5. Click **Finish Sketch**, then **Model › Boss / Add › Extrude** with **Depth 25** mm and click **OK**. The three circles become through-holes.
6. Click **Model › Features › Fillet**, enter **Radius 3**, click the four vertical corner edges, then click **Done**.
7. Press **Ctrl+S** and save it as `pen-stand.metprt`.



Finished three-pen desk stand, 60 x 20 x 25 mm

Challenge: Your markers are thicker? Double-click the sketch in the Design Explorer, change the three diameters to **12** mm and watch the stand update.

See also

- Sketch overview
- Extrude Boss
- Design Explorer

Sample jobs (Demo) included with the program Standard

Open the sample Camera, PET Bottle, Modern Chair, Robot 6 Axis and CAM jobs.

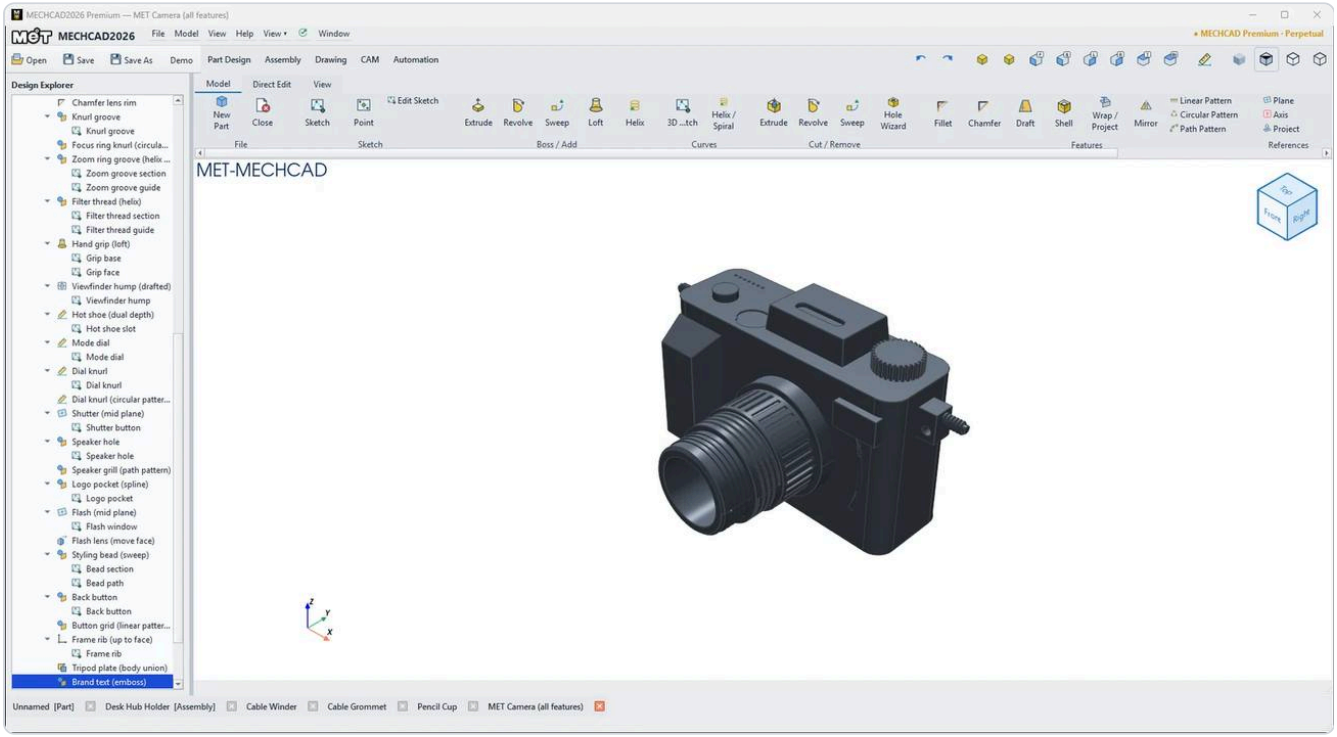
What it does

The **Demo** button on the top toolbar opens sample jobs for the tab you are using. It is a good way to learn every command from real work.

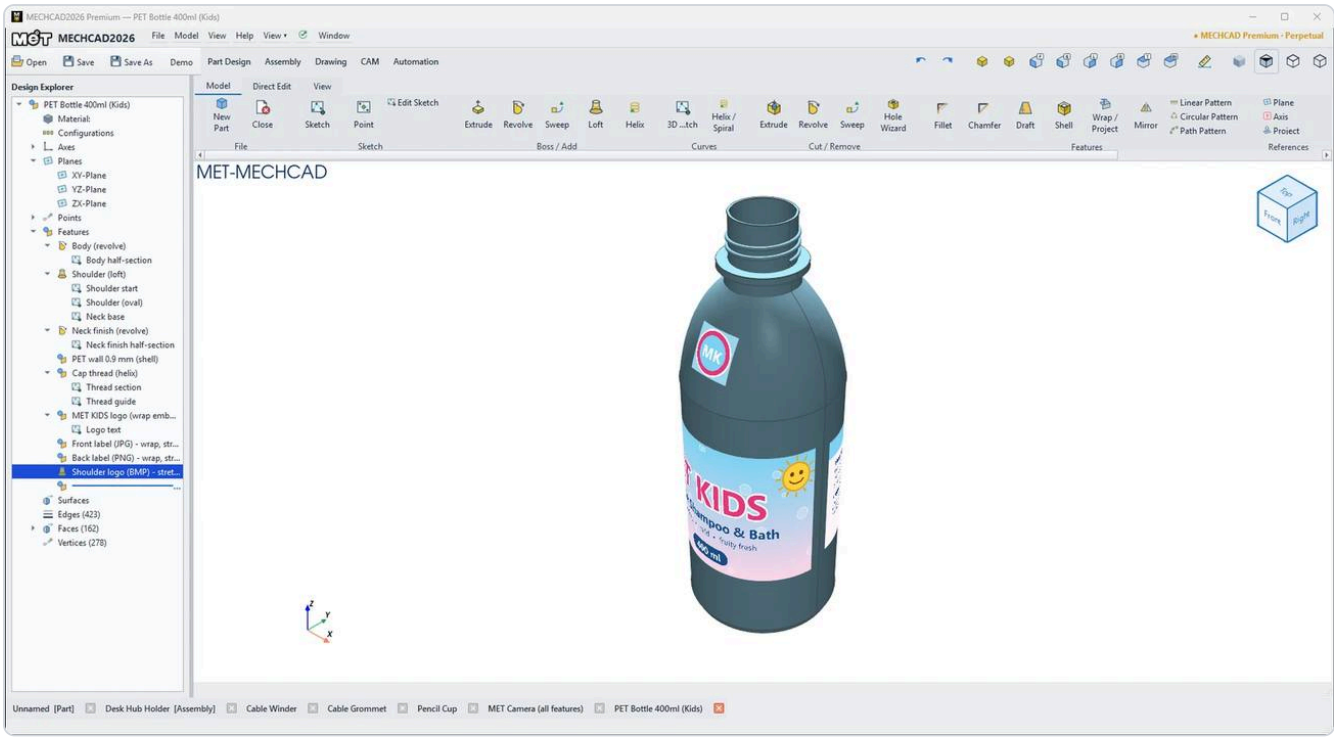
Where to find it

Top bar › **Demo** (next to Open / Save / Save As). The list depends on the tab that is open.

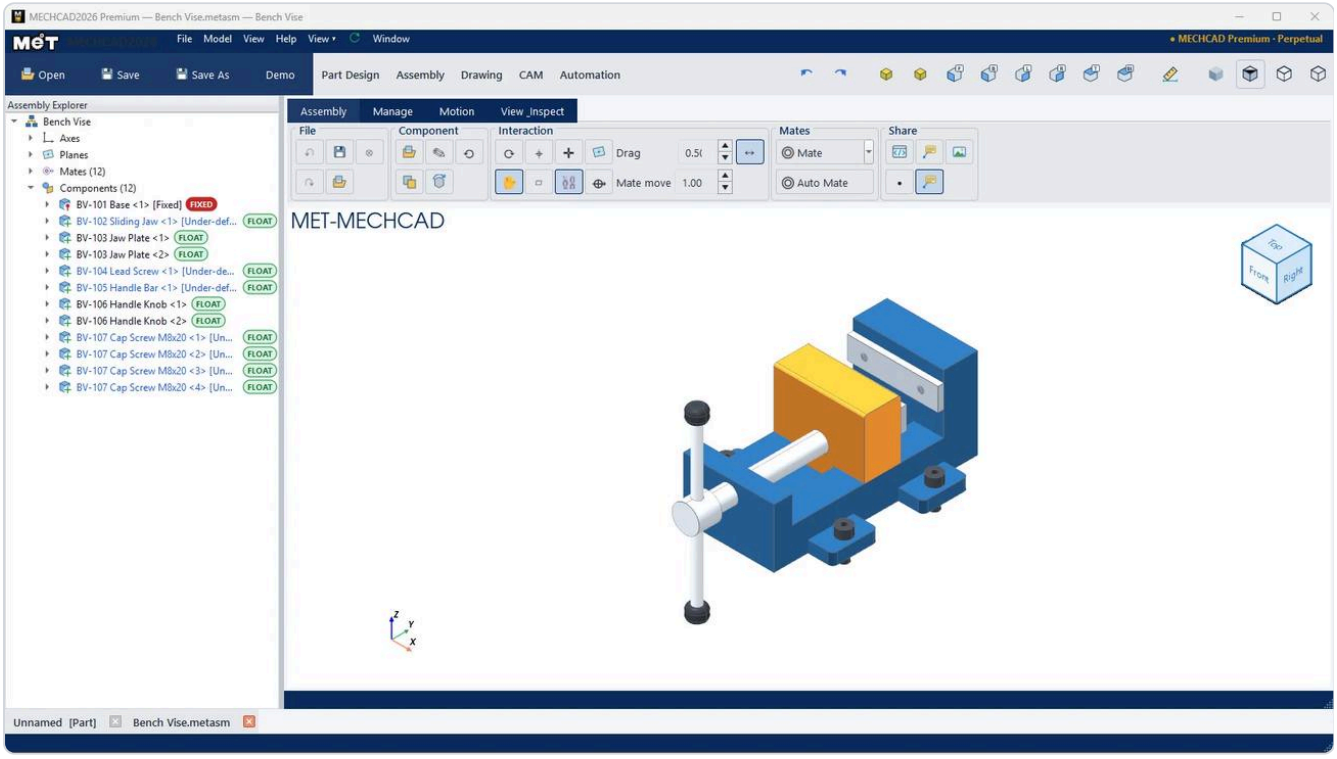
Sample	Tab	What you learn	Edition
MET Camera (all features)	Part Design	Every feature type in a single part	Standard
PET Bottle (revolve, loft, thread, wrap labels)	Part Design	Revolve, Loft, threads, Wrap emboss, label artwork	Standard
Modern Chair (parts, assembly, drawing, BOM, render)	Part Design / Assembly / Drawing	Assembly, Drawing with BOM, Render	Standard
Robot 6 Axis	Assembly	An assembly with joint mates	Expert
Mill demos / Lathe demos (5 each)	CAM	Every milling and turning operation	Premium



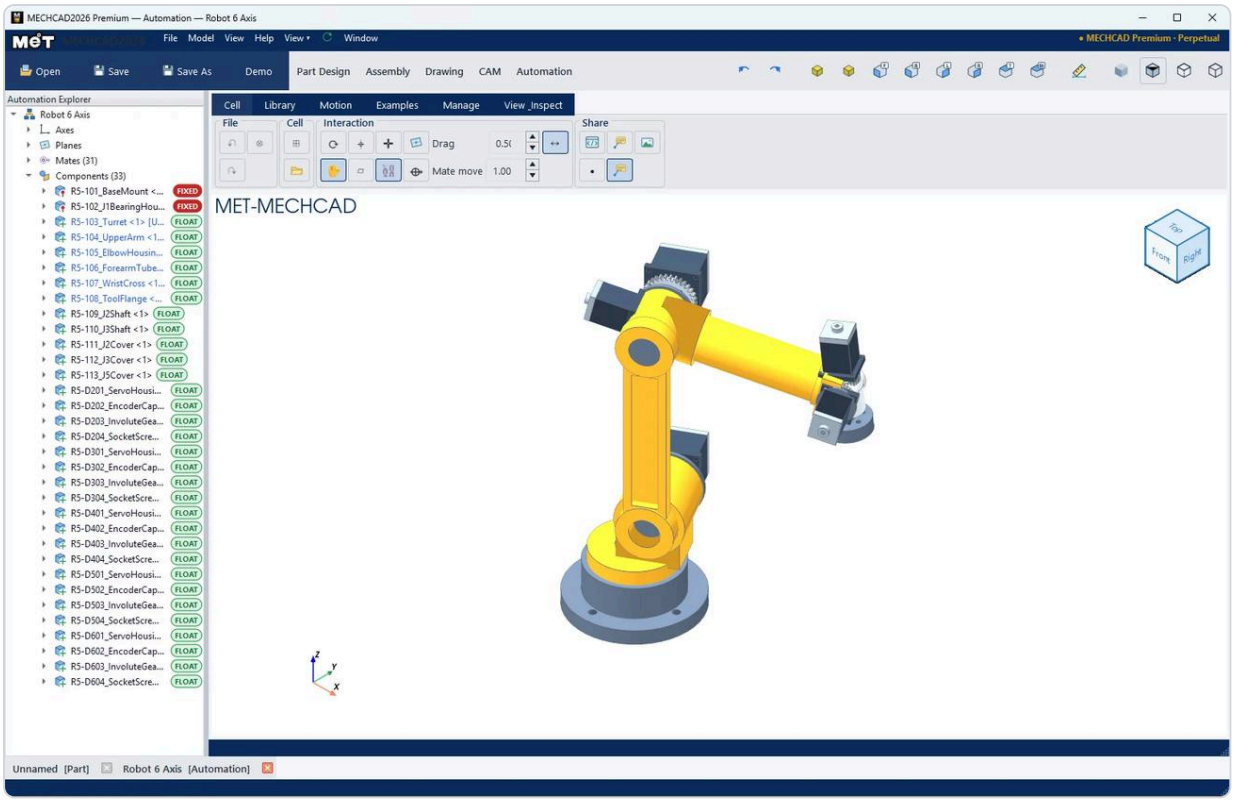
MET Camera: a sample that uses every feature type



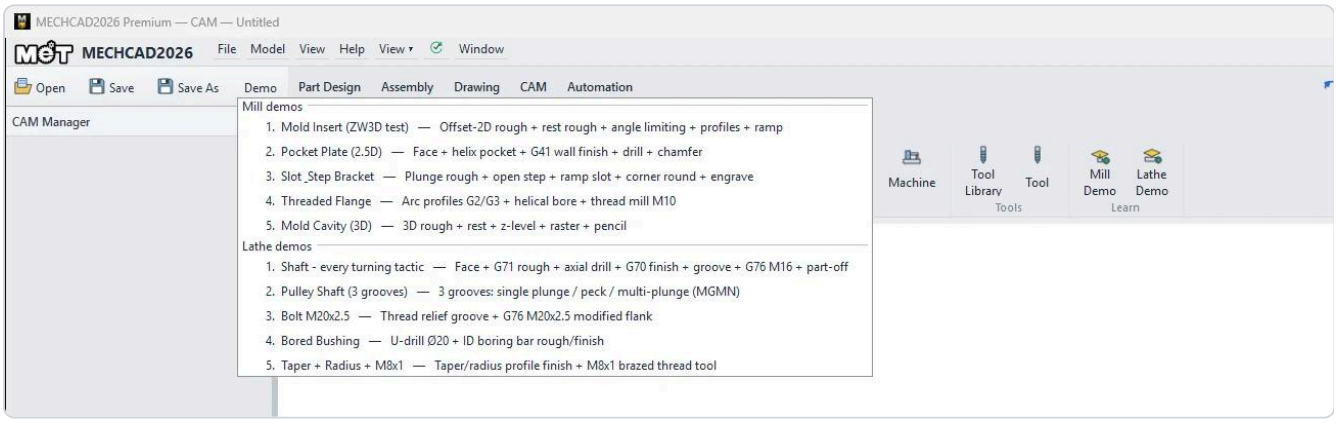
PET Bottle: Revolve, Loft, thread and label artwork



Modern Chair in the Assembly tab



Robot 6 Axis demo



Sample CAM jobs menu (Mill demos / Lathe demos)

Steps

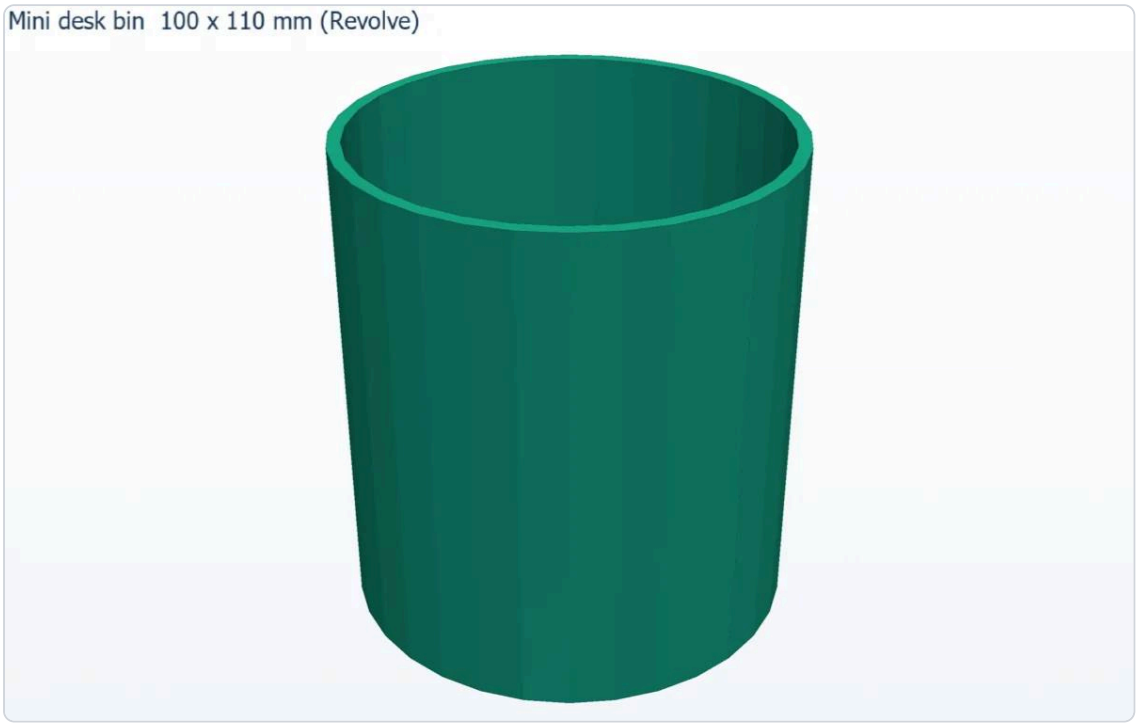
1. Choose a tab (Part Design, Assembly, Drawing, CAM or Automation).
2. Click **Demo** and choose a sample from the list.
3. The sample file is copied to `Documents\MECHCAD Demos` and opened in a new tab. You can edit it without affecting the original.

> **Note:** The sample CAM jobs are read-only. To keep one, use **Save CAM As** and save it to another folder.

Try it: Mini desk bin with the PET Bottle trick

The PET Bottle demo is built around one idea: draw half the outline, then spin it around an axis. Borrow the trick to make a 100 mm desk bin for paper scraps.

1. On the **Part Design** tab, click **Demo** and open **PET Bottle**. In the Design Explorer, right-click its Revolve feature and choose **Edit Child Sketch**: you see only half the outline, sitting next to the axis. Click **Finish Sketch** without changing anything.
2. Press **Ctrl+N** for a new part and start a sketch on the **ZX** plane (X points right, Z points up).
3. Press **L** (Line), click the origin, and type each segment as length, **Tab**, angle, **Enter**: **50 / 0°**, **110 / 90°**, **3 / 180°**, **106 / 270°**, **47 / 180°**. Finish by clicking the origin to close the outline (the last 4 mm line forms the bottom).
4. Click **Finish Sketch**, then **Model › Boss / Add › Revolve**. Choose **Axis Z**, **Angle 360**, and click **OK**.
5. Save it as `desk-bin.metprt`.



Mini desk bin made with Revolve, 100 mm across and 110 mm tall

Challenge: Edit the Revolve and set **Angle 180** to cut the bin in half and see the 3 mm wall and 4 mm floor.

See also

- Create your first part in 5 minutes
- CAM overview

Chapter 1 quiz: Getting started

Choose the one best answer. Answers are in Appendix A

1. **You install MECHCAD on a laptop that has two graphics cards. Which installer option should you turn on?**

   A. Create a desktop shortcut
   B. Prefer high-performance graphics for MECHCAD
   C. Launch MECHCAD
   D. Register Explorer thumbnails only

2. **How many times can the free 30-day Premium trial be used?**

   A. Once per month on the same computer
   B. Unlimited, as long as you enter a new name
   C. Once per computer and per e-mail
   D. Twice per computer

3. **Your licence is activated but the computer has no internet connection. How long can you keep working?**

   A. Only 5 minutes
   B. 1 day
   C. 30 days
   D. Within the 7-day offline grace period

4. **In the Start window (Recent Documents), what does Remove from History do?**

   A. Removes the entry from the list; the actual file is not deleted
   B. Deletes the file from the disk
   C. Moves the file to the Recycle Bin
   D. Pins the document to the top of the list

5. **You sketch a rectangle with a circle inside it and Extrude the sketch. What happens to the circle?**

   A. It becomes a separate cylinder
   B. It becomes a hole automatically
   C. It is ignored; you must cut it with Extrude Cut
   D. Extrude refuses to run until the circle is deleted

6. **Which Edition includes the CAM workbench (toolpaths, machining simulation, G-code)?**

   A. Standard and above
   B. Expert and Premium
   C. Premium only (and the trial)
   D. None; CAM is a separate program

CHAPTER 2

Interface & settings

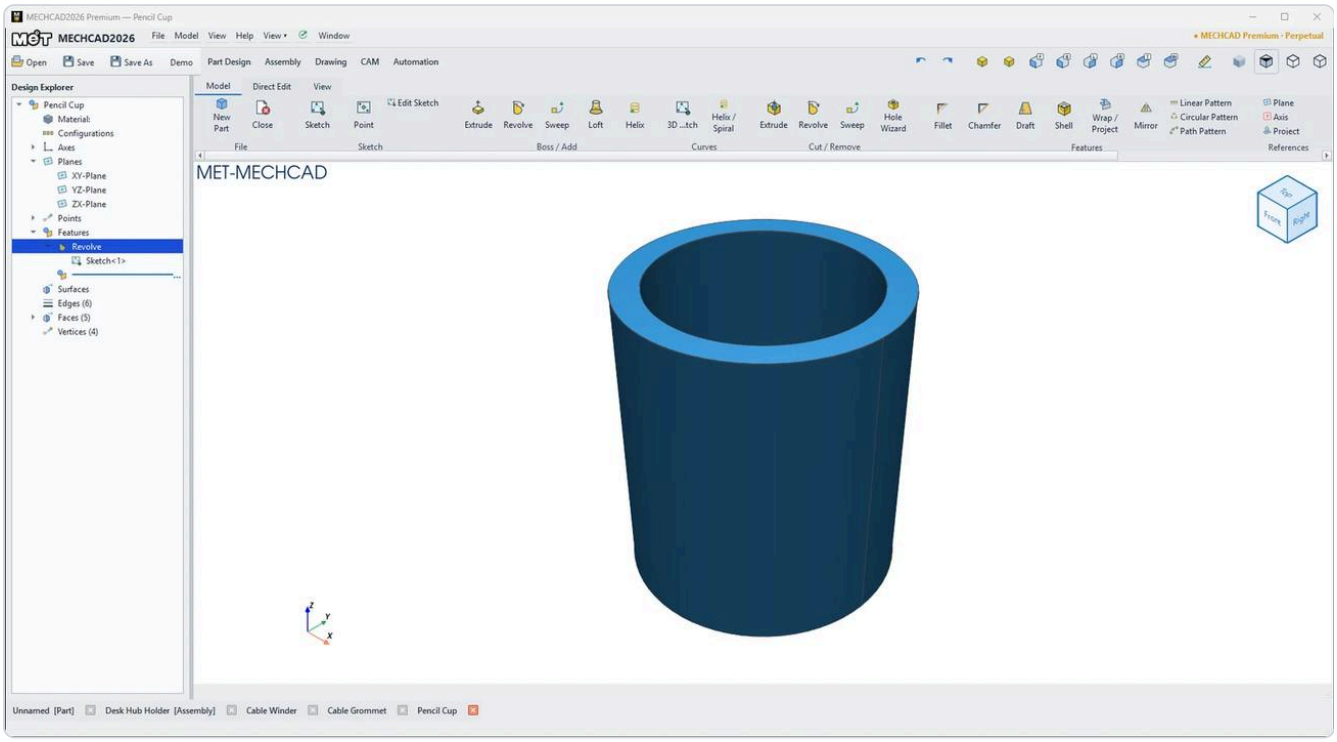
1. The MECHCAD Main Screen
   The parts of the main window: workbench tabs, ribbon, Design Explorer, 3D view and document bar.
2. Main Menus (File, Model, View, Help)
   The commands in the MECHCAD top menu bar.
3. Ribbon and Command Search (F3)
   How to use the ribbon tabs and groups, and how to find commands with F3.
4. Design Explorer (part tree)
   The part tree structure, the right-click menu, and editing features from the Design Explorer.
5. Mouse Use and View Rotation
   Mouse buttons for rotate, pan and zoom, and how to set them to match Alibre or SolidWorks.
6. Standard Views, View Cube and Normal To
   Turn the view with the standard view buttons, the View Cube and the Normal To command.
7. Display modes and the Display panel
   Switch between Shaded, Shaded + Edges, Hidden edges and Wireframe, and adjust lighting, shadow and background.
8. Realistic Render and Save Render
   Turn on a studio-style realistic view and save a high-resolution PNG image.
9. System Options (Program Settings)
   Every System Options page: General, Mouse Interaction, Keyboard Shortcuts, Sketching, Design Viewing and Color Scheme.
10. Keyboard Shortcuts
    The most-used keyboard shortcuts in every MECHCAD workbench.
11. Measure (Distance, Angle and Size)
    Measure length, diameter, distance between objects and angle in both parts and assemblies.

The MECHCAD Main Screen Standard

The parts of the main window: workbench tabs, ribbon, Design Explorer, 3D view and document bar.

What it does

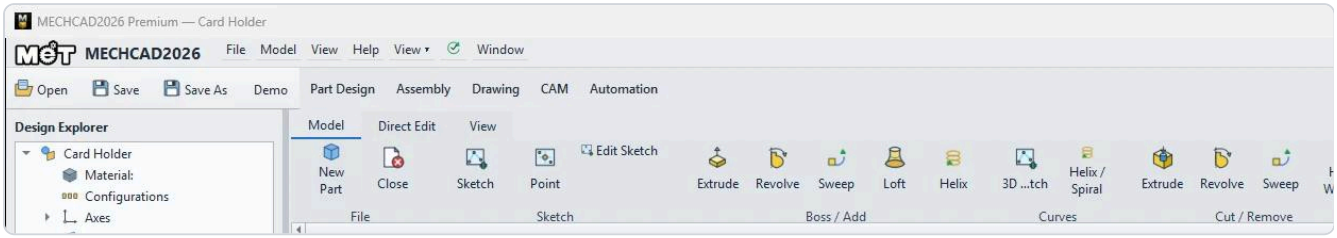
This page explains the parts of the main window. Every workbench (Part Design, Assembly, Drawing, CAM, Automation) uses the same layout.



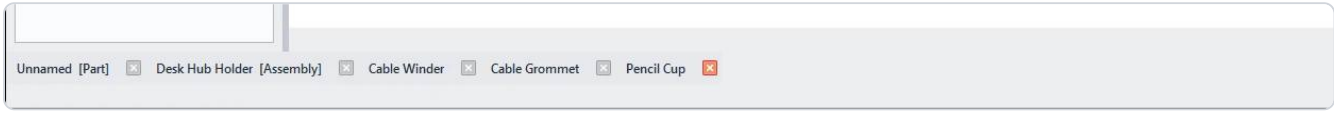
MECHCAD main window with the Pencil Cup part open

Part	Location	Purpose
Logo and menu bar	Top	The menus **File, Model, View, Help, View ▾, Rebuild, Window**. See Main menus.
Edition badge	Top right	Shows the edition and licence term. Click it to open the Welcome window.
Open / Save / Save As / Demo	Second row, left	File commands for the document on screen
Workbench tabs	Second row	**Part Design, Assembly, Drawing, CAM, Automation**
Shared view buttons	Second row, right	Undo/Redo, Zoom to fit, standard views, Measure, display modes. See Standard views and the View Cube.
Search / Manual / Report Bug	Second row, far right	The **Search tools... (F3)** box, the 📖 **Manual** button (online manual) and 🐞 **Report Bug**. See Report a Bug.
Ribbon	Below the tabs	The workbench commands, divided into sub-tabs and groups. See Ribbon and Command Search.
Design Explorer	Left	The part tree: planes, features, sketches, faces. See Design Explorer.
3D view	Center	The work area, with the View Cube at the top right and the XYZ axes at the bottom left

Part	Location	Purpose
Document bar	Bottom	List of all open documents. Click to switch, click ✕ to close.
Status bar	Bottom	Messages that guide the next step of the active command



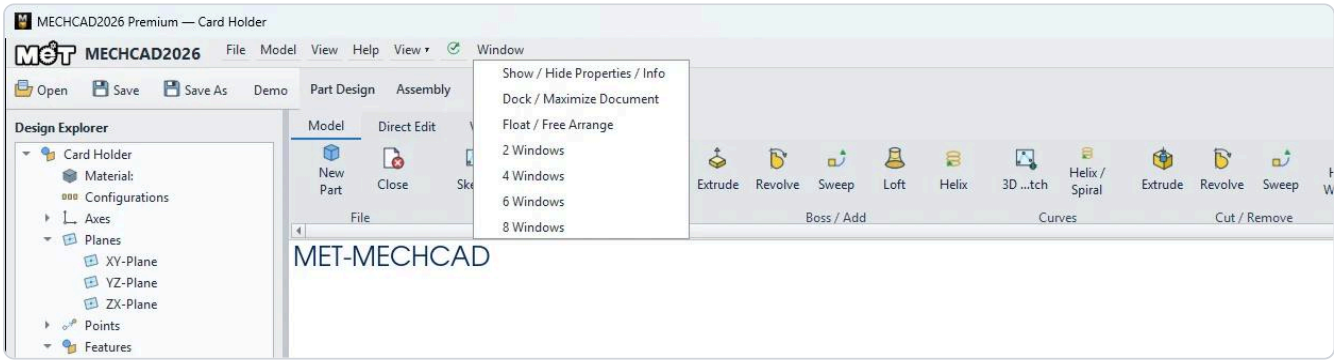
Workbench tabs



Document bar at the bottom: all open documents

Working with several documents

- Opening a new file (Open / double-click in Explorer) opens it in a new tab. If the file is already open, the program switches to that tab.
- Right-click a document tab for **Dock / Maximize Document**, **Float / Free Arrange**, **2 / 4 / 6 / 8 Windows** or **Close Document**.
- The **Window** menu has the same commands, plus **Show / Hide Properties / Info**.



Window menu

> **Note:** If you close a document that has unsaved changes, the program asks *Save changes to …?*. Choose Save, Discard or Cancel.

Small screens and large text

The main window works on a 1024 px wide screen and with Windows text size set to 125–175 %.

- **Ribbon wider than the window:** ‹ / › arrows appear at the left and right edges of the ribbon. Click to scroll, hold to keep scrolling, or turn the mouse wheel over the ribbon to scroll sideways. The » button at the far right lists every command that is not fully on screen, grouped by ribbon group. See Ribbon and Command Search.

- **Narrow workbench bar:** the **Search tools... (F3)** box gets shorter and the Manual / Report Bug buttons show only their 📖 / 🐞 icons, so the window never gets wider than the screen.

Floating windows

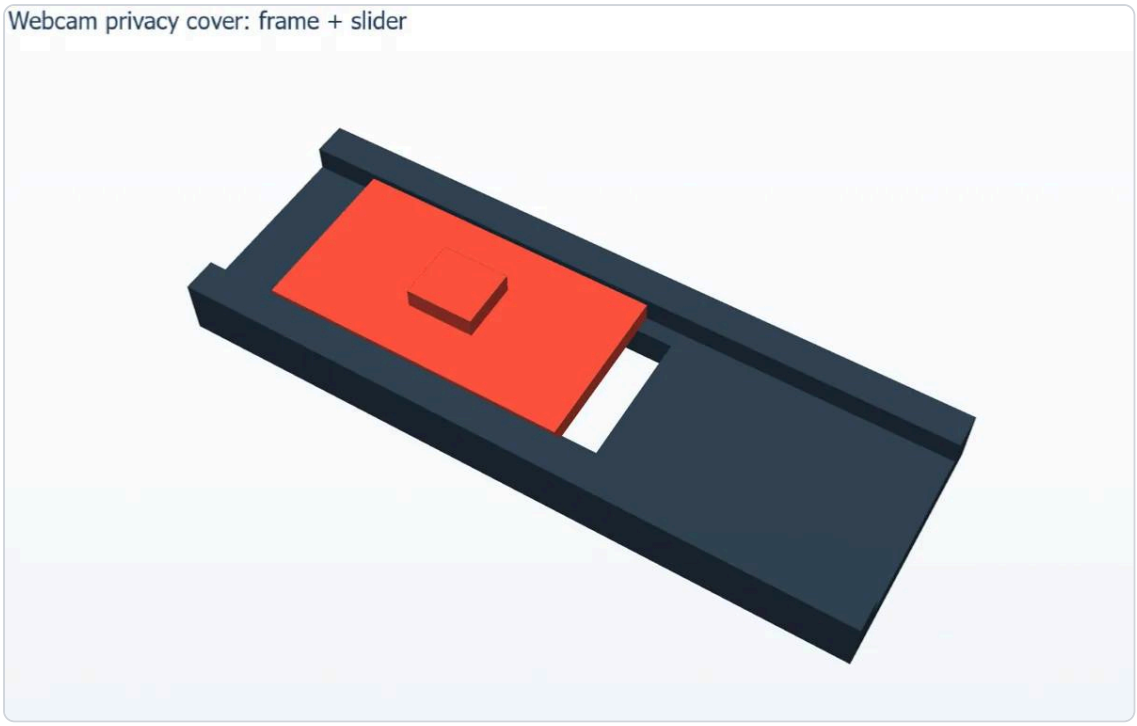
Floating windows and boxes (for example the feature parameter box or the *More colors* picker) all behave the same way.

- **Draggable:** drag them by the box header (or the window title bar).
- **Kept on screen:** the program pulls a box back on screen when it opens, moves, resizes, or when the screen or scaling changes. Boxes that float over the 3D view stay inside the view, so the title bar can always be grabbed.
- **Fits the screen:** if the content is taller than the screen, the body scrolls while the OK / Cancel buttons stay visible at the bottom.
- **Remembers its place:** the position you drag a window to is remembered per window type and used the next time it opens.

Try it: Webcam privacy cover, two documents side by side

A thin frame with a sliding shutter for your laptop camera. It is two parts, which lets you practise the document bar and window layouts.

1. Part 1, frame: sketch on **XY** a rectangle **40 × 14**, **Extrude 2** mm. On the top face, draw a window **8 × 8**, **16** mm from the left edge and **3** mm from the front edge, and cut it **Through All**. On the top face again, draw two strips **40 × 2** along the long edges and **Extrude** them **1.5** mm: these are the rails. Save as `cover-frame.metprt`.
2. Part 2, shutter: press **Ctrl+N**. Sketch a rectangle **18 × 9.6** and **Extrude 1.2** mm. On its top face, add a **4 × 3.6** knob and **Extrude** it **1.2** mm. Save as `cover-slider.metprt`.
3. Look at the **document bar** at the bottom: both parts are listed. Click a name to switch.
4. Right-click a document tab › **2 Windows**: both parts are shown side by side.
5. Choose **Window › Show / Hide Properties / Info** to make more room, and read the **status bar** while you draw: it tells you the next click.



Webcam privacy cover: frame with rails and the red slider

Challenge: Open the **Assembly** tab, insert both parts and slide the shutter between the rails.

See also

- Main menus
- Mouse use and view rotation
- System Options

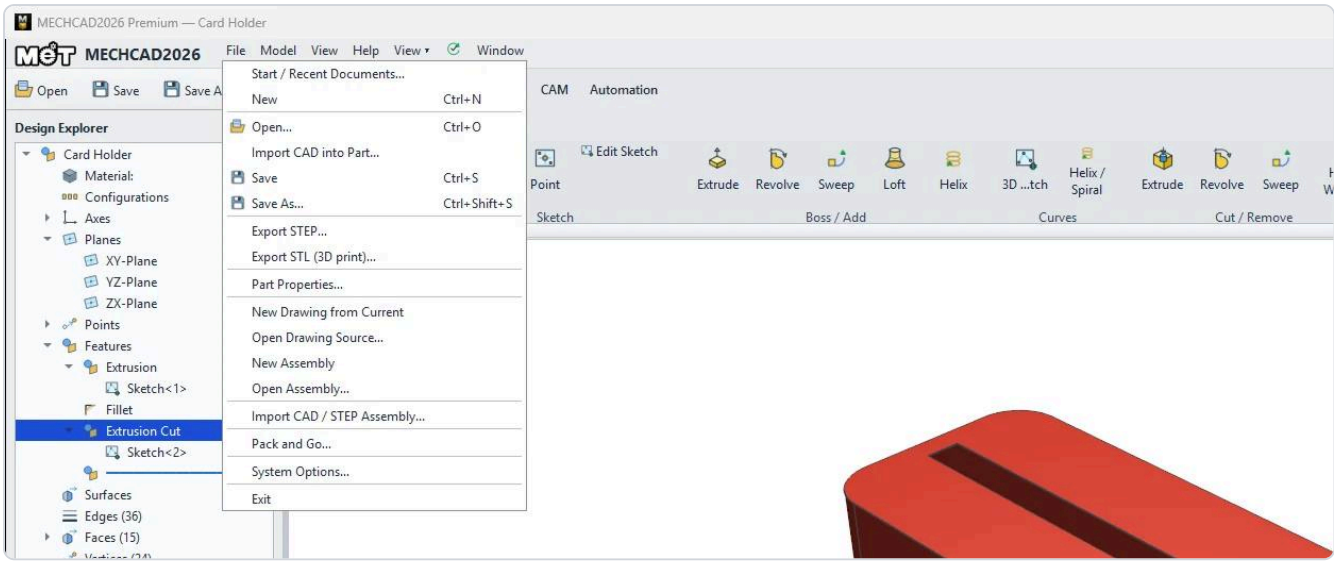
Main Menus (File, Model, View, Help) Standard

The commands in the MECHCAD top menu bar.

What it does

The menu bar collects the commands for files, import/export, settings and licence. The menu commands are the same as the buttons on the ribbon.

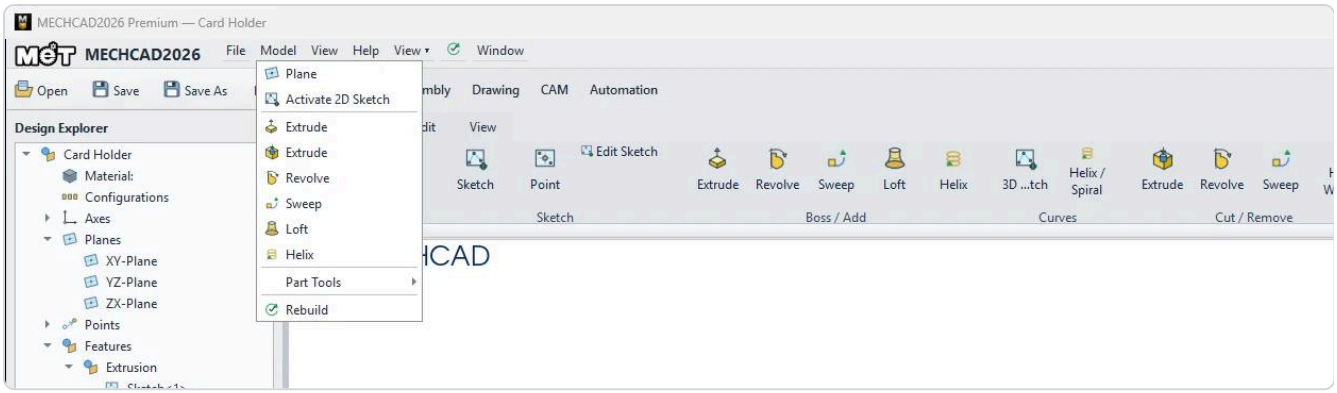
File



File menu

Command	Shortcut	Meaning
Start / Recent Documents…		Opens the Start Center
New	Ctrl+N	New document for the active tab
Open…	Ctrl+O	Opens a MECHCAD file or another CAD file
Import CAD into Part…		Imports STEP/IGES/X_T/SolidWorks/Alibre as a part. See Import CAD
Save / Save As…	Ctrl+S / Ctrl+Shift+S	Saves the document on screen
Export STEP… / Export STL (3D print)…		Exports the part. See Export STEP / STL
Part Properties…		Part data for the title block and BOM
New Drawing from Current / Open Drawing Source…		Creates a drawing from a part/assembly
New Assembly / Open Assembly…		Opens an Assembly tab
Import CAD / STEP Assembly…		Imports an assembly
Pack and Go…		Bundles an assembly and its components into one file. See Pack and Go
System Options…		Program settings. See System Options
Exit		Exits the program

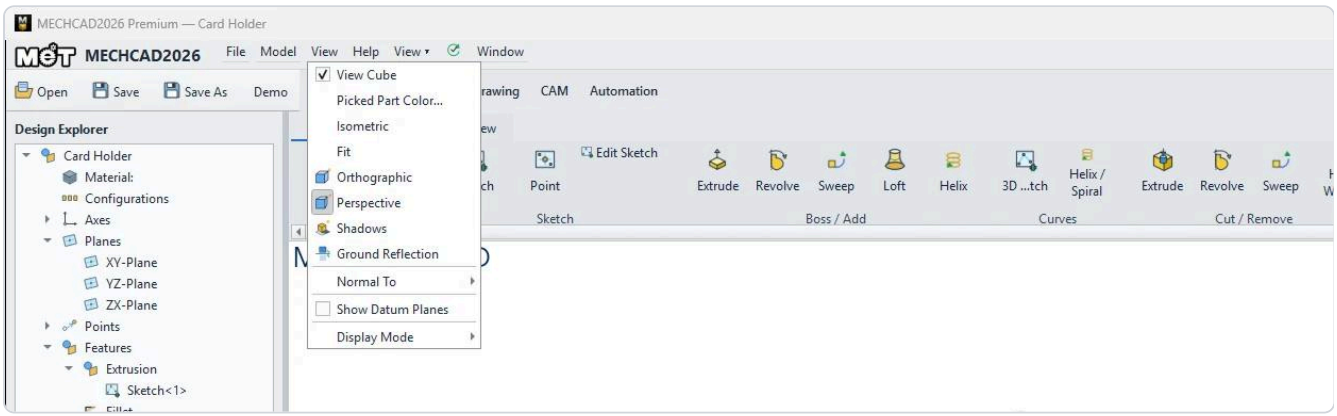
Model



Model menu

The main part-building commands: Plane, Activate 2D Sketch, Extrude (Boss/Cut), Revolve, Sweep, Loft, Helix, the **Part Tools** submenu (Fillet, Chamfer, Shell, Hole Wizard, Mirror, Pattern, Part Color, Union/Subtract/Intersect/Scale) and Rebuild.

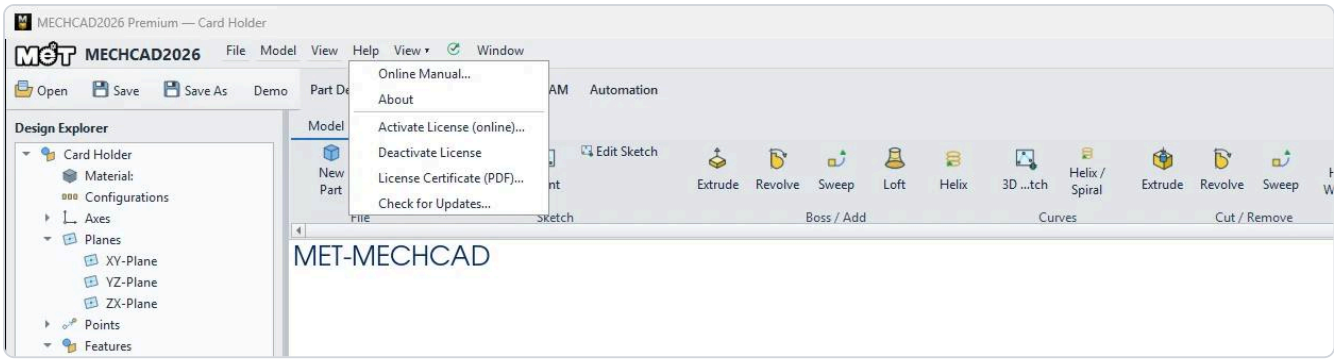
View



View menu

Command	Meaning
View Cube	Shows/hides the View Cube
Picked Part Color…	The color used to highlight the selected component (default blue)
Isometric / Fit	Isometric view / zoom to fit
Orthographic / Perspective	Projection type
Shadows / Ground Reflection	Shadows and ground reflection
Normal To › XY / YZ / ZX	Looks perpendicular to the plane
Show Datum Planes	Shows the reference planes (Ctrl+Shift+P)
Display Mode	Shaded / Shaded + Edges / Hidden Edges

Help



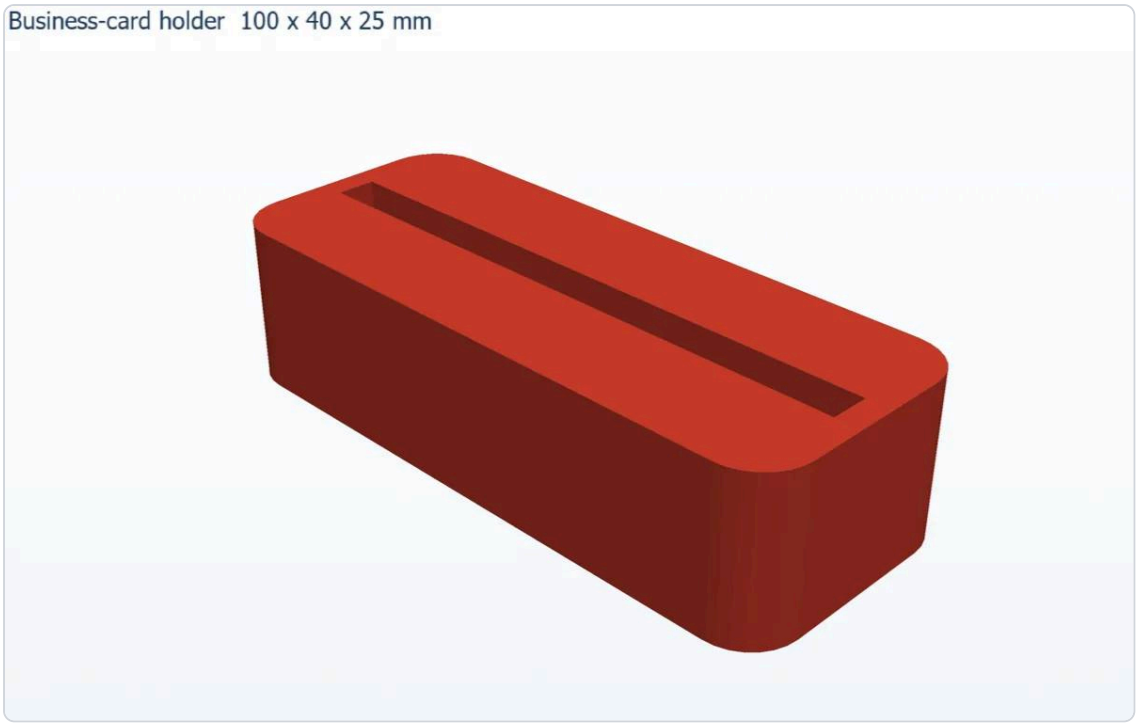
Help menu

Command	Meaning
Online Manual…	Opens the online manual
About	Version information
Activate License (online)…	Opens the Welcome window / enter a license key
Deactivate License	Releases this computer's seat to the server (before moving to another computer)
License Certificate (PDF)…	Creates a licence certificate as a PDF
Check for Updates…	Checks for a new version (MA customers)

Try it: Business-card holder using only the menu bar

Make a stand that keeps your business cards upright on the desk, and use the menus at the top for every main step, from New to the STL file.

1. Choose **File › New** on the Part Design tab.
2. Choose **Model › Activate 2D Sketch**, pick **XY**, and draw a rectangle **100 × 40** from the origin. Click **Finish Sketch**.
3. Choose **Model › Extrude** (Boss) with **Depth 25**.
4. Choose **Model › Part Tools › Fillet**, set **Radius 8** and click the four vertical corner edges, then **Done**.
5. Right-click the top face › **New Sketch on this face**. Draw a rectangle **92 × 6**, **4** mm from the left edge and **17** mm from the front edge. **Finish Sketch**, then choose **Model › Extrude** (Cut) with **Depth 18**. A standard 90 × 55 mm card stands in the slot with its top sticking out.
6. Choose **File › Part Properties…** and enter the description *Business card holder*.
7. Choose **File › Export STL (3D print)…** and save `card-holder.stl`.



Business-card holder, 100 x 40 x 25 mm, with a 6 mm card slot

Challenge: Use **View › Normal To › XY** to look straight down and check that the slot is centred.

See also

- Ribbon and Command Search
- Keyboard shortcuts

Ribbon and Command Search (F3) Standard

How to use the ribbon tabs and groups, and how to find commands with F3.

What it does

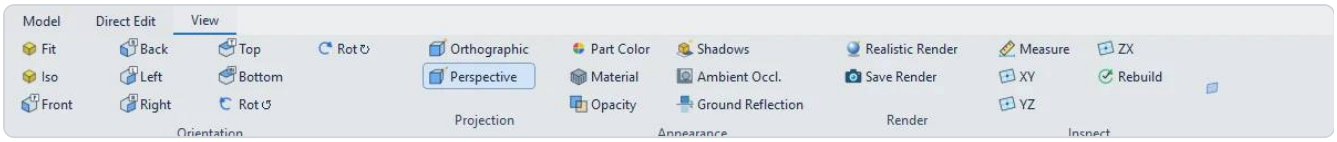
The ribbon shows the commands of the open workbench, divided into sub-tabs and groups. Part Design has three tabs: **Model**, **Direct Edit** and **View**. If the window is narrow, buttons shrink to icons only (hover to see the name).



Model tab: File, Sketch, Boss / Add, Curves, Cut / Remove, Features, References, Regenerate, Render



Direct Edit tab: Edit, Solid, Create



View tab: Orientation, Projection, Appearance, Render, Inspect

Tab › Group	Commands
Model › File	New Part, Close
Model › Sketch	Sketch, Point, Edit Sketch
Model › Boss / Add	Extrude, Revolve, Sweep, Loft, Helix
Model › Curves	3D Sketch, Helix / Spiral
Model › Cut / Remove	Extrude, Revolve, Sweep, Hole Wizard
Model › Features	Fillet, Chamfer, Draft, Shell, Wrap / Project, Mirror, Linear / Circular / Path Pattern
Model › References	Plane, Axis, Project, Parameters (see Parameters)
Model › Regenerate	Regen, Roll Back, Roll Fwd, Roll to End
Model › Render	Realistic Render, Save Render
Direct Edit › Edit	Undo, Redo, Suppress, Delete
Direct Edit › Solid	Move Face, Simplify, Scale, Union / Add, Subtract, Intersect
Direct Edit › Create	Text, Picture, Recognize Features, Gear, Feed Screw
View › Orientation	Fit, Iso, Front, Back, Left, Right, Top, Bottom, Rot ↺, Rot ↻
View › Projection	Orthographic, Perspective
View › Appearance	Part Color, Material, Opacity, Shadows, Ambient Occl., Ground Reflection
View › Inspect	Measure, XY, YZ, ZX, Rebuild

When the ribbon is wider than the window

On a small screen (for example 1024 px wide), or with Windows text size set to 125–175 %, the right-hand groups of the ribbon may not fit in the window. Use these to reach the rest of the commands. They work on the Part Design, Sketch, Assembly, Drawing and Automation ribbons.

Button / action	Result
‹ left edge / › right edge	Scrolls the ribbon left or right. Hold to keep scrolling. The arrow turns grey at the end.
Mouse wheel over the ribbon	Scrolls the ribbon left or right
» far right	A menu of every command that is not fully on screen, with a heading for each ribbon group. Click a command to run it. (When everything fits, it shows *Every command is on screen*.)

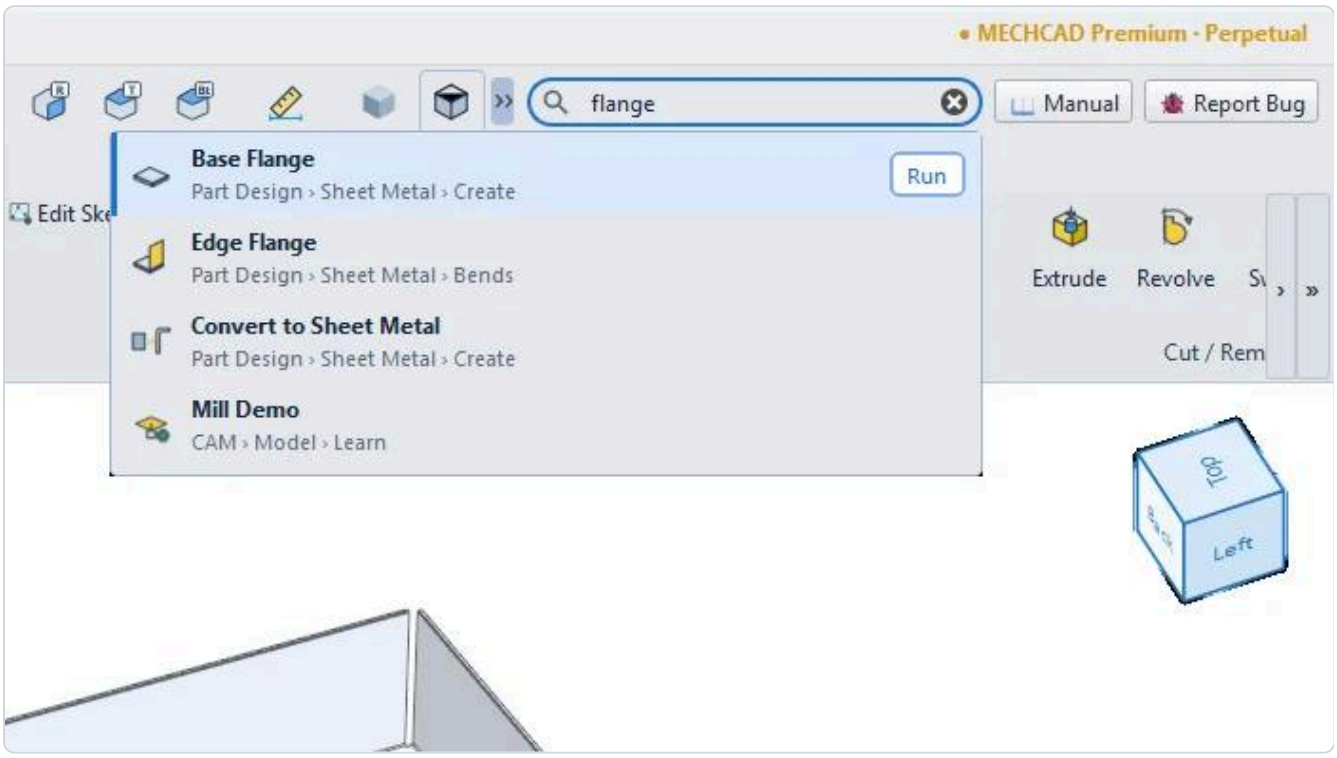
The arrows and the » button appear only while the ribbon is wider than the window, and they never cover a command button. In a narrow window the **Search tools... (F3)** box on the workbench bar also gets shorter,

and the Manual / Report Bug buttons show only their icons.

Command Search (F3)

Find a command by name instead of looking for its button.

1. Press **F3**, or click the **Search tools... (F3)** box on the right of the workbench bar.
2. Type a command name, such as *extrude*, *bevel* or *scale*. Synonyms work too (for example bevel = Chamfer, move = Move Face, resize = Scale). Use the up and down arrow keys to pick a result.
3. Press **Enter** to be taken to that button (the program switches workbench and tab, scrolls the ribbon and flashes a blue outline around the button), or press **Ctrl+Enter** to run the command at once.



Search tools (F3) box: typing flange lists the matching commands and where each one lives on the ribbon

> **Note:** Commands that cannot be used in the current state (for example Fillet before there is a part) are shown in grey.

Try it: Desk cable grommet, found with F3

A grommet for a Ø60 mm desk hole, with a side slot so the cables can come out. Run every command from **F3** instead of hunting for buttons.

1. Start a sketch on the **ZX** plane (X right, Z up). Press **L**, click a point on the X axis and draw: **4 / 0°**, **20 / 90°**, **5 / 0°**, **3 / 90°**, **9 / 180°**, then click the start point to close it.
2. Press **D** and set the distance from the origin to the start point to **26** mm (the inner radius). **Finish Sketch**.
3. Press **F3**, type *revolve*, choose **Revolve Boss** and press **Ctrl+Enter**. Choose **Axis Z**, **Angle 360** and click **OK**.

4. Press **F3**, type *bevel*, press **Ctrl+Enter**: Chamfer opens. Set **Distance 0.8** and click the top outer edge of the flange, then **Done**.
5. Right-click the top face of the flange › **New Sketch on this face**. Draw a rectangle **16 × 14** across the ring wall on one side.
6. **Finish Sketch**, press **F3**, type *extrude cut*, press **Ctrl+Enter** and choose **Through All**. You now have the cable slot.



Desk cable grommet for a 60 mm hole, with a cable slot

Challenge: Press **F3** before you make any part and type *fillet*: the command is grey because there is no body yet.

See also

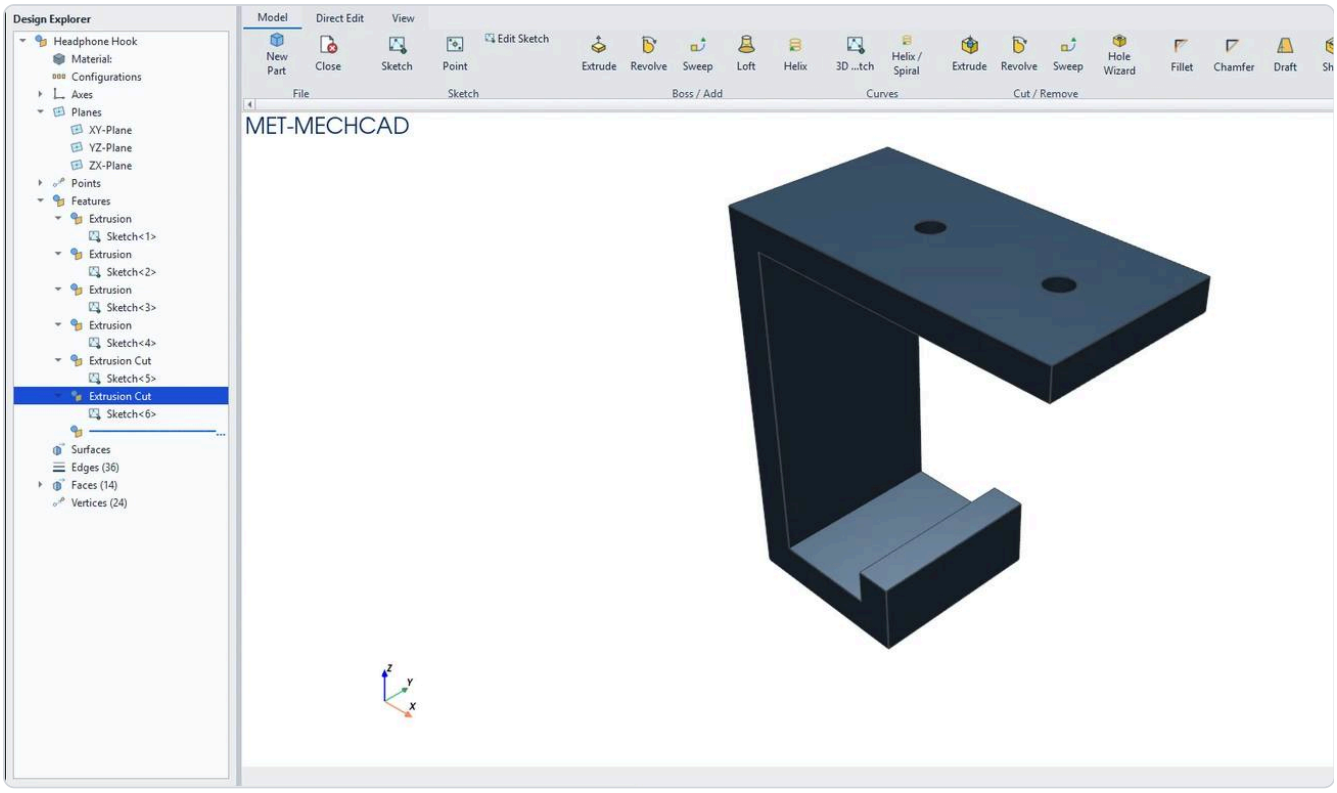
- Keyboard shortcuts
- The MECHCAD main screen

Design Explorer (part tree) Standard

The part tree structure, the right-click menu, and editing features from the Design Explorer.

What it does

The **Design Explorer** on the left shows everything in the part in the order it was built. Use it to select, edit, hide/show, and roll back the history.



Design Explorer of the Headphone Hook part

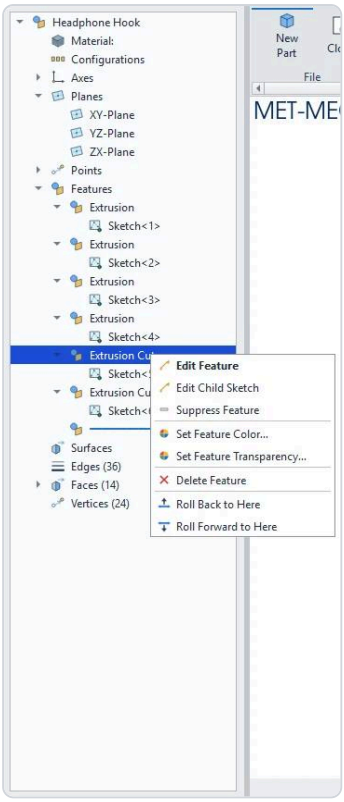
Nodes in the tree

Node	Description
Part name	The root of the tree
Material:	The material. Double-click to choose a material.
Configurations	Part configurations
Axes	X/Y/Z axes and reference axes Axis
Planes	XY-, YZ-, ZX-Plane and reference planes Plane
Points	Origin and Center of Mass
Features	Features in order, with each feature's child sketch and the blue **Rollback bar**
Surfaces / Edges (n) / Faces (n) / Vertices (n)	Lists of surfaces, edges, faces and points (● marks a face with its own colour)

Double-click

- Plane › creates a new sketch on that plane
- Feature › opens the feature edit box (Feature Parameters)
- Sketch › edits the sketch
- Material › opens the material menu

Right-click menu



Right-click a feature

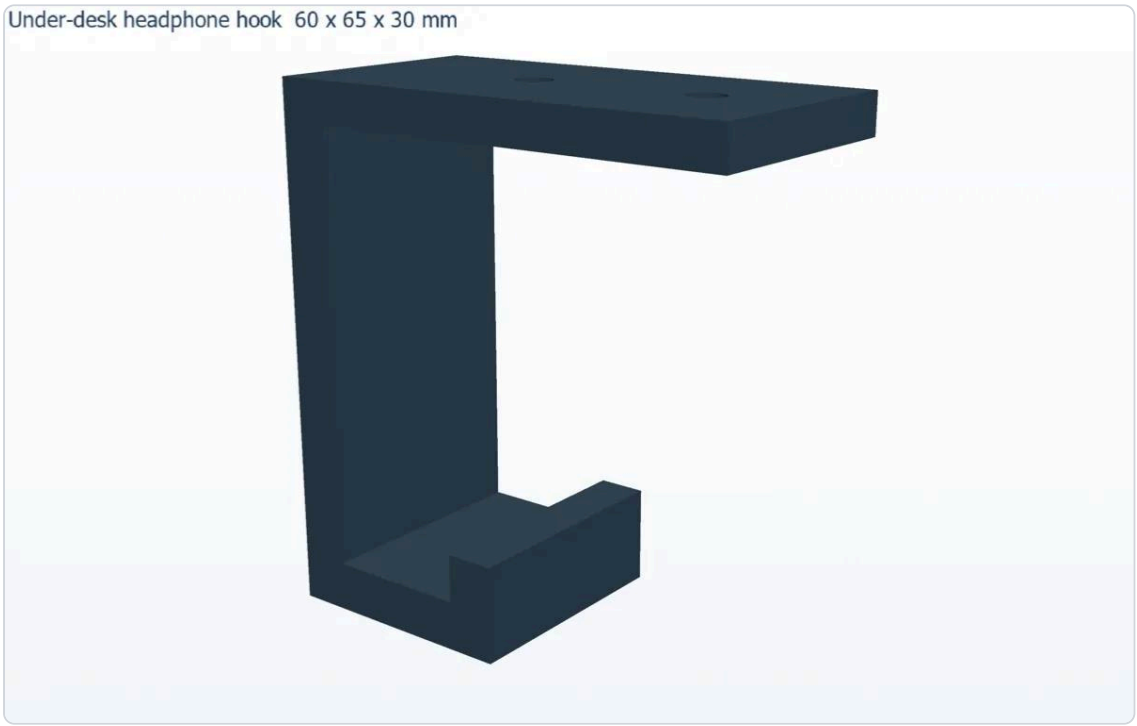
Menu	Applies to	Description
Edit Feature	Feature	Edits the feature values (the main command).
Edit Child Sketch	Feature	Edits the sketch used to create the feature.
Suppress / Unsuppress Feature	Feature	Turns calculation off/on temporarily (the name then starts with "(Suppressed)").
Set Feature Color… / Set Feature Transparency…	Feature	Sets the colour / transparency of that feature only.
Delete Feature	Feature	Deletes the feature (the sketch remains).
Roll Back to Here / Roll Forward to Here	Feature	Moves the Rollback bar. See Rollback and Regenerate.
Edit Sketch / Suppress Sketch / Delete Sketch	Sketch	Manages the sketch (deleting a sketch also deletes the features that use it).
New Sketch on this face	Face	Creates a sketch on the face.
Edit Plane… / New Sketch on … / Insert Offset Plane…	Plane	Manages reference planes.
Show / Hide	Planes, Axes, Points	Shows/hides reference objects.

> **Note:** You can rename a feature in the **Label** field of the Extrude, Revolve and Loft boxes.

Try it: Under-desk headphone hook

Screw this hook under the edge of your desk and your headphones are always within reach. Building it in three features gives you a small tree to practise suppress, edit and rollback on.

1. Start a sketch on the **ZX** plane. Press **L**, click the origin and type these segments (length, **Tab**, angle, **Enter**): **30 / 0°**, **12 / 90°**, **6 / 180°**, **6 / 270°**, **18 / 180°**, **54 / 90°**, **54 / 0°**, **5 / 90°**, **60 / 180°**. Click the origin to close the J shape.
2. **Finish Sketch**, then **Extrude** with **Type Mid Plane** and **Depth 30**. In **Label**, type `Hook body` and click **OK**.
3. Right-click the top face of the 60 mm top plate and choose **New Sketch on this face**. Draw two circles of **Ø4.5** on the centre line, **25** and **48** mm from the back face. **Finish Sketch**, then **Cut / Remove › Extrude** with **Through All** and label it `Screw holes`.
4. Click **Fillet**, **Radius 2**, and click the two top edges of the small front lip, then **Done**.
5. Now use the tree: right-click **Fillet 1** › **Suppress Feature** (it shows "(Suppressed)"), then **Unsuppress Feature**.
6. Right-click **Hook body** › **Edit Child Sketch**, double-click the **54** dimension of the vertical arm and change it to **74** for a thicker headband. **Finish Sketch**: the holes and fillets follow.
7. Right-click **Screw holes** › **Roll Back to Here** to see the part without its holes, then **Roll Forward to Here**.



Under-desk headphone hook with two screw holes

Challenge: Double-click the **Material:** node and pick **POM Black**, then check the new mass in Part Properties.

See also

- Feature Parameters (feature edit box)

- Rollback and Regenerate

Mouse Use and View Rotation Standard

Mouse buttons for rotate, pan and zoom, and how to set them to match Alibre or SolidWorks.

What it does

Explains how to control the 3D view with the mouse and keyboard. The MECHCAD default is: **right button rotates, left button selects/pans, wheel zooms**.

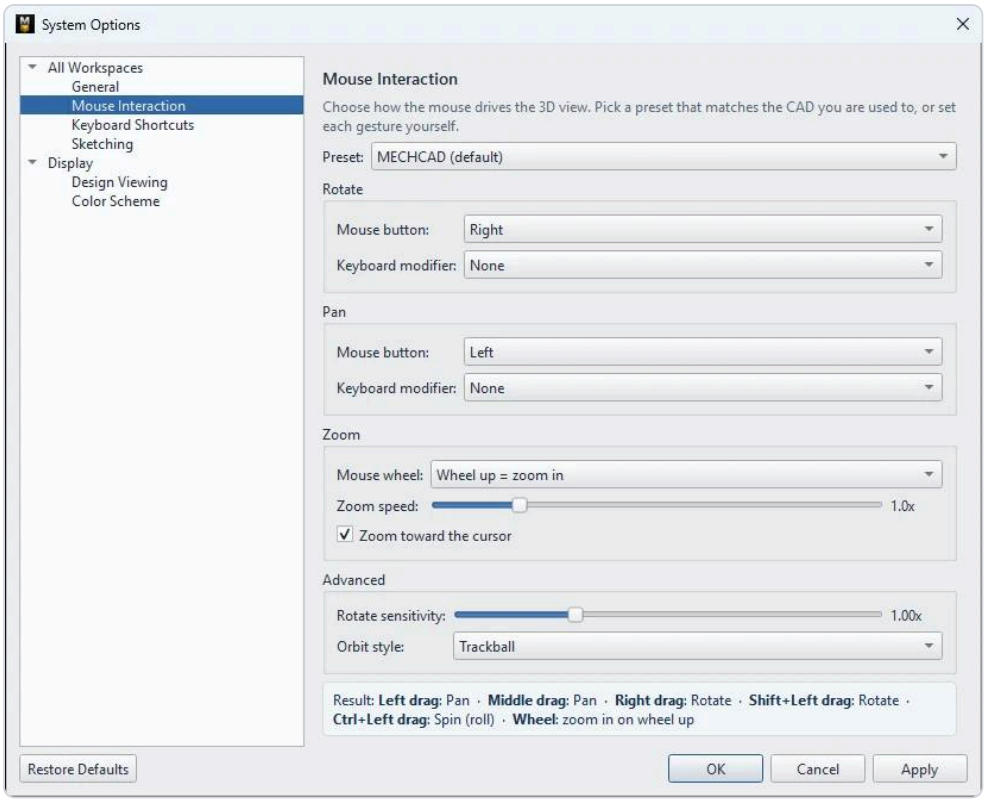
Defaults (preset: MECHCAD)

Action	Result
Left-click	Selects an object (face, edge, feature, component)
Drag with the left button on empty space	Pans the view. In an assembly, drags a selection box.
Drag with the right button	Rotates the view around the point under the cursor
Quick right-click	Opens the right-click menu
Drag with the middle button	Pans the view
Shift + drag with the left button	Rotates the view
Ctrl + drag with the left button	Rolls the view around the line of sight
Turn the wheel	Zooms in/out, toward the cursor position

> **Note:** While you select edges or faces (for example for Fillet or Shell), the left button only selects. You can still drag with the right button to rotate. While you draw a sketch, panning is turned off, but you can rotate and zoom.

Change the mouse settings

1. **File › System Options…** › **All Workspaces › Mouse Interaction**
2. Choose a **Preset**: *MECHCAD (default)*, *Alibre Design*, *SolidWorks* or *Custom*.
3. Adjust the details as in the table, then click **OK** or **Apply** (it takes effect in every view immediately).



System Options › Mouse Interaction

Option	Meaning
Preset	MECHCAD: rotate = right, pan = left. Alibre Design: rotate = right, pan = middle. SolidWorks: rotate = middle, pan = Ctrl+middle.
Rotate › Mouse button / Keyboard modifier	The button and modifier key (None / Shift / Ctrl) for rotating
Pan › Mouse button / Keyboard modifier	The button and modifier key for panning
Mouse wheel	Wheel up = zoom in, or Wheel up = zoom out
Zoom speed	0.1–5.0x
Zoom toward the cursor	Zooms toward the cursor
Rotate sensitivity	0.25–3.0x
Orbit style	Trackball / Terrain / Joystick

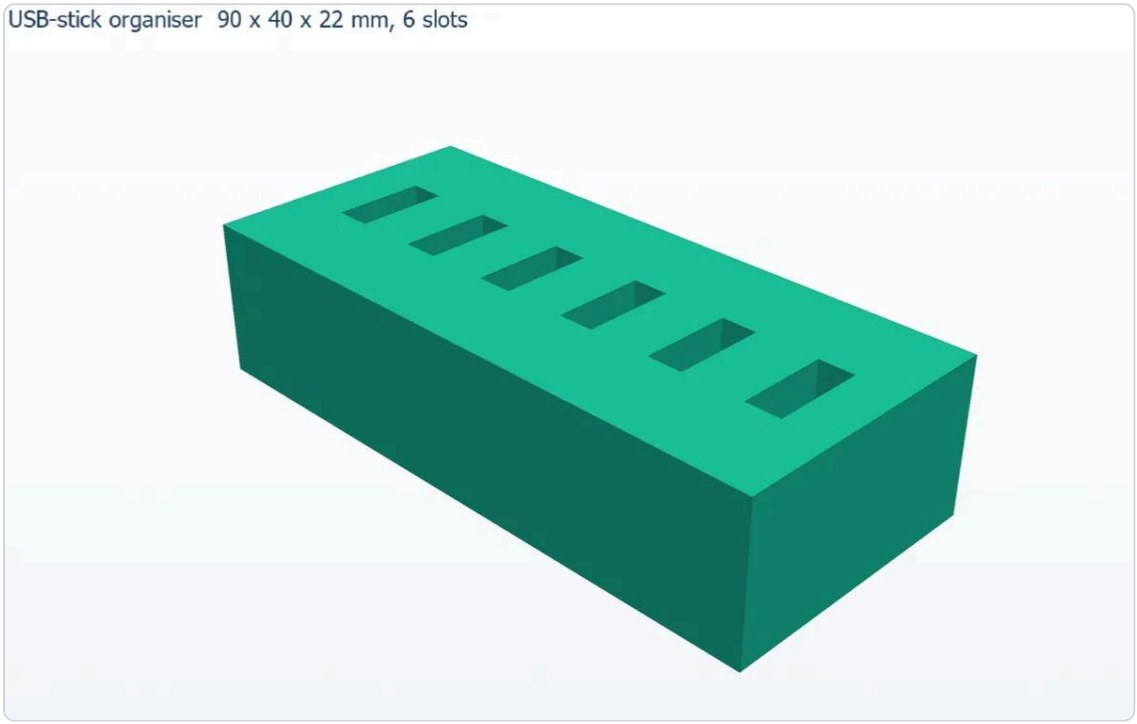
The **Result:** line at the bottom summarizes what each button does with your settings.

Try it: Look inside a USB-stick organiser

Six slots that hold your USB sticks upright. The slots are small and deep, so you have to rotate, zoom and pan to check them: good practice for your mouse hand.

1. Sketch on **XY** a rectangle **90 × 40**, then **Extrude 22** mm.
2. Right-click the top face › **New Sketch on this face**. Draw a rectangle **5 × 13**, **8** mm from the left edge and **13.5** mm from the front edge. **Finish Sketch**, then **Cut / Remove › Extrude 12** mm.

3. Click **Model › Features › Linear Pattern**: **Count 6**, **Spacing 14**, **Direction X**, then **OK**.
4. Now navigate:
   - Point at the first slot and **turn the wheel**: the view zooms toward the cursor.
   - **Drag with the right button**: the view rotates around the point under the cursor, so you can look down into the slot.
   - **Drag with the middle button** (or the left button on empty space): pan to the last slot.
   - **Ctrl + drag with the left button**: roll the view.
   - A **quick right-click** opens the right-click menu instead of rotating.
5. Open **File › System Options… › Mouse Interaction**, choose the **SolidWorks** preset and click **Apply**. Try rotating with the middle button, then choose **MECHCAD (default)** again.



USB-stick organiser, 90 x 40 x 22 mm, with 6 slots

Challenge: Set **Mouse wheel** to *Wheel up = zoom out* if that is what your other CAD program does, and check the **Result:** line.

See also

- Standard views and the View Cube
- System Options

Standard Views, View Cube and Normal To Standard

Turn the view with the standard view buttons, the View Cube and the Normal To command.

What it does

Turns the view quickly to a standard direction. These buttons are on the top bar and work in every workbench.

Shared view buttons (top bar, right)



Undo/Redo, Zoom to fit, Isometric, Front, Back, Left, Right, Top, Bottom, Measure and the 4 display modes

Button	Meaning
Undo (Ctrl+Z) / Redo (Ctrl+Y)	Undo / redo
Zoom to fit	Zooms to show the whole part
Isometric view	3D view (using the projection type you set)
Front, Back, Left, Right, Top, Bottom	Flat orthographic views
Measure	Measure tool. See Measure
View Display (4 buttons)	Shaded, Shaded + Edges, Hidden edges, Wireframe. See Display modes

View Cube



View Cube at the top right

- Click a face of the cube › the flat view of that face (Top, Front, Right …)
- Drag the cube › rotates the view

- Right-click › Top, Bottom, Front, Back, Left, Right, Isometric and **Hide View Cube**
- You can show it again from **View › View Cube**

Normal To

- **View › Normal To › XY / YZ / ZX** or **View ▾ › XY / YZ / ZX** looks perpendicular to a main plane in orthographic view.
- In an assembly, right-click a component › **View Normal to Face** looks perpendicular to the face you click.
- **Rot ↺ / Rot ↻** (View › Orientation) rotates the view 90° in the screen plane.

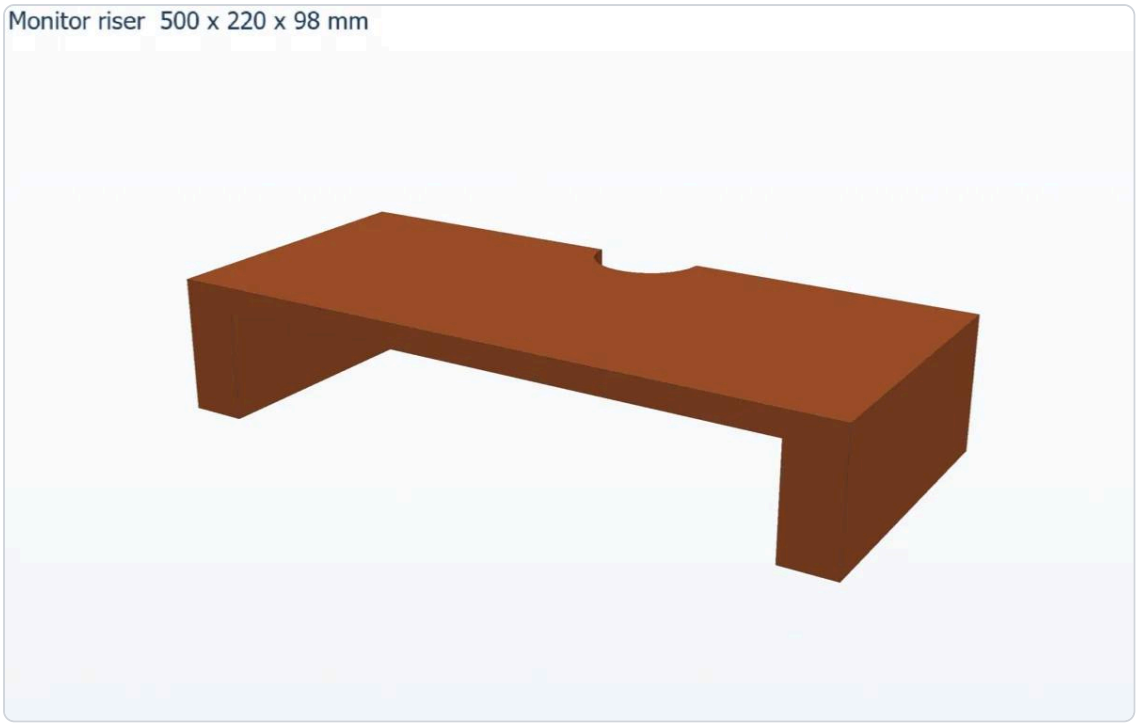
Orthographic / Perspective

View › Projection › Orthographic (no shrinking with distance) or **Perspective** (with depth). Your choice applies to every 3D view. You can set the default in **System Options › Design Viewing**.

Try it: Monitor riser, checked from every side

A 500 mm riser that lifts your screen 98 mm and leaves room for a keyboard underneath. Use the standard views to check every size.

1. Sketch on **XY** a rectangle **500 × 220** and **Extrude 18** mm.
2. Right-click the View Cube › **Bottom**. Right-click the bottom face › **New Sketch on this face** and draw two rectangles **40 × 220**, one at each end. **Finish Sketch** and **Extrude 80** mm (the legs grow downward from the face).
3. Click **Top**. Sketch on the top face a circle **Ø80** centred on the middle of the back edge, then cut it with **Through All**. This is the cable notch.
4. Click **Front**: you see the legs and the 98 mm overall height. Click **Right**: the 220 mm depth.
5. Click **Isometric**. Then use **View › Normal To › XY** and **Rot ↻** to turn the top view 90° on the screen.
6. Choose **View › Projection › Perspective** and back to **Orthographic**: in **Front**, the orthographic view keeps the legs the same height.



Monitor riser, 500 x 220 x 98 mm, with a cable notch

Challenge: Click a face of the View Cube instead of the buttons, then hide the cube with right-click › **Hide View Cube** and show it again from **View › View Cube**.

See also

- Mouse use and view rotation
- Display modes

Display modes and the Display panel Standard

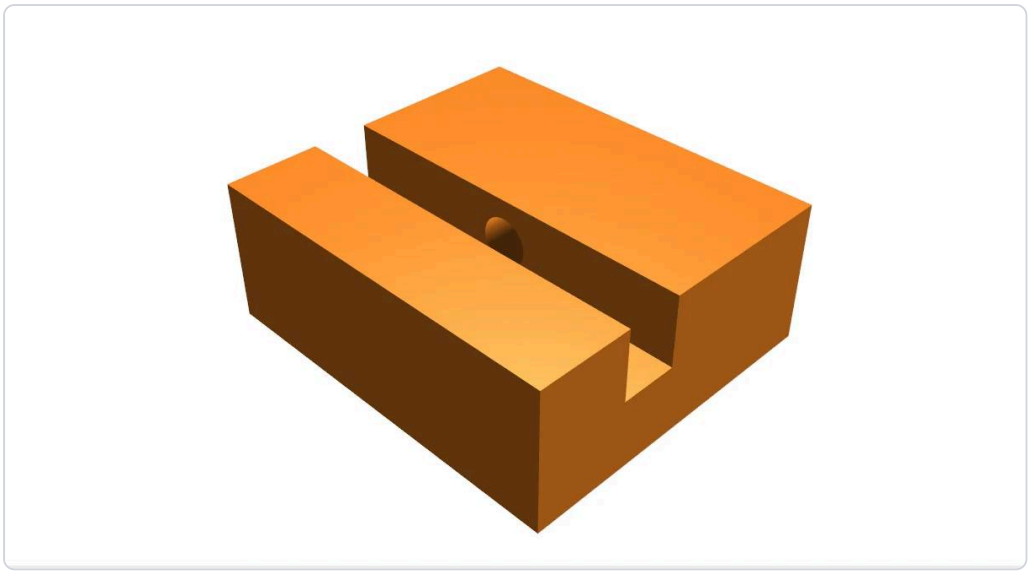
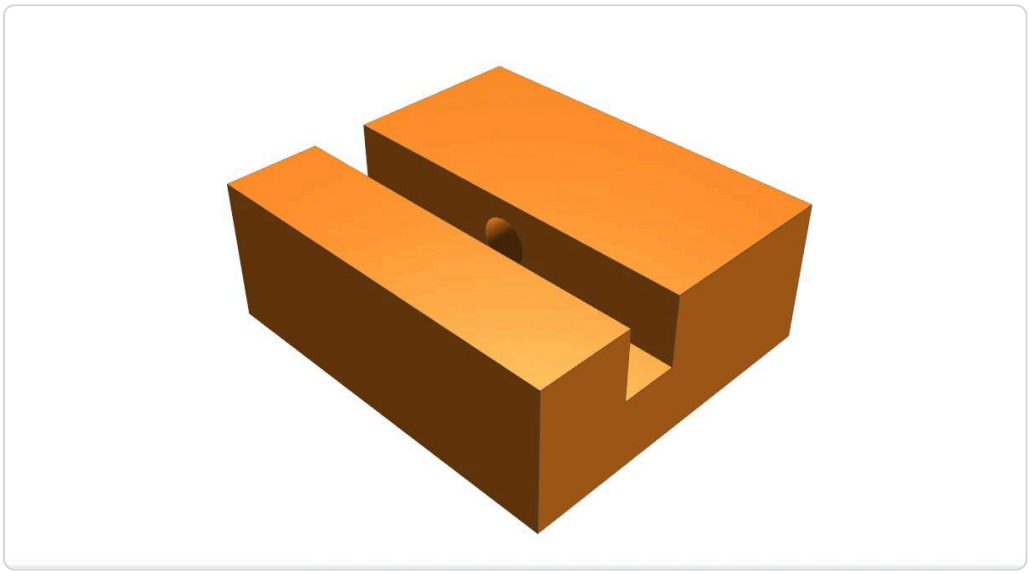
Switch between Shaded, Shaded + Edges, Hidden edges and Wireframe, and adjust lighting, shadow and background.

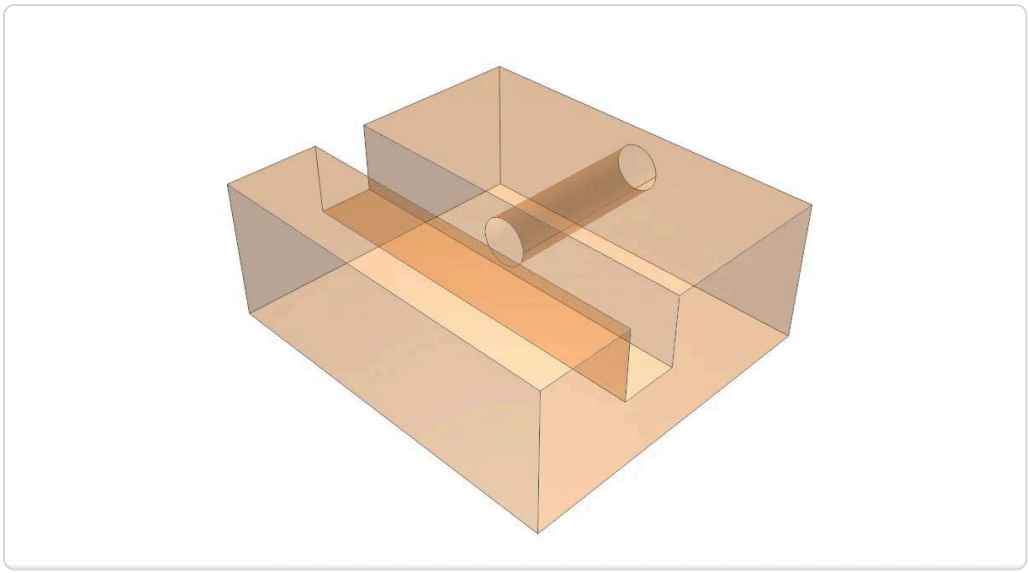
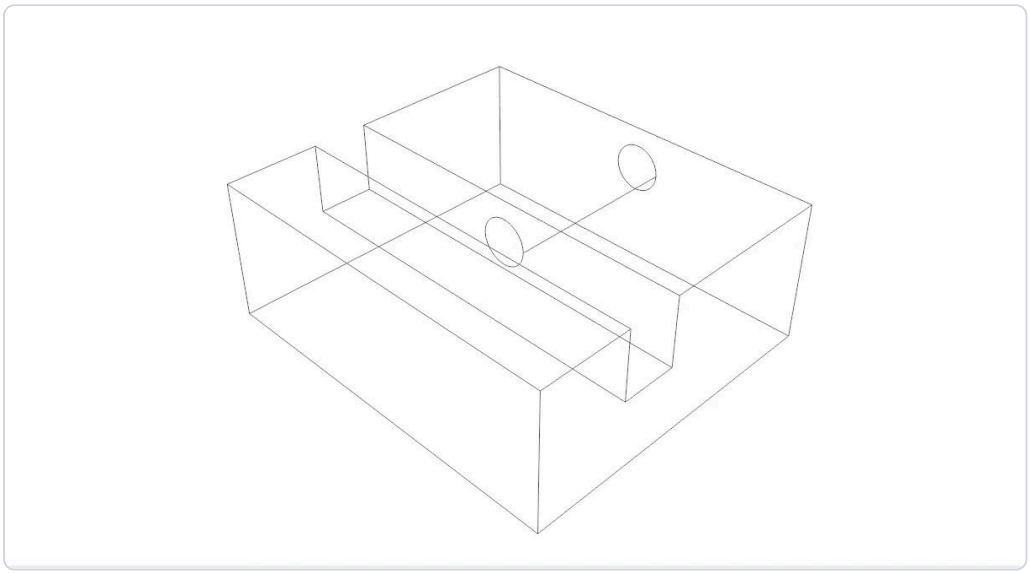
What it does

Chooses how the part is displayed and adjusts the lighting and background of the 3D view.

Where to find it

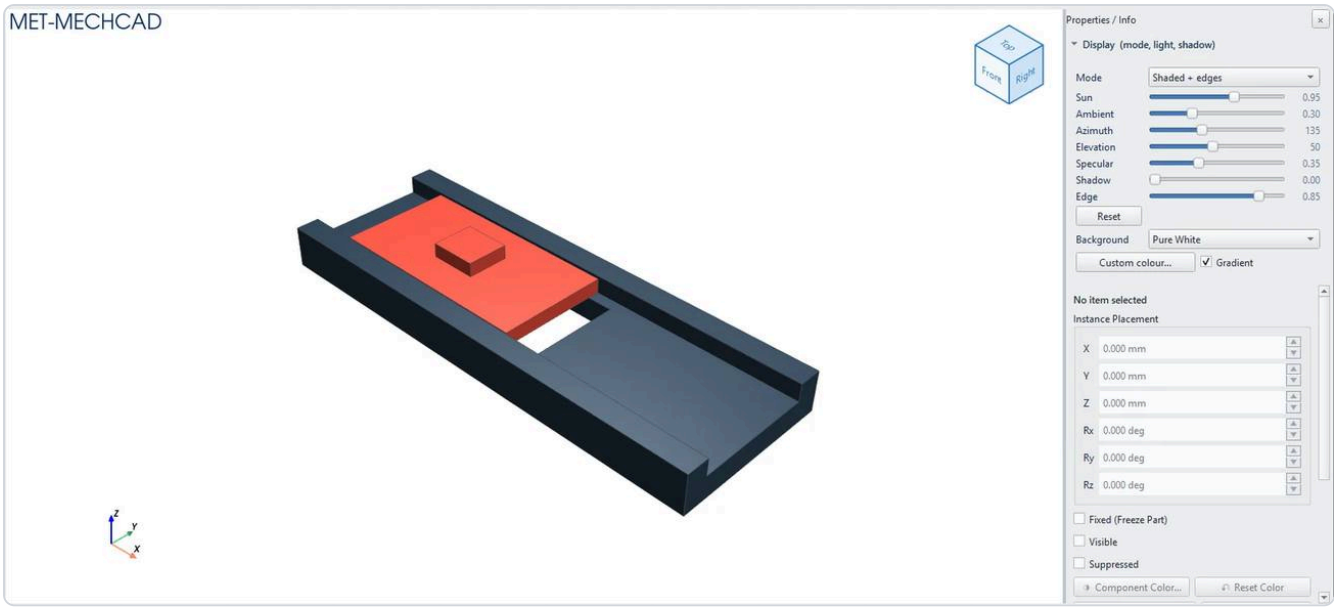
- The 4 buttons at the far right of the top bar (available in every workbench)
- **View › Display Mode**
- The **Display** panel in Assembly: **View & Inspect › View › Display**, or the **Display (mode, light, shadow)** section of the Properties / Info panel

Mode	Example
Shaded	 Shaded
Shaded + Edges (default)	 Shaded + Edges

Mode	Example
Hidden edges (see-through)	 Hidden edges
Wireframe	 Wireframe

Note: In Part Design the Wireframe button shows the same see-through view as Hidden edges; the line-only Wireframe is available in Assembly (the examples above were taken on the Assembly tab).

Display panel



Display panel in the Properties / Info panel on the right

Option	Range	Default	Description
Mode	–	Shaded + edges	Display mode
Sun	0–1.5	0.95	Brightness of the main light
Ambient	0–1	0.30	Ambient light
Azimuth	0–360	135	Horizontal light direction
Elevation	15–90	50	Light elevation angle
Specular	0–1	0.35	Glossiness
Shadow	0–0.8	0	Strength of the shadow on the floor
Edge	0.1–1	0.85	Edge line strength
Reset			Restores lighting, shadow and edges (keeps the mode)
Background		Pure White	Pure White, Light, Dark, Blueprint, Studio and the classic style
Custom colour… / Gradient			Your own background colour and gradient

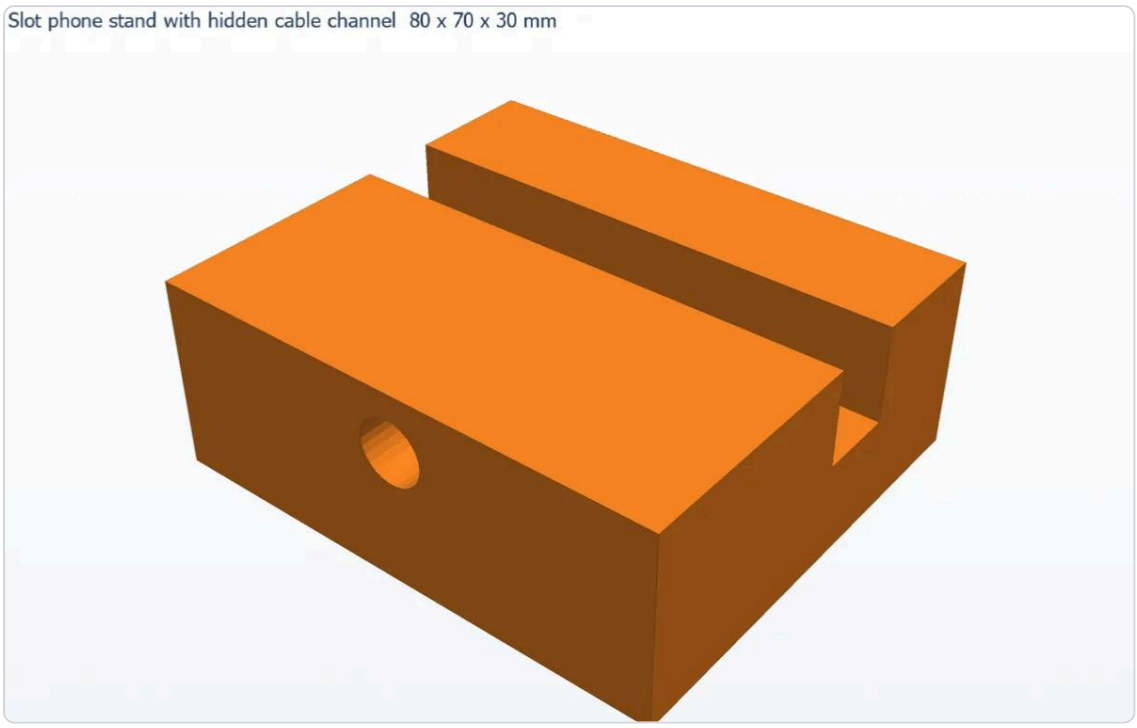
> **Note:** Assemblies with many parts (more than about 1,500) turn shadows off automatically for speed: *Shadow is off for large assemblies*.

Try it: Slot phone stand with a hidden cable channel

A solid block with a slot holds your phone upright, and a tunnel at the back lets the charging cable reach it. The tunnel is hidden inside the part, which makes this the perfect model for trying all four display modes.

1. Sketch on **XY**: press **R**, click the origin, type **80**, **Tab**, **70**, **Enter**. **Finish Sketch** and **Extrude 30** mm.
2. Right-click the top face › **New Sketch on this face**. Draw a rectangle **80 × 12** right across the part, **20** mm from the front edge (use **D** for the 20). **Finish Sketch**, then **Cut / Remove › Extrude** with **Depth 15**.

3. Right-click the back face (80 × 30) › **New Sketch on this face**. Draw a **Ø10** circle, **40** mm from the side and **9** mm below the top edge. **Finish Sketch**, then cut it with **Depth 38**, so it stops exactly in the slot.
4. Click the four display buttons at the right of the top bar one by one:
   - **Shaded**: you see only a hole in the back.
   - **Hidden edges**: the tunnel now shows through the block all the way to the slot.
   - **Wireframe**: every edge, including the round channel.
   - **Shaded + Edges**: back to the normal working view.



Slot phone stand, 80 x 70 x 30 mm, with the cable hole at the back

Challenge: Your phone has a thick case? Edit the slot sketch to **14** mm and check the result in **Hidden edges**.

See also

- Realistic Render
- System Options › Color Scheme

Realistic Render and Save Render Standard

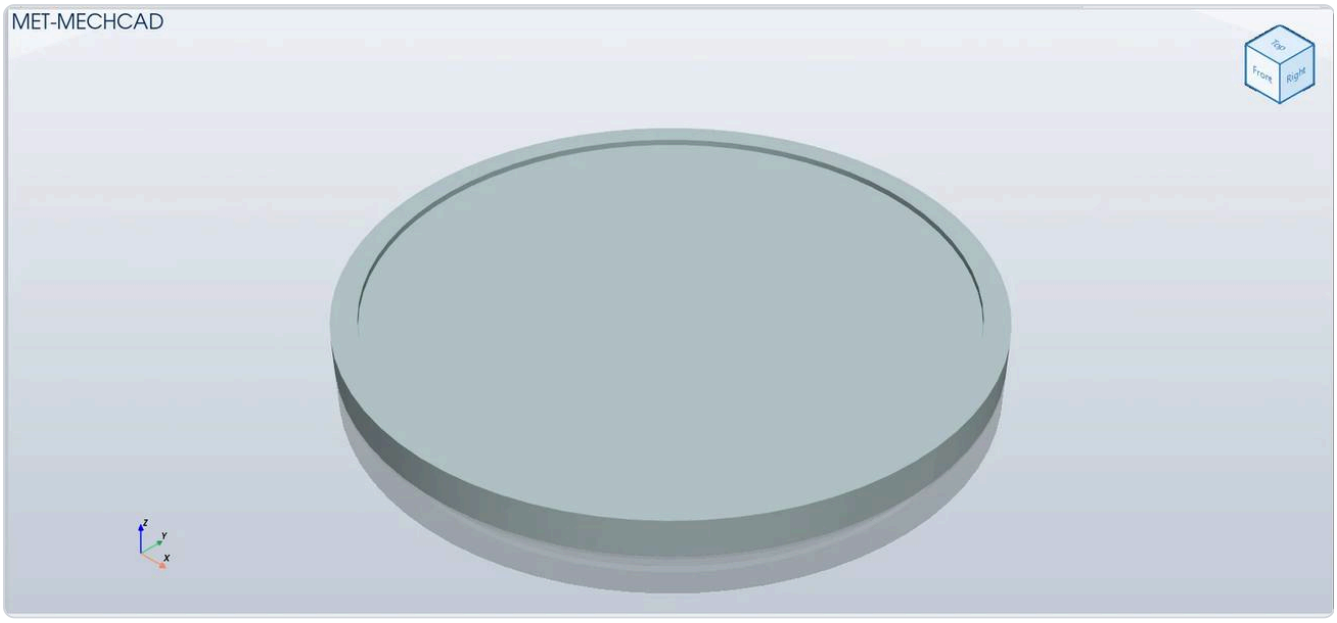
Turn on a studio-style realistic view and save a high-resolution PNG image.

What it does

Realistic Render changes the view to studio lighting with a soft background, shadows and a ground reflection. **Save Render** saves the current view as a PNG at twice the screen resolution.

Where to find it

- Part Design: **Model › Render** or **View › Render › Realistic Render** / **Save Render**
- Assembly: **View & Inspect › View › Realistic Render** / **Save Render**



Realistic Render mode

Steps

1. Set up the view the way you want.
2. Click **Realistic Render** (click it again to return to the working view).
3. Click **Save Render** and choose a `.png` file name.

Related option	Meaning
Shadows	Turns shadows on/off
Ambient Occl.	Shading in corners and crevices (contact shading)
Ground Reflection	Reflection on the ground
Material	Material (color, gloss, transparency). See Part Color / Material / Opacity

Try it: Studio shot of an aluminium desk coaster

A 100 mm coaster with a shallow recess for a cork insert. Turn it into a product photo you could post online.

1. Sketch on **XY** a circle **Ø100** at the origin and **Extrude 6** mm.
2. Right-click the top face › **New Sketch on this face**. Draw a circle **Ø92** on the same centre. **Finish Sketch**, then **Cut / Remove › Extrude 1** mm. This is the recess for the cork.
3. Choose **View › Appearance › Material › AL6061**.
4. Click the **Isometric** button, then rotate a little with the right mouse button until the recess catches the light.
5. Click **Realistic Render**. The view changes to studio lighting with shadows and a ground reflection.
6. Click **Save Render** and save `coaster.png`. The image is twice the screen resolution.

7. Click **Realistic Render** again to go back to the working view.



Desk coaster, Ø100 x 6 mm, with a 1 mm recess for cork

Challenge: Make a second render with **Black Anodized** and compare the two pictures side by side.

See also

- Display modes
- Part Color, Material and Opacity

System Options (Program Settings) Standard

Every System Options page: General, Mouse Interaction, Keyboard Shortcuts, Sketching, Design Viewing and Color Scheme.

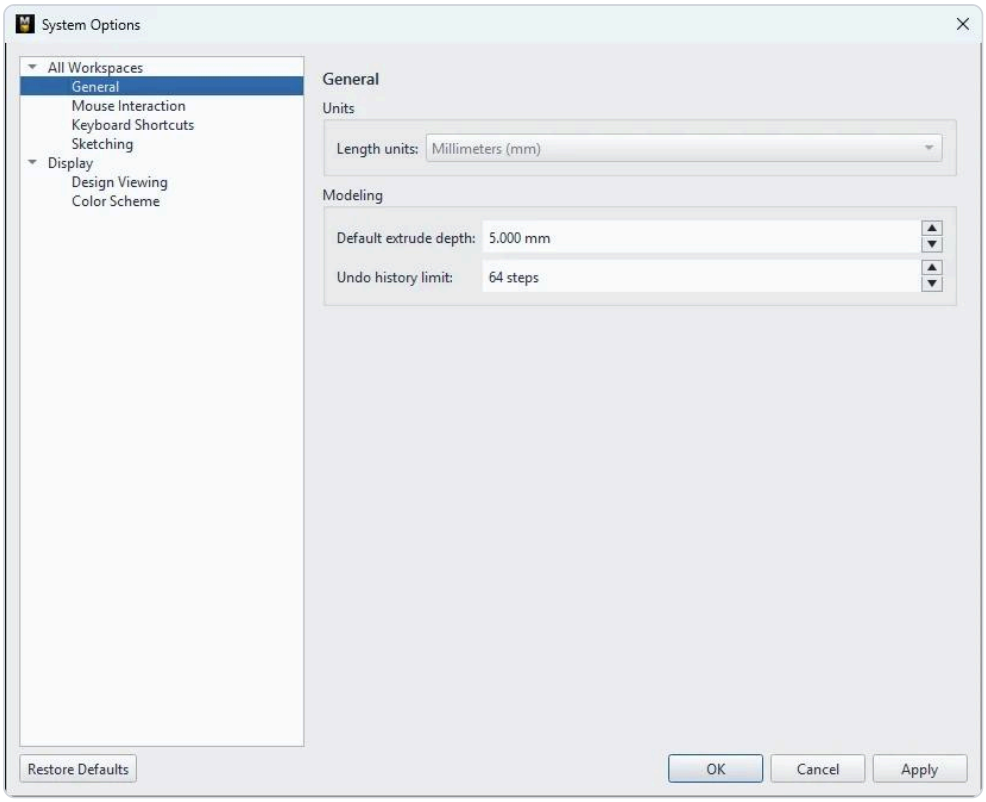
What it does

Sets how MECHCAD behaves. The values are saved and apply immediately to every open view.

Where to find it

File › System Options... Buttons at the bottom: **OK**, **Cancel**, **Apply** and **Restore Defaults** (restores all defaults; takes effect when you click OK or Apply).

All Workspaces › General

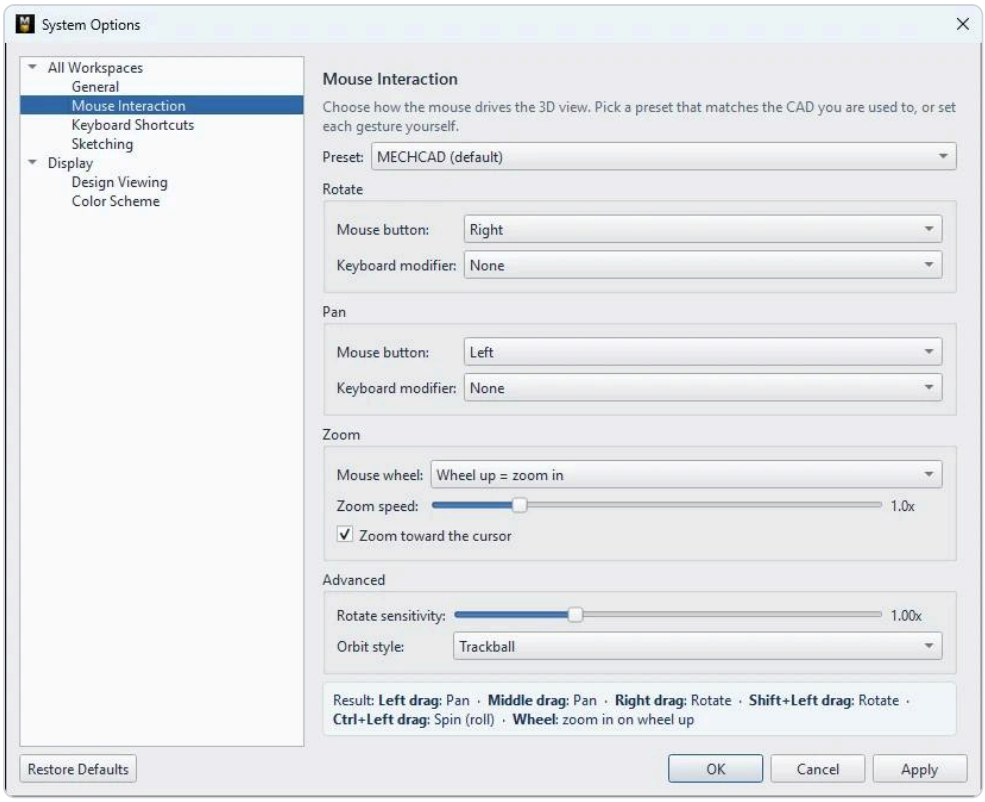


General

Option	Default	Meaning
Length units	Millimeters (mm)	Length unit. MECHCAD always uses millimeters (it cannot be changed).
Default extrude depth	5.0 mm	Default depth of an Extrude
Undo history limit	64 steps	Number of steps you can undo (5–500)

All Workspaces › Mouse Interaction

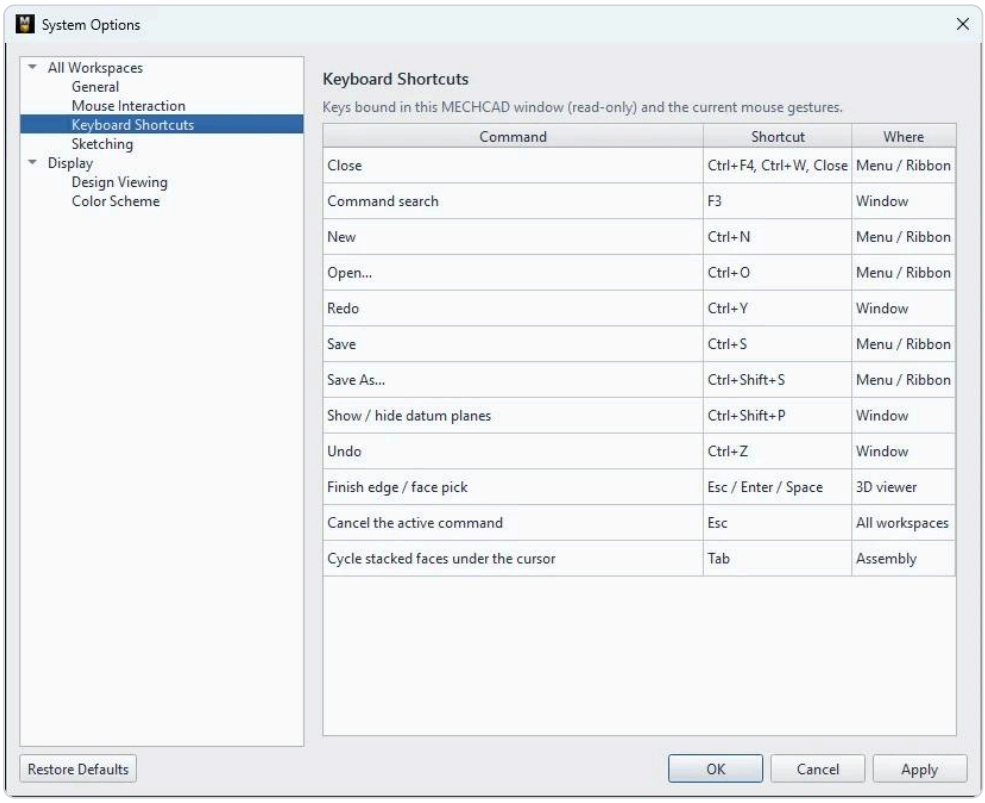
For details, see Mouse use and view rotation.



Mouse Interaction

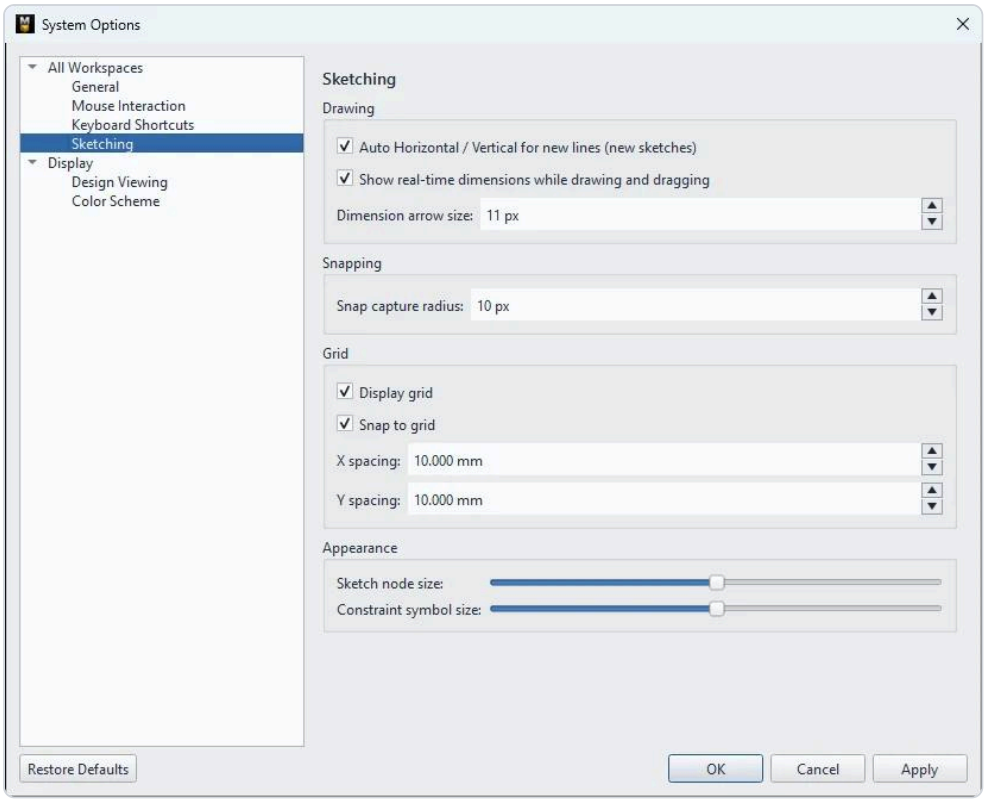
All Workspaces › Keyboard Shortcuts

A read-only table of keyboard shortcuts. See Keyboard shortcuts.



Keyboard Shortcuts

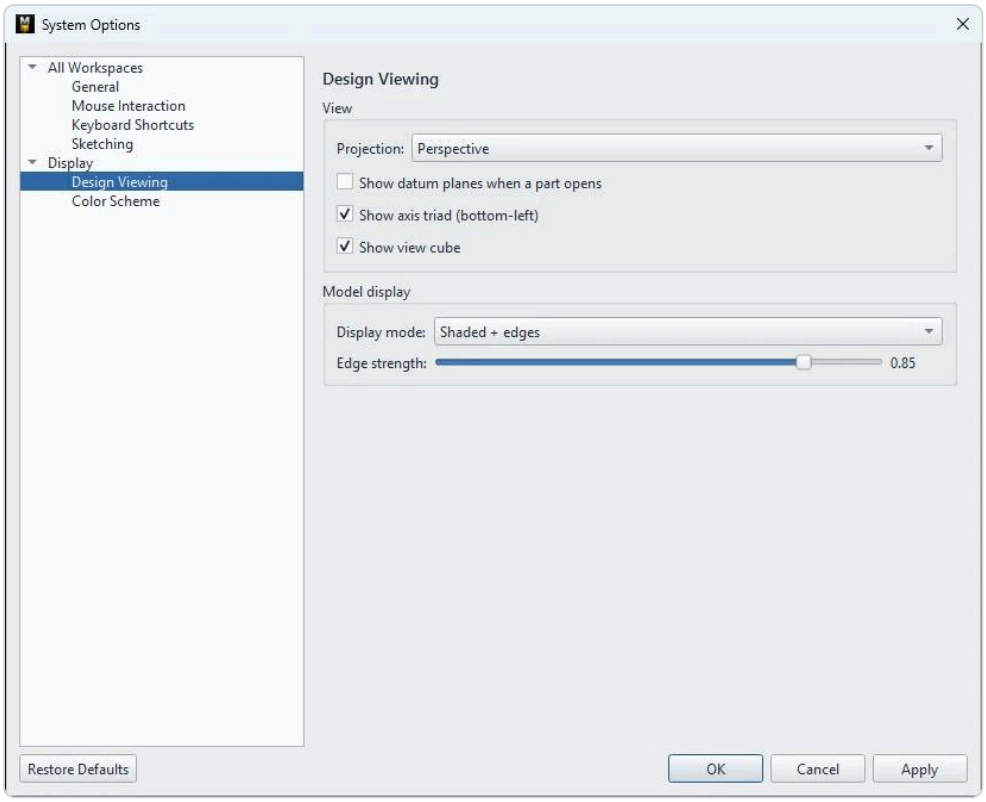
All Workspaces › Sketching



Sketching

Option	Default	Meaning
Auto Horizontal / Vertical for new lines (new sketches)	On	Lines that are nearly horizontal/vertical are straightened and get a constraint
Show real-time dimensions while drawing and dragging	On	Shows sizes while you draw/drag
Dimension arrow size	11 px	Size of the dimension arrowheads (4–40)
Snap capture radius	10 px	Distance at which the cursor snaps to a snap point (2–40)
Display grid / Snap to grid	On	Shows the grid and snaps to it (used in the 2D editor; a sketch on a 3D face always hides the grid)
X spacing / Y spacing	10 mm	Grid spacing
Sketch node size / Constraint symbol size	2	Size of points and constraint symbols (1–3)

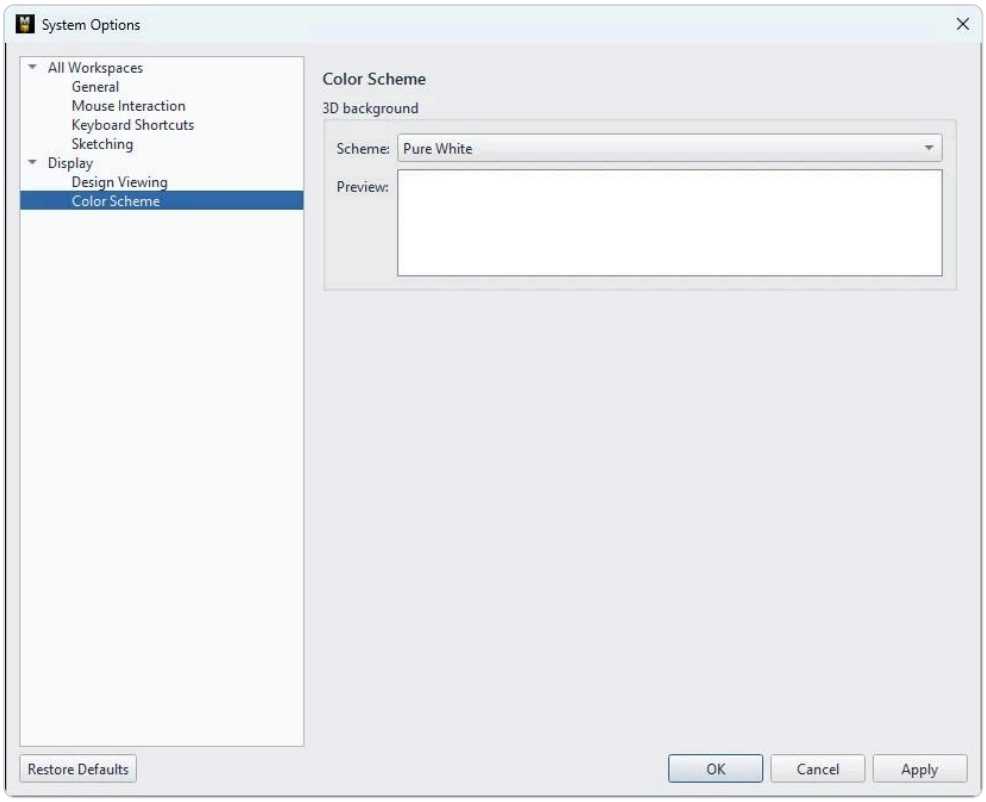
Display › Design Viewing



Design Viewing

Option	Default	Meaning
Projection	Perspective	Default projection type (Perspective / Orthographic)
Show datum planes when a part opens	Off	Shows the reference planes when a part opens
Show axis triad (bottom-left)	On	XYZ axes at the bottom left
Show view cube	On	View Cube at the top right
Display mode	Shaded + edges	Default display mode
Edge strength	0.85	Edge line intensity (0.1–1.0)

Display › Color Scheme



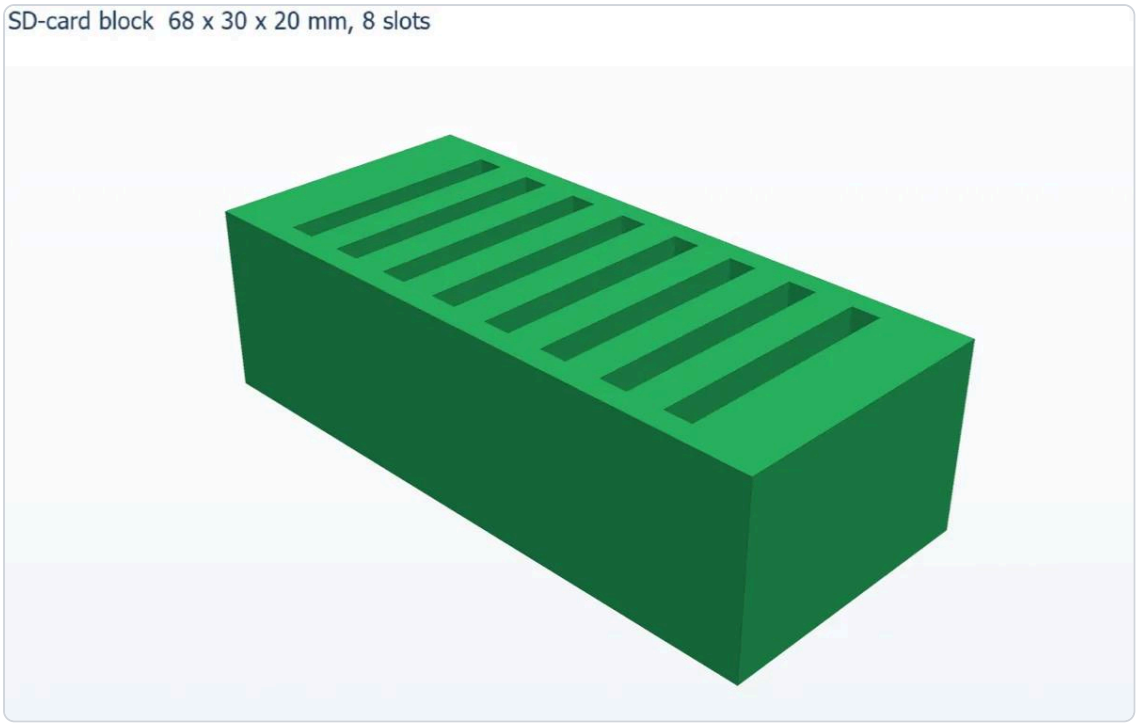
Color Scheme

Choose the 3D view background: Pure White (default), Default, Very Light Background, Light Background, Dark Background, Very Dark Background, Alibre Classic, MET Blue, with a preview image.

Try it: SD-card block with your own default settings

Set MECHCAD up for the way you work, then build an 8-slot SD-card block and see the settings take effect.

1. Open **File › System Options...**.
2. In **General**, set **Default extrude depth** to **20 mm** and **Undo history limit** to **200**.
3. In **Color Scheme**, choose **MET Blue**. Click **Apply**: the 3D view changes immediately. Click **OK**.
4. Sketch on **XY** a rectangle **68 × 30**, **Finish Sketch** and click **Extrude**: **Depth** is already **20**. Click **OK**.
5. Right-click the top face › **New Sketch on this face**: rectangle **3 × 25**, **8** mm from the left edge and **2.5** mm from the front edge. **Finish Sketch**.
6. Click **Cut / Remove › Extrude**. The box suggests 20, but that would cut through the bottom, so type **15**.
7. **Linear Pattern**: **Count 8**, **Spacing 7**, **Direction X**, **OK**.



SD-card block, 68 x 30 x 20 mm, with 8 slots

Challenge: Click **Restore Defaults** then **OK** and open a new Extrude: the depth is back to 5 mm.

See also

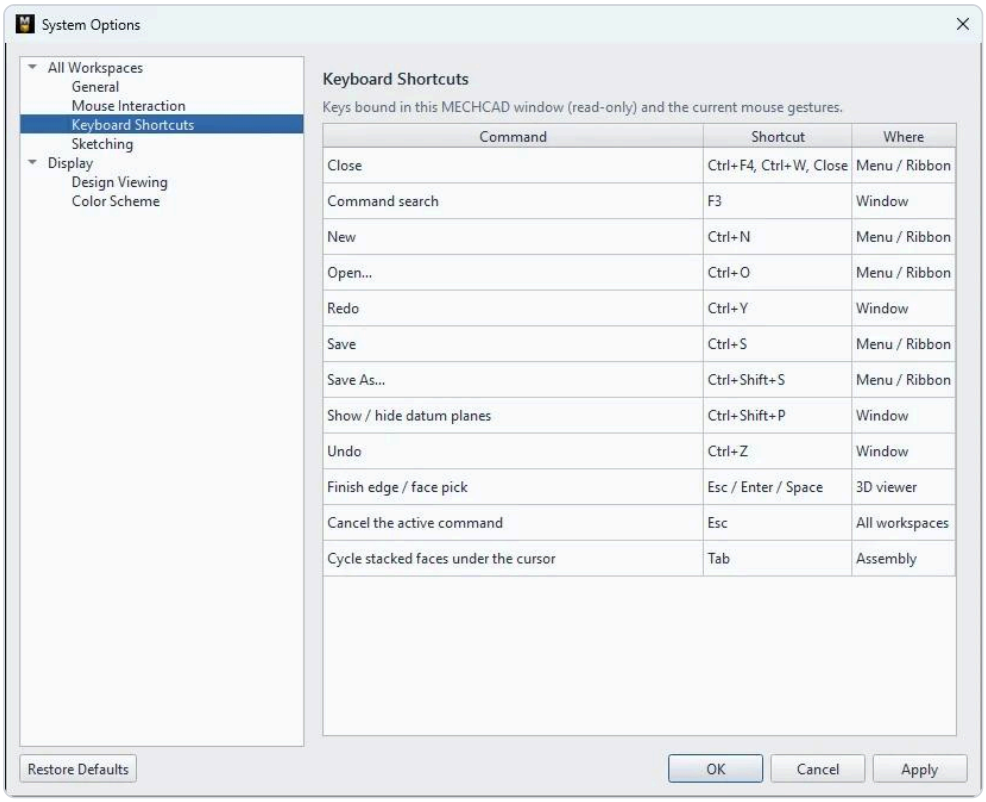
- Mouse use and view rotation
- Keyboard shortcuts
- Display modes

Keyboard Shortcuts Standard

The most-used keyboard shortcuts in every MECHCAD workbench.

What it does

Lists the MECHCAD keyboard shortcuts. The shortcuts are fixed. You can see the list at **File › System Options... › Keyboard Shortcuts**.



System Options › Keyboard Shortcuts

General

Key	Command
Ctrl+N	New
Ctrl+O	Open...
Ctrl+S	Save
Ctrl+Shift+S	Save As...
Ctrl+Z	Undo
Ctrl+Y / Ctrl+Shift+Z	Redo
F3	Command search
Ctrl+Shift+P	Show / hide datum planes
Esc	Cancel the active command
Esc / Enter / Space	Finish edge/face selection (Fillet, Chamfer, Shell ...)
Delete / Backspace	Delete the selection

Sketch

Key	Command
L	Line
C	Circle

Key	Command
R	Rectangle
D	Smart Dimension
Tab / Shift+Tab	Switch between input fields (Dynamic input)
Enter	Create the shape from the typed values / go to the next step of Mirror, Pattern
Esc (once)	Finish the line you are drawing
Esc (twice)	Return to Select
Hold Alt	Turn snaps off temporarily
R / D (while dimensioning a circle)	Switch between radius / diameter

Assembly

Key	Command
Tab	Cycle through stacked faces under the cursor while picking a mate
Esc	Cancel a drag / finish the Mate command
Shift + drag a component	Rotate the component
Ctrl + drag a component	Copy, then drag

Drawing

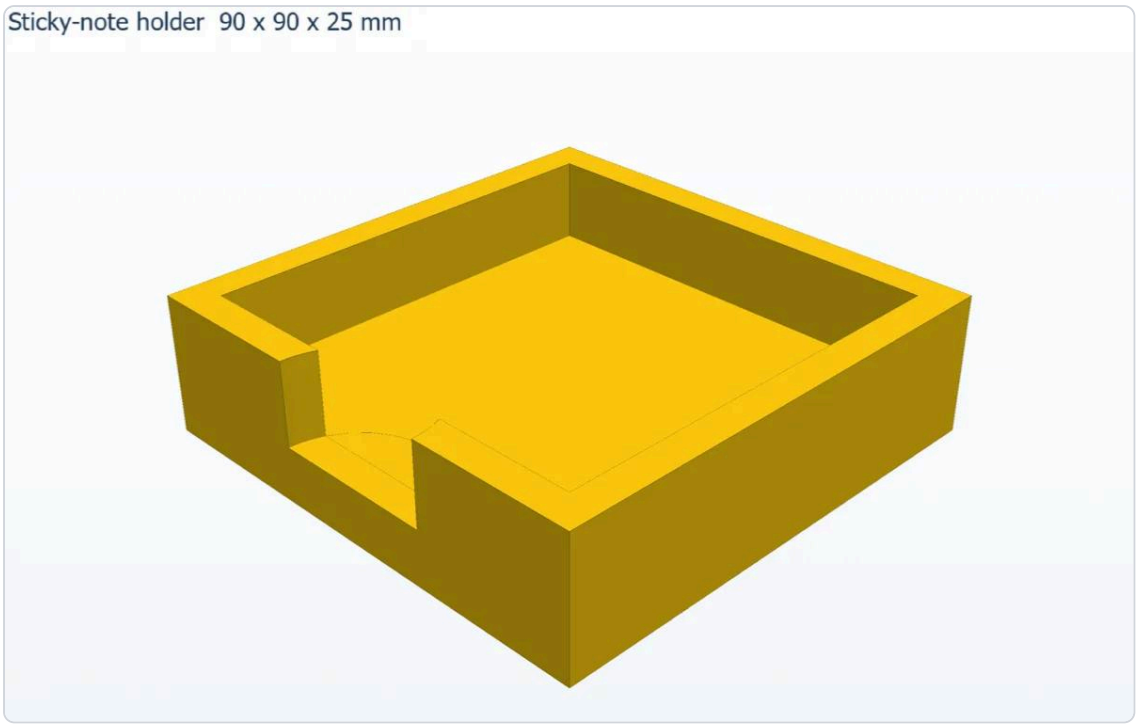
Key	Command
Esc	Deselect a dimension / exit the tool
Delete	Delete the selection
Ctrl+Z / Ctrl+Y	Undo / Redo (up to 60 steps)
Double-click text	Edit the text (Edit Text)

Try it: Sticky-note holder (almost without the mouse)

A tray for a standard 76 mm sticky-note pad, built mostly from the keyboard. Once these keys are in your fingers, every sketch gets faster.

1. Press **Ctrl+N**, then start a sketch on the **XY** plane.
2. Press **R**, click the origin, type **90**, **Tab**, **90**, **Enter**. Press **Esc** twice to go back to Select.
3. **Finish Sketch** and **Extrude 25** mm.
4. Right-click the top face › **New Sketch on this face**. Press **R**, click near the lower-left corner, type **78**, **Tab**, **78**, **Enter**, then **Esc** twice.
5. Press **D** and set the pocket **6** mm from the left edge and **6** mm from the front edge. **Finish Sketch**, then **Cut / Remove › Extrude** with **Depth 15**.
6. Sketch on the top face again. Press **C**, click the midpoint of the front edge, type **30** and press **Enter**. Cut it **15** mm deep: this is the finger notch for grabbing the pad.

7. Made a mistake? **Ctrl+Z** undoes, **Ctrl+Y** redoes. Finish with **Ctrl+S**.



Sticky-note holder, 90 x 90 x 25 mm, with a finger notch

Challenge: Press **F3**, type *bevel* and press **Enter** to run Chamfer, then put a 1 mm chamfer on the top outside edges.

See also

- Ribbon and Command Search
- Mouse use and view rotation

Measure (Distance, Angle and Size) Standard

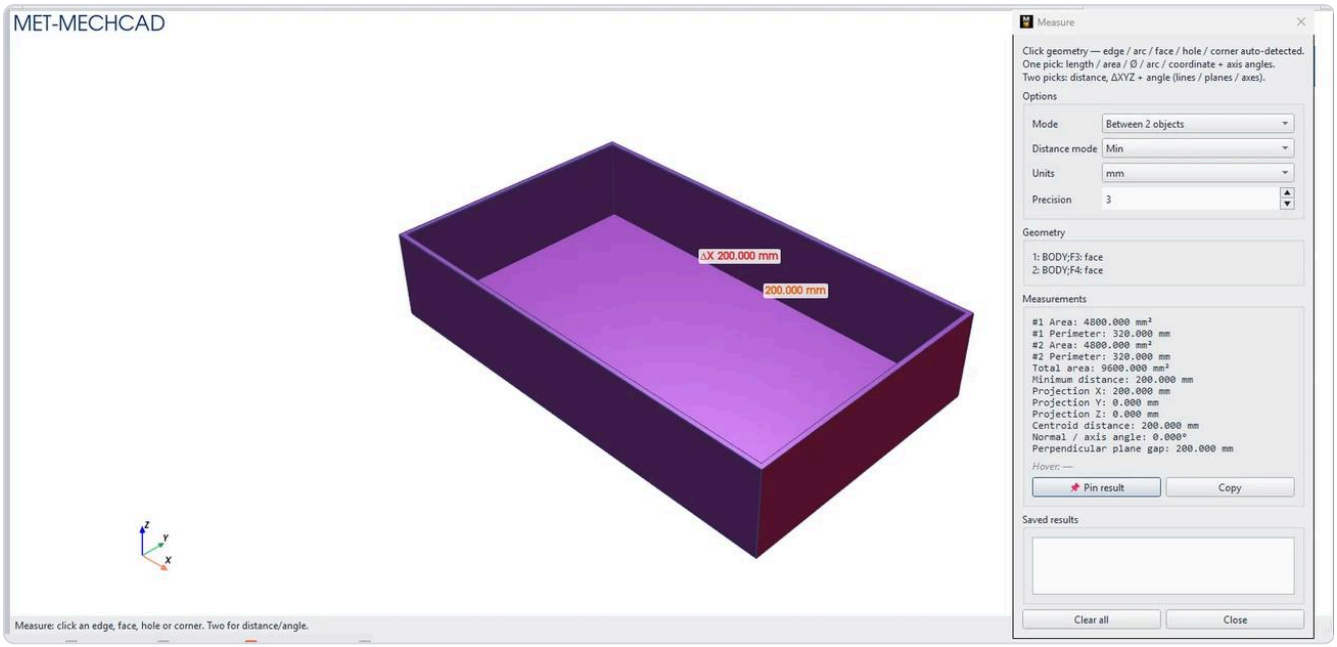
Measure length, diameter, distance between objects and angle in both parts and assemblies.

What it does

The measure tool detects the object type automatically (edge, face, hole, corner, point). One click gives the size of the object. Two clicks give the ΔX/ΔY/ΔZ distance and the angle.

Where to find it

- The **Measure** button on the top bar (every workbench)
- Part Design: **View › Inspect › Measure**, or right-click in the view › **Measure**
- Assembly: **View & Inspect › Inspect › Measure**

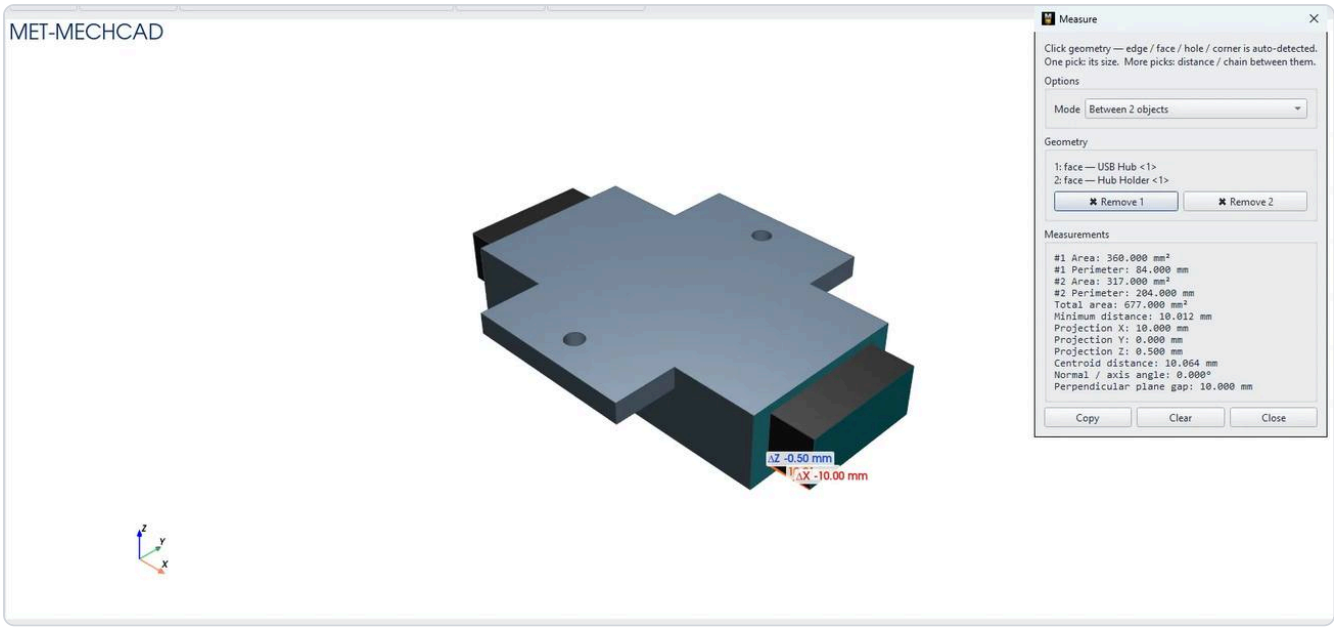


Measure panel in Part Design

Steps

1. Click **Measure**. The Measure panel opens at the top right.
2. Click the first object. The panel shows the length, area, Ø, arc or coordinates.
3. Click the second object. The panel shows the ΔX/ΔY/ΔZ distance and the angle.
4. Click 📌 **Pin result** to keep the result, or **Copy** to copy it.
5. Click **Close** when you are done.

Option	Meaning
Mode	**Between 2 objects** (measure between two objects) / **Chain** (continuous measuring) / **From First** (always measure from the first object)
Distance mode	Min / Max / Center-to-center / Perpendicular
Units	mm / cm / m / in
Precision	Number of decimals, 0–6 (default 3)
Saved results	Results you have pinned
Clear all	Clears all results



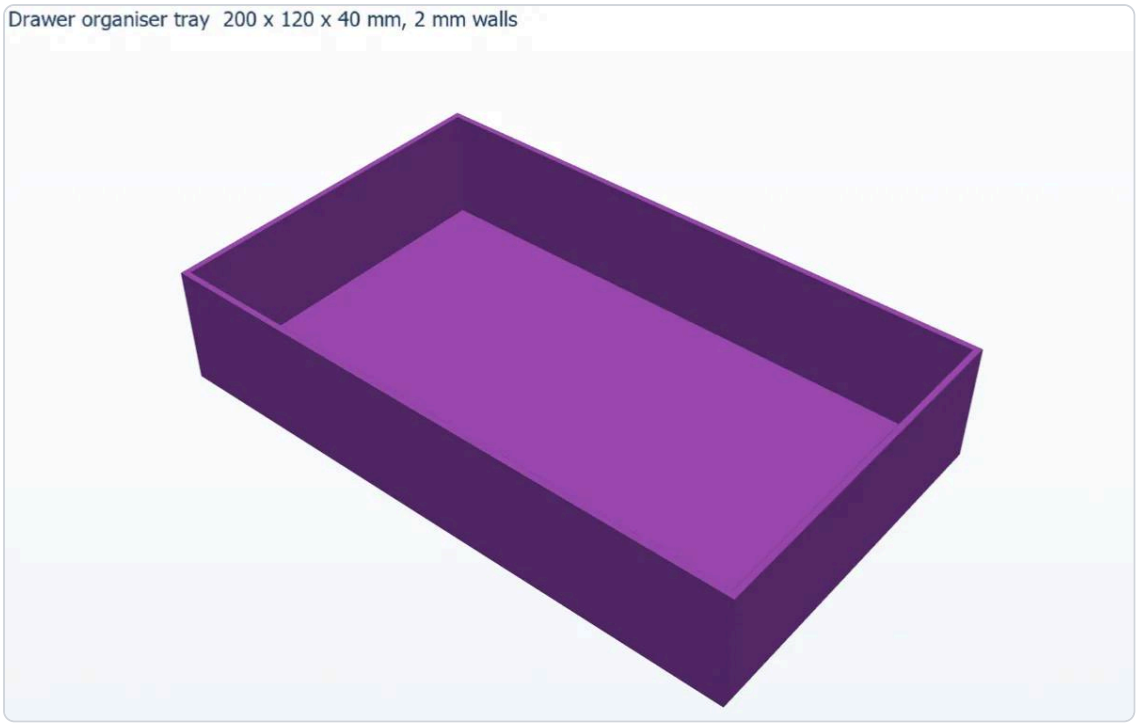
Measure panel in Assembly

> **Note: Size Box** (right-click in the view) shows the overall X × Y × Z size of the part, or of the selected component in an assembly.

Try it: Check a drawer organiser tray before you print it

Before you spend hours printing a tray, measure it: will it fit the drawer, and are the walls thick enough?

1. Sketch on **XY** a rectangle **200 × 120**, then **Extrude 40** mm.
2. Click **Model › Features › Shell**, click the top face, set **Wall thickness 2** and click **OK**.
3. Click **Measure**. Click the inside face of the left wall, then the inside face of the right wall. The panel shows **196.000** mm. Click 📌 **Pin result**.
4. Click the outer face and the inner face of the same wall: **2.000** mm.
5. Set **Mode** to **From First**, click the inside floor, then the top rim: the usable depth is **38.000** mm.
6. Change **Units** to **in**: 196 mm shows as about **7.717 in**. Switch back to **mm** and click **Close**.
7. Right-click in the view › **Size Box**: **200 × 120 × 40** mm, the size you need for the drawer.



Drawer organiser tray, 200 x 120 x 40 mm with 2 mm walls

Challenge: Measure the inside of your real drawer with a ruler and edit the Extrude sketch until **Size Box** leaves 2 mm of clearance on every side.

See also

- Interference and Clearance

Chapter 2 quiz: Interface & settings

Choose the one best answer. Answers are in Appendix A

1. **Which key opens Command Search so you can find a command by name?**

   A. F3 **B.** F1
   C. Ctrl+F **D.** F5

2. **With the default MECHCAD mouse preset, how do you rotate the 3D view?**

   A. Drag with the left button on empty space **B.** Drag with the right button
   C. Drag with the middle button **D.** Turn the mouse wheel

3. **What is the default Undo history limit in System Options › General?**

   A. 20 steps **B.** 32 steps
   C. 64 steps **D.** 500 steps

4. **Which shortcut shows or hides the datum planes?**

   A. Ctrl+P **B.** Ctrl+D
   C. F4 **D.** Ctrl+Shift+P

5. **In the Measure tool, what do you get after you click a second object?**

   A. The ΔX/ΔY/ΔZ distance and the angle **B.** The mass of both objects
   C. The total area of both faces **D.** The overall X × Y × Z size of the part

6. **Which display mode is the default?**

   A. Shaded **B.** Shaded + Edges
   C. Hidden edges **D.** Wireframe

CHAPTER 3

Sketch

1. Sketch Overview: Start, Finish and the Sketch Ribbon
   Start a sketch on a plane or face, the parts of the Sketch ribbon, and how to finish a sketch.
2. Sketch on a Face of the Part
   Create or edit a sketch on a flat face of a 3D part.
3. Line, Construction Line, Mid Line and Polyline
   Draw continuous straight lines, construction lines, lines from the midpoint, and polylines.
4. Rectangle, Center Rect and 3-Pt Rect
   Draw a rectangle from two corners, from the center, or from three corners (a tilted rectangle).
5. Circle, Center Arc and 3-Pt Arc
   Draw circles and arcs in a sketch.
6. Slot, Ellipse, 3-Pt Ellipse and Polygon
   Draw a slot, an ellipse and a polygon.
7. Spline, Point and Text in a Sketch
   Draw free-form curves, hole position points, and text as sketch shapes.
8. Dynamic Input: Type Sizes While You Draw
   Type a length, angle, width, height or diameter while drawing, and get the shape and dimension at once.
9. Smart Dimension and Smart Angle
   Dimension lines, circles, arcs, distances between objects, and angles.
10. Constraints (Geometric Relations)
    Apply Horizontal, Vertical, Coincident, Parallel, Perpendicular, Tangent, Concentric, Equal, Midpoint, Point On, Symmetric and Fix.
11. Trim, Extend, Fillet, Chamfer and Offset in a Sketch
    Trim, extend, round corners, chamfer and offset lines in a sketch.
12. Mirror, Linear Pattern and Circular Pattern in a Sketch
    Mirror and repeat shapes in a sketch along a line or around a center point.
13. Snaps and Auto H/V
    Snapping to end points, midpoints, centers, intersections and alignment guide lines.
14. Sketch Right-Click Menu

Commands you can call by right-clicking empty space, a line, a point or a dimension.

15. Project, Project All and Project Face

    Project the edges of the part or another sketch into a sketch as reference lines or real lines.

16. 3D Sketch (3D Paths)

    Create a 3D polyline or spline path for sweeping pipes, cables and frames.

17. Helix / Spiral curve

    Create a constant or variable pitch helix to use as a Sweep path for springs and threads.

Sketch Overview: Start, Finish and the Sketch Ribbon Standard

Start a sketch on a plane or face, the parts of the Sketch ribbon, and how to finish a sketch.

What it does

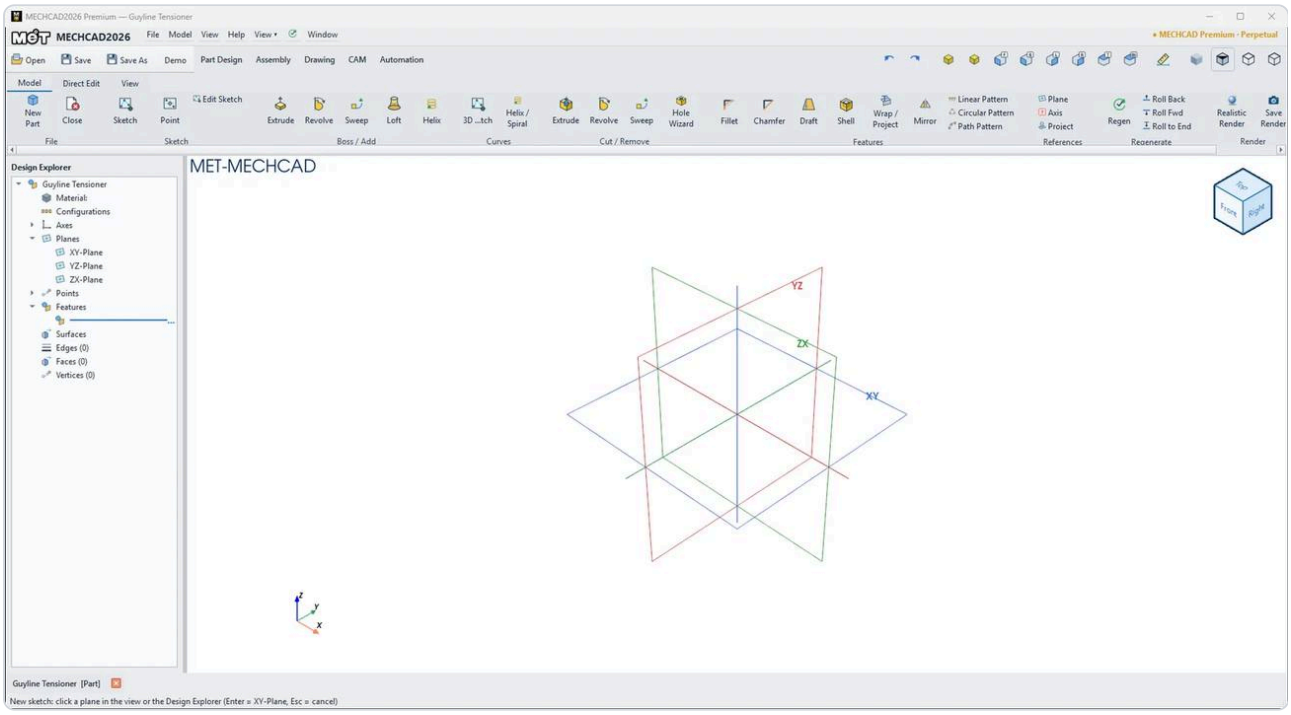
A sketch is a 2D shape on a plane or face. It is the profile for Extrude, Revolve, Sweep, Loft and other features. In MECHCAD you draw the sketch **directly in the 3D view** (sketch-in-place). The part stays visible, and you can rotate the view while you draw.

Where to find it

- **Model › Sketch › Sketch** (the *Activate 2D Sketch* command)
- **Model › Sketch › Edit Sketch** — edits the selected sketch
- **Model › Sketch › Point** — starts a sketch with the Point tool, for Hole Wizard hole positions
- Double-click a plane in Design Explorer or in the view, or right-click a face › **Insert Sketch on Surface**

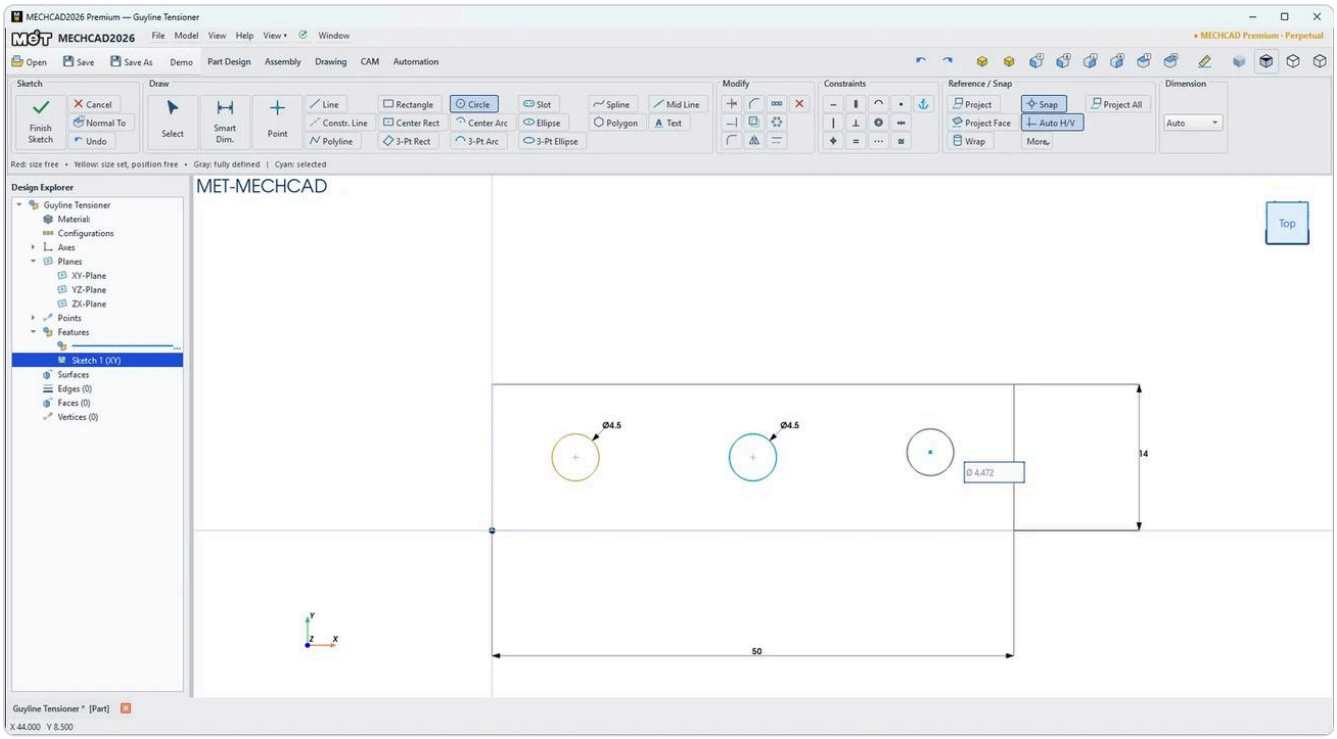
Start a sketch

1. Click **Sketch**. If you have not selected a plane, the main planes appear and the status bar shows *New sketch: click a plane in the view or the Design Explorer (Enter = XY-Plane, Esc = cancel)*.



Waiting for a plane for the new sketch

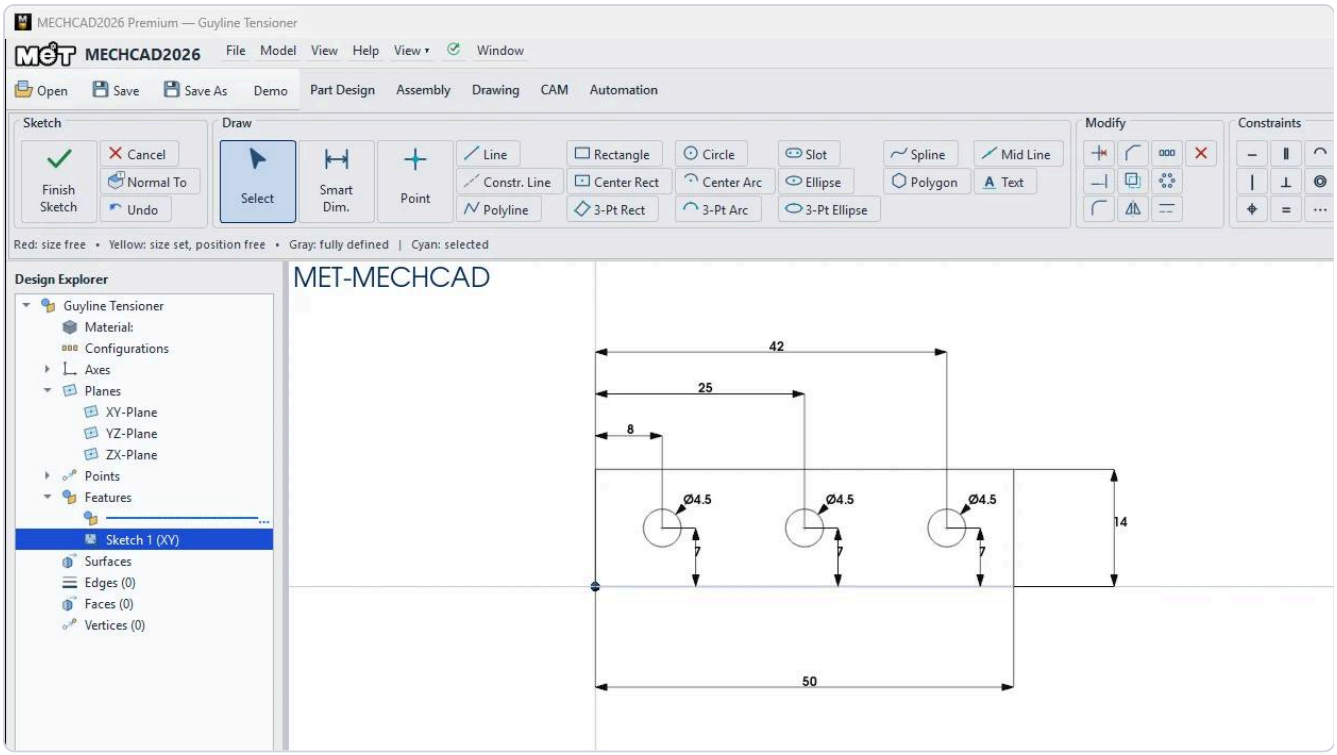
2. Click a plane in the view or in Design Explorer (press Enter = XY-Plane). The view turns perpendicular to the plane.
3. The ribbon changes to the Sketch ribbon.



Sketch ribbon and the shapes you can draw

Sketch ribbon

Group	Commands
Sketch	**Finish Sketch**, Cancel, Normal To, Undo
Draw	Select, Smart Dim., Point, Line, Constr. Line, Polyline, Rectangle, Center Rect, 3-Pt Rect, Circle, Center Arc, 3-Pt Arc, Slot, Ellipse, 3-Pt Ellipse, Spline, Polygon, Mid Line, Text
Modify	Trim, Extend, Fillet, Chamfer, Offset, Mirror, Linear Pattern, Circular Pattern, Toggle Construction, Delete Selection
Constraints	Horizontal, Vertical, Coincident, Parallel, Perpendicular, Tangent, Concentric, Equal, Midpoint, Point On, Symmetric, Fix
Reference / Snap	Project, Project All, Project Face, Wrap, Snap, Auto H/V, More
Dimension	Dimension type options: Auto / Size / Between / Between X / Between Y



Sketch group: Finish Sketch, Cancel, Normal To, Undo

Status colors (below the ribbon)

Color	Meaning
Red	Size is not set (size free)
Yellow	Size is set, but the position can still move
Grey	Fully defined
Cyan	Selected

Finish a sketch

- **Finish Sketch** — saves the sketch and returns to the Part ribbon
- **Cancel** — cancels this edit
- **Esc** — once finishes the line you are drawing, twice returns to Select

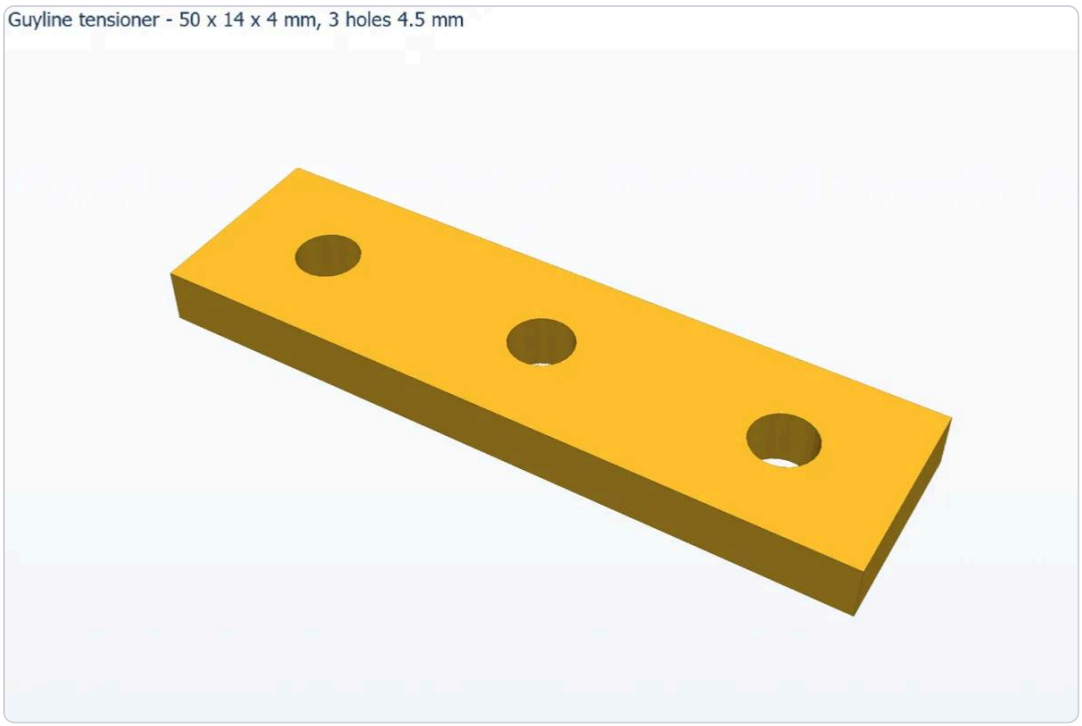
> **Note:** While you sketch, click = draw, drag = rotate the view, right-click = tool and constraint menu, type a number then Enter = create to size. The part does not move while you sketch.

Try it: Guyline tensioner

A three-hole guyline tensioner keeps a tent line tight without knots. It is the perfect first sketch: start, draw, check the colours, finish and extrude.

1. Click **Model › Sketch › Sketch** and press **Enter** to use the XY-Plane. The ribbon changes to the Sketch ribbon.
2. Press **R** (Rectangle), click the origin, type **50**, press Tab, type **14** and press Enter. Press Esc.
3. Press **C** (Circle) and draw three circles of Ø **4.5** inside the rectangle.

4. With **Smart Dimension**, place the centres **7** mm from the bottom edge and **8**, **25** and **42** mm from the left edge. Watch the status colour change from red to grey (Fully defined).
5. Click **Finish Sketch**, then **Model › Boss / Add › Extrude 4** mm.



Guyline tensioner: the first complete sketch-to-part workflow

Challenge: Before you finish the sketch, round the four corners with **Fillet** R5 so the tensioner does not catch on the tent fabric.

See also

- Sketch on a face of the part
- Dynamic input (type values while drawing)
- Smart Dimension

Sketch on a Face of the Part Standard

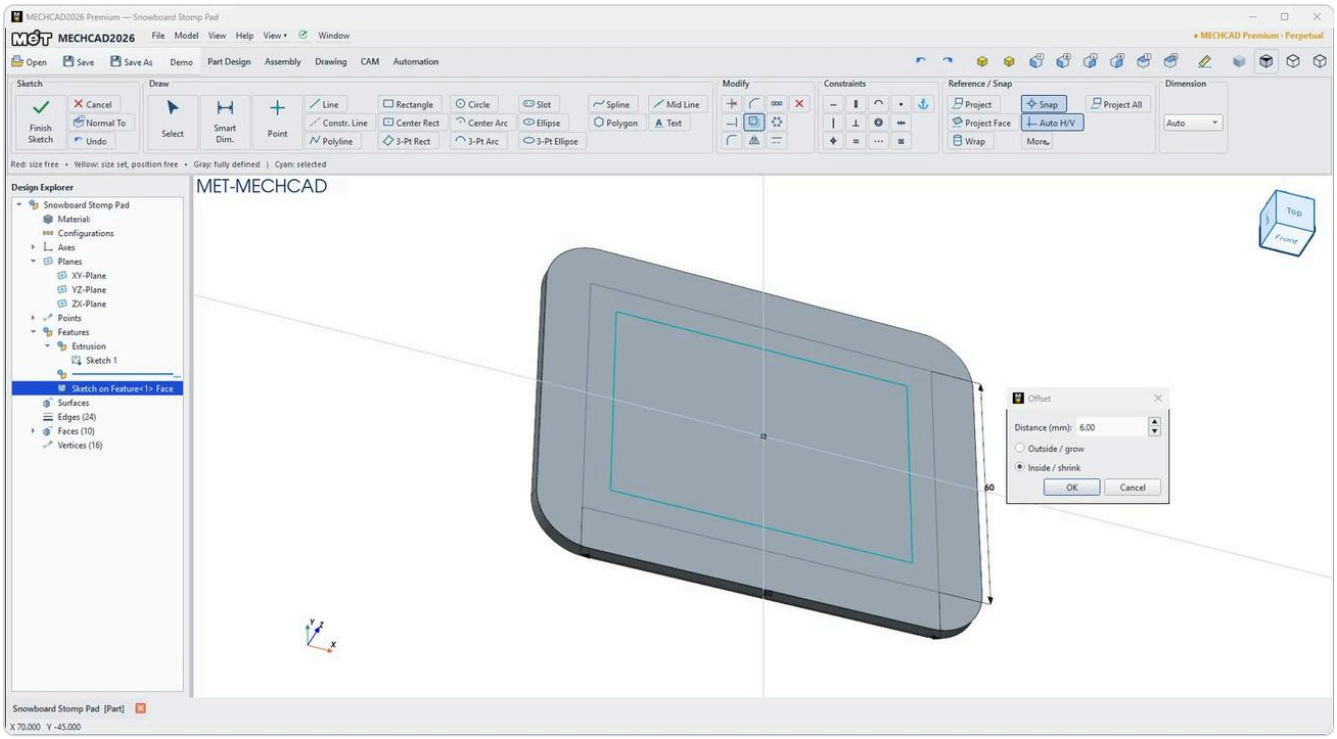
Create or edit a sketch on a flat face of a 3D part.

What it does

Draws a sketch on a flat face of an existing part, for example to make a groove, hole or boss on the top surface. The face becomes the sketch plane directly.

Where to find it

- Right-click a face in the view › **Insert Sketch on Surface**
- Design Explorer › **Faces** › right-click a face › **New Sketch on this face**
- Select a face, then click **Sketch**



Editing a sketch on the top face of a part

Steps

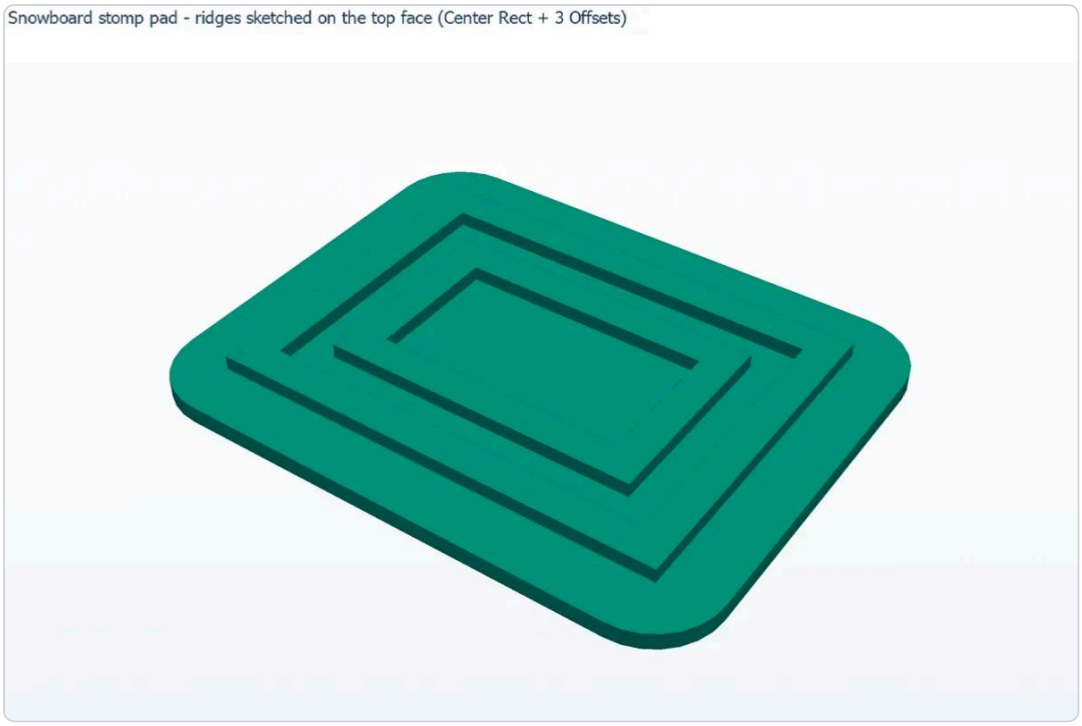
1. Right-click a flat face › **Insert Sketch on Surface**.
2. The view turns to look straight at the face. The edges of the face can be used as snap points right away.
3. Draw the shapes and add dimensions. Use **Project Face** if you want the face edges as dashed reference lines.
4. Click **Finish Sketch**, then create a feature such as Extrude Cut.

> **Note:** Right-clicking in the sketch area gives the command ⊥ **Plane View (look at the sketch)**, which turns the view perpendicular to the sketch.

Try it: Snowboard stomp pad

A stomp pad gives your free boot grip on the board. Sketch the raised frames straight on the top face of the pad. The face centre becomes the sketch origin, so everything is centred automatically.

1. Click **Sketch** and press Enter (XY-Plane). Press **R**, click the origin, type **100**, Tab, **80** and press Enter. Round the four corners with sketch **Fillet 12** mm. Click **Finish Sketch**, then **Extrude 3** mm.
2. Right-click the top face › **Insert Sketch on Surface**.
3. Choose **Center Rect**, click the centre of the face (snap), type **80**, Tab, **60** and press Enter.
4. Click **Offset**, click the rectangle, set Distance **6** and **Inside / shrink**, then click OK. Offset the new inner rectangle the same way, and then once more, so you have 4 nested rectangles.
5. Click **Finish Sketch**, then **Extrude 2.5** mm. The nested loops become two raised frames with grooves between them.



Snowboard stomp pad with ridges sketched on the top face

Challenge: Make a fifth nested loop (one more Offset) and see a small centre island appear.

See also

- Project to Sketch
- Sketch overview
- Smart Dimension
- Constraints

Line, Construction Line, Mid Line and Polyline Standard

Draw continuous straight lines, construction lines, lines from the midpoint, and polylines.

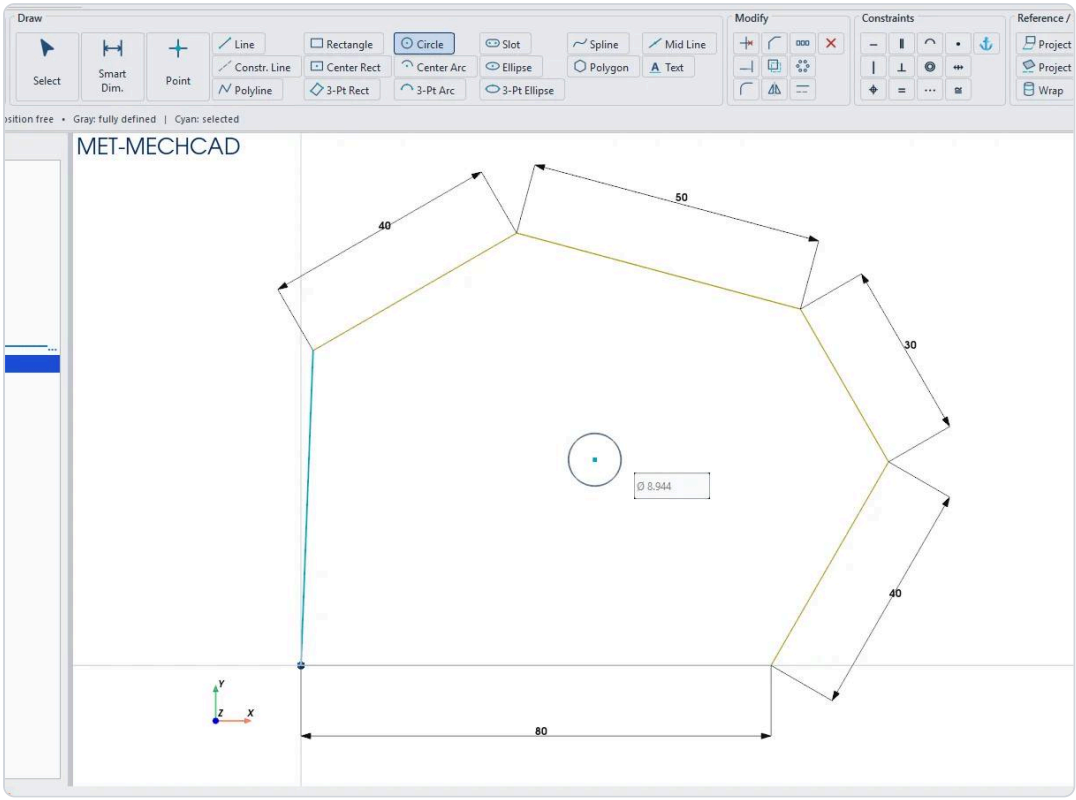
What it does

Draws straight lines in a sketch.

Tool	Shortcut	How to use it
Line	L	Click the start point, then click the next points one after another. Finish by closing the shape (*Profile closed*), double-clicking, clicking the last point again, or pressing Esc.
Constr. Line		Like Line, but it makes a construction line (dashed) that is not used to build a feature.
Mid Line		Click the midpoint, then click an end of the line (*Line by Middle*).
Polyline		Click vertices one after another. Double-click or press Esc to finish.

Where to find it

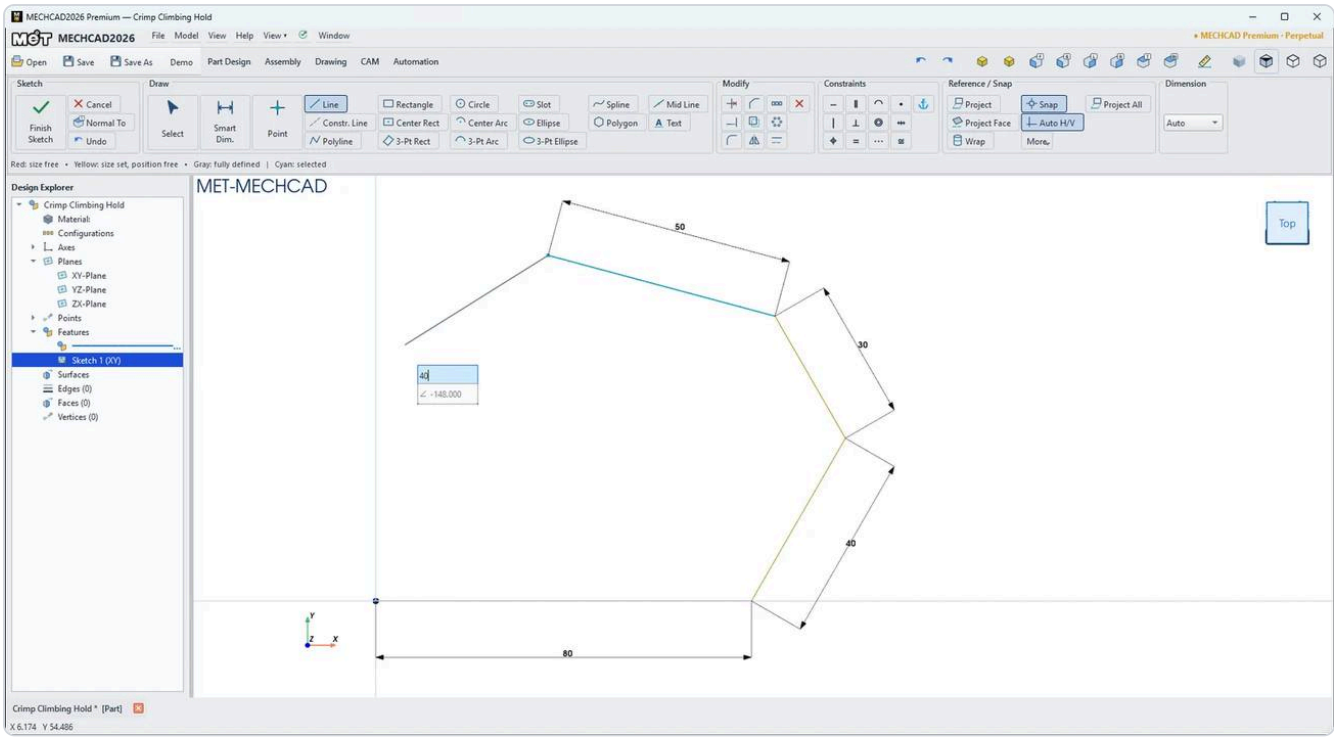
Sketch ribbon › **Draw**, or right-click on empty space



Draw group of the Sketch tab

Steps

1. Click **Line** (or press L).
2. Click the start point. While you drag, you see the **L** (length) and ∠ (angle) boxes. You can type values right away. See Dynamic input.
3. Click the next point. The new line continues from the end of the previous line.
4. Press Esc to end the chain. Press Esc again to return to Select.



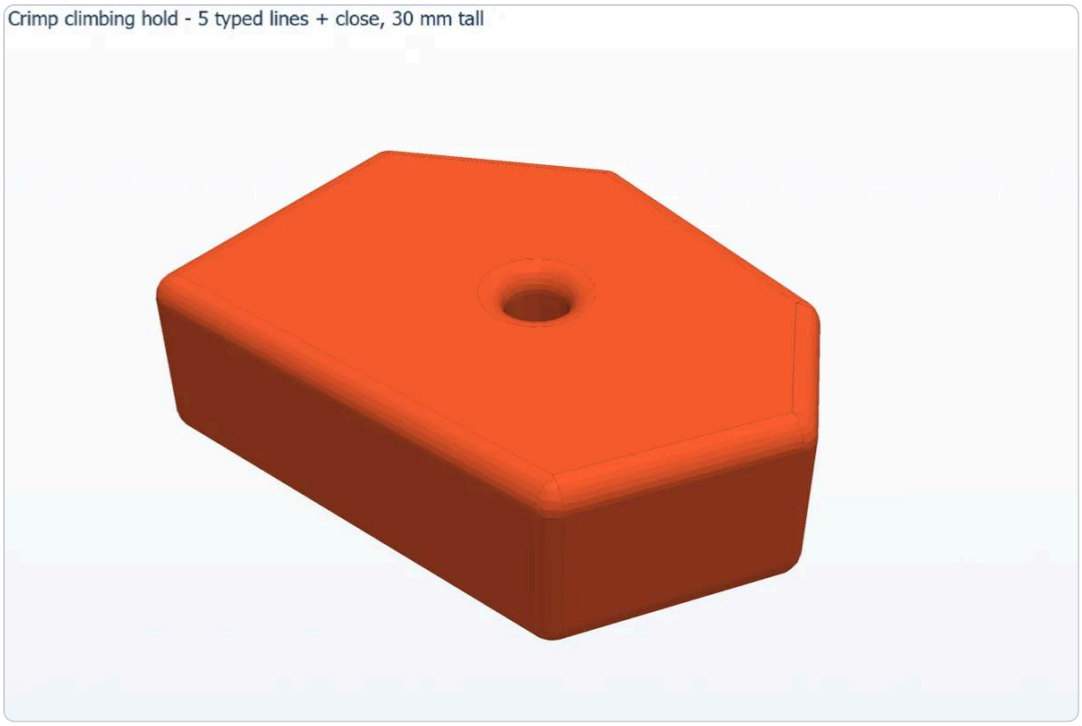
Drawing a continuous Line with the L and ∠ boxes

> **Note:** When **Auto H/V** is on, a line that is nearly horizontal or vertical is straightened and given a Horizontal/Vertical constraint automatically. End points also align with other points, with dashed guide lines.

Try it: Crimp climbing hold

A bouldering crimp is a faceted block with one sharp edge to pull on. With Line you can draw the whole outline in one chain by typing a length and an angle for each edge.

1. Click **Sketch** and press Enter (XY-Plane). Press **L** and click the origin.
2. For each edge, type the length, press Tab, type the angle and press Enter:
   - **80**, Tab, **0**, Enter
   - **40**, Tab, **60**, Enter
   - **30**, Tab, **120**, Enter
   - **50**, Tab, **165**, Enter
   - **40**, Tab, **210**, Enter
3. Click the start point (Endpoint snap) to close the outline. The status bar shows *Profile closed*.
4. Press **C**, click about (50, 35), type **10** and press Enter. This is the hole for the M10 hold bolt.
5. Click **Finish Sketch**, **Extrude 30** mm, then round all the edges with a **4** mm Fillet so the hold is easy on the hands.



Climbing hold drawn as one Line chain with typed lengths and angles

Challenge: Change the 165° edge to 150° and see how the crimp edge becomes sharper.

See also

- Sketch overview
- Smart Dimension
- Constraints

Rectangle, Center Rect and 3-Pt Rect Standard

Draw a rectangle from two corners, from the center, or from three corners (a tilted rectangle).

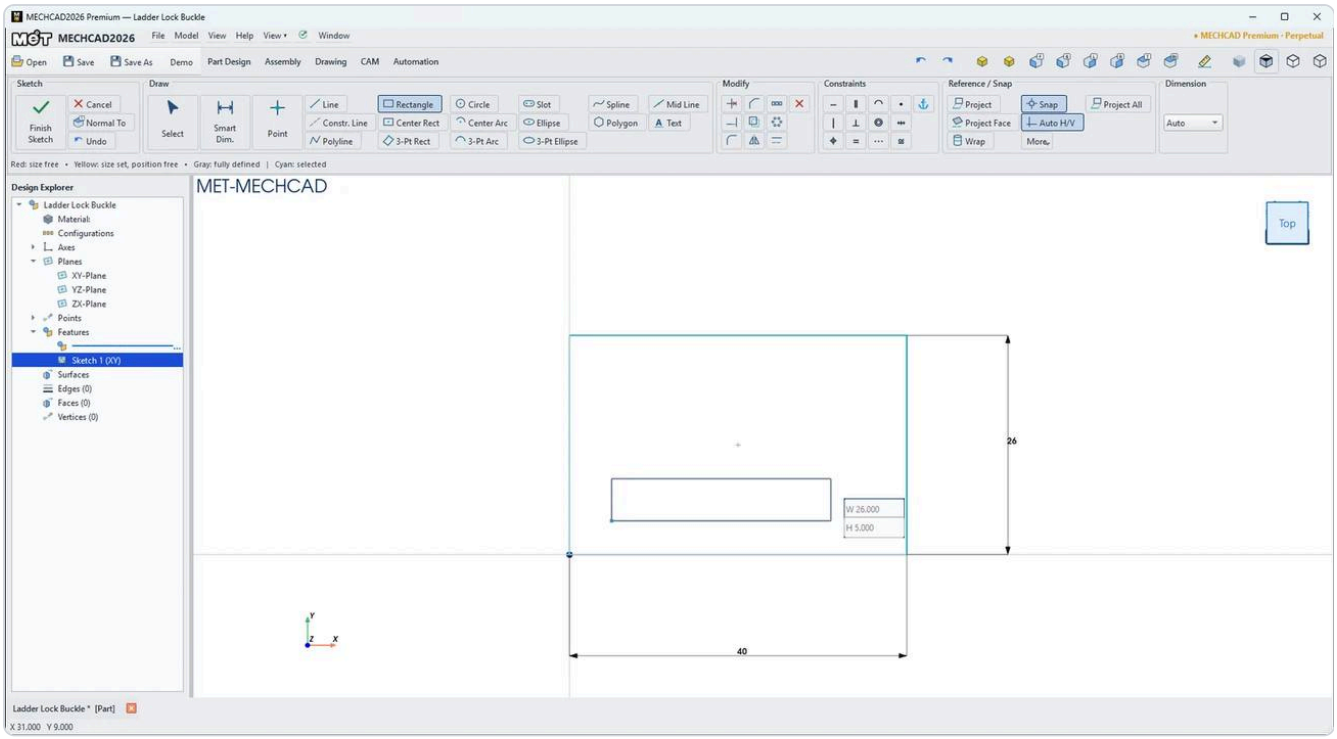
What it does

Draws a rectangle in three ways.

Tool	Shortcut	How to use it
Rectangle	R	Click the first corner, then click the opposite corner
Center Rect		Click the center point, then click a corner
3-Pt Rect		Click corner 1, corner 2 (sets the first side and the tilt angle), then corner 3

Where to find it

Sketch ribbon › **Draw**, or right-click › **Rectangle** (Rectangle by Corners / by Center / by 3 Corners)



Drawing a Rectangle

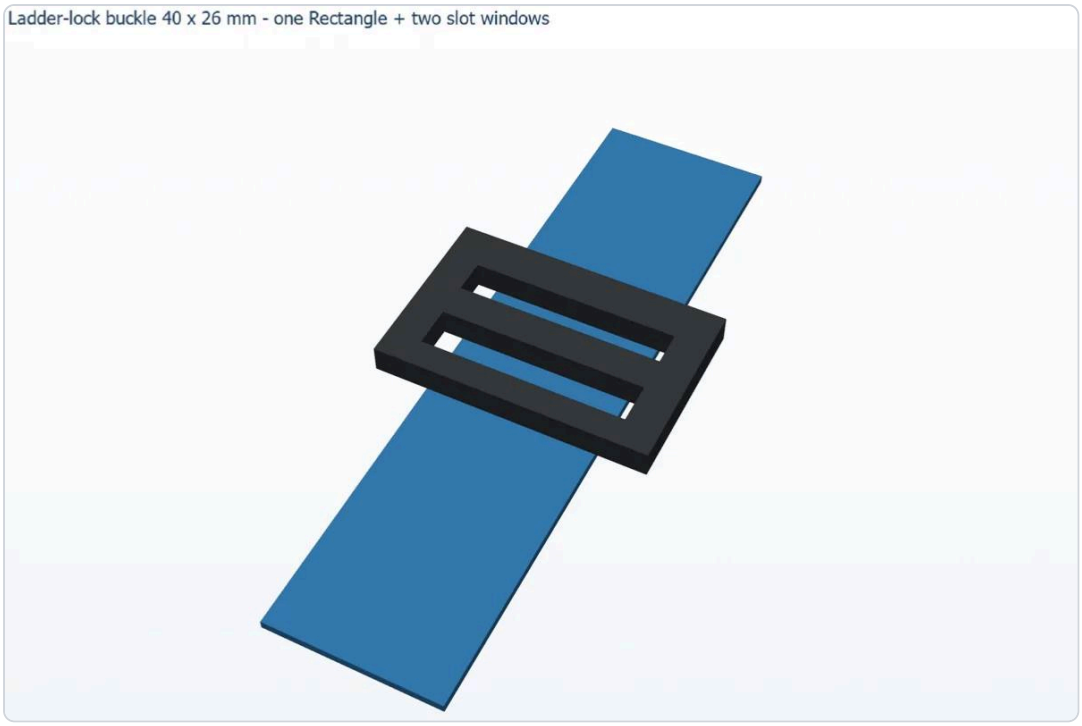
Steps

1. Click **Rectangle** (or press R).
2. Click the first corner. The **W** (width) and **H** (height) boxes appear next to the cursor.
3. Click the opposite corner, or type W, press Tab, type H, then press Enter to get the exact size with dimensions.

Try it: Ladder-lock buckle

The ladder-lock buckle on every backpack strap is just three rectangles: the frame and two webbing slots. It is a good way to try Rectangle and Center Rect.

1. Click **Sketch** and press Enter (XY-Plane). Press **R**, click the origin, type **40**, Tab, **26** and press Enter. This is the frame.
2. With **Rectangle** still active, click inside near (5, 4), type **30**, Tab, **6** and press Enter. This is the lower slot. Dimension its corner **5** mm from the left edge and **4** mm from the bottom edge.
3. Choose **Center Rect**, click near (20, 17.5), type **30**, Tab, **5** and press Enter. This is the upper slot. Dimension its centre **20** mm from the left edge and **17.5** mm from the bottom edge.
4. Click **Finish Sketch** and **Extrude 4** mm. The blue strap in the picture only shows the scale: 25 mm webbing.



Ladder-lock buckle: one Rectangle frame and two slot windows

Challenge: Use **3-Pt Rect** to add a pull tab tilted 15° on the bottom edge in a new sketch, and extrude it 2 mm.

See also

- Dynamic input
- Sketch overview
- Smart Dimension
- Constraints

Circle, Center Arc and 3-Pt Arc Standard

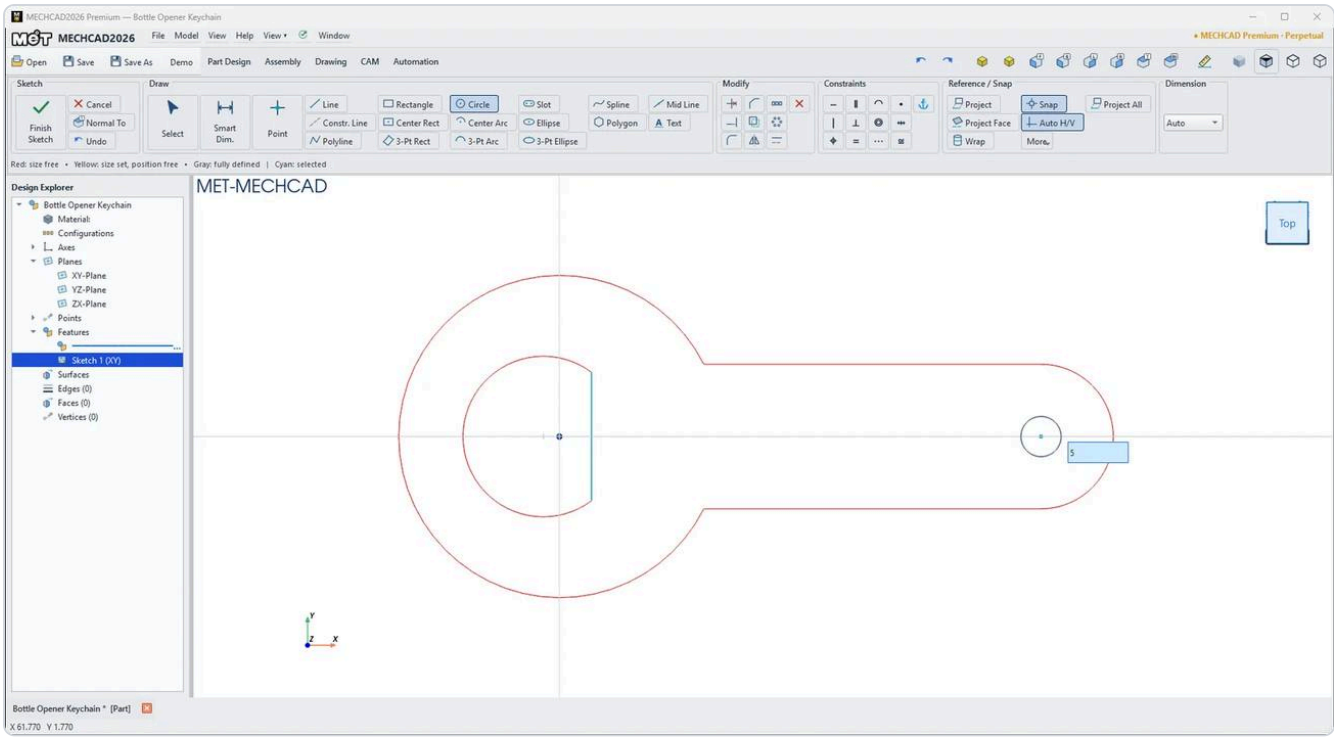
Draw circles and arcs in a sketch.

What it does

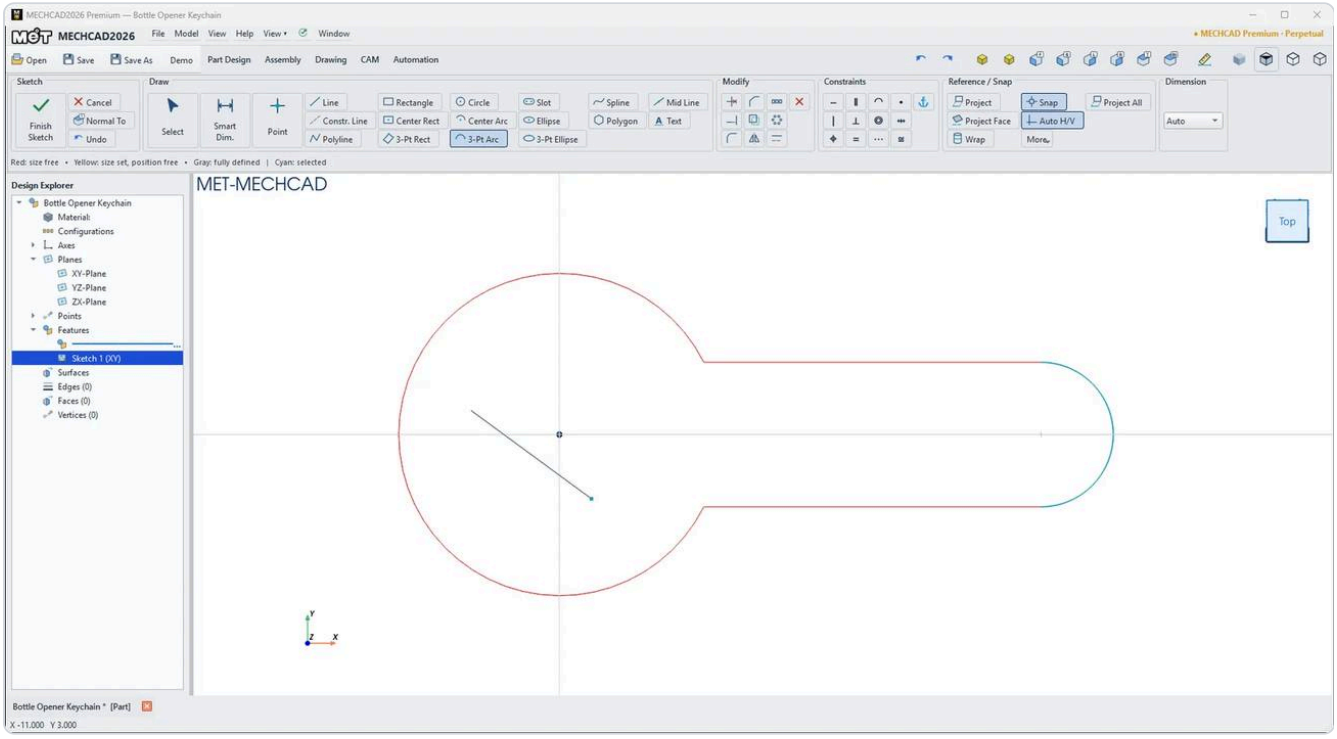
Tool	Shortcut	How to use it
Circle	C	Click the center point, then click a point on the circumference (you can type Ø)
Center Arc		Click the center point, the start point and the end point (*Arc by Center-Start-End*)
3-Pt Arc		Click the start point, the end point and a point on the arc (*Arc by 3 Points*)

Where to find it

Sketch ribbon › **Draw**, or right-click › **Circle** / **Arc**



Drawing a Circle



Drawing a 3-Pt Arc

Steps (Circle)

1. Click **Circle** (or press C).
2. Click the center point.
3. Drag outward and click, or type the **Ø** value and press Enter.

> **Note:** Apply Smart Dimension to a circle, then press **R** or **D** to switch between radius and diameter.

Try it: Bottle-opener keychain

This keychain bottle opener is made only from arcs, two lines and one circle. Cut it from 3 mm stainless steel and it really opens bottles.

1. Click **Sketch** and press Enter (XY-Plane). Choose **3-Pt Arc**. Click the start point at about (18, 9), the end point at about (18, −9), then a point on the arc at about (−20, 0), left of the origin. This is the round head, which is more than half a circle (Center Arc only draws the short way round, up to half a circle).
2. With **Line**, draw from the top end of the head arc to about (60, 9). Press Esc. Draw a second line from the bottom end to about (60, −9). Auto H/V makes both lines horizontal.
3. Choose **Center Arc** again. Click the centre at about (60, 0), then the end of the bottom line, then the end of the top line. This rounds off the tail.
4. Choose **3-Pt Arc**. Click (4, 8), then (4, −8), then a point at about (−12, 0). Close it with a **Line** from (4, −8) to (4, 8). This straight edge is the lip that grips the cap.
5. Press **C** (Circle), click the centre of the tail arc (Center snap), type **5** and press Enter. This is the key-ring hole.
6. With **Smart Dimension**, set the head arc to **R20**, the tail arc to **R9**, the distance between the two arc centres to **60** and the 3-Pt arc to **R10**. Add **Tangent** between the tail arc and each line if they are not tangent yet.
7. Click **Finish Sketch**, then **Extrude** the profile **3** mm.



Bottle-opener keychain made from 3-Pt Arcs, a Center Arc and a Circle

Challenge: Change the head to R22 and see the lines and the tail follow, because the dimensions and constraints drive the shape.

See also

- Sketch overview

- Smart Dimension
- Constraints

Slot, Ellipse, 3-Pt Ellipse and Polygon Standard

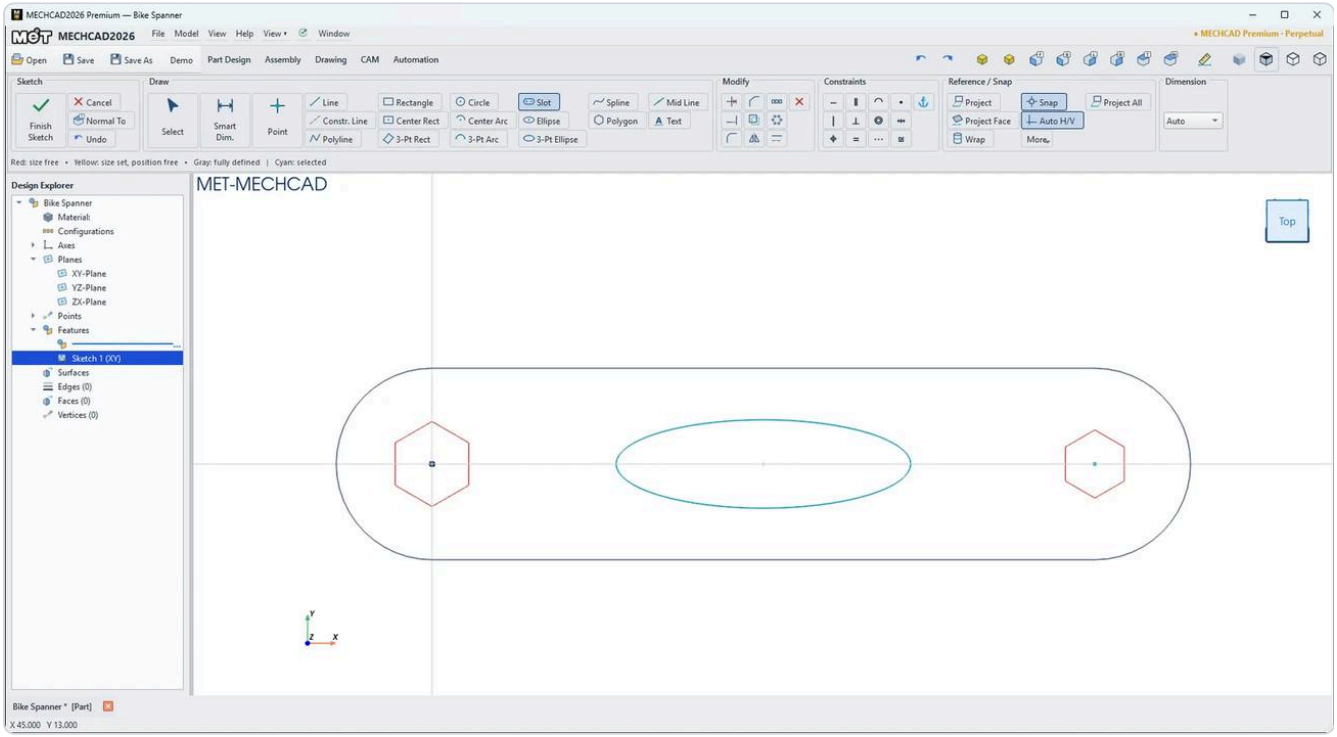
Draw a slot, an ellipse and a polygon.

What it does

Tool	How to use it
Slot (*Straight Slot*)	Click the center of one end, then the other end, then click a third time to set the width (the axis locks to horizontal/vertical when Auto H/V is on).
Ellipse (*Ellipse by Center*)	Click the center, an end of one axis, then a point that sets the size of the other axis.
3-Pt Ellipse	Click 3 points to define the ellipse.
Polygon	Click the center, then click a radius point. You get an inscribed hexagon.

Where to find it

Sketch ribbon › **Draw**, or right-click › **Ellipse**



Drawing a Slot (the image shows an ellipse and a hexagon already drawn)

> **Note:** In Part Design, Polygon is always a hexagon. If you need another number of sides, draw it with Line and add an Equal constraint.

Try it: Bike multi-tool spanner

A pocket spanner for bike trips needs a rounded outline, two hex sockets and a light grip window. That is exactly Slot, Polygon and Ellipse.

1. Click **Sketch** and press Enter (XY-Plane). Choose **Slot**. Click the origin, click (90, 0) and move the mouse to set the width, then click. With **Smart Dimension**, set **90** between the end centres and **R13** on an end arc (26 mm wide).
2. Choose **Polygon**. Click the centre of the left end arc (Center snap), then click a radius point. Dimension one side to **5.77** mm. A hexagon's side equals its corner radius, so this gives a **10 mm** socket across the flats.
3. Do the same at the right end with a side of **4.62** mm. This gives an **8 mm** socket.
4. Choose **Ellipse**. Click the centre at (45, 0), click an axis end 20 mm to the right at (65, 0), then click a point 6 mm above the centre. This is the grip window.
5. Click **Finish Sketch** and **Extrude 5** mm.



Bike spanner: Slot outline, two hexagons and an Ellipse window

Challenge: Change the 8 mm socket to 13 mm (side **7.51** mm) and widen the slot to **R16** so the bigger socket still has enough wall.

See also

- Sketch overview
- Smart Dimension
- Constraints

Spline, Point and Text in a Sketch Standard

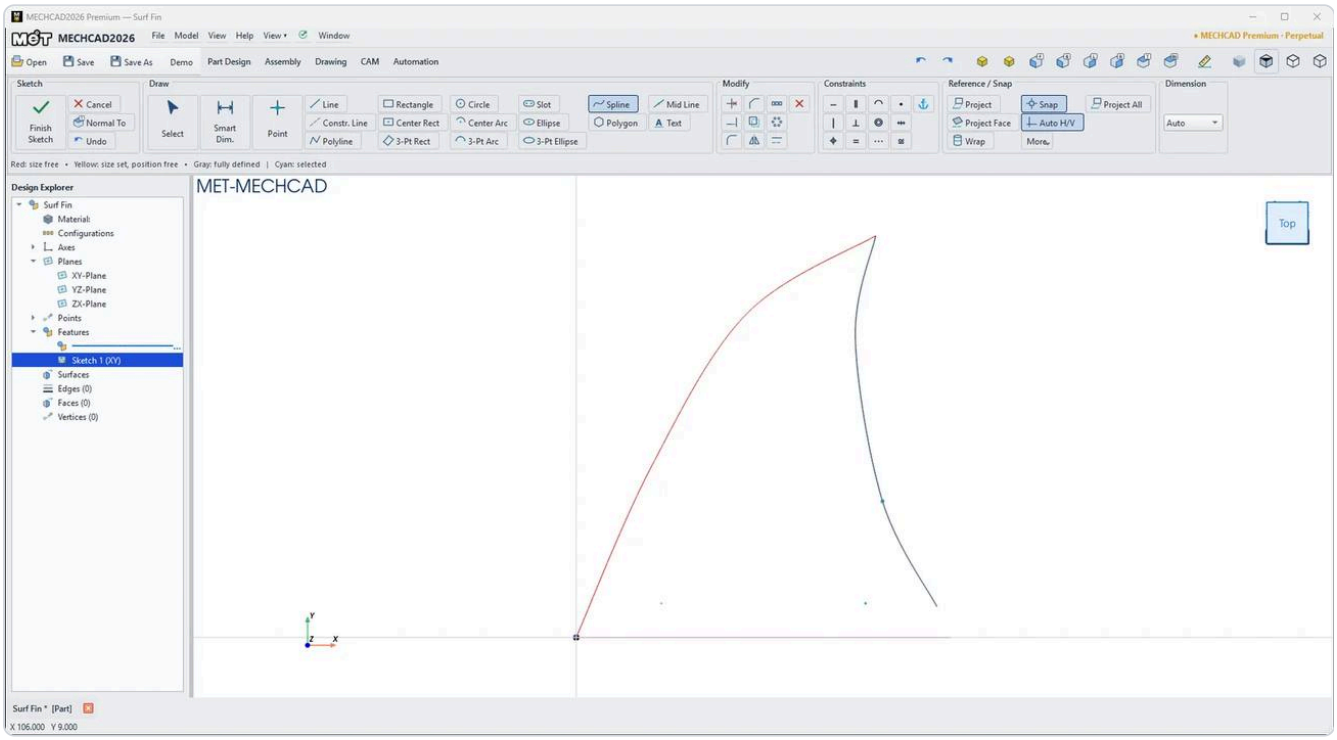
Draw free-form curves, hole position points, and text as sketch shapes.

What it does

Tool	How to use it
Spline	Click control points one after another. Double-click or press Esc to finish.
Point	Click once per point. Use it to set hole positions for the Hole Wizard (*Position drills at every Point*).
Text	Creates text as outlines around the letters. You can use Extrude / Cut / Wrap with it. The settings panel has Text (default "MET"), Font (DejaVu Sans / Serif / Sans Mono), Height (10), Centre X, Centre Y and Angle.

Where to find it

Sketch ribbon › **Draw** › Spline / Point / Text, or Part ribbon › **Model › Sketch › Point**



Drawing the second Spline of the fin (2 Points mark the screw holes)

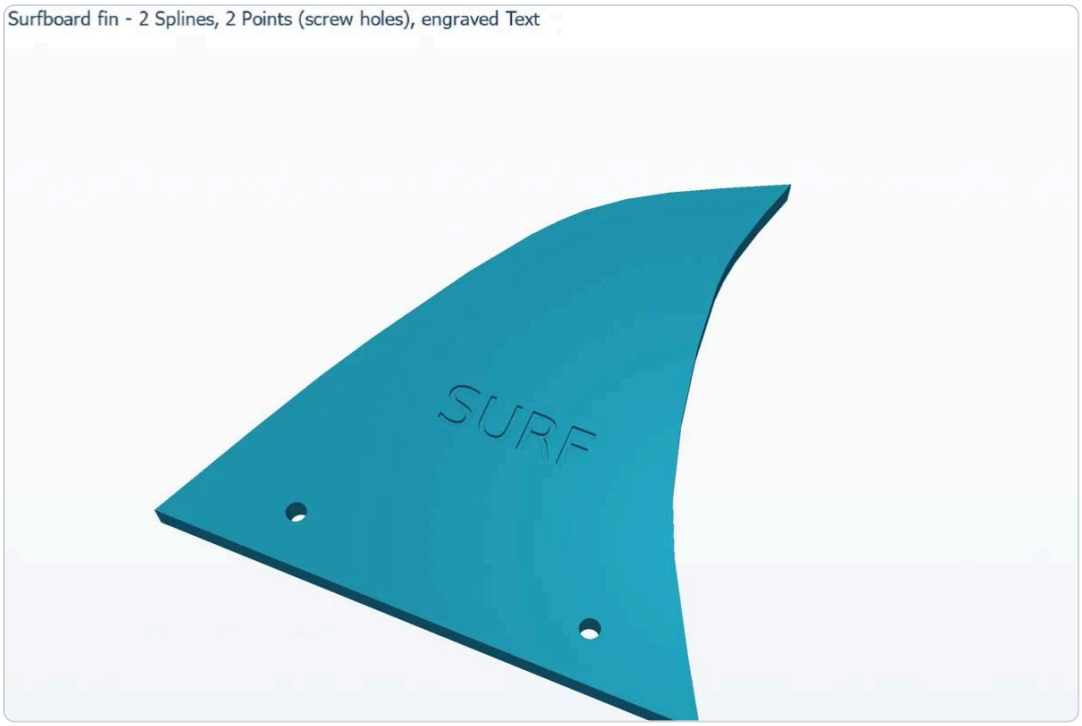
Steps (Spline)

1. Click **Spline**.
2. Click the control points one at a time. The curve passes through every point.
3. Double-click the last point, or press Esc, to finish.
4. You can adjust the shape by dragging points in Select mode.

Try it: Surfboard fin

A surf fin needs a smooth, swept-back curve that straight lines and arcs cannot give you. Use two splines for the edges, two points for the screw holes and your own text on the side.

1. Click **Sketch** and press Enter (XY-Plane). Draw a **Line** from (110, 0) to the origin. This is the fin base.
2. Choose **Spline** and click (0, 0) [Endpoint], (22, 50), (50, 95) and (88, 118), then press Esc. This is the leading edge.
3. Choose **Spline** again and click (88, 118) [Endpoint], (82, 88), (90, 40) and (110, 0) [Endpoint], then press Esc. This is the trailing edge.
4. Choose **Point** and click (25, 10) and (85, 10). Click **Finish Sketch**, then **Extrude 6** mm.
5. Click **Hole Wizard** and under **Position** choose - *all 2 points (Drill)* for two 4 mm holes.
6. Right-click the top face › **Insert Sketch on Surface** › **Text**. Set Text **SURF**, Font DejaVu Sans, Height **12**, Centre X **0**, Centre Y **0** (on a face, the sketch origin is the centre of the face) and Angle **0**. Click **Finish Sketch**, then **Extrude Cut 1** mm.



Surfboard fin with spline edges, screw holes at Points and engraved text

Challenge: In Select mode, drag the (82, 88) spline point to (75, 90) for a more raked fin. The extrude updates with it.

See also

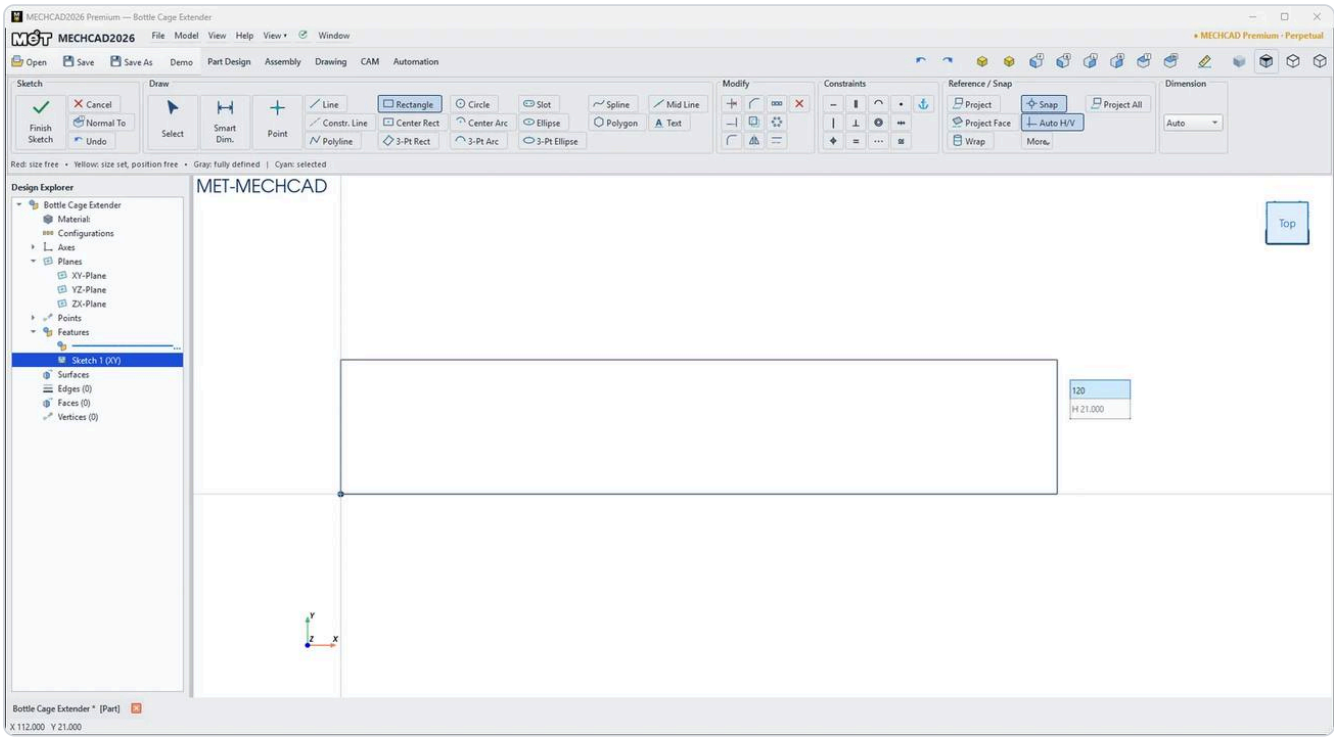
- Hole Wizard
- Text (Emboss / Deboss on a face)
- Sketch overview
- Smart Dimension
- Constraints

Dynamic Input: Type Sizes While You Draw Standard

Type a length, angle, width, height or diameter while drawing, and get the shape and dimension at once.

What it does

After you click the first point of a Line, Rectangle, Center Rect or Circle, input boxes follow the cursor. Type a value and press Enter to get a shape of exactly that size, with **driving dimensions**, so you do not need to add a Smart Dimension afterward.



W/H boxes while drawing a Rectangle: typed values turn blue (locked)

Tool	Input boxes
Line, Constr. Line	**L** length, ∠ angle
Rectangle, Center Rect	**W** width, **H** height
Circle	**Ø** diameter
Dragging a line end point	L / ∠
Dragging a single point	X / Y

Steps

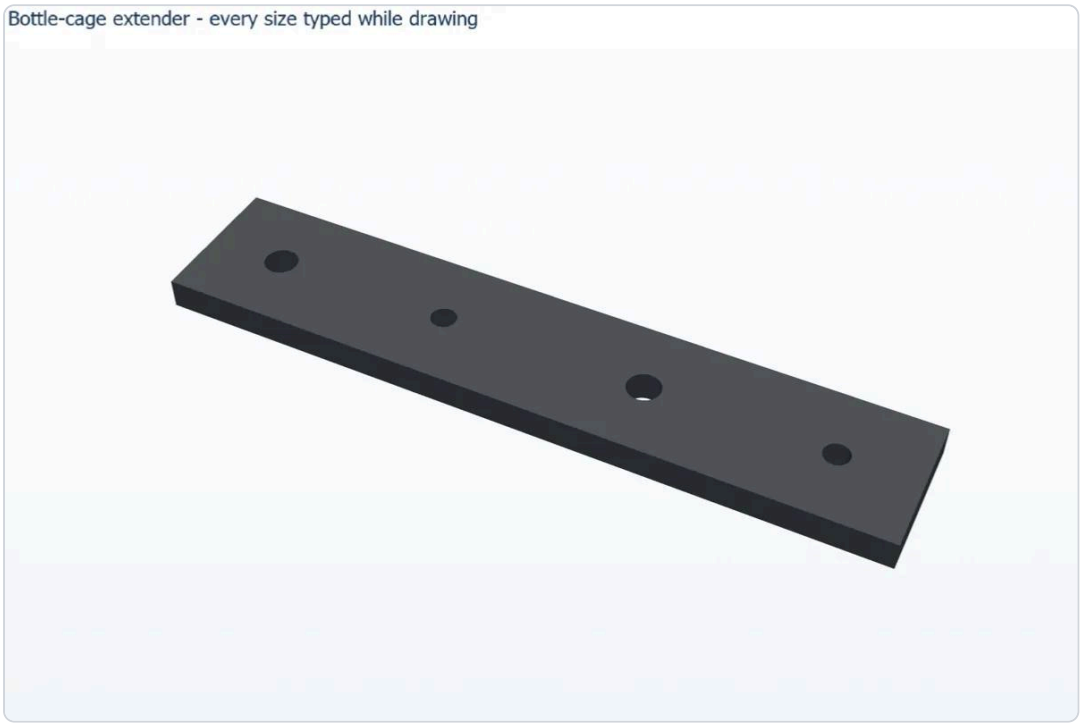
1. Choose a tool (for example Rectangle) and click the first point.
2. Type a number. The value goes into the first box and that box is **locked** (blue).
3. Press **Tab** / **Shift+Tab** to go to the next box, then type a value.
4. Press **Enter** to create the shape. The tool stays active.
5. Press **Esc** to cancel.

> **Note:** Typed values take priority over snaps. The result always matches the numbers.

Try it: Bottle-cage extender

Want your water bottle 30 mm higher on the frame? This extender plate bolts onto the standard bottle bosses, which are 64 mm apart, and holds the cage higher up. Every size is typed while you draw, so you never need to add a dimension afterward.

1. Click **Sketch** and press Enter (XY-Plane). Press **R** (Rectangle) and click the origin. Move the mouse up and to the right, type **120**, press Tab, type **24** and press Enter. Press Esc.
2. Press **C** (Circle). Click inside the plate, type **5.5** and press Enter. Do this again for a second 5.5 mm circle. Then make two circles of **4.2** mm (tap size for M5).
3. Press Esc to go back to Select. Drag the centre of the first 5.5 mm circle. When the **X / Y** boxes appear, type **13**, press Tab, type **12** and press Enter. Move the other centres the same way: second 5.5 mm circle to **77 / 12**, and the 4.2 mm circles to **43 / 12** and **107 / 12**.
4. Click **Finish Sketch** and **Extrude 5** mm.



Bottle-cage extender with every size typed while drawing

Challenge: Make a 50 mm extender. Move the cage holes to X **63** and **127** and the rectangle to **140** long.

See also

- Sketch overview
- Smart Dimension
- Constraints

Smart Dimension and Smart Angle Standard

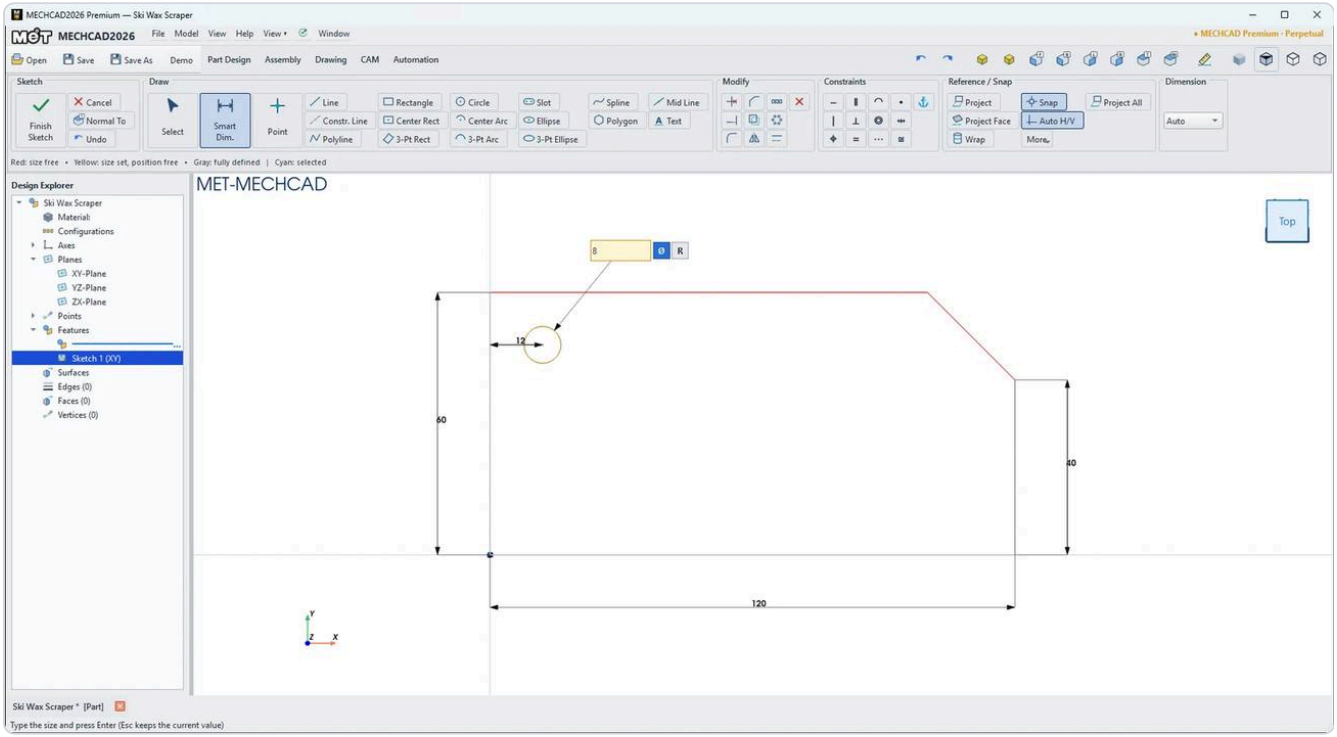
Dimension lines, circles, arcs, distances between objects, and angles.

What it does

Smart Dimension chooses the dimension type automatically from the object you click (length, horizontal, vertical, Ø, R, distance). **Smart Angle** measures the angle between two lines. The dimensions you add are **driving dimensions**: change the value and the sketch changes with it.

Where to find it

- Sketch ribbon › **Draw › Smart Dim.** (shortcut **D**)
- Right-click on empty space › **Smart Dimension** / **Smart Angle**
- **Reference / Snap › More › Dimensions**: Dim Aligned, Horizontal, Vertical, Radius, Diameter, Angle, Point-Line, Ellipse Major, Ellipse Minor
- The **Dimension** box at the right of the ribbon: Auto / Size / Between / Between X / Between Y

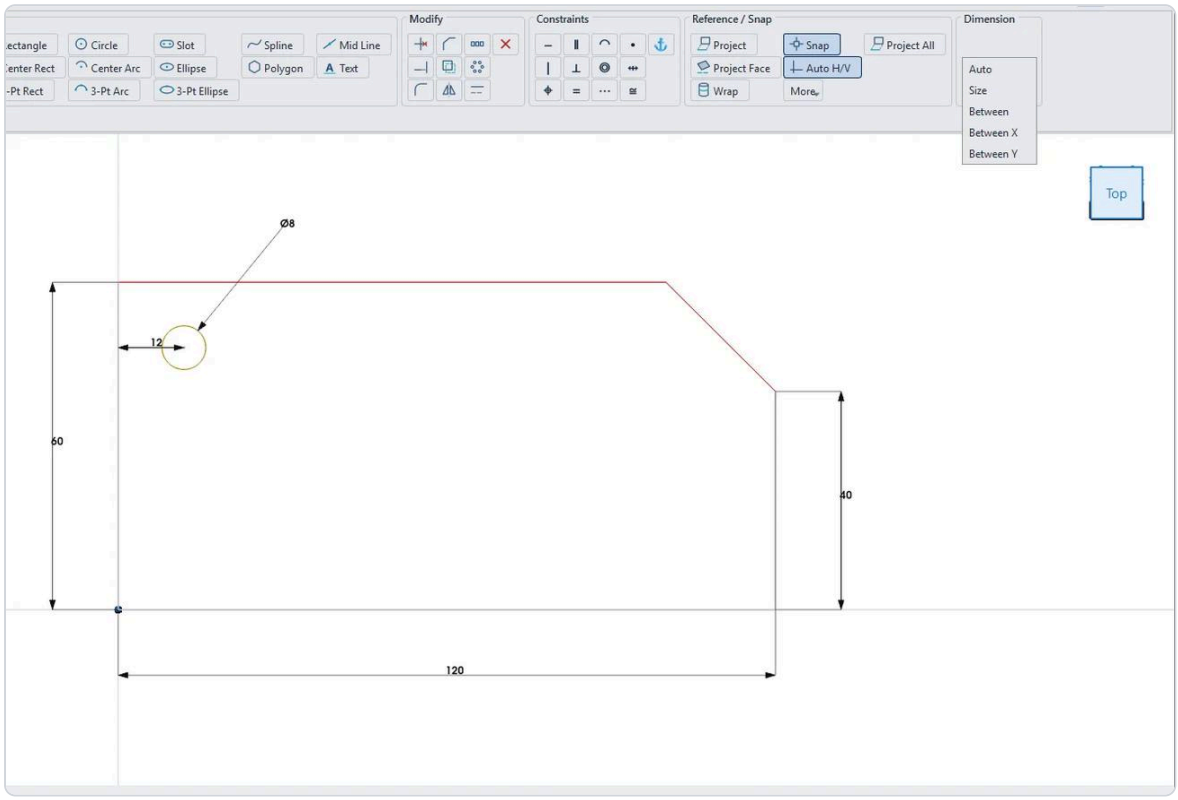


Smart Dimension on a circle: value box with Ø and R buttons

Steps

1. Click **Smart Dim.** (or press D).
2. Click an object: a line (length), a circle (Ø), an arc (R) or a point. You can click a second object to measure the distance between them (circles and ellipses are measured from the center).
3. Move the mouse to choose the dimension direction (for a single line: parallel to the line, horizontal or vertical, depending on the cursor position), then click in empty space to place it.
4. The value box opens. Type the value and press Enter (Esc = use the current value). Type R or Ø/D before the number to switch between radius and diameter.

Action	Result
Double-click a dimension (Select mode)	Edit the dimension value
Right-click a dimension › Edit Dimension	Edit the dimension value
Right-click a dimension › Show as Diameter / Show as Radius	Switch the style
Right-click an angle dimension › Flip Angle (acute/obtuse)	Switch between the acute and obtuse angle
Delete	Delete the dimension



Dimension group: choose the dimension type

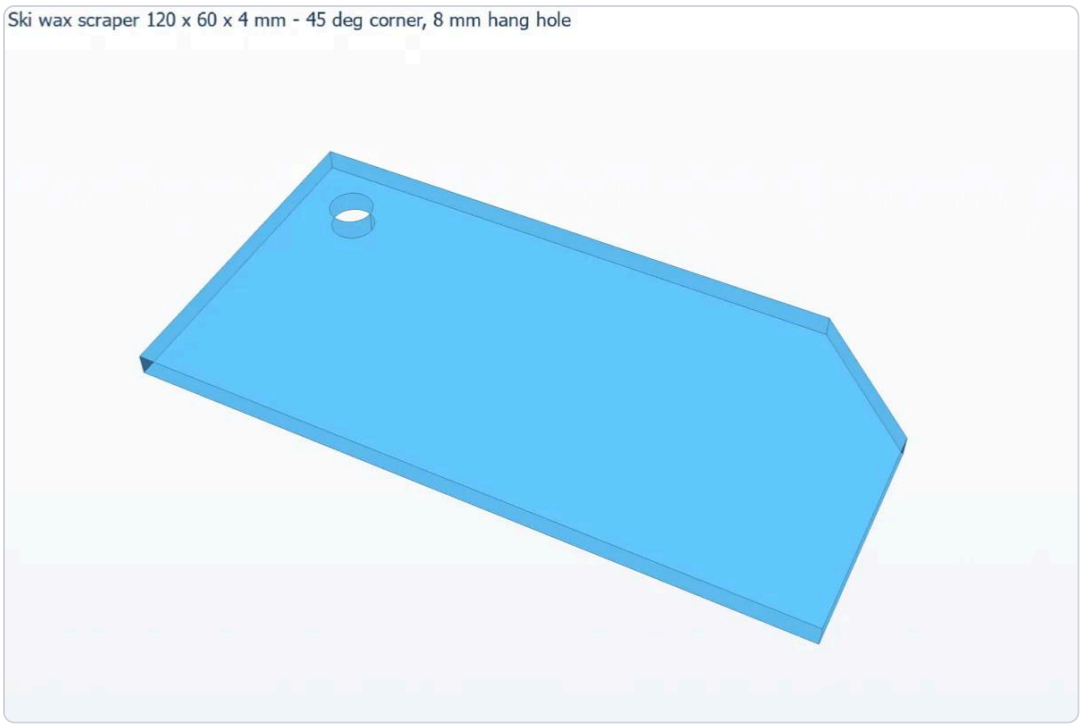
> **Note:** If that size is already defined, the new dimension is a **Reference (driven)** dimension that only shows the value. If the value you enter makes the sketch conflict, you see *Cannot make it X: the sketch would conflict*.

Try it: Ski wax scraper

A wax scraper is a simple plate, but the 45° corner is what gets wax out of the ski's edge groove. Draw the rough outline, then let Smart Dimension make it exact.

1. Click **Sketch** and press Enter (XY-Plane). With **Line**, draw a five-sided outline near (0, 0) → (120, 0) → (120, 40) → (100, 60) → (0, 60) and back to the start.
2. Press **D** (Smart Dim.): click the bottom edge and type **120**; click the left edge and type **60**; click the right edge and type **40**.
3. Right-click › **Smart Angle**, click the slanted edge and the right edge, then type **135**. If the angle shows the acute side, right-click the dimension › **Flip Angle (acute/obtuse)** first.

4. Draw a **Circle**, click it with Smart Dim. and type **8** (Ø). Then click the circle and the left edge and type **12**, and click the circle and the top edge and type **12**.
5. Check that the sketch turns grey (Fully defined), click **Finish Sketch** and **Extrude 4** mm.



Ski wax scraper sized entirely with Smart Dimension

Challenge: Double-click the 40 dimension and change it to 30. The 135° corner stays and the slanted edge gets longer.

See also

- Constraints
- Dynamic input

Constraints (Geometric Relations) Standard

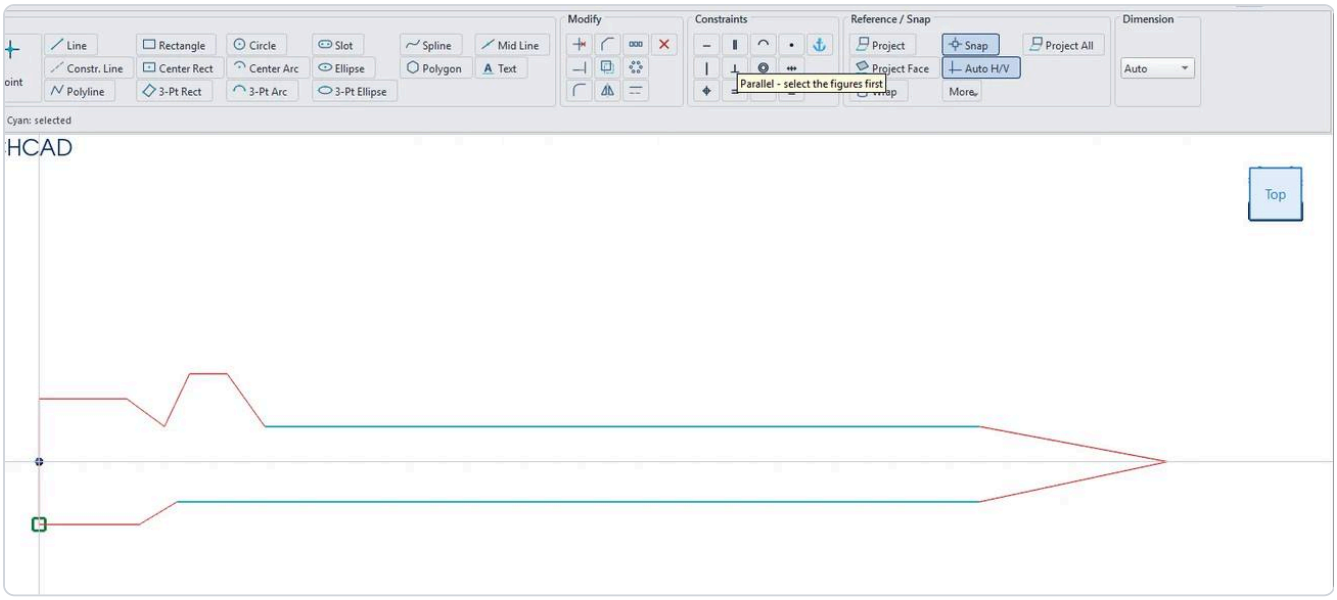
Apply Horizontal, Vertical, Coincident, Parallel, Perpendicular, Tangent, Concentric, Equal, Midpoint, Point On, Symmetric and Fix.

What it does

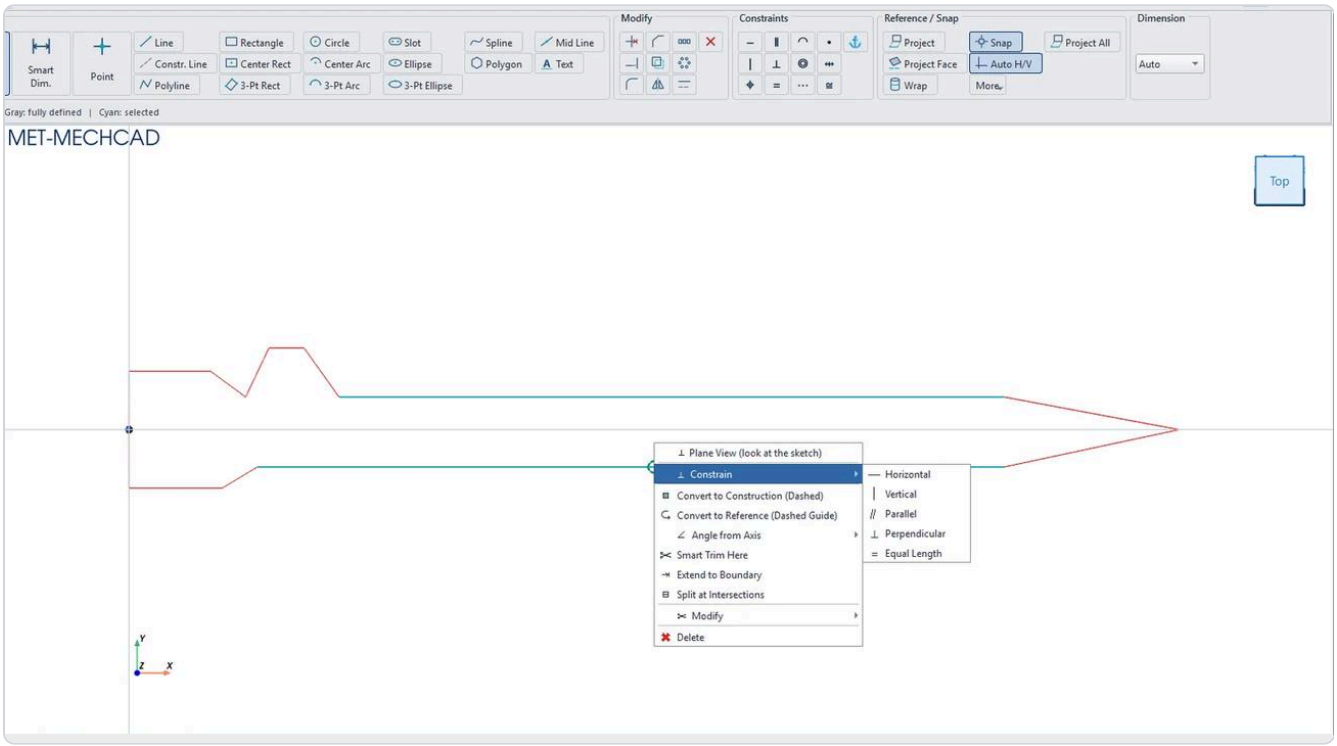
A constraint sets a geometric relation, such as parallel, perpendicular or tangent, so the sketch keeps its shape when you change a dimension.

Where to find it

- Sketch ribbon › **Constraints** group (icons; hover to see the name: - *select the figures first*)
- Right-click a line › **Constrain** (shows only the constraints that fit the selected objects)



Constraints group



Right-click a line › Constrain

Steps

1. In **Select** mode, click the objects (Ctrl+click to select several, or drag a box).
2. Click the constraint button you want.
3. If you selected the wrong type of object, you see *Invalid selection for …*.

Constraint	What to select	Result
Horizontal / Vertical	One or more lines, or 2 points	Makes them horizontal / vertical
Coincident	2 points or 2 lines	Points coincide (2 lines: joins the nearest ends)

Constraint	What to select	Result
Parallel	2 lines	Parallel to each other
Perpendicular	2 lines	Perpendicular to each other
Equal	2 lines	Equal length
Tangent	Line + circle, or 2 circles	Tangent to each other
Concentric	2 circles	Share the same center
Equal Radius	2 circles	Equal radius
Midpoint	Point + line	The point is at the midpoint of the line
Point On	Point + line or circle	The point lies on the object
Symmetric	2 points + line	Symmetric about the line
Fix / Unfix (⚓)	Point	Fixes / releases the position

Automatic constraints

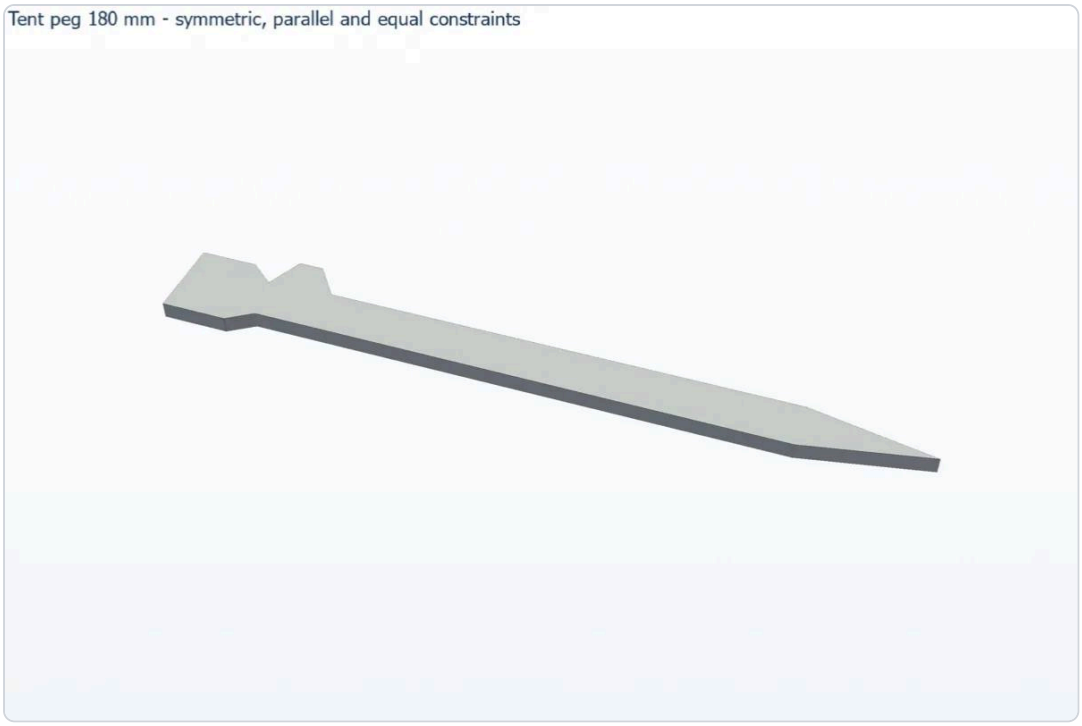
- **Auto H/V** (Reference / Snap group): lines that are nearly horizontal/vertical get H/V automatically (the default is set in System Options › Sketching).
- End points placed on a snap get Coincident automatically.

> **Note:** Use the line colors to see whether the sketch is fully defined: red = size not set, yellow = size set but can still move, grey = Fully defined.

Try it: Tent peg

A tent peg has to be perfectly symmetric, otherwise it twists when you hammer it into hard ground. Draw it roughly, then let the constraints straighten it out.

1. Click **Sketch** and press Enter (XY-Plane). Draw a **Constr. Line** from the origin to about (190, 0). This is the centre line.
2. With **Line**, click around the outline near these points and finish on the first point: (0, −10) → (16, −10) → (22, −6) → (150, −6) → (180, 0) → (150, 6) → (36, 6) → (30, 14) → (24, 14) → (20, 5) → (14, 10) → (0, 10). It does not need to be exact.
3. Select the long bottom shaft edge and click **Horizontal**. Then select both long shaft edges and click **Parallel**.
4. Ctrl+click the two shaft corners near (150, 6) and (150, −6) and the centre line, then click **Symmetric**.
5. Select the tip point and the centre line and click **Point On**. Select the two short tip edges and click **Equal**.
6. Add **Smart Dimension**s: **180** from the left edge to the tip, **12** between the two shaft edges and **30** for the tip length (horizontal).
7. Click **Finish Sketch** and **Extrude** the peg **4** mm.



Tent peg kept symmetric by constraints

Challenge: Change the shaft width from 12 to 10. The peg stays centred on the centre line and the tip stays sharp, because Symmetric and Point On hold it.

See also

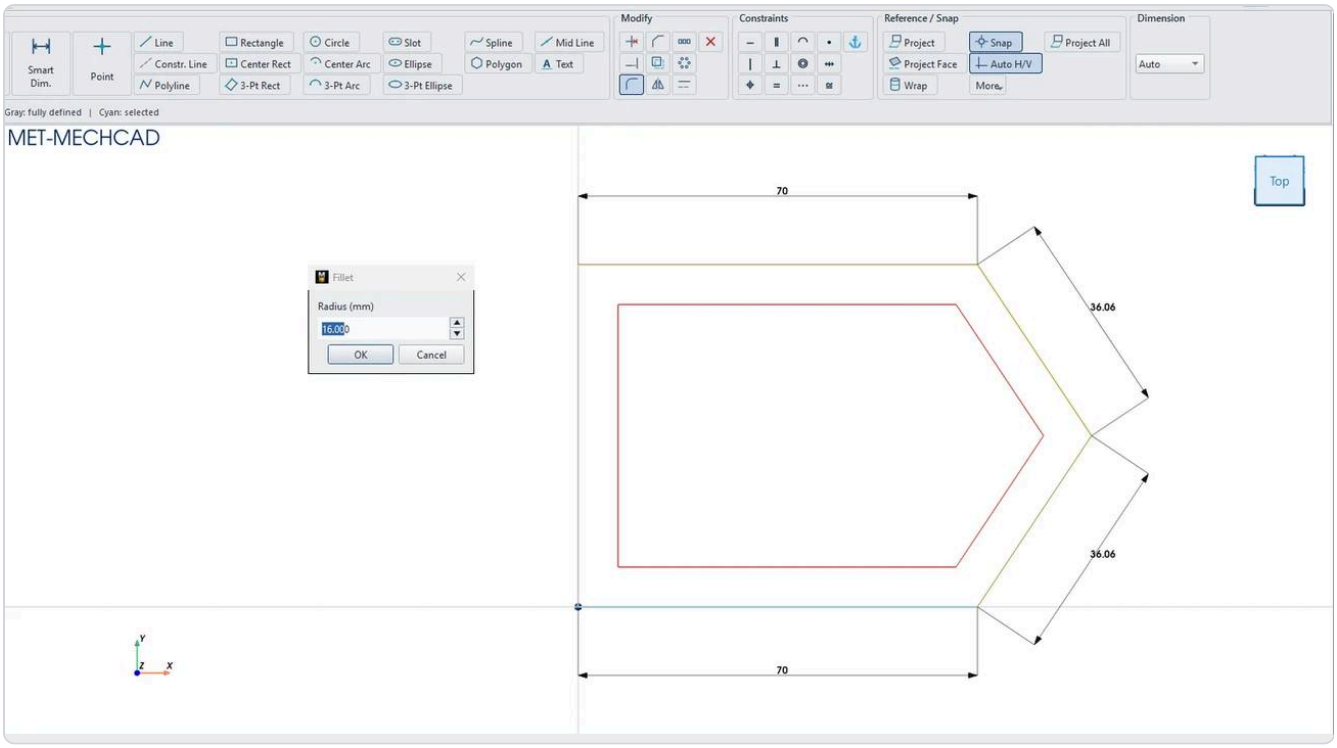
- Smart Dimension
- Sketch overview

Trim, Extend, Fillet, Chamfer and Offset in a Sketch Standard

Trim, extend, round corners, chamfer and offset lines in a sketch.

What it does

Shape-editing tools in the **Modify** group of the Sketch ribbon (icons; hover to see the name).

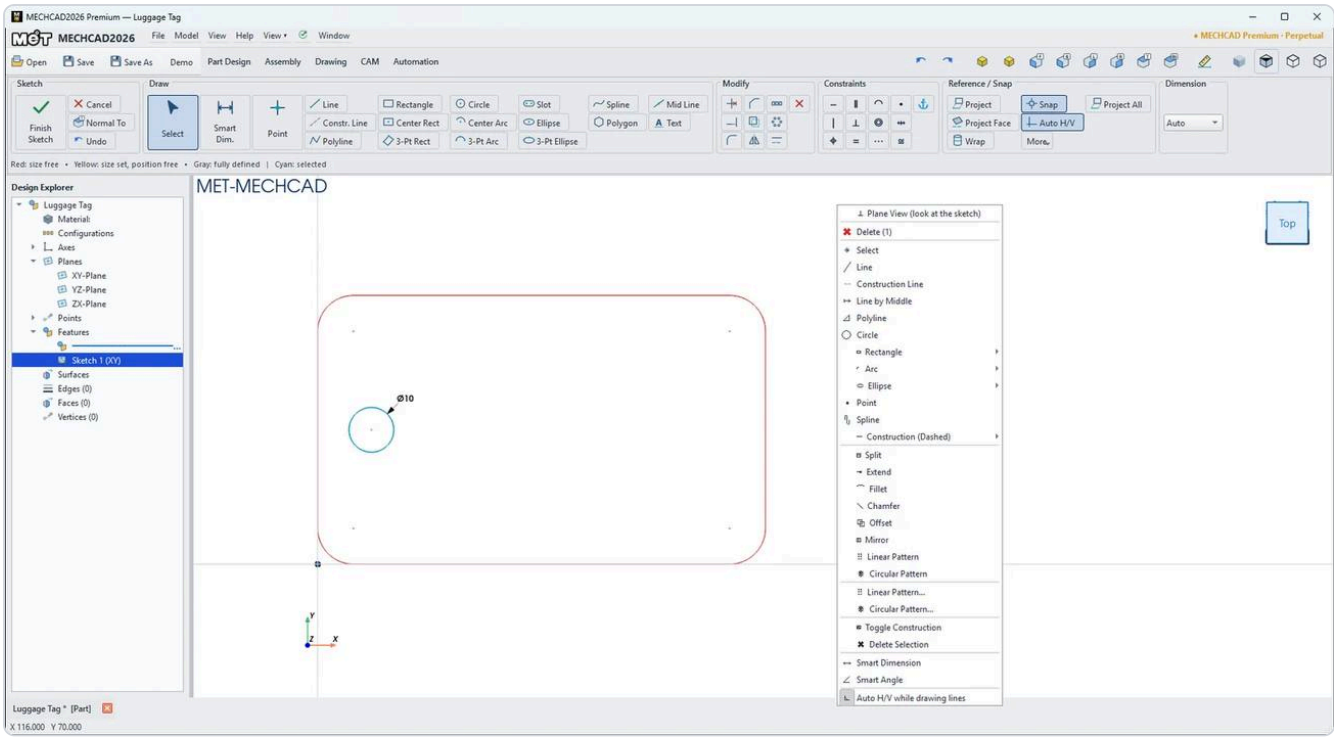


Modify group

Tool	How to use it	Options
Trim / Split	Click the segment of a line that an intersection bounds. That segment is removed (lines, circles, arcs).	–
Extend	Click near the end of a line. The line extends to the next boundary.	–
Fillet	Click two lines, then enter the **Radius (mm)**.	Default 5
Chamfer	Click two lines, then choose the type.	**Distance / Distance** or **Distance / Angle**; Distance 1, Distance 2, Angle (deg) default 5 / 5 / 45°
Offset	Click a line, circle or arc, then enter the **Distance (mm)**.	**Outside / grow** or **Inside / shrink**; default 5
Toggle Construction	Switches the selected objects between construction and normal lines.	–
Delete Selection	Deletes the selected objects.	–

More commands from right-clicking a line

- **Smart Trim Here** — trims the segment you click
- **Extend to Boundary** — extends to the boundary
- **Split at Intersections** — splits the line at intersections
- **Convert to Construction (Dashed)** / **Convert to Normal Geometry**
- **Angle from Axis** › From X-Axis / From Y-Axis — adds an angle dimension relative to an axis



Right-click menu on empty space in a sketch

> **Note:** There are no Move / Rotate / Scale commands in a sketch. Drag objects in Select mode, or use dimensions to set positions.

Try it: Keychain carabiner

A keychain carabiner is a thick ring with big, smooth radii. Draw only the outer outline. Offset makes the inside, and Fillet rounds all ten corners.

1. Click **Sketch** and press Enter (XY-Plane). Press **L**, click the origin and type each edge as length Tab angle Enter: **70**/**0**, **36.06**/**56.31**, **36.06**/**123.69**, **70**/**180**. Then click the start point to close the outline.
2. Click **Offset**, click any edge of the outline, set Distance **7** and **Inside / shrink**, then click OK. The whole closed chain is copied 7 mm inside.
3. Click **Fillet** and click the two lines at each outer corner with Radius **16** (5 corners). Then do the inner corners with Radius **9**.
4. Click **Finish Sketch** and **Extrude 5** mm.



Keychain carabiner: Offset 7 mm inside, then Fillet R16 / R9

Challenge: Make a gate gap. Draw two short lines 8 mm apart across the upper slanted side, then **Trim** away the ring pieces between them so the profile becomes one C-shaped loop.

See also

- Mirror and Pattern in a sketch
- Sketch overview
- Smart Dimension
- Constraints

Mirror, Linear Pattern and Circular Pattern in a Sketch Standard

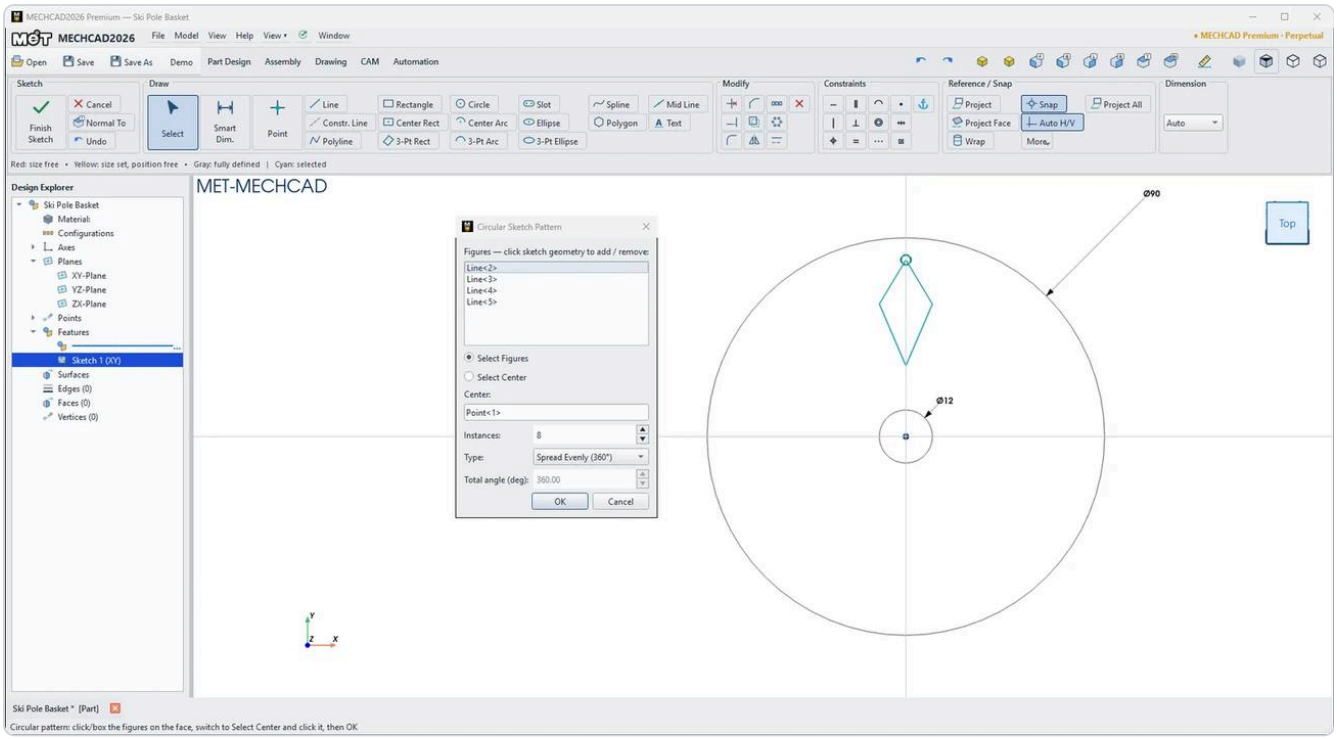
Mirror and repeat shapes in a sketch along a line or around a center point.

What it does

Creates repeated shapes in a sketch without redrawing them.

Where to find it

Sketch ribbon › **Modify** › Mirror / Linear Pattern / Circular Pattern, or right-click › **Linear Pattern...** / **Circular Pattern...**



Circular Sketch Pattern window

Mirror

1. Click **Mirror**, then click the objects to mirror.
2. Press **Enter**.
3. Click the line to use as the mirror axis.
4. Confirm *Mirror N object(s) across this line?*

Linear / Circular Pattern

1. Click **Linear Pattern** or **Circular Pattern**. The Pattern window stays open.
2. In **Select Figures** mode, click or drag a box around the objects (click again to remove one).
3. Choose **Select Direction Line** (Linear) or **Select Center** (Circular), then click the direction line or center point.
4. Enter the values and click **OK**.

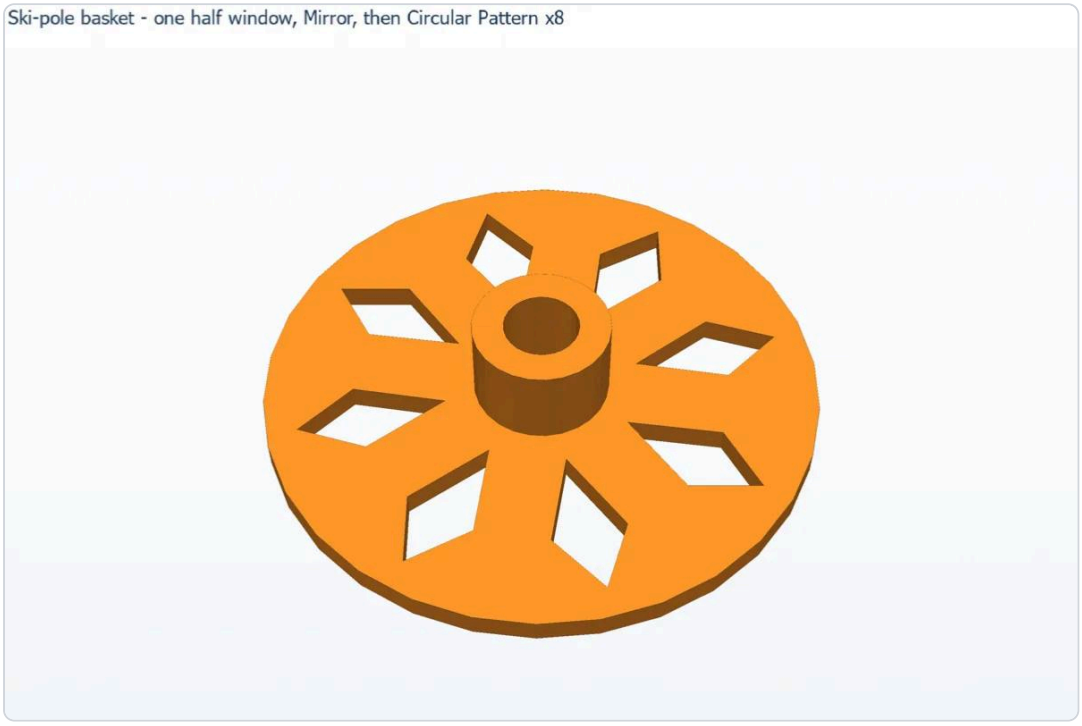
Option	Meaning
Figures	List of objects to repeat
Instances	Number of copies (default 6)
Spacing (mm)	Distance between copies (Linear, default 20)
Type	Spread Evenly (360°) / Fixed total angle (Circular)
Total angle (deg)	Total angle when Fixed total angle is chosen

> **Note:** Right-click an item in the pattern › **Explode Pattern** (separates it into independent objects) or **Delete Whole Pattern**.

Try it: Ski-pole basket

A powder basket stops your ski pole from sinking into the snow. You draw only half of one window. Mirror makes the full window, and Circular Pattern places all eight.

1. Click **Sketch** and press Enter (XY-Plane). Draw two circles at the origin: Ø **90** and Ø **12**.
2. Draw a **Constr. Line** from the origin straight up to about (0, 45). This is the mirror line.
3. With **Line**, draw (0, 16) → (6, 30) → (0, 40) and press Esc. Start and end on the construction line.
4. Click **Mirror** and click the two lines. Press Enter, click the construction line and confirm *Mirror 2 object(s)*…. You now have a kite-shaped window.
5. Click **Circular Pattern**. In **Select Figures** mode, click the four window lines. Choose **Select Center** and click the origin. Set Instances **8**, Type **Spread Evenly (360°)** and click **OK**.
6. Click **Finish Sketch** and **Extrude 4** mm. For the hub, sketch circles Ø **22** and Ø **12** on the XY-Plane and **Extrude** them **18** mm.



Ski-pole basket: one half window, Mirror, then Circular Pattern x8

Challenge: Right-click one window › **Explode Pattern**, then delete every second window to get a lighter 4-window basket.

See also

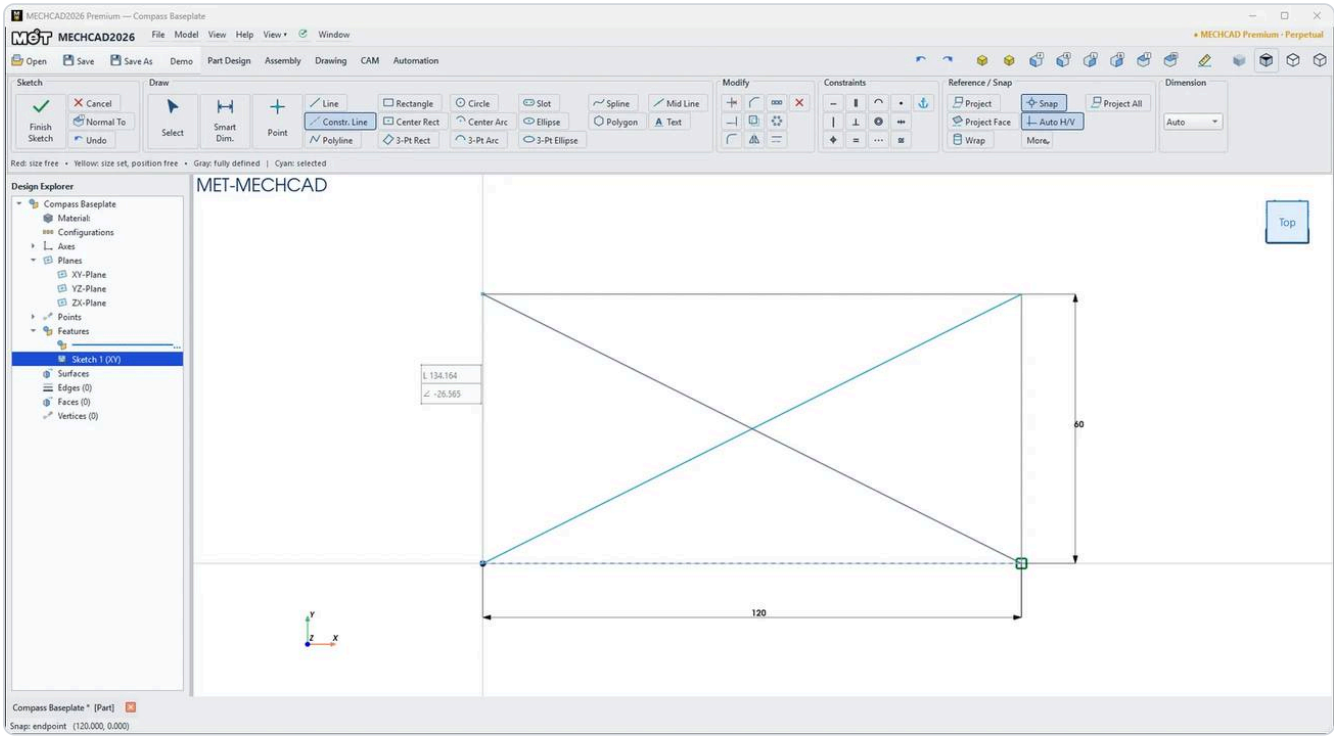
- Linear / Circular Pattern of a feature
- Sketch overview
- Smart Dimension
- Constraints

Snaps and Auto H/V Standard

Snapping to end points, midpoints, centers, intersections and alignment guide lines.

What it does

Snaps help you click exactly on key points of existing shapes. When the cursor gets close, you see a symbol and the status bar shows *Snap: {type} (x, y)*.

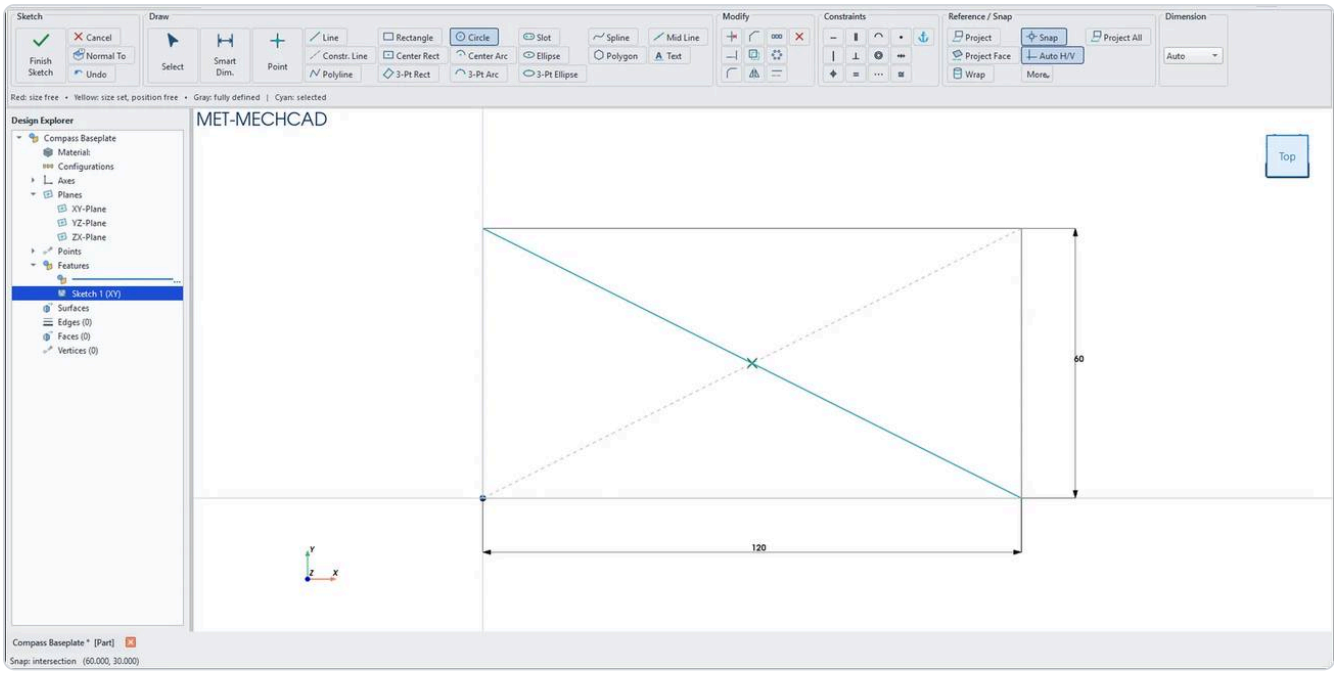


Snap at the corner of a rectangle while drawing a Constr. Line

Snap type	Symbol
Origin	Filled circle
Endpoint	Square
Intersection	X
Midpoint	Triangle
Center / Quadrant	Circle
Tangent, Nearest	–
Corners, midpoints and center of the face you are sketching on	Available when you sketch on a face

Where to find it

Sketch ribbon › **Reference / Snap**



Reference / Snap group: Project, Project All, Project Face, Wrap, Snap, Auto H/V, More

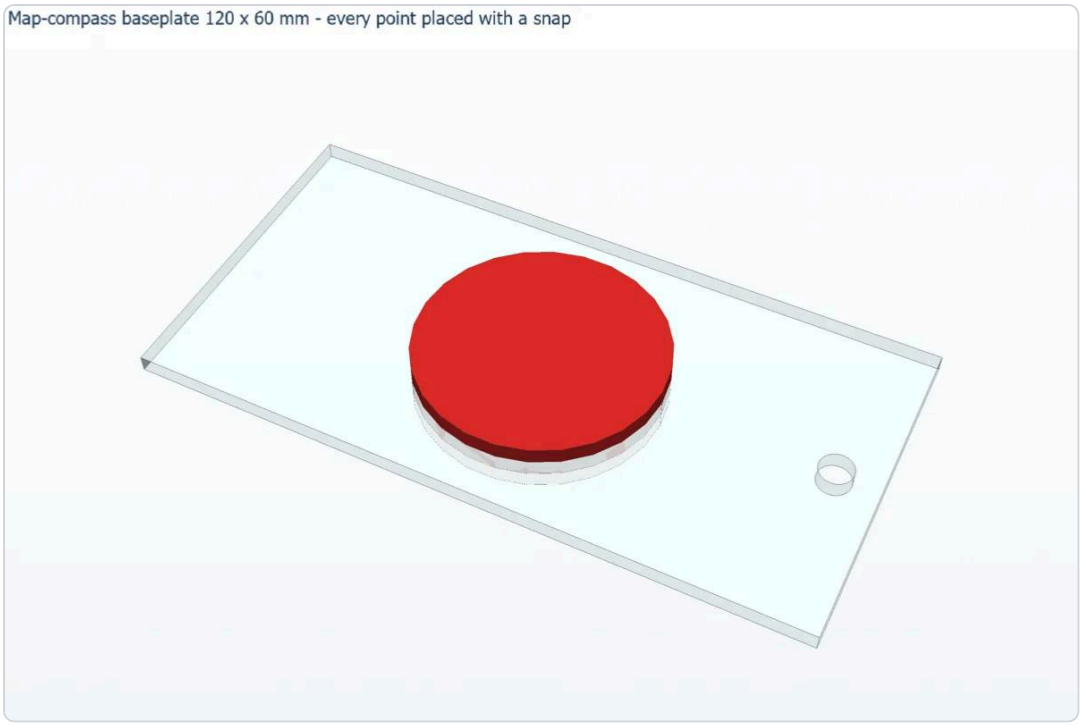
Button	Meaning
Snap	Turns all snaps on/off
Auto H/V	New lines that are nearly horizontal/vertical are straightened and get a constraint (*Auto H/V on new lines: ON/OFF*)
Hold Alt	Turns snaps off temporarily while drawing

> **Note:** You can set the snap capture radius in **System Options › Sketching** (default 10 px). A sketch on a 3D face does not show a grid and does not snap to a grid.

Try it: Map-compass baseplate

An orienteering compass is a clear baseplate with the capsule exactly in the middle. You will not type a single coordinate for the centre: the snaps find it for you.

1. Click **Sketch** and press Enter (XY-Plane). Press **R**, snap to the **Origin** (filled circle), type **120**, Tab, **60** and press Enter.
2. Draw a **Constr. Line** from the bottom-left corner to the top-right corner (Endpoint snaps, square). Draw another one from top-left to bottom-right.
3. Press **C**, hover over the crossing until you see the **Intersection** snap (X), click, type **44** and press Enter. This is the capsule hole.
4. Draw a **Constr. Line** from the **Midpoint** (triangle) of the left edge to the Midpoint of the right edge. Put a Ø **6** circle on that line and dimension it **8** mm from the right edge. This is the lanyard hole.
5. Click **Finish Sketch** and **Extrude 3** mm. The red capsule in the picture is only there for display.



Compass baseplate with every point placed using a snap

Challenge: Hold **Alt** while you place a circle and see how the snaps switch off.

See also

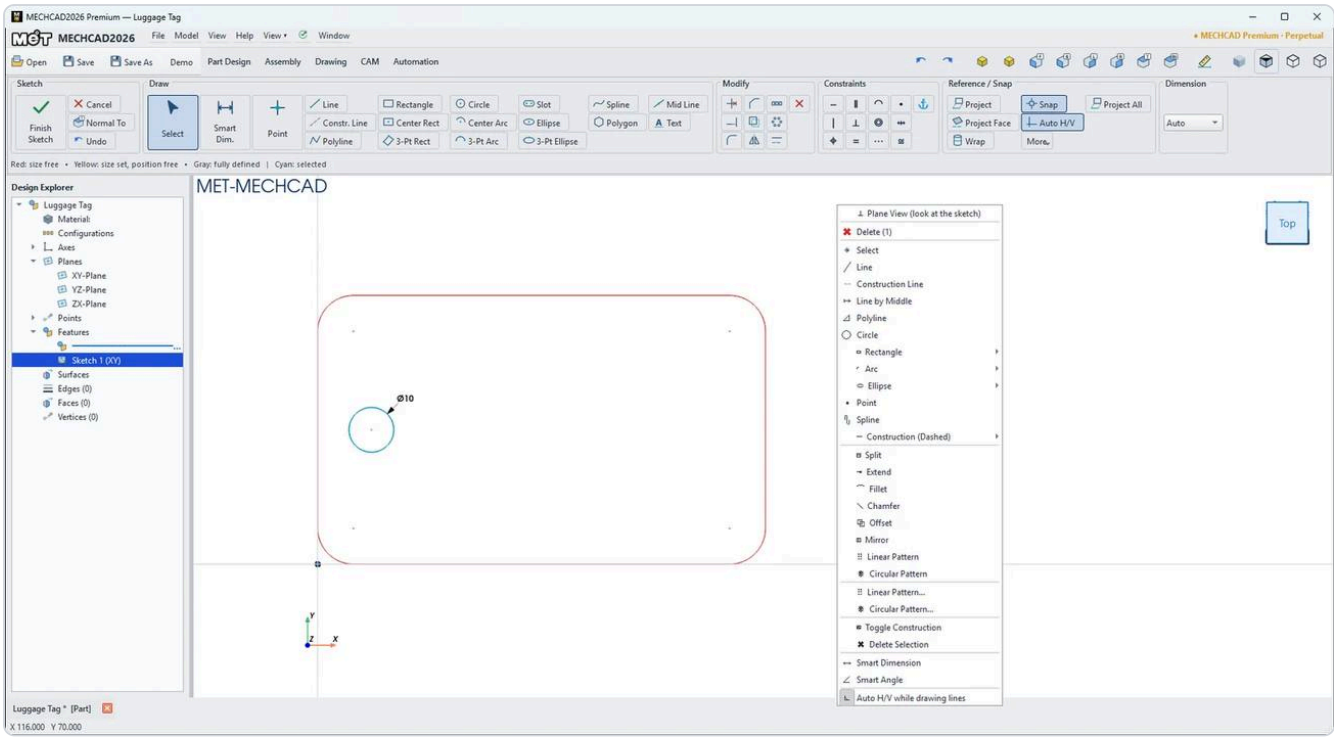
- Project to Sketch
- Sketch overview
- Smart Dimension
- Constraints

Sketch Right-Click Menu Standard

Commands you can call by right-clicking empty space, a line, a point or a dimension.

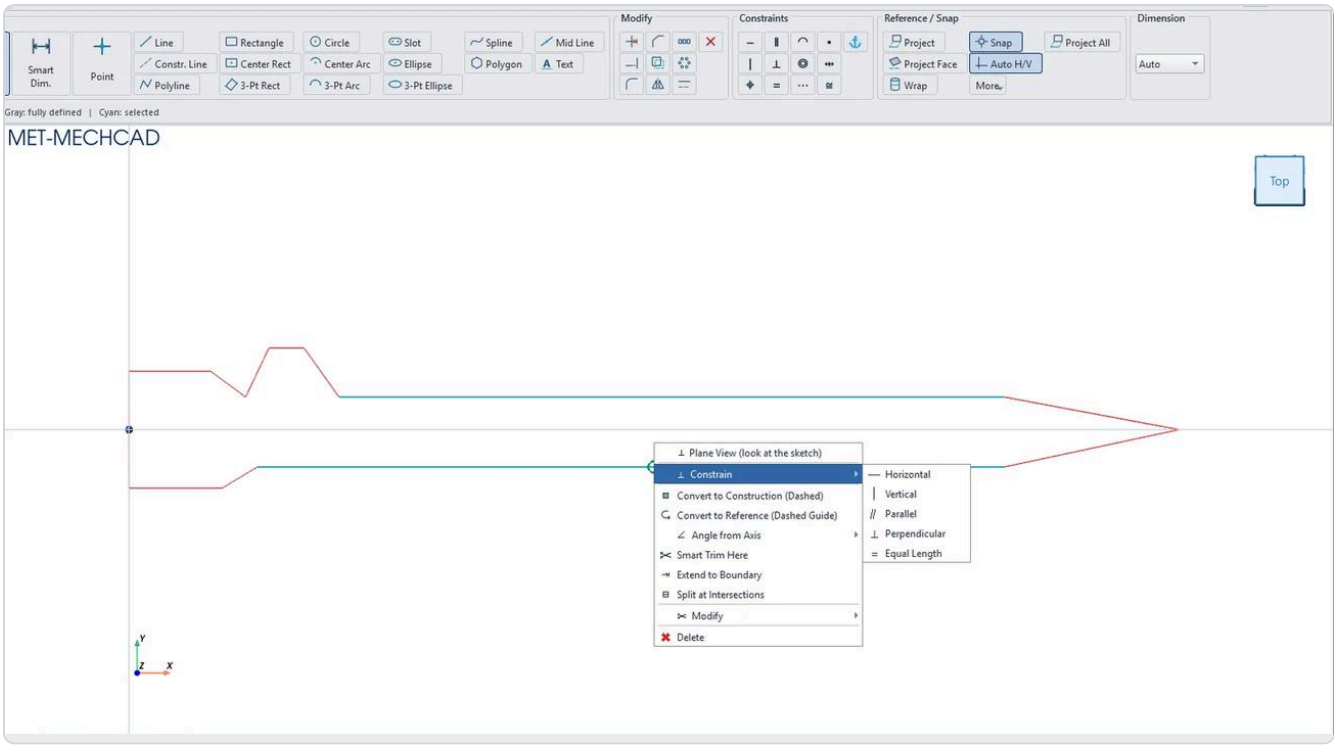
What it does

Right-click while sketching to call tools quickly without going to the ribbon. Every menu starts with ⊥ **Plane View (look at the sketch)**.



Right-click menu on empty space

Right-click on	Commands
Empty space	Delete (N), Select, Line, Construction Line, Line by Middle, Polyline, Circle, Rectangle ›, Arc ›, Ellipse ›, Point, Spline, Construction (Dashed) ›, Split, Extend, Fillet, Chamfer, Offset, Mirror, Linear/Circular Pattern, Toggle Construction, Delete Selection, Smart Dimension, Smart Angle, Auto H/V while drawing lines
Line	Constrain ›, Convert to Construction (Dashed), Convert to Reference (Dashed Guide), Angle from Axis ›, Smart Trim Here, Extend to Boundary, Split at Intersections, Modify ›, Delete
Point	Modify ›, Fix Point / Unfix Point, Coincident (when 2 points are selected)
Dimension	Edit Dimension, Flip Angle (acute/obtuse), Show as Diameter (Ø width) / Show as Radius (R), Delete
An item in a pattern	Explode Pattern, Delete Whole Pattern



Right-click a line › Constrain

Try it: Luggage tag

Make a luggage tag without touching the ribbon once. Every command comes from the right-click menu, which is the fastest way to sketch.

1. Click **Sketch** and press Enter (XY-Plane).
2. Right-click empty space › **Rectangle** › by corners. Click the origin, type **100**, press Tab, type **60** and press Enter.
3. Right-click › **Fillet**. Click two lines that meet at a corner and enter Radius **8**. Repeat for all four corners.
4. Right-click › **Circle**. Click near (12, 30), type **10** and press Enter. This is the strap hole.
5. Right-click › **Smart Dimension**. Dimension the circle centre **12** mm from the left edge and **30** mm from the bottom edge.
6. Right-click › ⊥ **Plane View** to look straight at the sketch and check it. Then click **Finish Sketch** and **Extrude 3** mm.



Luggage tag drawn using only the right-click menu

Challenge: In a new sketch on the front face, right-click › **Rectangle** for a 60 x 30 mm name window and **Extrude Cut** it 1 mm deep.

See also

- Sketch overview
- Smart Dimension
- Constraints

Project, Project All and Project Face Standard

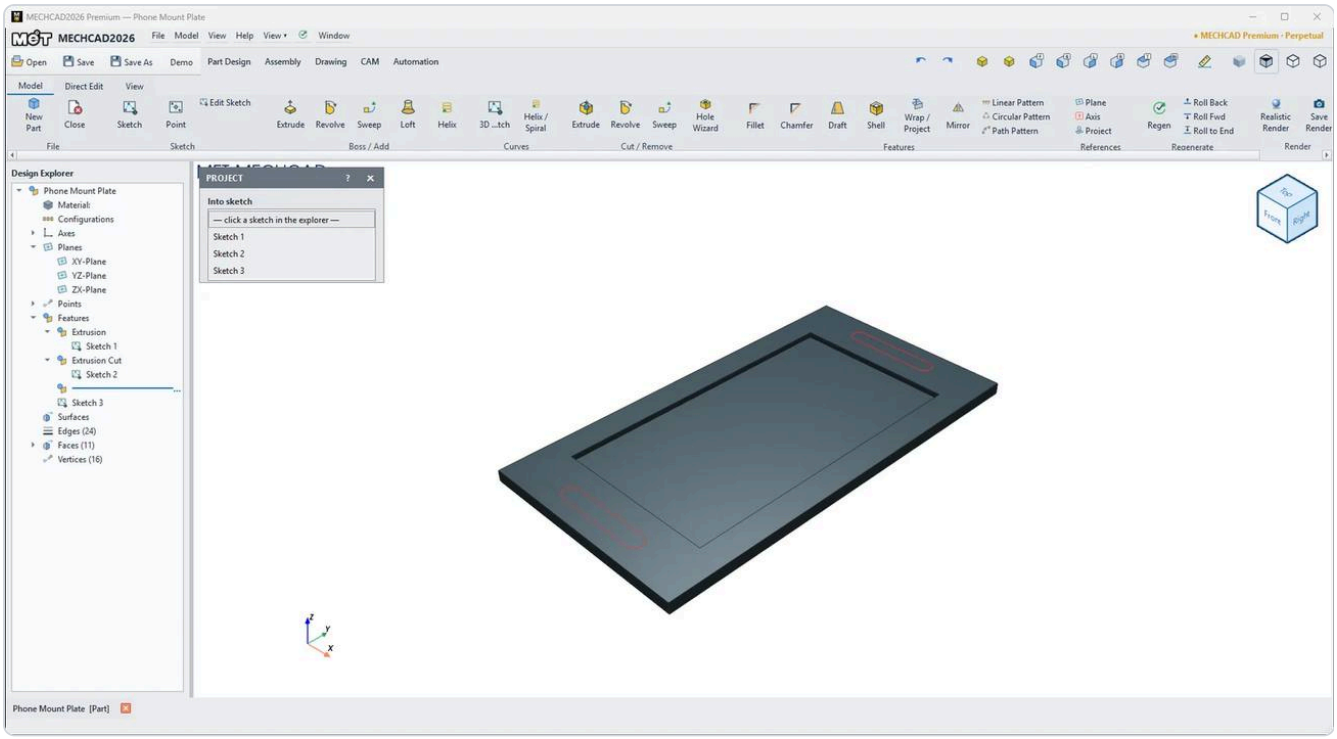
Project the edges of the part or another sketch into a sketch as reference lines or real lines.

What it does

Brings the edges of the part or of another sketch into the current sketch. Use them as snap points, to add dimensions, or as real shapes for a feature.

Where to find it

- While sketching: **Reference / Snap › Project / Project All / Project Face**
- Part ribbon: **Model › References › Project** (if no sketch is open yet, it first asks which sketch to project into)



PROJECT box, step 1: choose the target sketch

Command	Meaning
Project	Choose what to project: Body outline (silhouette), Body edges (all), Support face edges and other sketches
Project All	Projects all edges, the outer outline and all other sketches as dashed reference lines in one click
Project Face	Copies the edges of the face you are sketching on as reference lines
Wrap	Finishes the sketch and opens the Wrap / Project command

Project to Sketch box

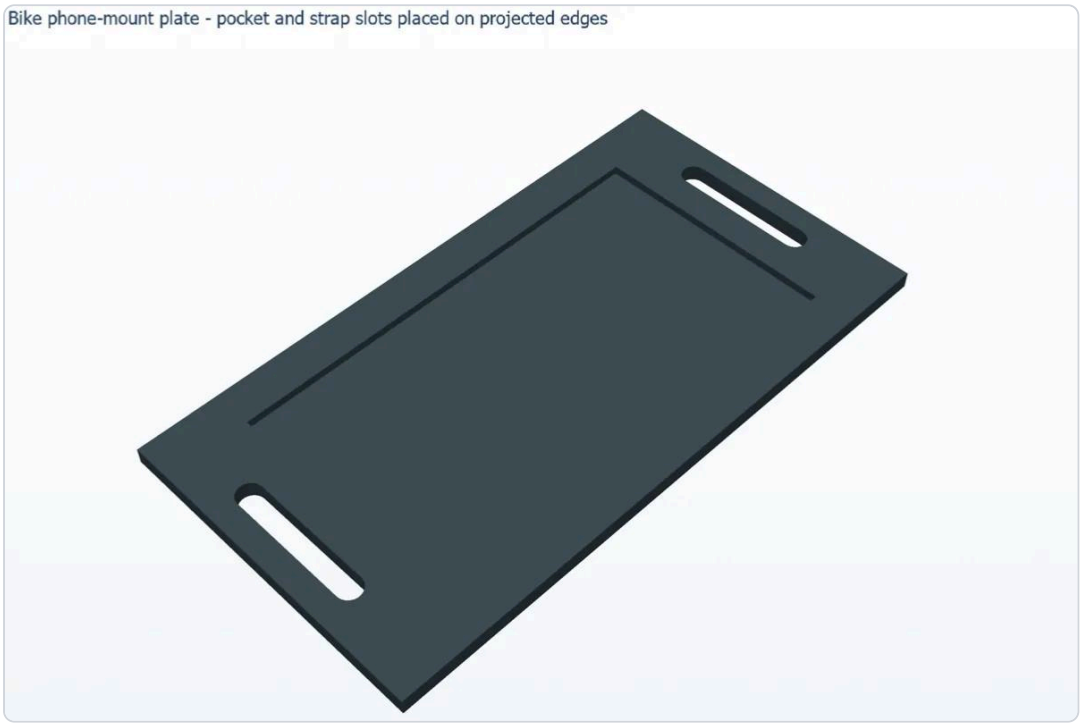
Option	Meaning
Geometry / Sketches	Tick the items to project
Select All	Selects everything
Pick faces in the view	Click faces in the view to select them
Maintain Association	The projected lines update with the part (default: on)
Project As	**References (dashed)** reference lines / **Figures** real lines

> **Note:** Projected lines can be trimmed or deleted right away. Right-click a reference › **Convert to Normal (Detach Reference)** breaks the link to the part and leaves an editable dashed construction line. To use it as a profile edge, right-click it again › **Convert to Normal Geometry**. Lines projected as **Figures** become the profile only while the sketch has no shapes of your own. As soon as you draw your own shapes, Finish Sketch uses only those, and the projected lines stay as guides.

Try it: Bike phone-mount plate

A handlebar phone mount needs a pocket for a grip pad and two slots for the straps. Project Face gives you the plate edges as dashed guides, so everything lines up with the outline without any measuring from the origin.

1. Click **Sketch** and press Enter (XY-Plane). Choose **Center Rect**, click the origin, type **80**, press Tab, type **150** and press Enter. Click **Finish Sketch**, then **Extrude** the plate **4** mm.
2. Right-click the top face › **Insert Sketch on Surface**, then click **Reference / Snap › Project Face**. The four plate edges appear as dashed reference lines.
3. Choose **Center Rect**, click the centre of the face, type **60**, Tab, **110** and press Enter. Click **Finish Sketch**, then **Extrude Cut 2** mm. The dashed references are only guides. Only your rectangle is cut, so you get the pocket.
4. Right-click the top face again › **Insert Sketch on Surface** › **Project Face**. Draw a horizontal **Slot** near each short edge. With **Smart Dimension**, set each slot to **30** mm between the end centres, **R3** on an end arc (6 mm wide), **9** mm from the slot centre line to the dashed short edge and **25** mm from the left end centre to the dashed long edge.
5. Click **Finish Sketch**, then **Extrude Cut** with **Through All**.



Phone-mount plate with a pocket and strap slots placed from projected edges

Challenge: Change the plate to 90 x 160 mm. Because the pocket and slots are dimensioned to the projected edges (Maintain Association is on), they follow the new outline.

See also

- Sketch on a face
- Snaps and Auto H/V

3D Sketch (3D Paths) Standard

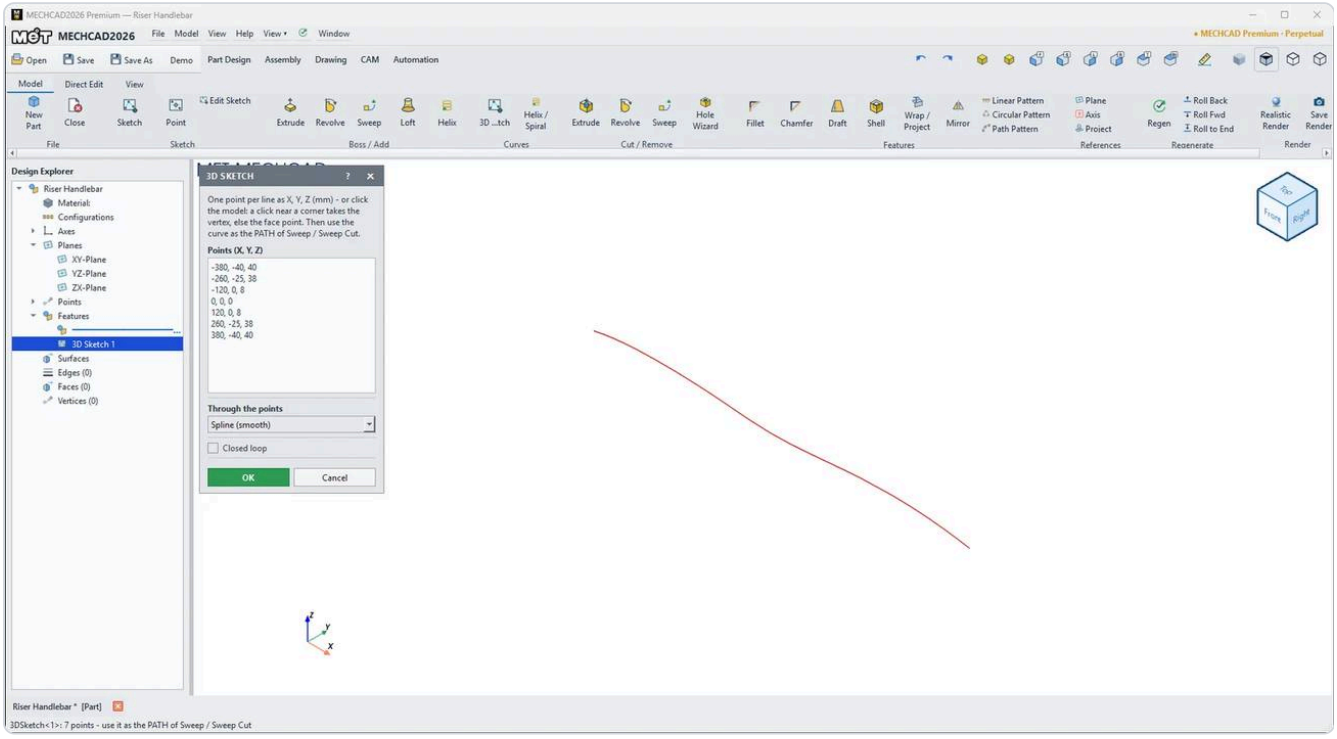
Create a 3D polyline or spline path for sweeping pipes, cables and frames.

What it does

3D Sketch creates a line through 3D points. Use it as the **Path** for Sweep / Sweep Cut, for example for pipes, cables or steel frames.

Where to find it

Model › Curves › 3D Sketch



3D SKETCH box

Steps

1. Click **3D Sketch**.
2. Type the points in the **Points (X, Y, Z)** box, one point per line, for example `0, 0, 0` (separate values with `,` `;` Tab or a space; `#` starts a comment). Or click vertices or faces on the part to add points.
3. Choose **Through the points** and **Closed loop** as needed.
4. Click **OK**. You get `3DSketch<n>` in Design Explorer.

Option	Meaning
Points (X, Y, Z)	List of points (at least 2 different points)
Through the points	**Polyline (straight segments)** / **Spline (smooth)**
Closed loop	Connects the last point back to the first

Try it: Riser handlebar

A mountain-bike riser bar bends in three directions at once: it rises, sweeps back and stays flat in the middle. A single 3D Sketch spline gives you the path for that shape.

1. Click **Model › Curves › 3D Sketch** and type these points in **Points (X, Y, Z)**, one per line:

```
-380, -40, 40
-260, -25, 38
-120, 0, 8
0, 0, 0
120, 0, 8
260, -25, 38
380, -40, 40
```

2. Set **Through the points** to **Spline (smooth)**, leave **Closed loop** off and click **OK**. You get `3DSketch<1>`.
3. Click **Sketch**, pick the **YZ-Plane**, draw a **Circle** at the origin with Ø **22.2** and click **Finish Sketch**.
4. Click **Model › Boss / Add › Sweep**. Choose the circle as **Profile** and `3DSketch<1>` as **Path**, then click **OK**. Sweep moves the circle to the start of the path for you.
5. For the stem clamp area, sketch another circle on the **YZ-Plane** with Ø **31.8** and **Extrude** it with **Mid Plane 100** mm.



760 mm riser handlebar swept along a 3D Sketch spline

Challenge: Switch **Through the points** to **Polyline (straight segments)** and compare. Then change the two Z = 40 values to 20 for a lower rise.

See also

- Sweep and Sweep Cut
- Helix / Spiral

Helix / Spiral curve Standard

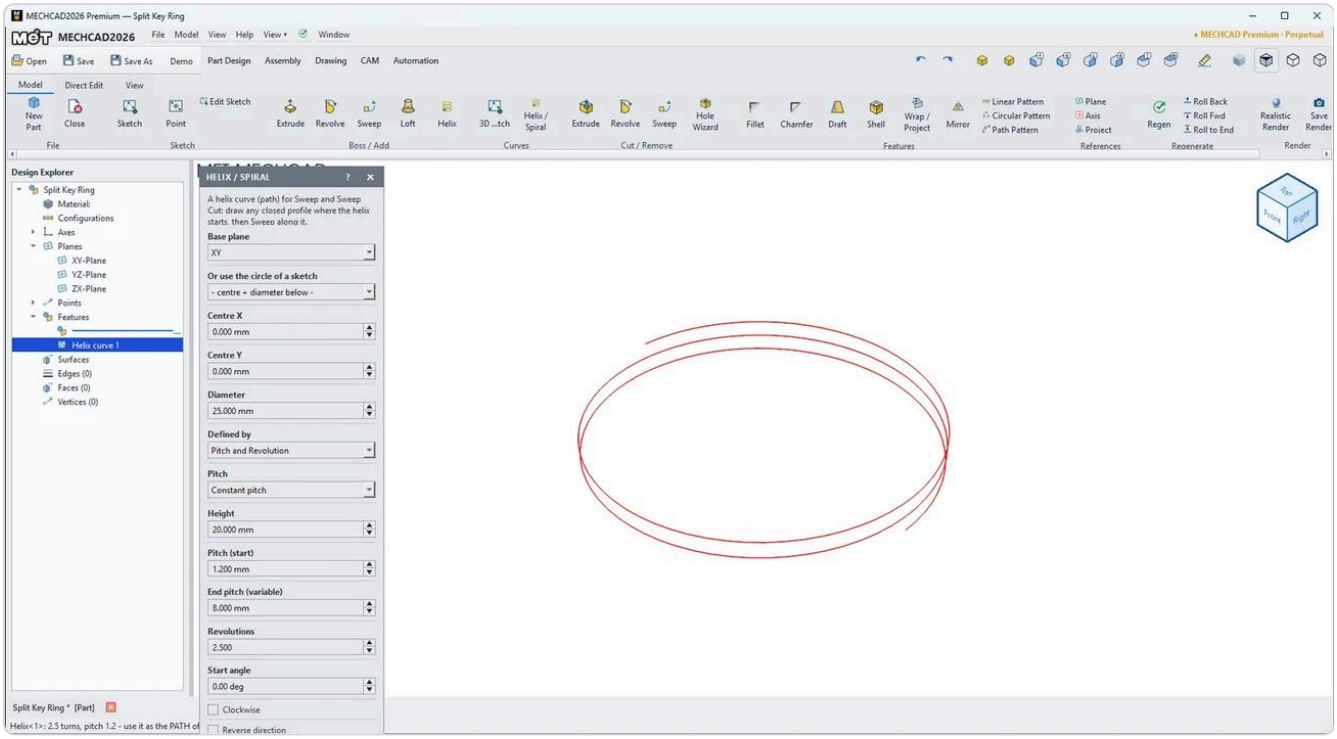
Create a constant or variable pitch helix to use as a Sweep path for springs and threads.

What it does

Creates a helical curve (like SolidWorks) with constant or variable pitch, clockwise or counter-clockwise. Use it as the path for Sweep and Sweep Cut with any closed profile, for example springs and threads.

Where to find it

Model › Curves › Helix / Spiral



HELIX / SPIRAL box

Option	Default	Description
Base plane	XY	Base plane (XY / YZ / ZX / reference plane)
Or use the circle of a sketch	–	Uses a circle in a sketch for the centre and size
Centre X / Centre Y	0	Centre point
Diameter	25 mm	Diameter
Defined by	Pitch and Revolution	Pitch and Revolution / Height and Revolution / Height and Pitch
Pitch	Constant pitch	Constant pitch / Variable pitch
Height	20 mm	Height
Pitch (start)	4 mm	Pitch
End pitch (variable)	8 mm	End pitch (when variable)
Revolutions	5	Number of turns
Start angle	0°	Start angle (±360°)
Clockwise	Off	Turns clockwise
Reverse direction	Off	Reverses the direction

Steps

1. Click **Helix / Spiral** and enter the values in the table.
2. Click **OK** to create a `Helix<n>` curve.
3. Draw a closed profile at the start of the helix, then use Sweep and choose this curve as the Path.

Try it: Split key ring

A split key ring is nothing more than 2.5 turns of thin wire wound on a helix. Build one in two minutes and you also know how every spring and thread is made.

1. Click **Model › Curves › Helix / Spiral** and enter: Base plane **XY**, Centre X **0**, Centre Y **0**, Diameter **25** mm, Defined by **Pitch and Revolution**, Pitch **Constant pitch**, Pitch (start) **1.2** mm, Revolutions **2.5**, Start angle **0**. Click **OK**. You get `Helix<1>`.
2. Click **Sketch**, pick the **ZX-Plane**, press **C** (Circle), click the origin, type **1.1** and press Enter. Click **Finish Sketch**. The wire (1.1 mm) is thinner than the pitch (1.2 mm), so the turns do not fuse together.
3. Click **Model › Boss / Add › Sweep**. Choose the circle sketch as **Profile (section)** and `Helix<1>` as **Path (spine)**, then click **OK**. The circle is not at the start of the helix, so Sweep moves it there for you.



Split key ring made from a Helix and a Sweep

Challenge: Change Pitch to **Variable pitch** (1.2 to 3 mm) and Revolutions to **6**. Now you have a tapered spring.

See also

- Helix (spring/thread feature)
- Sweep and Sweep Cut

Chapter 3 quiz: Sketch

Choose the one best answer. Answers are in Appendix A

1. **While sketching, a line is shown in grey. What does that mean?**

   A. Its size is not set
   B. Its size is set but it can still move
   C. The sketch is fully defined
   D. It is selected

2. **What does pressing Esc twice do while you are drawing in a sketch?**

   A. Finishes and saves the sketch
   B. Cancels the whole sketch
   C. Deletes the last line
   D. Returns to Select

3. **How do you turn snaps off temporarily while drawing?**

   A. Hold Alt
   B. Hold Shift
   C. Hold Ctrl
   D. Press F3

4. **In the Smart Dimension value box for a circle, how do you switch between radius and diameter?**

   A. Ctrl+click the circle again
   B. Type R or Ø/D before the number
   C. It is not possible; circles always use diameter
   D. Right-click › Flip Angle

5. **You add a dimension to a size that is already defined. What do you get?**

   A. A Reference (driven) dimension that only shows the value
   B. The old dimension is replaced
   C. The sketch is deleted with an error
   D. Both dimensions become driving dimensions

6. **How do you start a sketch directly on a flat face of an existing part?**

   A. File › New Sketch
   B. Right-click the face › Insert Sketch on Surface
   C. Double-click the face with Shift held
   D. Model › References › Plane, then Extrude

CHAPTER 4

Part

1. Part Design Overview
   The workflow for building a parametric 3D part and the commands on the Part Design tab.
2. Feature Parameters: the feature edit box
   Edit an existing feature with an Alibre-style floating box, and the behaviour shared by all command boxes.
3. Extrude (Boss)
   Add material by pushing a closed sketch perpendicular to its plane using To Depth, Mid Plane, Two Directions or Up To Face.
4. Extrude Cut
   Remove material with a closed sketch using To Depth, Mid Plane, Two Directions, Up To Face or Through All.
5. Revolve and Revolve Cut
   Add or remove material by revolving a profile around an axis.
6. Sweep and Sweep Cut
   Sweep a closed profile along a path to add or remove material.
7. Loft
   Create a solid that blends through two or more profiles on planes at different levels.
8. Helix (springs and threads)
   Sweep a cross-section along an automatically generated helix for springs and threads.
9. Hole Wizard (standard holes)
   Drill standard M3–M12 holes (Simple, Counterbore, Countersink, Tapped or Tapered) by clicking a face or a Point.
10. Fillet and Chamfer
    Round (Fillet) or bevel (Chamfer) part edges by clicking them.
11. Draft
    Tilt side walls so the part can be released from a mould.
12. Shell
    Hollow out a part to a wall thickness, leaving the selected faces open.
13. Wrap / Project (Emboss / Deboss on a Face)
    Wrap a sketch around a cylinder or project it onto a face, then raise or recess it.
14. Mirror

Mirror a body about a plane or a flat face and merge it into one part.

15. Linear Pattern and Circular Pattern

    Repeat a feature along a line (one or two directions) or around an axis.

16. Path Pattern

    Repeat a feature along a selected edge or curve.

17. Reference Plane and Axis

    Create offset reference planes, and reference axes from a hole, an edge or two points.

18. Move Face and Simplify (Direct Edit)

    Push or pull a flat face directly, and heal and merge the faces of an imported part.

19. Union, Subtract, Intersect, Scale and Rotate Datum

    Combine, subtract or intersect with another part, scale a part, and rotate it about the origin.

20. Text (Raised or Engraved Lettering on a Face)

    Write raised or recessed text on a flat face.

21. Picture (Image on a Face)

    Place a PNG/JPG image on a flat or cylindrical face, and adjust its size, rotation and mirroring.

22. Recognize Features (Convert an Imported Part to Features)

    Import CAD and automatically build an editable feature history (base Extrude + holes).

23. Gear Wizard

    Create parametric Spur, Rack, Internal, Helical, Worm and Bevel gears.

24. Feed Screw Wizard (bottle feed screw)

    Create a bottle feed/timing screw from the bottle shape, pitch and thread hand.

25. Part Color, Material and Opacity

    Set the colour, material (with density) and transparency of a part.

26. Rollback, Regen and Suppress

    Step back through the history with the Rollback bar, recalculate the part, and turn features off temporarily.

27. Parameters (named values and equations)

    Name the values you use often, such as width = 50, and write equations that refer to them. When you change a value, the part rebuilds automatically.

28. Part Properties

    Enter the part number, description, material, finish, revision, vendor and cost for the title block and BOM. Mass is computed from the material.

29. Surface Modeling

    Create and edit surfaces from sketches, edges or faces, then knit, thicken or cut with them to get a solid.

30. Sheet Metal

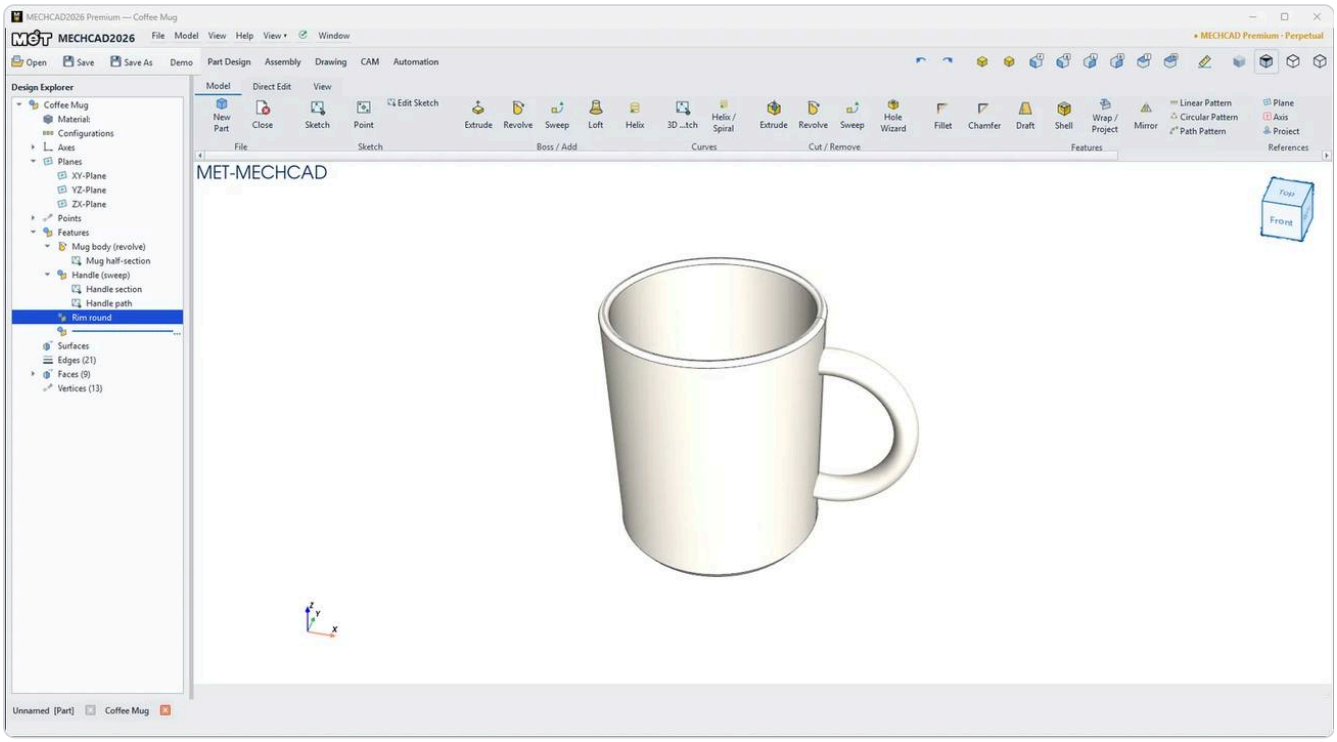
    Build sheet metal parts with Base Flange, Edge Flange, Hem, Jog and Sketched Bend, then unfold to a Flat Pattern and export a DXF for the laser.

Part Design Overview Standard

The workflow for building a parametric 3D part and the commands on the Part Design tab.

What it does

Part Design builds parametric 3D parts. Every feature is recorded as history in Design Explorer. You can edit a value at any time and the part recalculates.

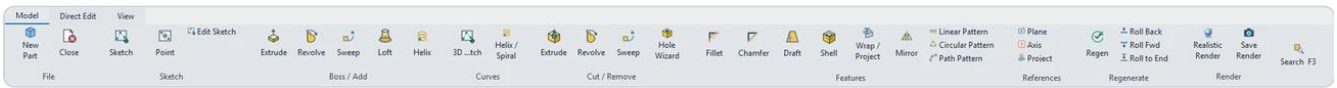


Part Design with the Coffee Mug part open

Typical workflow

1. Create a Sketch on a plane or a face.
2. Add material with a feature: Extrude, Revolve, Sweep, Loft, Helix.
3. Remove material: Extrude Cut, Revolve Cut, Sweep Cut, Hole Wizard.
4. Finish: Fillet / Chamfer, Draft, Shell, Wrap / Project.
5. Repeat: Mirror, Linear / Circular Pattern, Path Pattern.
6. Set the material and color and the Part Properties for drawings and BOM.
7. Save as `.metprt`.

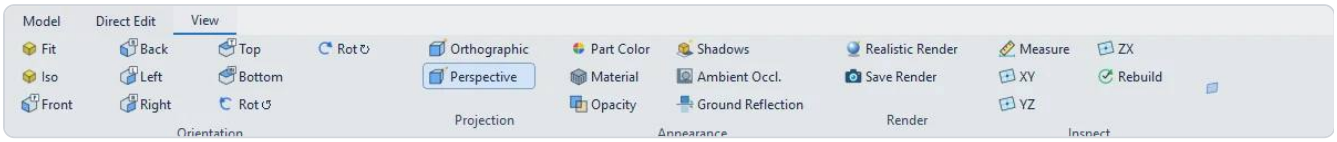
Part Design ribbon



Model tab



Direct Edit tab



View tab

For details of every group, see Ribbon and Command Search.

Right-click in the view

- Empty space: Measure, Size Box, Sketch, Edit Sketch, Boss (Extrude/Revolve/Sweep/Loft/Helix), Cut (Extrude/Revolve/Hole Wizard), Fillet, Chamfer, Shell, Mirror, Linear/Circular Pattern, Plane, Rebuild
- Face: Measure, Insert Sketch on Surface, feature color, Edit Feature, Edit Child Sketch, Suppress/Delete Feature, Insert Offset Plane from This Face..., Fillet/Chamfer Edges of This Face, Edit Face Color, Edit Solid Color

See also

- Design Explorer
- Feature Parameters

Feature Parameters: the feature edit box Standard

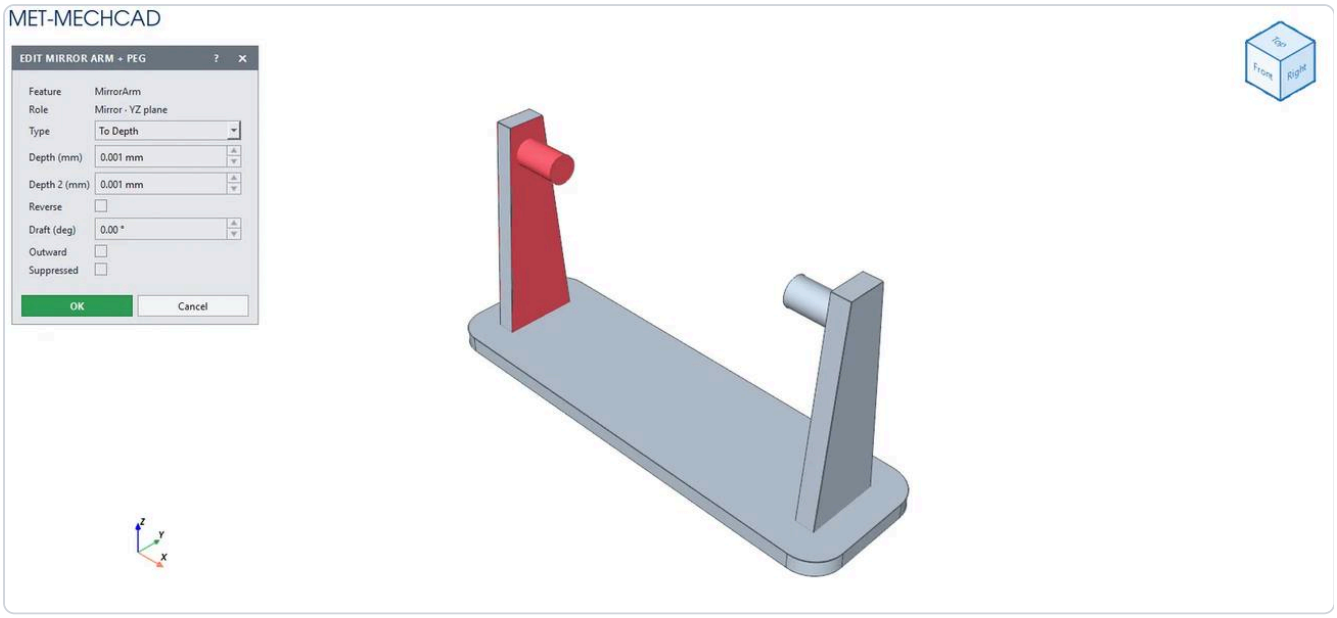
Edit an existing feature with an Alibre-style floating box, and the behaviour shared by all command boxes.

What it does

Every feature can be edited later through a floating box that is pre-filled with the existing values. Click OK and the part recalculates at the feature's original position in the history.

Where to find it

- Double-click a feature in the Design Explorer
- Right-click a feature (in the Design Explorer or on its face) › **Edit Feature**



Feature edit box (example: Mirror arm + peg on a paper towel holder)

Behaviour shared by all command boxes

Part	Description
Title bar (dark)	The command name in capitals, the **?** and ✕ buttons. Drag to move.
Pick fields (Sketch, Face, Edge)	Click in the view or the Design Explorer to fill them.
Orange text ▸	What you still need to select before you can click OK.
Red box	An error. The box stays open so you can correct it.
OK / Done (green)	Confirms.
Cancel	Cancels and restores the previous values.

Fields in the Edit box (general features)

Field	Description
Feature / Role	Name and role of the feature
Type	Distance type (To Depth, Through All …)
Depth (mm) / Depth 2 (mm) / Reverse	Depth and direction
Draft (deg) / Outward	Taper angle
Pattern features, Count, Spacing, Axis, Full 360°, Angle, Axis centre	Pattern settings
Turns, Pitch (mm), End pitch (0=const), Right-hand (CCW)	Helix settings
Suppressed	Turns calculation off

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** is saved as one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not yet met, the OK button is grey and an orange message tells you what to select.

See also

- Design Explorer
- Rollback and Regenerate

Extrude (Boss) Standard

Add material by pushing a closed sketch perpendicular to its plane using To Depth, Mid Plane, Two Directions or Up To Face.

What it does

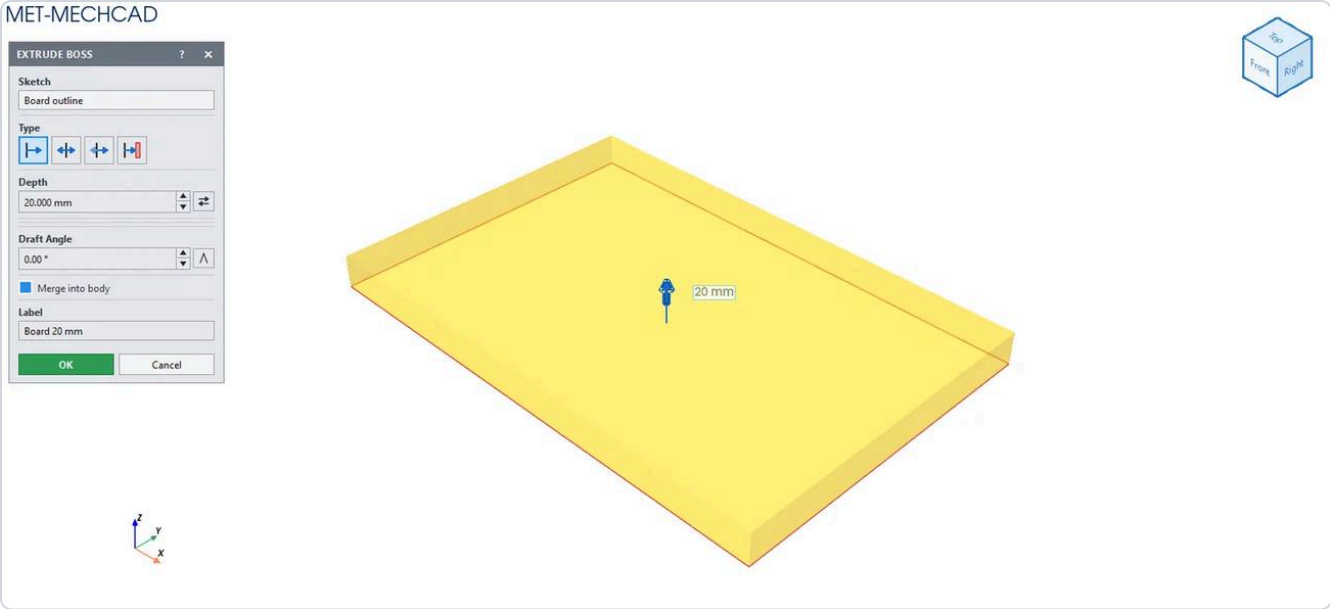
Extrude adds material by pushing a closed sketch out perpendicular to the sketch plane.

Where to find it

- **Model › Boss / Add › Extrude**
- **Model › Extrude** (menu), or right-click in the view › Boss › Extrude

Steps

1. Select an unused closed sketch (or click Extrude first, then click the sketch in the Design Explorer in step 1 and click **Next**).
2. The **EXTRUDE BOSS** box opens with a yellow translucent preview and a depth arrow.
3. Choose the **Type** and enter the **Depth**, or drag the arrow in the view.
4. Click **OK** to create a feature named *Extrusion Boss N*.

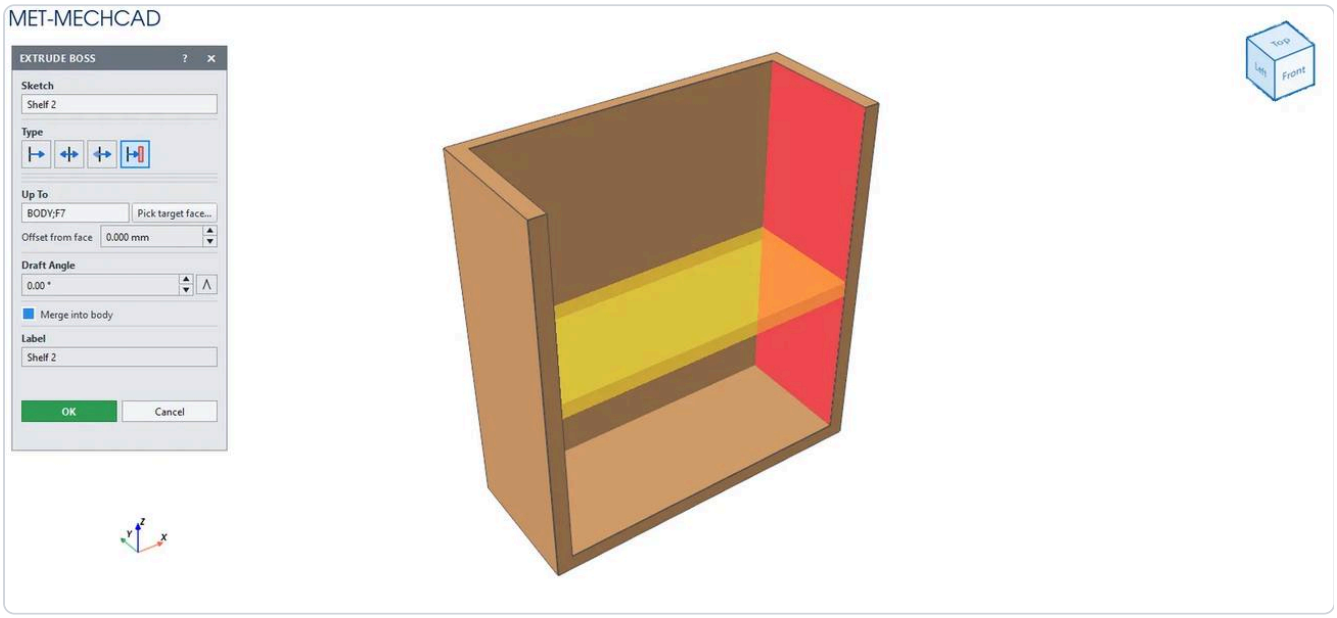


EXTRUDE BOSS with To Depth

Option	Description
Sketch	The sketch used (read-only)
Type › To Depth	Extrudes in one direction by the distance
Type › Mid Plane	Extrudes symmetrically on both sides of the plane
Type › Two Directions	Extrudes in two directions, with Depth and Depth 2 set separately
Type › Up To Face / Plane	Extrudes up to a selected face. Click **Pick target face...**, then click a face. **Offset from face** keeps a gap from the face.
Depth / Depth 2	Distance (mm). The default comes from System Options (5 mm). The ⇄ button reverses direction (or drag the arrow through the sketch).
Draft Angle	Wall taper angle ±59°. The Λ button switches between outward and inward.
Merge into body	Merges with the existing body (on by default)
Label	Feature name (leave blank for an automatic name)



Two Directions (dual depth)



Up To Face: choose the target face and Offset

> **Note:** With Up To Face, the OK button is disabled until you choose a target face. If the result cannot be calculated, the system rolls back and leaves the box open for you to correct.

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** is saved as one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not yet met, the OK button is grey and an orange message tells you what to select.

See also

- Extrude Cut
- Sketch overview

Extrude Cut Standard

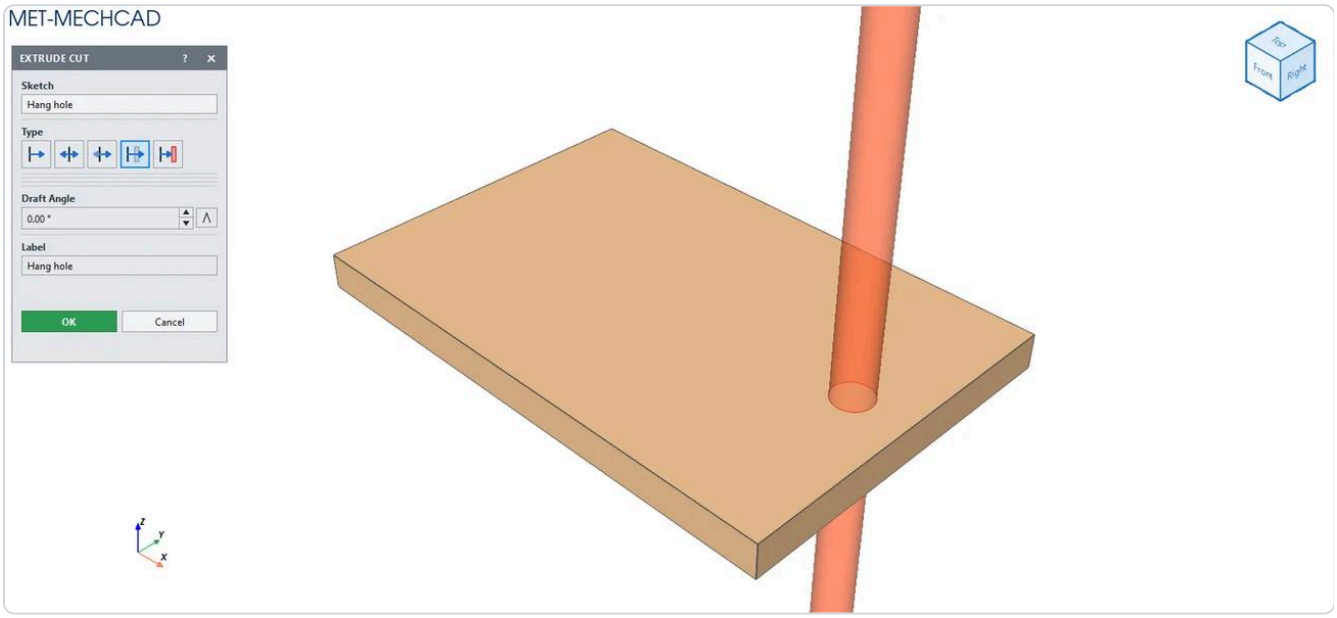
Remove material with a closed sketch using To Depth, Mid Plane, Two Directions, Up To Face or Through All.

What it does

Extrude Cut removes material from the part in the shape of a closed sketch. A body must exist first (*Create a body first*).

Where to find it

Model › Cut / Remove › Extrude, or right-click in the view › Cut › Extrude



EXTRUDE CUT with Through All

Steps

1. Draw a sketch on a face or plane, then click **Finish Sketch**.
2. Click **Cut / Remove › Extrude**.
3. Choose the Type and enter the depth. The preview is shown in red (a sketch on a face cuts into the material automatically).
4. Click **OK** to create the feature *Extrusion Cut N*.

Option	Description
Type › To Depth / Mid Plane / Two Directions / Up To Face	Same as Extrude
Type › **Through All**	Cuts through the whole part (Cut only)
Depth / Depth 2 / ⇄	Distance and direction
Draft Angle	Taper angle of the hole walls
Label	Feature name



Two Directions (shown on an Extrude Boss; Extrude Cut uses the same options)

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** is saved as one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not yet met, the OK button is grey and an orange message tells you what to select.

See also

- Hole Wizard
- Extrude

Revolve and Revolve Cut Standard

Add or remove material by revolving a profile around an axis.

What it does

Revolve revolves a sketch around an axis to add material (a shaft, wheel or bottle). **Revolve Cut** revolves it to remove material (a groove).

Where to find it

- **Model › Boss / Add › Revolve**
- **Model › Cut / Remove › Revolve**



REVOLVE box with live preview (coffee mug body, Angle 270°)

Steps

1. Draw a sketch of half the cross-section on one side of the axis.
2. Click **Revolve** and select the sketch.
3. Choose the **Axis** and enter the **Angle**.
4. Click **OK**.

Option	Meaning
Sketch	The profile to use
Axis	One of the two main axes in the sketch plane (for example XY › X, Y), a reference axis Axis, or *Reference line:* (the first line of another sketch)
Angle	0.1–360° (default 360)
Label	Feature name

> **Note:** If the feature cannot be calculated, you see *Revolve failed:* …. Check that the profile does not cross the axis.

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** saves one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not complete, the OK button is grey and an orange message tells you what to select.

See also

- Reference planes and axes

Sweep and Sweep Cut Standard

Sweep a closed profile along a path to add or remove material.

What it does

Sweep sweeps a closed profile along a path, for example a pipe, a rail or a trim edge. **Sweep Cut** sweeps to cut a groove.

Where to find it

- **Model › Boss / Add › Sweep**
- **Model › Cut / Remove › Sweep**



SWEEP box (mug handle: section + path)

Steps

1. Create a (closed) profile sketch and a path sketch (or a 3D Sketch / Helix / Spiral).
2. Click **Sweep** and choose the **Profile (section)** and the **Path (spine)**.
3. Click **OK**.

Option	Meaning
Profile (section)	The cross-section sketch (closed)
Path (spine)	The sketch / 3D Sketch / Helix used as the path

> **Note:** The profile and the path must be different sketches (*Profile and path must be different sketches*). Sweep Cut needs a body first.

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** saves one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not complete, the OK button is grey and an orange message tells you what to select.

See also

- 3D Sketch
- Helix / Spiral

Loft Standard

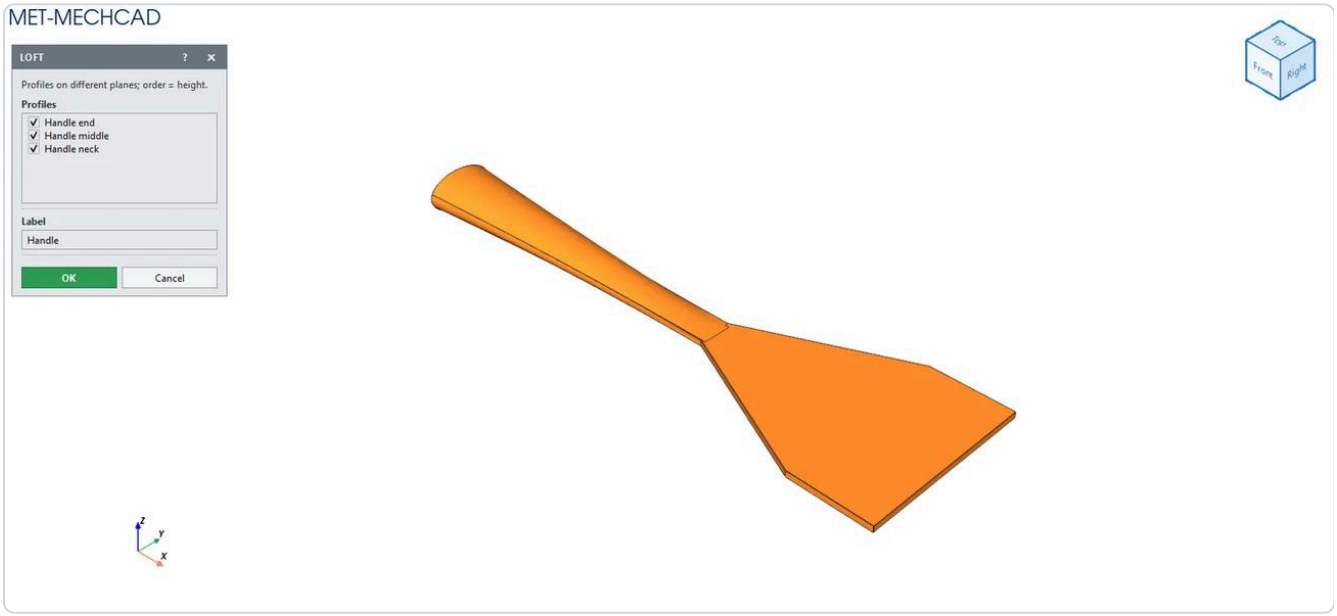
Create a solid that blends through two or more profiles on planes at different levels.

What it does

Loft creates a shape whose cross-section changes smoothly through several profiles, such as a handle or a bottle neck.

Where to find it

Model › Boss / Add › Loft



LOFT box (spatula handle from three profiles)

Steps

1. Create reference planes at different levels and draw a profile on each plane.
2. Click **Loft** and tick at least two sketches in the **Profiles** list.
3. Enter a name in the **Label** box and click **OK**.

Option	Meaning
Profiles	List of sketches (at least 2), sorted automatically by plane height.
Label	Feature name.

> **Note:** Loft only adds material (there is no Loft Cut).

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** saves one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not complete, the OK button is grey and an orange message tells you what to select.

See also

- Reference planes and axes

Helix (springs and threads) Standard

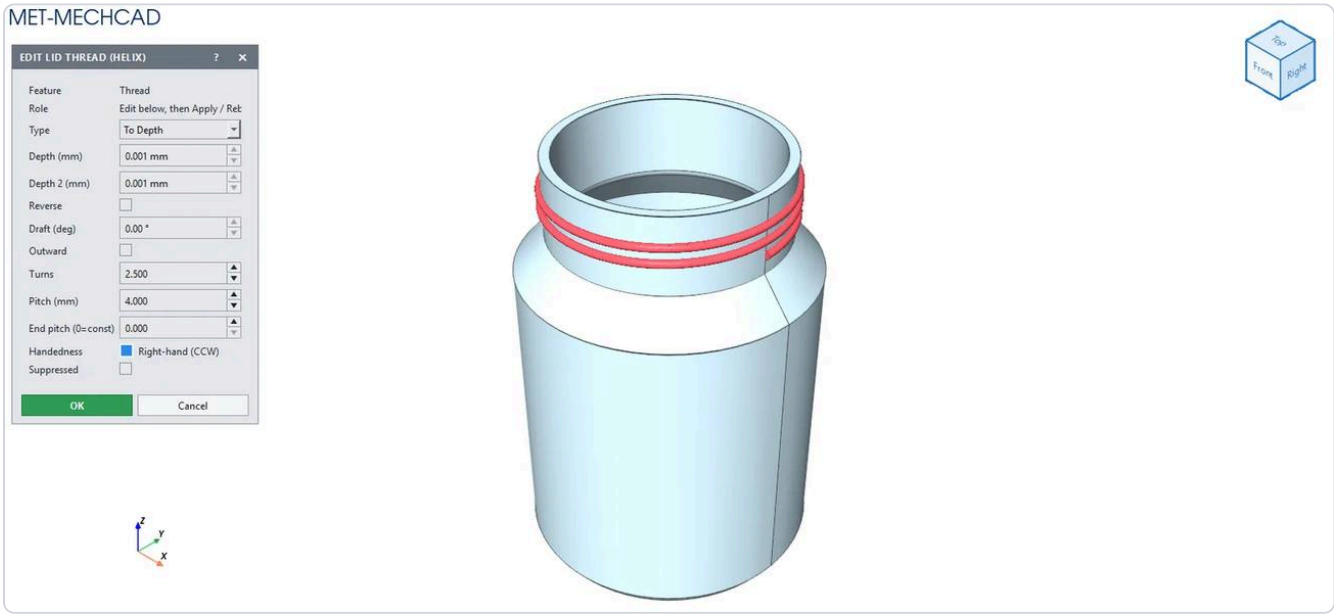
Sweep a cross-section along an automatically generated helix for springs and threads.

What it does

Helix creates a spring or thread in one command. It sweeps a cross-section sketch along a helix made from a guide circle.

Where to find it

Model › Boss / Add › Helix



Editing a Helix (lid thread on a jam jar)

Steps

1. Draw two separate sketches: a guide circle (the spring diameter) and the wire cross-section.
2. Click **Helix** and choose the Wire section and the Guide circle.
3. Enter Turns and Pitch, then click **OK**.

Option	Default	Description
Wire section (sketch)	–	Wire cross-section
Guide circle (spring Ø) (sketch)	–	Circle that sets the diameter
Turns	5	Number of turns
Pitch	10 mm	Distance per turn
End pitch (0 = constant)	0	Distance per turn at the end (variable)
Right-hand (CCW)	On	Right-hand thread

> **Note:** The Wire section and the Guide circle must be in different sketches. For a helix with a complex profile, use Helix / Spiral + Sweep.

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** is saved as one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not yet met, the OK button is grey and an orange message tells you what to select.

Hole Wizard (standard holes) Standard

Drill standard M3–M12 holes (Simple, Counterbore, Countersink, Tapped or Tapered) by clicking a face or a Point.

What it does

Creates standard holes with sizes from a table. You can place them in two ways: click a face (one click per hole), or drill every Point in a sketch.

Where to find it

Model › Cut / Remove › Hole Wizard (a body must exist)



HOLE WIZARD box

Steps

1. (To place holes at points) Draw a sketch with the **Point** tool at the hole positions.
2. Click **Hole Wizard** and choose the **Standard** and **Type**.
3. Under **Position**, choose *Click on a face (one hole per click)* and click the face at the hole position (it snaps to the centre of a circular edge or the centre of the face), or choose *- all N points (Drill)*.
4. Click **Done** (or **Drill**). Press Esc to finish.

Option	Default	Description
Standard	M6	Custom, M3, M4, M5, M6, M8, M10, M12 (fills in the Ø automatically)
Type	Simple	Simple / Counterbore / Countersink / Tapped / Tapered Bore
Hole Ø (mm)	6.6	Hole size (clearance hole) or tap drill (Tapped)
Through all	On	Drills all the way through
Blind depth (mm)	10	Depth when not through
C'bore Ø / C'bore depth (mm)	11 / 6	Counterbore size and depth
C'sink head Ø (mm)	12.5	Countersink head size (90°)
Thread depth (mm)	0	Thread depth (0 = full depth of the hole)
Cosmetic thread note on the feature	On	Adds the thread to the feature name
Show thread (cosmetic helix)	On	Shows the thread as lines
Taper angle (deg)	3	Taper angle (Tapered Bore)
Position	Click on a face	How the hole is placed

Standard	Clearance hole (mm)	Tap drill (mm)
M3	3.4	2.5
M4	4.5	3.3
M5	5.5	4.2
M6	6.6	5.0
M8	9.0	6.8
M10	11	8.5
M12	13.5	10.2

> **Note:** To edit a hole later, use Edit Feature. It opens the same *Edit* box. Deleting a hole deletes the hole and its whole sketch. In a Drawing, use **Hole Callout** to show hole sizes automatically.

See also

- Spline, Point and Text
- Dimensions in a Drawing

Fillet and Chamfer Standard

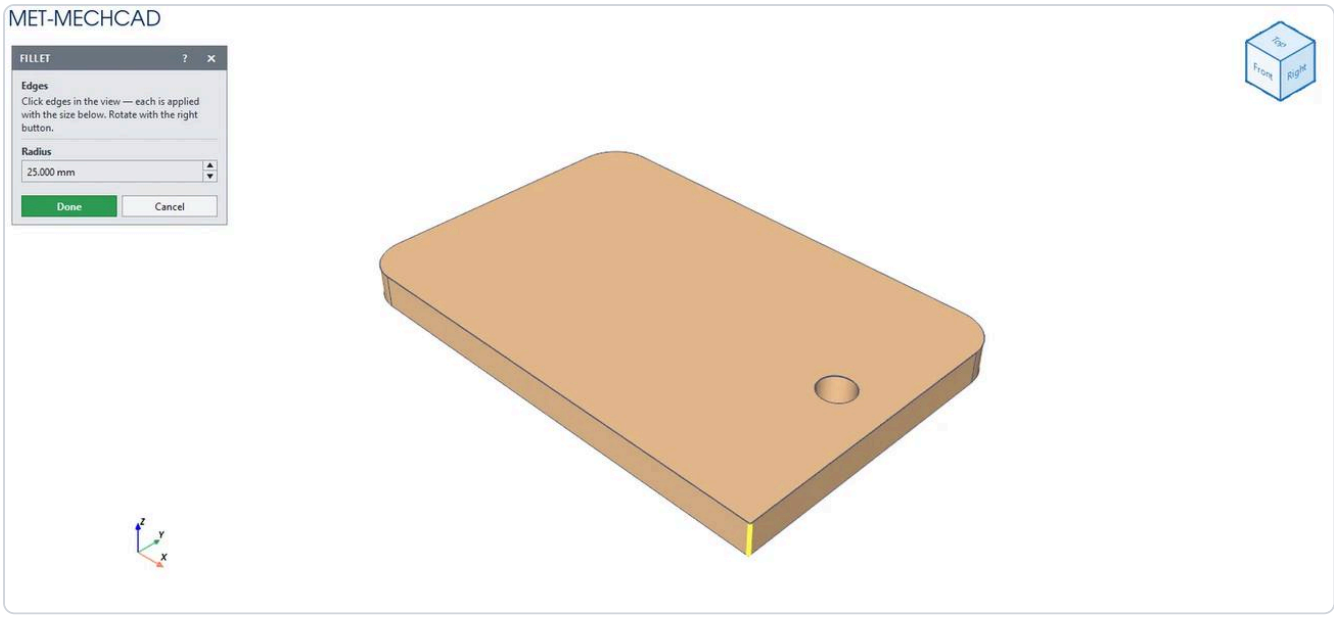
Round (Fillet) or bevel (Chamfer) part edges by clicking them.

What it does

Fillet rounds an edge with a radius. **Chamfer** bevels a corner by a distance.

Where to find it

- **Model › Features › Fillet / Chamfer** (a body must exist)
- Right-click a face › **Fillet Edges of This Face** / **Chamfer Edges of This Face** (applies to every edge of that face)



FILLET box

Steps

1. Click **Fillet** (or **Chamfer**). A floating box opens beside the view.
2. Enter the **Radius** (or **Distance**).
3. Click edges in the view to build an **Edge set**. Selected edges turn orange, and the Edge set row shows *N edge(s) selected*.
4. Check the preview. It is recalculated every time you add or remove an edge or change the size.
5. Click **OK** to create **one** feature (Fillet N / Chamfer N) for the whole edge set, or click **Cancel** / press Esc to discard it. The part goes back to how it was.

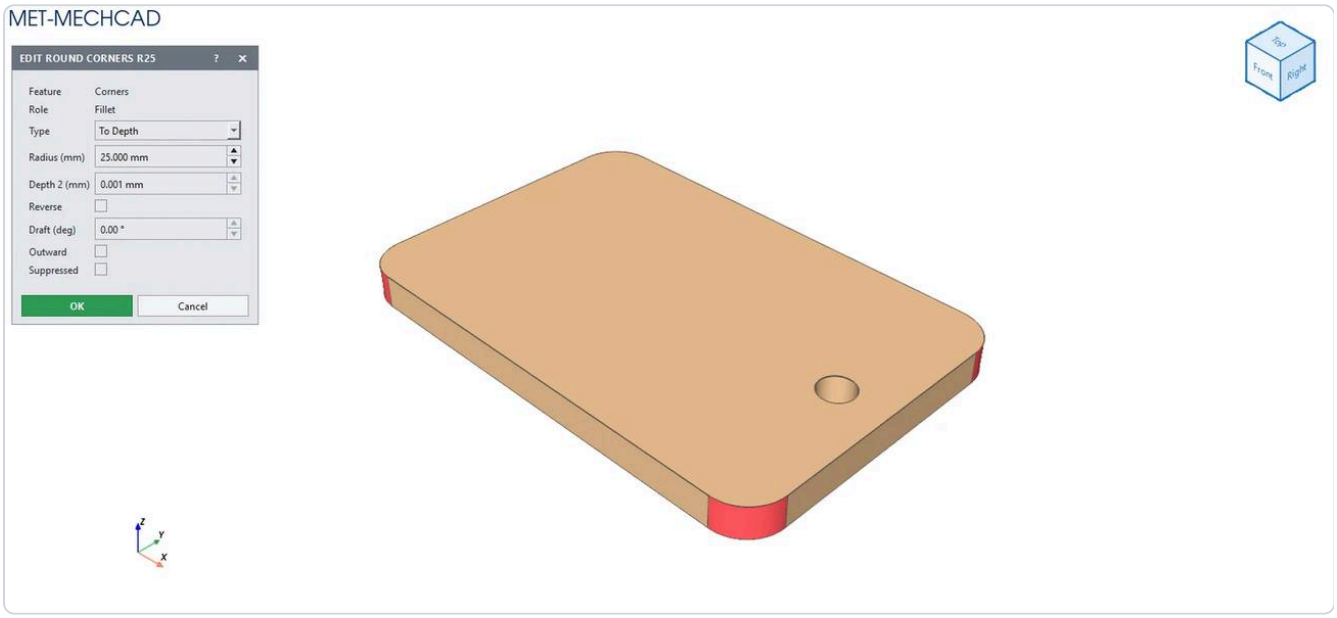
Selecting an edge set

Click	Result
Click an edge	Adds that edge to the set. Click the same edge again to remove it.
Click a face	Adds every edge of that face at once (round a whole rim in one click). Click the same face again to remove those edges.
Tab	Cycles through overlapping edges under the cursor
Right-drag	Rotates the view to reach back edges while you select

Option	Default	Description
Edge set	–	Number of edges currently selected
Radius (Fillet)	3 mm	Fillet radius
Distance (Chamfer)	3 mm	Chamfer distance



CHAMFER box



Editing an existing fillet

> **Note:** If the size is too large for the edges in the set, the preview shows the original part with a warning in the box, and OK shows *could not be built on this edge set*. Reduce the size or remove the problem edge.

> **Note:** Every edge in the set uses the same size. For different sizes, create separate features. To change the size later, right-click the feature › **Edit Feature** (**Radius (mm)** field). The edge set itself cannot be re-picked; to change the edges, delete the feature and create it again.

See also

- Feature Parameters

Draft Standard

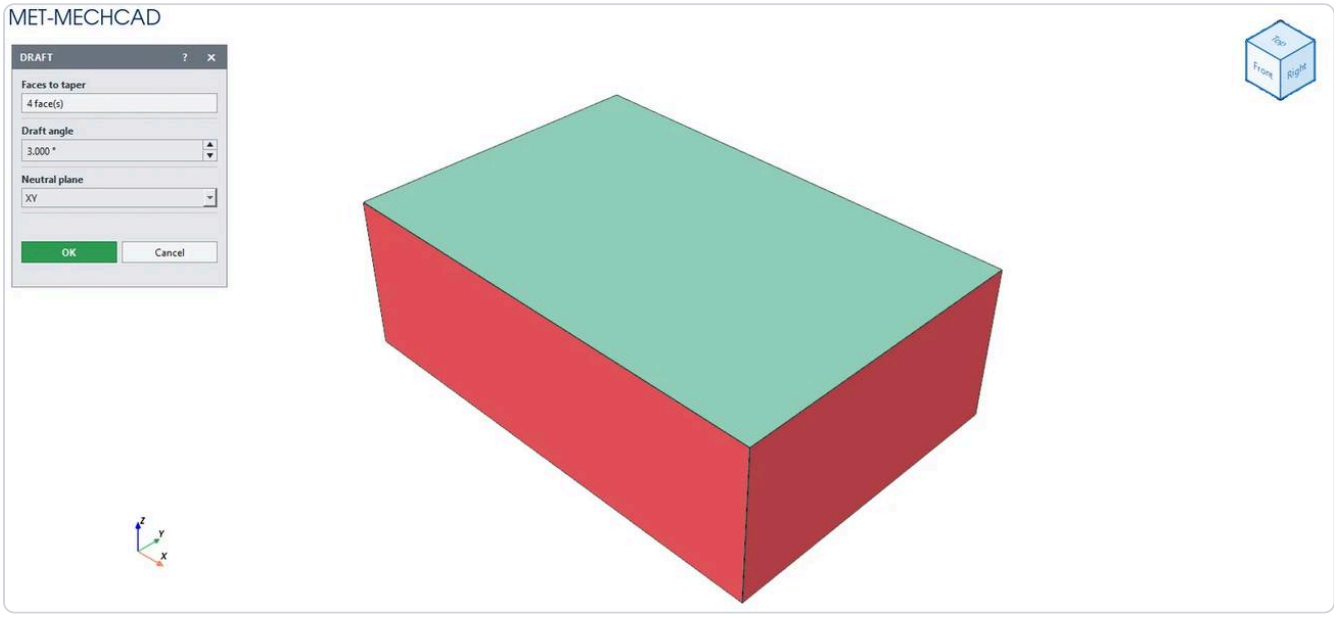
Tilt side walls so the part can be released from a mould.

What it does

Draft tilts side walls about a reference (neutral) plane, for cast or moulded parts.

Where to find it

Model › Features › Draft



DRAFT box

Steps

1. Click **Draft**.
2. Click the side walls you want to tilt (click again to remove one).
3. Enter the **Draft angle** and choose the **Neutral plane**.
4. Click **OK**.

Option	Default	Description
Faces to taper	–	The wall faces to tilt
Draft angle	3°	0.001–88.999°
Neutral plane	XY	XY / YZ / ZX

> **Note:** If you see *Draft failed (try a smaller angle or a side wall)*, reduce the angle or select only side walls. The box stays open so you can correct it.

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** is saved as one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not yet met, the OK button is grey and an orange message tells you what to select.

Shell Standard

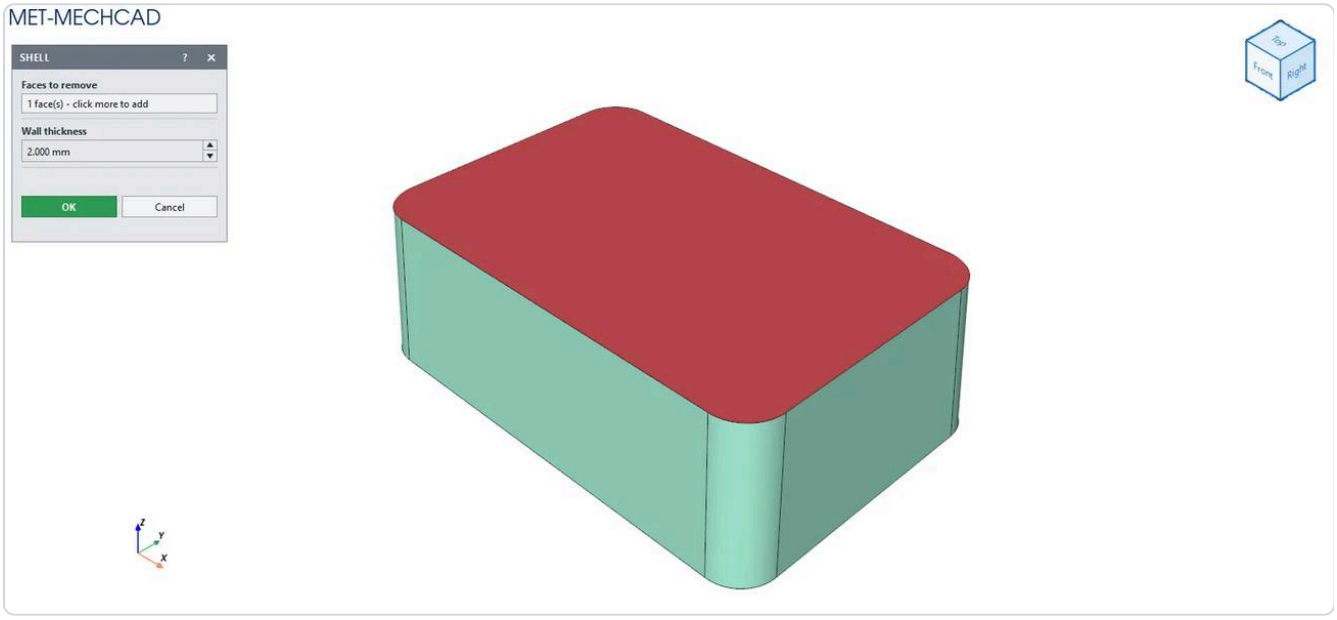
Hollow out a part to a wall thickness, leaving the selected faces open.

What it does

Shell removes the material inside a part, leaving walls of a set thickness, and opens the faces you select. Use it for boxes and covers.

Where to find it

Model › Features › Shell



SHELL box

Steps

1. Click **Shell**.
2. Click the faces you want to open (at least 1).
3. Enter the **Wall thickness**.
4. Click **OK** (or press Esc/Enter).

Option	Default	Meaning
Faces to remove	–	The faces to open
Wall thickness	2 mm	Wall thickness

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** saves one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not complete, the OK button is grey and an orange message tells you what to select.

Wrap / Project (Emboss / Deboss on a Face) Standard

Wrap a sketch around a cylinder or project it onto a face, then raise or recess it.

What it does

Places a closed sketch on a face, then makes it **raised (Emboss)** or **recessed (Deboss)**. **Wrap** mode rolls the sketch around a cylinder and keeps its length. **Project** projects it straight onto the face, for example a logo on a bottle.

Where to find it

- **Model › Features › Wrap / Project**
- While sketching: **Reference / Snap › Wrap** (finishes the sketch and opens this command right away)



WRAP / PROJECT box

Steps

1. Draw a closed sketch (for example text or a logo).
2. Click **Wrap / Project** and choose the **Sketch**.
3. Click the **Target face** (clicking a cylindrical face switches to Wrap mode automatically).
4. Choose the **Mode**, **Type** and **Depth**, then click **OK**.

Option	Default	Meaning
Sketch	–	The closed sketch to place
Target face	–	The target face
Mode	Wrap	**Wrap (cylinder, keep length)** / **Project (along sketch normal)**
Type	Emboss	**Emboss (raise)** raised / **Deboss (recess)** recessed
Depth	1 mm	Height/depth (0.01–1000)



LEMON logo wrapped as an emboss on a bottle

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** saves one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not complete, the OK button is grey and an orange message tells you what to select.

See also

- Text (Emboss / Deboss)
- Picture (image on a face)

Mirror Standard

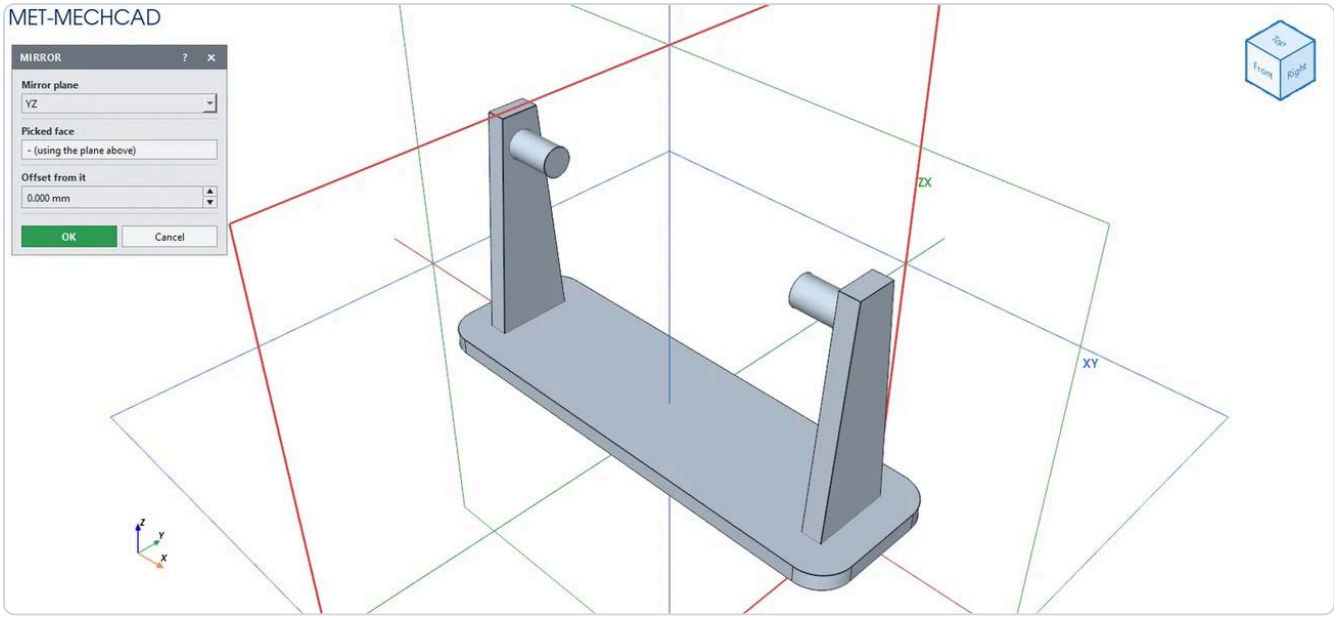
Mirror a body about a plane or a flat face and merge it into one part.

What it does

Mirror reflects a body about a plane, then fuses the copy with the part. Use it for symmetric parts.

Where to find it

Model › **Features** › **Mirror**, or right-click in the view › Mirror



MIRROR box

Steps

1. Click **Mirror**.
2. Select a **Mirror plane**, or click a flat face in the view.
3. Enter **Offset from it** if you want to shift the mirror plane.
4. Click **OK**.

Option	Default	Meaning
Mirror plane	YZ	YZ / ZX / XY / reference plane / clicked face
Picked face	–	The flat face used as the plane
Offset from it	0	Distance to shift from the plane

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** saves one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not complete, the OK button is grey and an orange message tells you what to select.

See also

- Mirror a part in an assembly

Linear Pattern and Circular Pattern Standard

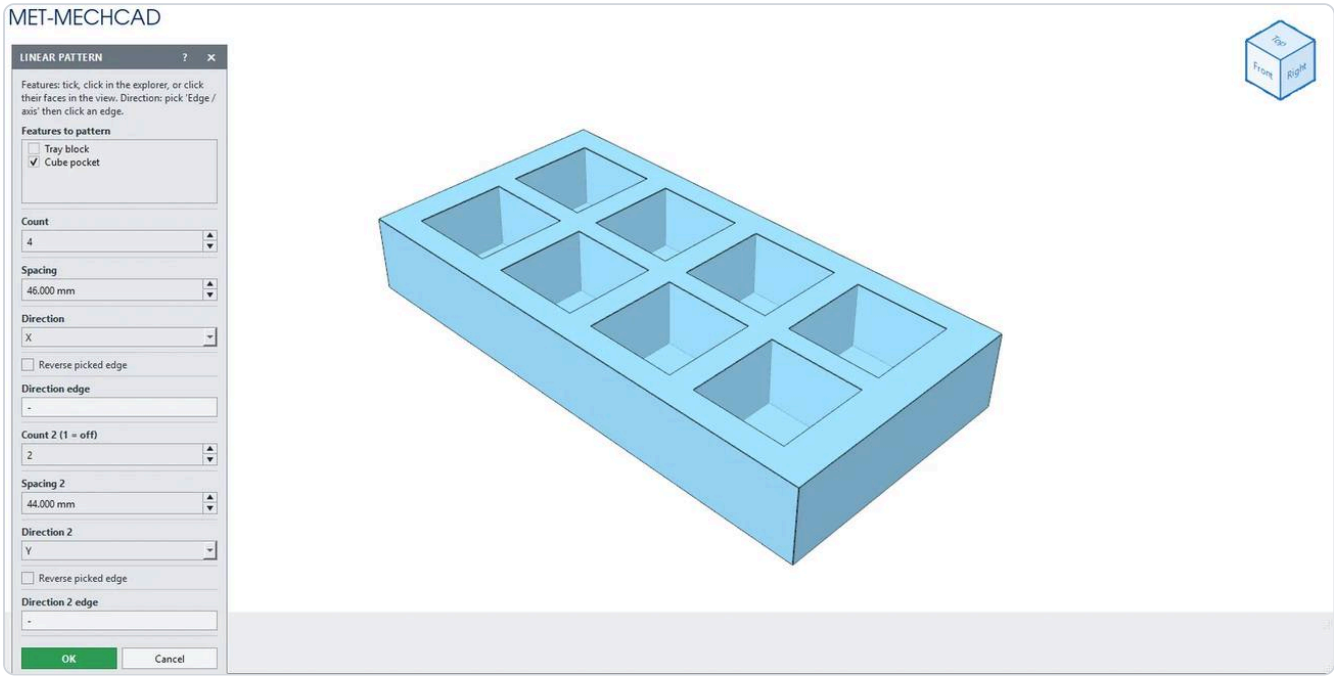
Repeat a feature along a line (one or two directions) or around an axis.

What it does

Repeats the selected feature (Extrude, Revolve, Loft, Sweep, Helix). **Linear Pattern** arranges copies along one or two directions. **Circular Pattern** arranges copies around an axis.

Where to find it

Model › Features › Linear Pattern / Circular Pattern



LINEAR PATTERN box (ice cube tray pockets, 4 × 2)

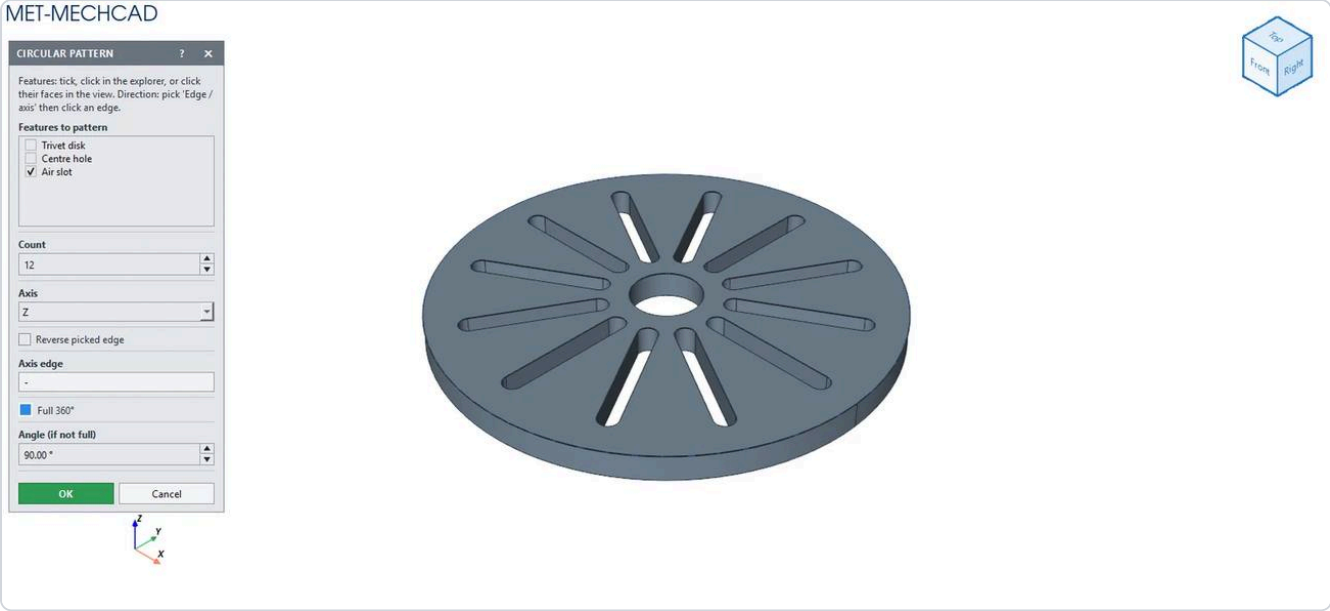
Steps

1. Click **Linear Pattern** (or **Circular Pattern**).
2. Tick features in the **Features to pattern** list, or click a feature in Design Explorer, or click one of its faces in the view. The latest feature is ticked for you.
3. Set the direction: choose X/Y/Z, or choose *Edge / axis* and click an edge (a straight edge = the edge direction, a circular edge = its axis, two clicked vertices = the direction between them).
4. Enter the count and spacing, then click **OK**.

Linear Pattern

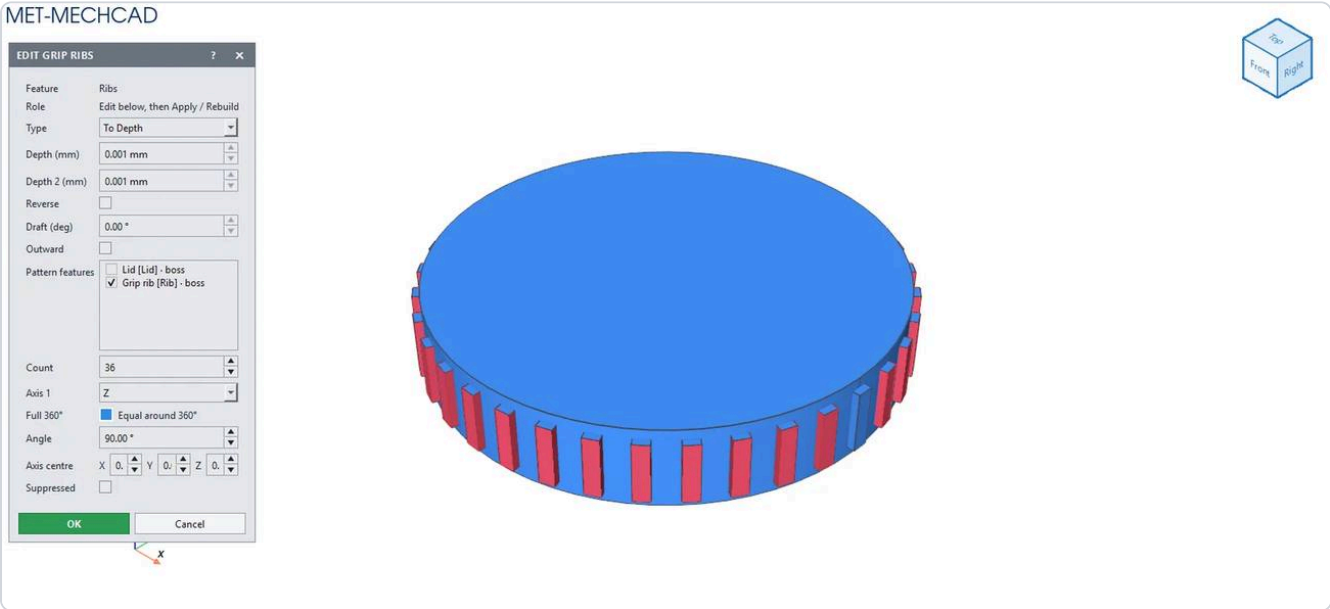
Option	Default	Meaning
Features to pattern	Latest feature	The features to repeat
Count	4	Number of instances (2–9999)
Spacing	30 mm	Distance between instances
Direction	X	X, Y, Z, -X, -Y, -Z, a reference axis, or a clicked edge
Reverse picked edge	Off	Reverses the edge direction
Count 2 (1 = off) / Spacing 2 / Direction 2	1 / 30 / Y	The second direction

Circular Pattern



CIRCULAR PATTERN box

Option	Default	Meaning
Count	6	Number of instances
Axis	Z	Rotation axis, or a clicked edge/axis
Full 360°	On	Spreads the instances evenly around the circle
Angle (if not full)	90°	Total angle when not 360°



Edit Circular Pattern (grip ribs around a jar lid)

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** saves one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not complete, the OK button is grey and an orange message tells you what to select.

See also

- Path Pattern
- Mirror and Pattern in a sketch

Path Pattern Standard

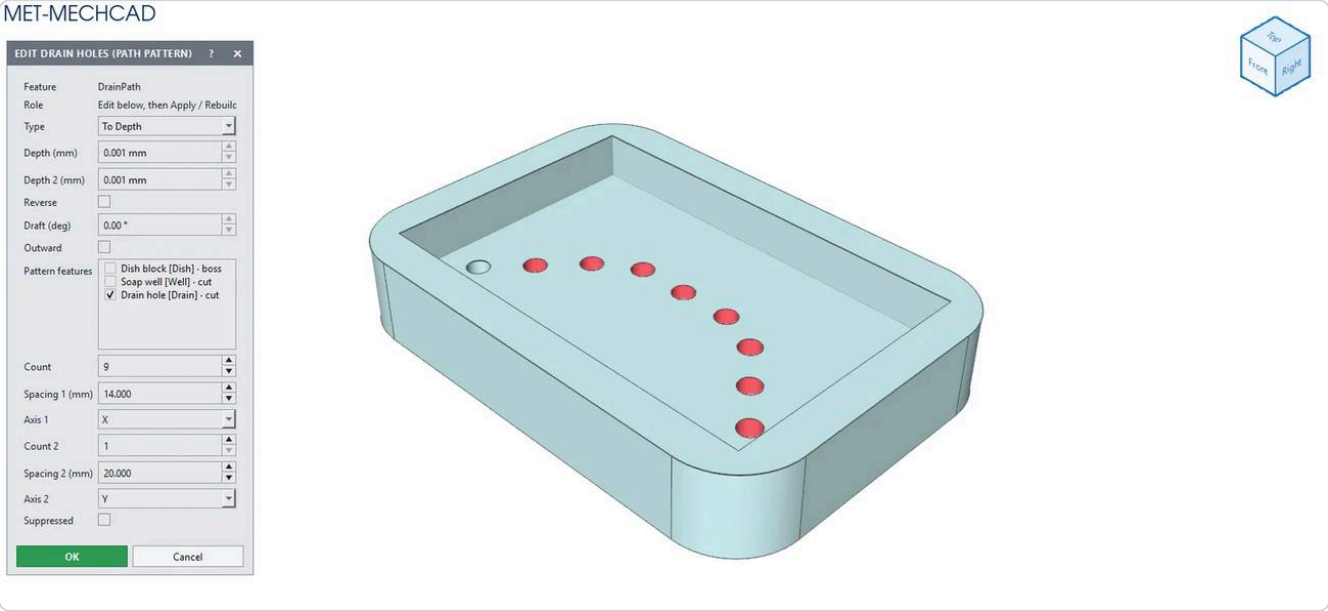
Repeat a feature along a selected edge or curve.

What it does

Path Pattern repeats a feature along an edge or curve. Spacing = path length ÷ (count − 1).

Where to find it

Model › Features › Path Pattern



Edit Path Pattern (drain holes along a path in a soap dish)

Steps

1. Click **Path Pattern**.
2. Tick the features in **Features to pattern**.
3. Click the edge to use as the path (**Path edge**).
4. Enter the **Count** and click **OK**.

Option	Default	Meaning
Features to pattern	–	The features to repeat
Path edge	–	The edge/curve used as the path
Count	4	Number of instances

> **Note:** When you edit a Path Pattern later, it asks for the spacing directly.

> **Note:** The command box is floating (it does not lock the screen). Drag its title bar to move it. The preview updates immediately when you change a value. **OK** saves one Undo step. **Cancel**, ✕ or Esc cancels. If the conditions are not complete, the OK button is grey and an orange message tells you what to select.

Reference Plane and Axis Standard

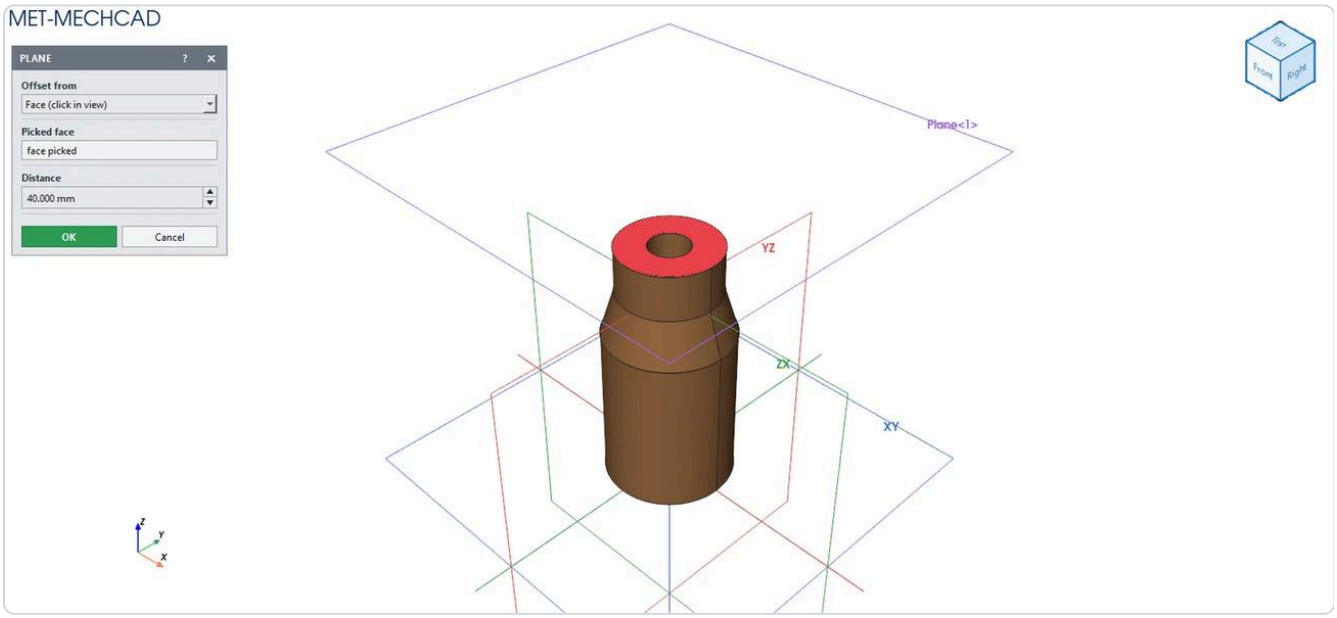
Create offset reference planes, and reference axes from a hole, an edge or two points.

What it does

Plane creates a reference plane parallel to a main plane or a face. Use it to draw sketches at different levels (for example for a Loft). **Axis** creates a reference axis for Revolve, Circular Pattern and Mate.

Where to find it

- **Model › References › Plane / Axis**
- Right-click a face › **Insert Offset Plane from This Face...**
- Right-click **Planes** in Design Explorer › **Insert Plane...**



PLANE box

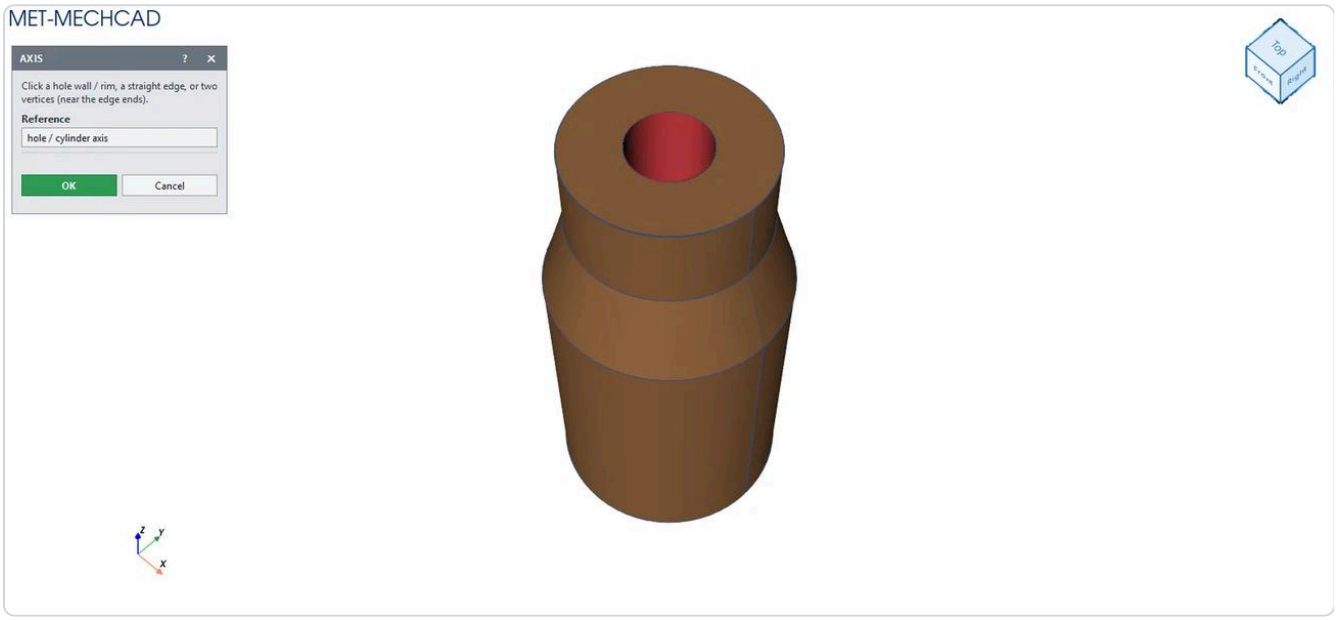
Plane

1. Click **Plane**.
2. Choose **Offset from** (XY / YZ / ZX / a reference plane), or click a flat face.
3. Enter the **Distance** and click **OK**. You get `Plane<n>`. Double-click it to start a sketch on that plane.

Option	Default	Meaning
Offset from	XY	Base plane
Picked face	–	A flat face (used instead of the base plane)
Distance	20 mm	Offset distance

To change the distance later, right-click the plane › **Edit Plane...** (Offset box). Features that use this plane recalculate.

Axis



AXIS box

1. Click **Axis** (a body must exist).
2. Click the wall or edge of a hole/cylinder (the axis of the hole), a straight edge, or two vertices.
3. Click **OK**. You get `Axis<n>`.

See also

- Loft
- Revolve

Move Face and Simplify (Direct Edit) Standard

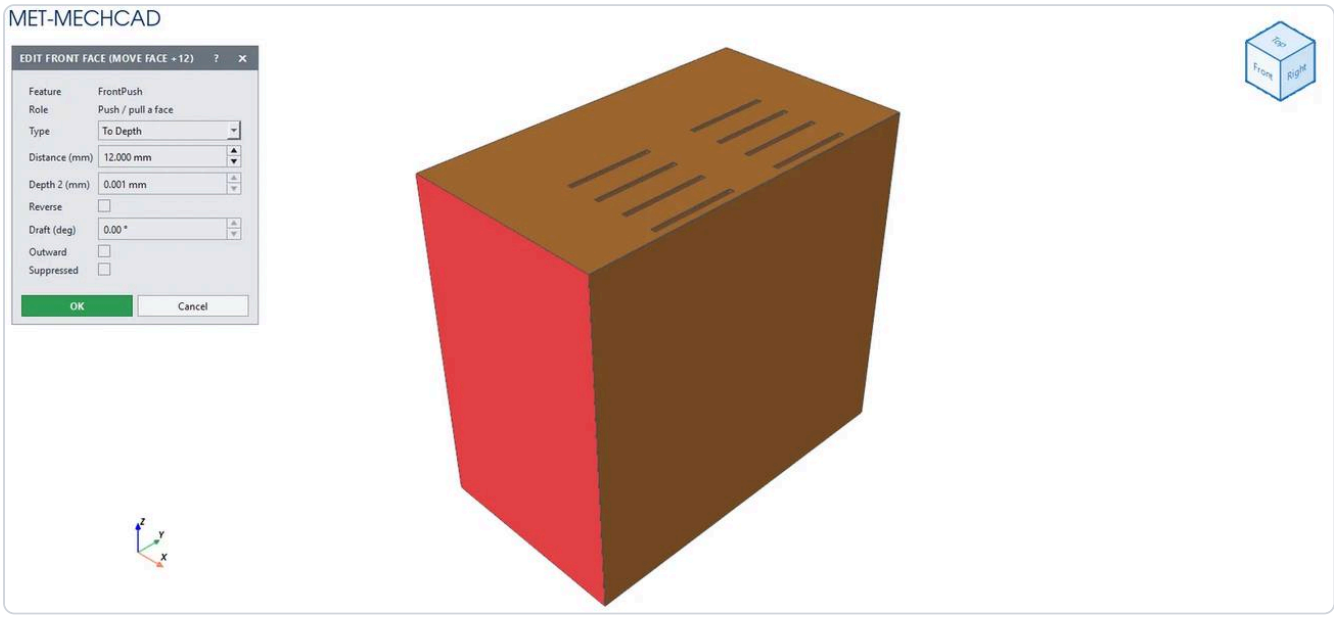
Push or pull a flat face directly, and heal and merge the faces of an imported part.

What it does

- **Move Face** pushes or pulls a flat face by a distance (+ adds material, − removes material). It suits imported parts that have no feature history.
- **Simplify** repairs import errors and merges faces that lie on the same plane.

Where to find it

Direct Edit › Solid › Move Face / Simplify



Edit Move Face (front face of a knife block pushed +12 mm)

Move Face

1. Click **Move Face**.
2. Click one flat face.
3. Enter **Distance (mm)** (default 5; enter a negative value to push inward), then click OK (Esc = cancel).

Simplify

1. Click **Simplify**.
2. Confirm *Simplify / Heal: Heal merged A → B faces. Replace the feature history…?*

> **Note:** Simplify **replaces the whole feature history** with a single solid named *Simplified Solid*. Use Undo if you want to go back.

See also

- Import CAD
- Recognize Features

Union, Subtract, Intersect, Scale and Rotate Datum Standard

Combine, subtract or intersect with another part, scale a part, and rotate it about the origin.

What it does

These commands work on the whole body (a body must exist).

Command	Description
Union / Add	Merge with another part (`.metprt`).
Subtract	Remove the shape of another part.
Intersect	Keep only the overlapping region.
Scale	Scale about the origin (*Uniform scale about origin*, 0.001–1000).
Rotate Datum	Rotate the body about the origin (choose axis X/Y/Z, then the angle; default 90°) while keeping the feature history.

Where to find it

Direct Edit › Solid › Union / Add, Subtract, Intersect, Scale and the **Model › Part Tools** menu



Editing a body-union feature (coat hook)

Steps (Union / Subtract / Intersect)

1. Click the command.
2. Select the tool part file (*Select tool part (shared XYZ coordinates)*).
3. The part recalculates immediately.

> **Note:** Both parts share the same XYZ coordinates (there is no positioning step). Place the tool part at the correct coordinates first.

See also

- Derived Part in an Assembly

Text (Raised or Engraved Lettering on a Face) Standard

Write raised or recessed text on a flat face.

What it does

Text creates raised (Emboss) or engraved (Deboss) text on a flat face, such as a model name or a lettered logo.

Where to find it

Direct Edit › Create › Text



Edit "SUGAR" emboss text on a canister

Steps

1. Click **Text**.
2. Click the flat face where you want the text.
3. Enter the values in the **Text** window and click OK.

Option	Default	Meaning
Text	MET	The text
Height (mm)	8	Letter height
Depth (mm)	1	Raised height / depth
Style	Emboss (raised)	Emboss (raised) / Deboss (engraved)

> **Note:** If you want text on a curved surface, use the **Text** tool in a sketch, then use Wrap / Project.

See also

- Spline, Point and Text in a sketch

Picture (Image on a Face) Standard

Place a PNG/JPG image on a flat or cylindrical face, and adjust its size, rotation and mirroring.

What it does

Picture (*Insert Picture on Face*) places an image on a flat or cylindrical face, such as a product label or a machine nameplate. It is for display and rendering only and does not change the shape.

Where to find it

- Part Design: **Direct Edit › Create › Picture**
- Assembly: **Share › Picture** (see 3D Notes and Pictures)



EDIT PICTURE box on a dish-soap bottle label

Steps

1. Click **Picture**, then click **Browse picture...** (png, jpg, jpeg, bmp, gif, webp).
2. Click the face to place the picture on.
3. Adjust the size, rotation and position, check the live preview, then click **OK**.

Option	Meaning
Browse picture... / Replace picture...	Choose the image file

Option	Meaning
Face	The face the picture is placed on
Width / Height (0 = keep picture ratio)	Size (Height 0 = keep the proportions)
Fit	Auto (flat / wrap cylinder) / Stretch fit onto curved face
Rotation	Rotate ±360°
Move X / Move Y	Shift the position
Offset from face	Distance above the surface (default 0.2)
Mirror horizontal / Mirror vertical	Flip the image
Opacity	0.05–1

See also

- Realistic Render

Recognize Features (Convert an Imported Part to Features) Standard

Import CAD and automatically build an editable feature history (base Extrude + holes).

What it does

Recognize Features reads a solid from a CAD file (for example STEP) and rebuilds it as an editable feature tree, such as a base Extrude, steps, holes, Fillets and Chamfers.

Where to find it

Direct Edit › Create › Recognize Features



Direct Edit tab: Create group

Steps

1. Click **Recognize Features**.
2. Select a file in the *Recognize Features from CAD* window.
3. The result opens in a new Part tab, with a message summarizing the conversion.

> **Note:** If the rebuilt shape does not match the original after verification, the program uses the original solid (with no history) instead, to stay accurate. A part with a complex shape may not get all its features.

See also

- Import CAD
- Move Face and Simplify

Gear Wizard Standard

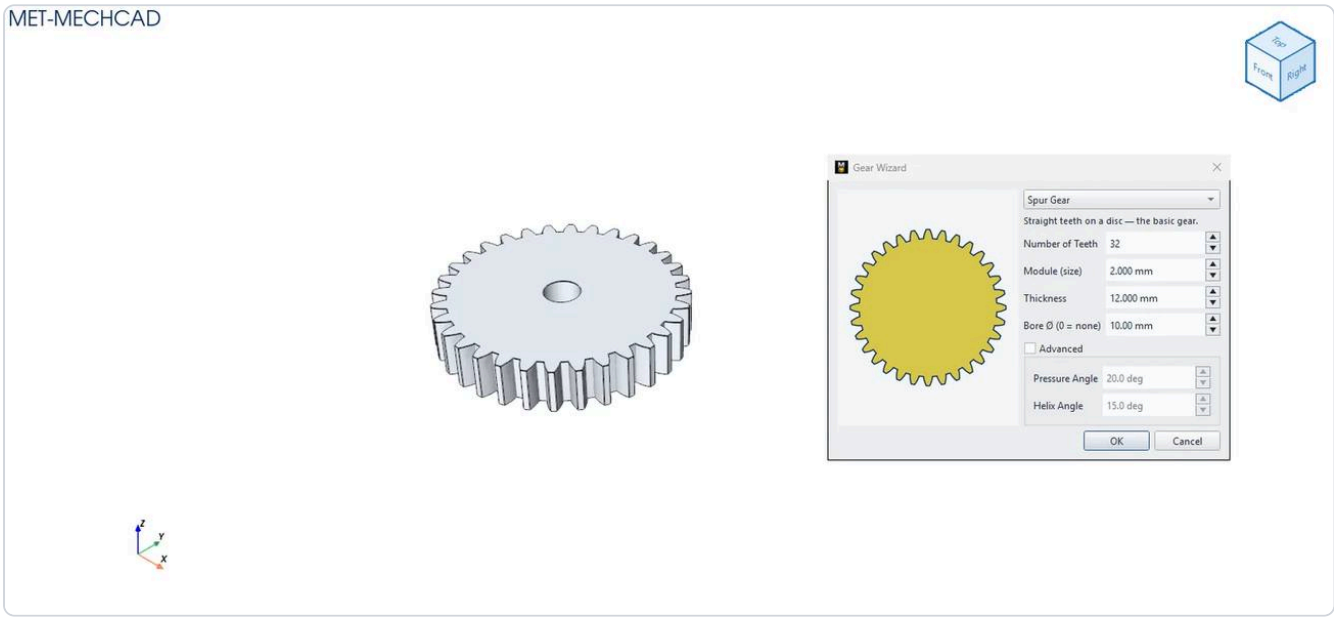
Create parametric Spur, Rack, Internal, Helical, Worm and Bevel gears.

What it does

Gear Wizard creates a gear part from standard values (number of teeth, module, thickness), with a 2D preview.

Where to find it

Direct Edit › Create › Gear



Gear Wizard window

Steps

1. Click **Gear**.
2. Choose the gear type and enter the values.
3. Click **OK**. The result **replaces the body of this part** (it is best to create it in a new part).
4. To edit later: right-click the part in the Design Explorer › **Edit Gear (Wizard)...**

Option	Default	Description
Type	Spur Gear	Spur / Rack / Internal / Helical / Worm / Bevel
Number of Teeth	20	3–400 (Rack: *Teeth along length*)
Module (size)	1.5 mm	Tooth size

Option	Default	Description
Thickness	6 mm	Thickness
Bore Ø (0 = none)	0	Shaft bore (Spur, Helical, Bevel, Worm)
Advanced › Pressure Angle	20°	5–35°
Advanced › Helix Angle	15°	0–45° (Helical)

> **Note:** In an assembly, use a **Gear Mate** so the gears turn together. The ratio is calculated from the pitch radius (Module × number of teeth ÷ 2).

See also

- Mate types
- Feed Screw Wizard

Feed Screw Wizard (bottle feed screw) Standard

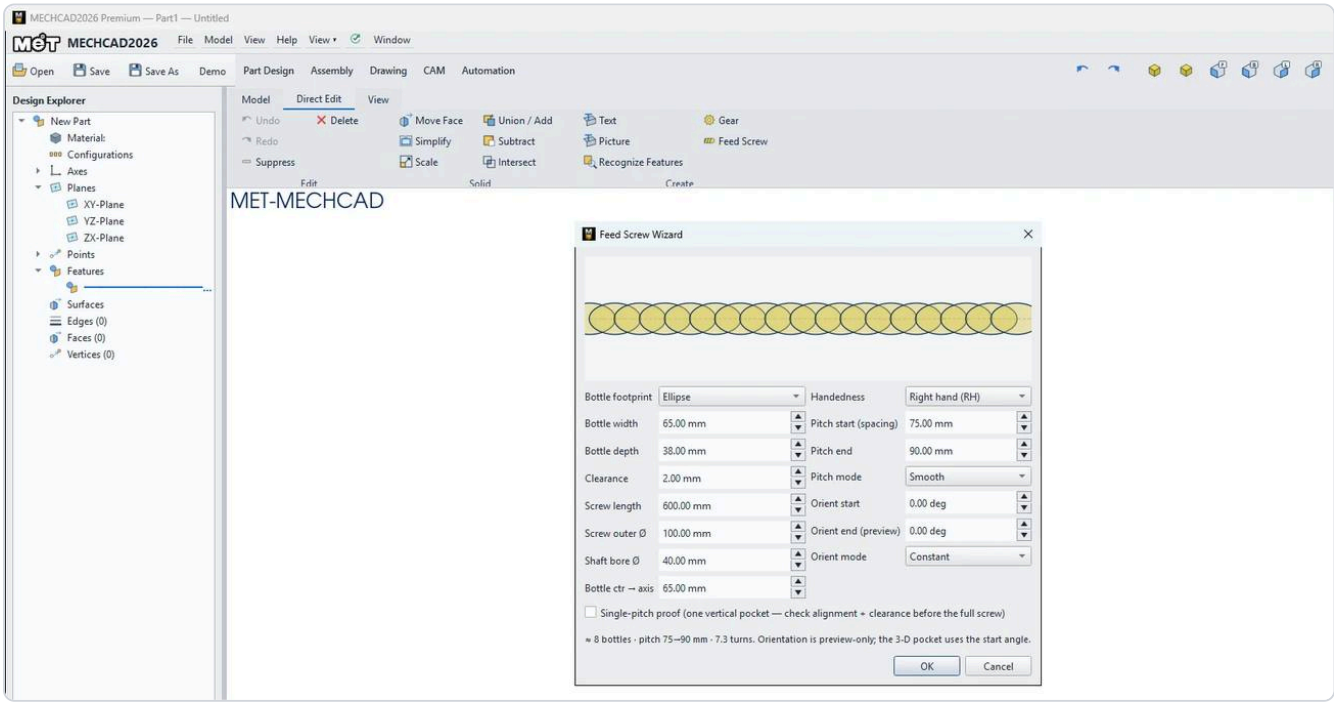
Create a bottle feed/timing screw from the bottle shape, pitch and thread hand.

What it does

Feed Screw Wizard creates a bottle feed screw (timing screw) for production lines, from the bottle base size, the start/end pitch and the thread hand.

Where to find it

Direct Edit › Create › Feed Screw



Feed Screw Wizard window

Option	Default	Description
Bottle footprint	–	Circle / Ellipse / Asymmetric Oval / Rounded Rectangle
Bottle width / Bottle depth	65 / 38	Bottle base size (depth is not used for Circle)
Clearance	2	Clearance allowance
Screw length	600	Screw length
Screw outer Ø	100	Outer diameter
Shaft bore Ø	40	Shaft bore
Bottle ctr → axis	65	Distance from the bottle centre to the screw axis
Handedness	–	Right hand (RH) / Left hand (LH)
Pitch start (spacing) / Pitch end	75 / 90	Start/end pitch
Pitch mode	Smooth	Constant / Linear / Smooth
Orient start / Orient end / Orient mode	–	Bottle rotation (used in the preview)
Single-pitch proof	Off	Creates a single pocket for testing

The information line at the bottom shows the number of bottles and the number of turns. To change the screw later, right-click › **Edit Feed Screw (Wizard)...**.

See also

- Gear Wizard

Part Color, Material and Opacity Standard

Set the colour, material (with density) and transparency of a part.

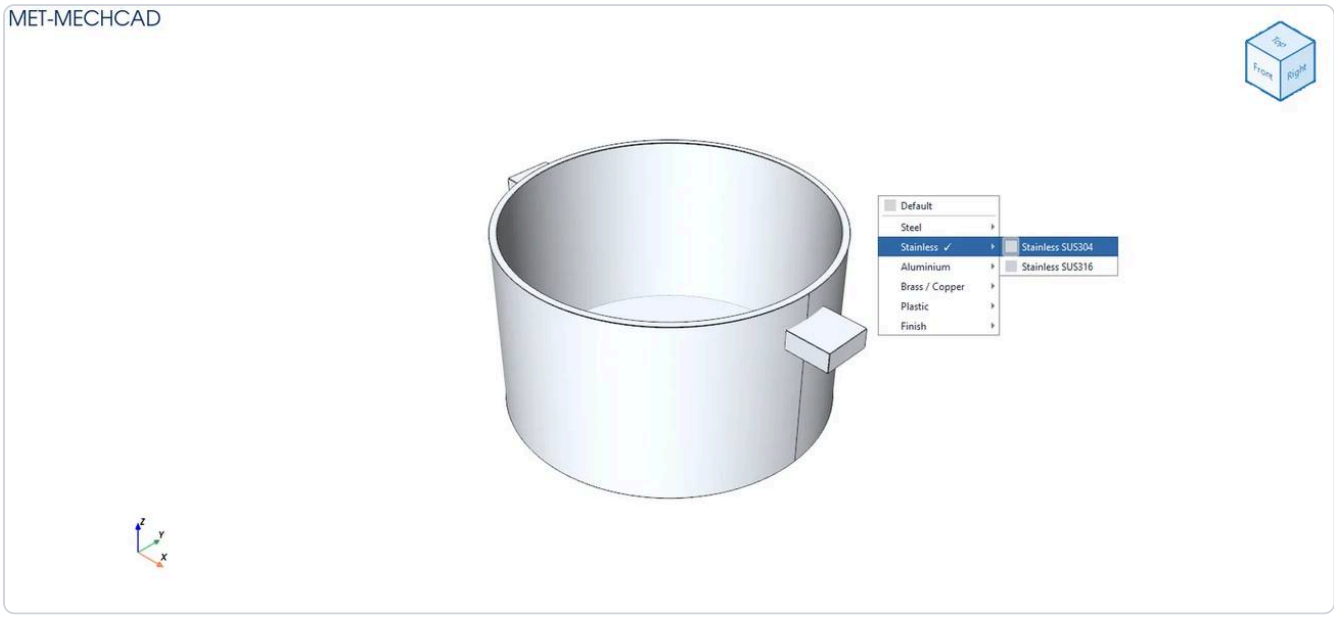
What it does

Sets how a part is displayed and which material is used for mass calculations.

Where to find it

- **View › Appearance › Part Color / Material / Opacity**
- Double-click the **Material:** node in the Design Explorer
- Right-click a face › **Edit Face Color** / **Edit Solid Color**; right-click a feature › **Set Feature Color...** / **Set Feature Transparency...**

Material

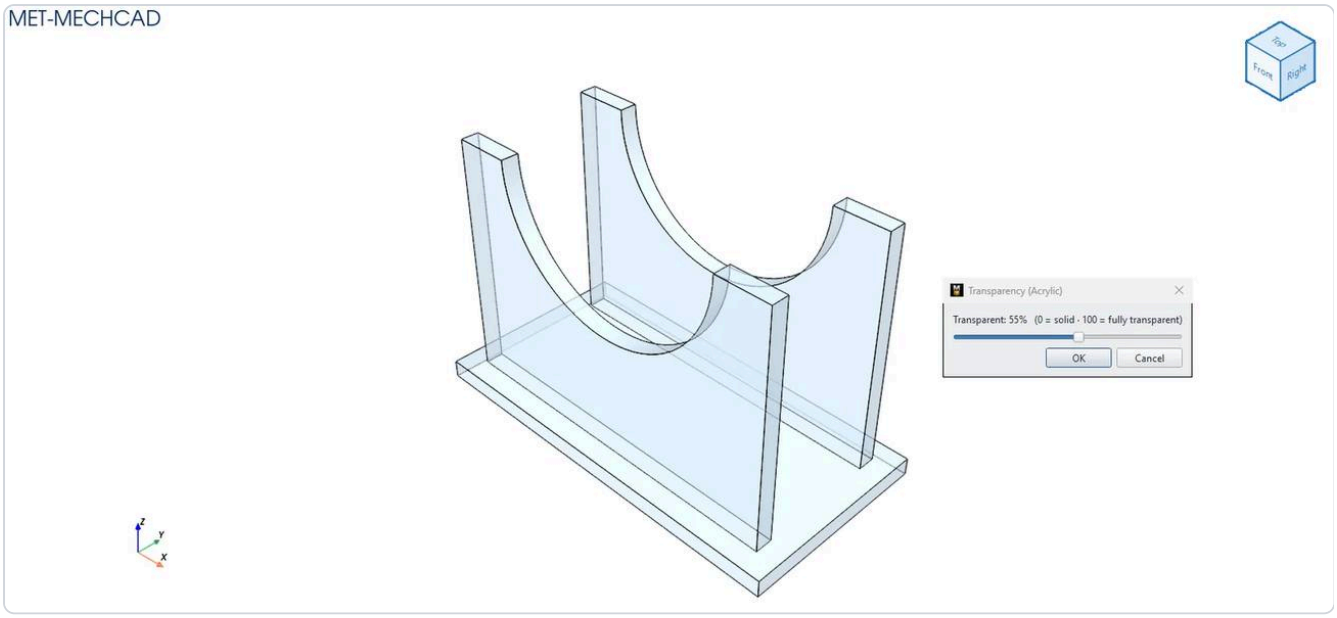


Material menu

Group	Materials
Steel	Steel SS400, Carbon Steel S45C
Stainless	SUS304, SUS316
Aluminium	AL6061, AL5083, Black Anodized
Brass / Copper	Brass C3604, Copper C1100
Plastic	POM White/Black, MC Nylon Blue, ABS Black, PVC Grey, Matte Plastic, Clear Acrylic (PMMA)
Finish	Painted

Choosing a material sets the colour, transparency and density (used for mass). It also writes the material grade into the Material field of Part Properties, unless you have already typed one. Choose **Default** to clear it.

Opacity



Transparency window (Acrylic)

Drag the slider *Transparent: N% (0 = solid · 100 = fully transparent)*. The result updates live. Click OK to confirm.

Part Color

Opens the *Edit Solid Color* colour picker. The chosen colour applies to the whole part.

See also

- Realistic Render
- Part Properties

Rollback, Regen and Suppress Standard

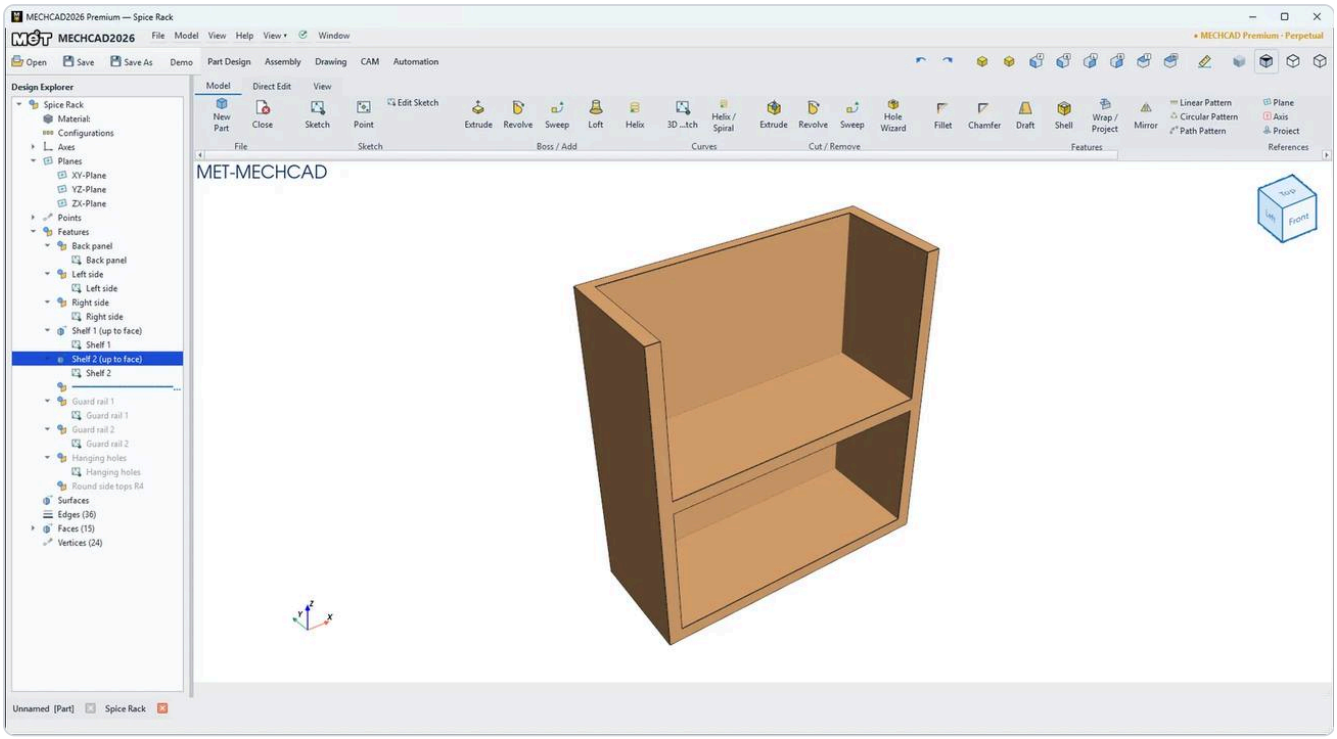
Step back through the history with the Rollback bar, recalculate the part, and turn features off temporarily.

What it does

- The **Rollback bar** (the blue line in Design Explorer) rolls the part back to any point in the feature history. Features below the bar are grey and are not calculated. Use it to insert a new feature before an existing one.
- **Regen** recalculates every feature (without the cache) and reports the time it took.
- **Suppress** turns a feature or sketch off temporarily.

Where to find it

- **Model › Regenerate › Regen, Roll Back, Roll Fwd, Roll to End**
- Drag the Rollback bar in Design Explorer (double-click the bar = Roll to End)
- Right-click a feature › **Roll Back to Here / Roll Forward to Here**
- **Direct Edit › Edit › Suppress / Delete**



Rollback by 4 features: features below the bar are grey

Command	Meaning
Roll Back	Moves the bar up by one feature
Roll Fwd	Moves the bar down by one feature
Roll to End	Moves the bar to the end, so every feature is active
Regen	Recalculates everything
Rebuild	Applies edited values and recalculates (**Rebuild** on the top menu)
Suppress	Turns a feature on/off (the name starts with "(Suppressed)")
Delete	Deletes a feature (the sketch stays), or deletes a sketch (and the features that use it)

> **Note:** While rolled back, the status bar shows *Rolled back: N feature(s) below the bar are not calculated*. Remember to **Roll to End** before you save or send the part to an assembly.

See also

- Design Explorer

Parameters (named values and equations) Standard

Name the values you use often, such as width = 50, and write equations that refer to them. When you change a value, the part rebuilds automatically.

What it does

Keeps the key values of a part as names, such as `width = 50` and `wall = 3`, and lets you write other values as equations, such as `len = 2*width + wall`. When you change one value, every value that refers to it is recalculated at once, and features bound to a parameter change size after the rebuild.

Where to find it

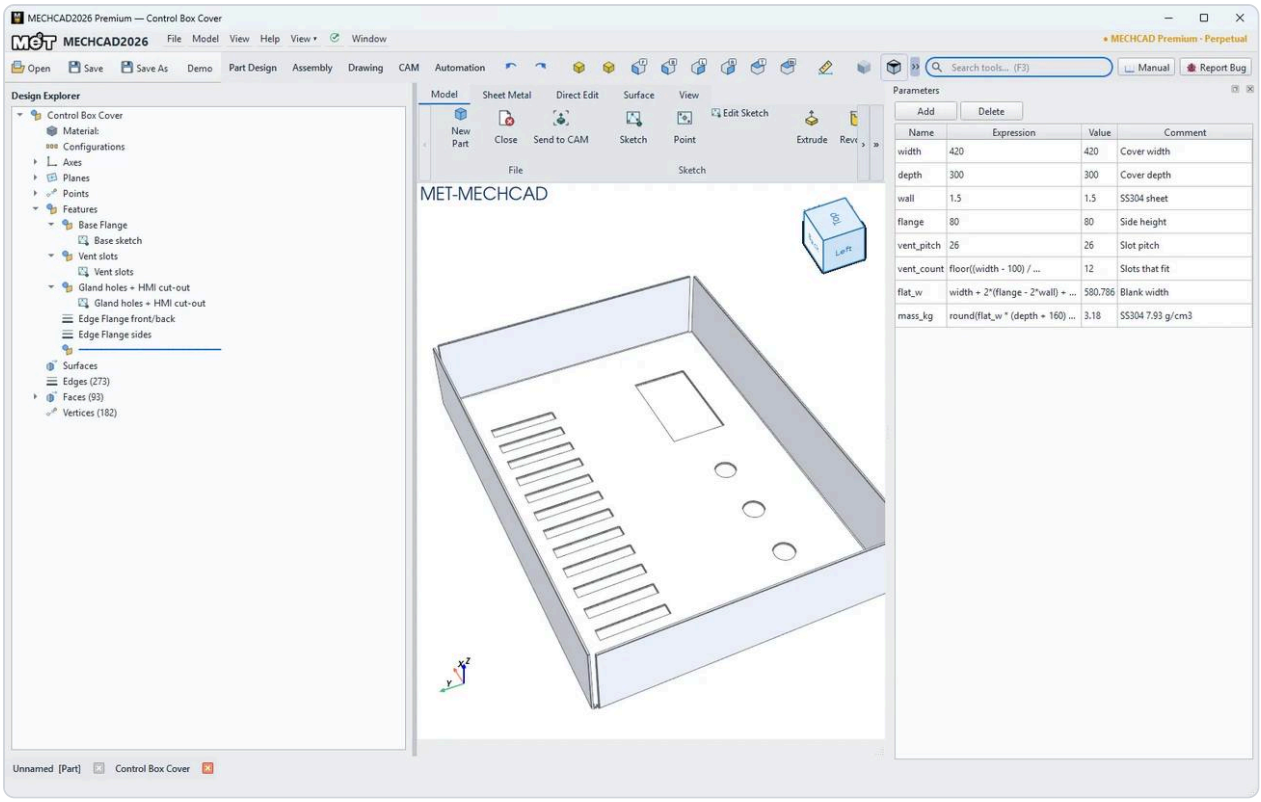
Model › References › Parameters in Part Design. A part must be open or created first; otherwise the message *Open or create a part first.* appears.

The **Parameters** panel opens docked on the right of the window, so you can keep working on the part while it is open. Click the button again to bring the panel back after you switch to another part.

Steps

1. Click **Parameters** in the References group.
2. Click **Add**. A new row named `param1` with the value `0` appears.
3. Double-click the **Name** cell to rename it, for example `width`.
4. Double-click the **Expression** cell and type a number (`50`) or an equation (`2*width + wall`), then press Enter.
5. Check the **Value** column. If the equation is valid, the part rebuilds at once.
6. Select a row and click **Delete** to remove that parameter.

Column	Editable	Meaning
Name	Yes	The parameter name. Use English letters, digits and `_`; it cannot start with a digit.
Expression	Yes	A number, or an equation that refers to other parameters
Value	No	The calculated result. Shows *error* when it cannot be calculated.
Comment	Yes	A note, such as *wall thickness* (editing it does not rebuild the part)



Parameters panel docked beside a control box cover: values, equations and comments

Writing equations

Allowed	Example
`+ - * /` and parentheses	`(width - 2*wall) / 2`
Power `**`, floor division `//`, remainder `%`	`side**2`, `len // 25`
Constants `pi`, `e`, `tau`	`pi * dia`
Functions `sqrt`, `abs`, `min`, `max`, `floor`, `ceil`, `round`	`max(wall, 2)`, `round(len / 3, 1)`
Trigonometry `sin`, `cos`, `tan`, `asin`, `acos`, `atan`, `atan2`	`h * tan(rad(30))`
Angle conversion `rad(degrees)` and `deg(radians)`	`deg(atan2(h, w))`

- The trigonometric functions work in **radians**. Wrap an angle in degrees with `rad()`, for example `sin(rad(30))` = 0.5.
- Names are not case-sensitive: `Width` and `width` are the same name. A name cannot be the same as a function or constant (`pi`, `sin`, `max` …).
- When you rename a parameter, every equation that uses the old name is updated to the new name automatically.
- If an equation is wrong, uses a name that does not exist, or refers back to itself in a loop (for example `a = b + 1` and `b = a`), its row turns red and a message appears below the table. The part does not rebuild until you fix it.

Parameters and features

- Parameters, and the features bound to them, are saved in the Part file.

- When you open an Alibre **.AD_PRT** file as editable features, its Alibre parameters come into this table (Comment *Alibre*, lengths converted to mm) and stay bound to the values they drive, such as Extrude depth and draft angle, Revolve angle, Fillet / Chamfer size, Shell thickness and Pattern count. Change a value in the table and the part follows.
- A field bound to a parameter takes its value from the table at every rebuild. Change that size in the Parameters panel, not in the feature box.

> **Note:** In this version there is no command yet to bind a feature field or a sketch dimension to a parameter yourself. Bindings come from opening Alibre files as described above. Parameters you create yourself can still hold values and calculate equations in full.

See also

- Open SolidWorks and Alibre files in MECHCAD
- Rollback, Regen and Suppress
- Part Properties

Part Properties Standard

Enter the part number, description, material, finish, revision, vendor and cost for the title block and BOM. Mass is computed from the material.

What it does

The information you enter appears in the drawing title block and in the BOM table.

Where to find it

File › Part Properties...



Part Properties window

Field	Meaning
Part Number	Part number (shown in the BOM)
Description	Description
Material	Choose a material from the list (group — material, for example *Steel — Carbon Steel S45C*, *Stainless — Stainless SUS304*). Each material carries its own density; the part color and mass follow your choice. **Default (no material)** = no material set.
Finish	Surface finish / plating
Revision	Revision number
Vendor / Maker	Manufacturer / brand
Mass	Mass (kg), **computed and read-only** = volume × material density. It updates as soon as you change Material in this window. With no material it shows — *(set a material with density)*.
Cost	Cost

> **Note:** Empty fields are not saved.

Mass properties

Every time the part rebuilds or its material changes, the part Properties panel (**Body result**) shows values computed from the actual solid:

Value	Meaning
Volume	Volume (mm^3)
Mass	Mass (kg) = volume × material density, with the material name in brackets. With no material it shows — *(pick a material)*.
CoG	Centre of gravity X, Y, Z (mm), assuming uniform density
Bounding box	Size of the box around the part, X × Y × Z (mm)
Faces / Triangles	Number of faces and display triangles

Example densities (g/cm^3): Steel SS400 / S45C 7.85, SUS304 7.93, SUS316 7.98, AL6061 2.70, AL5083 2.66.

> **Note:** The computed mass is written to the part's Mass field automatically; you do not need to type it.

See also

- Title Block and Sheet
- Auto BOM

Surface Modeling Expert

Create and edit surfaces from sketches, edges or faces, then knit, thicken or cut with them to get a solid.

What it does

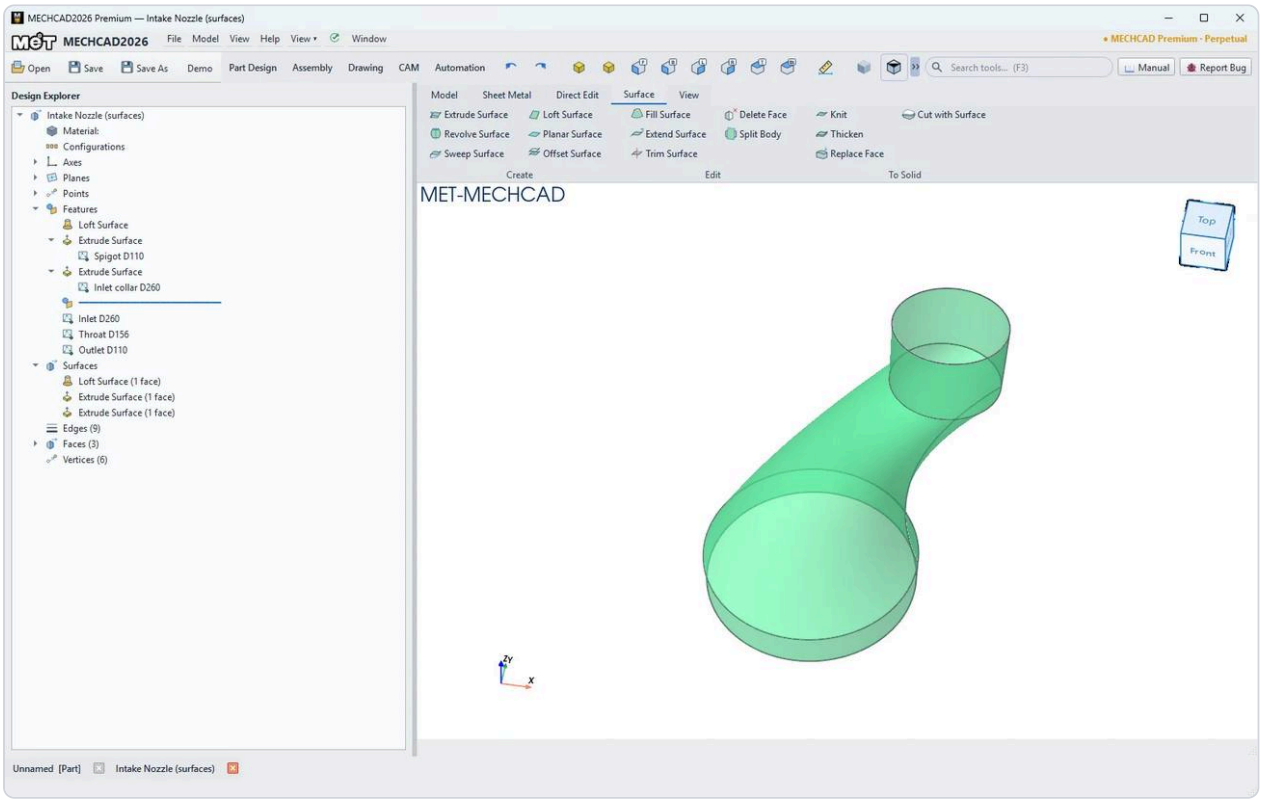
Use surfaces for parts with complex curved shapes that are hard to build with solid features, such as covers, handles or shapes whose faces must blend smoothly. A surface is a thin sheet with no thickness. You build the surfaces first, then turn them into a solid with the **To Solid** group. All surfaces are listed under the **Surfaces** node in the Design Explorer.

Every command opens the floating feature box with a live preview. **OK** creates one feature (one undo step). To change it later, right-click › **Edit Feature** reopens the same box with its values filled in.

Where to find it

The **Surface** tab of the ribbon in the Part window has three groups:

Group	Buttons
Create	Extrude Surface, Revolve Surface, Sweep Surface, Loft Surface, Planar Surface, Offset Surface
Edit	Fill Surface, Extend Surface, Trim Surface, Delete Face, Split Body
To Solid	Knit, Thicken, Replace Face, Cut with Surface



Surface tab with an intake nozzle: a Loft Surface through three sections and two Extrude Surface collars, listed under Surfaces in the Design Explorer

Steps

1. Prepare the sketch (or a body with the edges/faces to pick) that the command needs.
2. Click the command on the **Surface** tab. The feature box opens.
3. Make the picks: choose a sketch from the list in the box (or click the sketch in the Design Explorer), click faces or edges in the view (click again to remove), and click a surface in the view to fill a surface slot.
4. Adjust the values, check the preview, then click **OK**.

Create: new surfaces

Command	Needs	Options (default)
Extrude Surface	A profile sketch, open or closed	Depth (10 mm), Other side (0 = none), Mid plane (off), Reverse direction (off)
Revolve Surface	A profile sketch, open or closed	Axis (X axis / Y axis / Z axis, starts at X), Angle (360°)
Sweep Surface	A profile sketch and a different path sketch	Keep profile orientation (off)
Loft Surface	At least 2 section sketches, ticked in order	Ruled = straight between sections (off)
Planar Surface	A closed sketch, or click the edges of a closed planar loop	-
Offset Surface	Click faces of the solid or of a surface	Distance (5 mm, negative = inward)

Edit: change surfaces

Command	Needs	Options (default)
Fill Surface	Click boundary edges (one closed loop) and/or tick boundary sketches	Continuity to the faces: C0 (position) / G1 (tangent) / G2 (curvature) (starts at C0)
Extend Surface	Click one face of a surface, and optionally the edges to extend past (none = every side)	Distance (5 mm), Extension: Natural (follow the surface) / Linear (straight on) (starts at Natural)
Trim Surface	The surface to trim and a trimming tool	Trim with: Another surface / Datum plane / Sketch (projected) (starts at Another surface), Flip side (off), Keep the trimming surface (on)
Delete Face	Click faces of the solid or of a surface	Mode: Delete (body becomes a surface) / Delete & Patch (heal the solid) / Untrim (full natural face) (starts at Delete)
Split Body	A splitting surface, or choose a plane	Keep the splitting surface (on)

To Solid: turn surfaces into a part

Command	Needs	Options (default)
Knit	Tick (or click) the surfaces to sew along shared edges	Gap tolerance (0.01 mm), Try to form a solid (on), Merge a formed solid into the body (on)
Thicken	Click the surface to thicken	Thickness (2 mm), Both sides = half each way (off), Reverse side (off), Merge into the body (on), Keep the surface (off)

Command	Needs	Options (default)
Replace Face	Click the solid face(s) to replace, then the replacing surface	Keep the surface (off)
Cut with Surface	Click the cutting surface	Flip side = keep the other part (off), Keep the surface (on)

> **Note:** Surface modeling is available in MECHCAD **Expert** and **Premium**. On Standard the buttons are visible but disabled, with an upgrade hint. When Knit closes the surfaces completely, the result becomes a solid. For Replace Face, the replacing surface must cover the whole face.

See also

- Sweep
- Loft
- Feature Parameters
- MECHCAD editions

Sheet Metal Expert

Build sheet metal parts with Base Flange, Edge Flange, Hem, Jog and Sketched Bend, then unfold to a Flat Pattern and export a DXF for the laser.

What it does

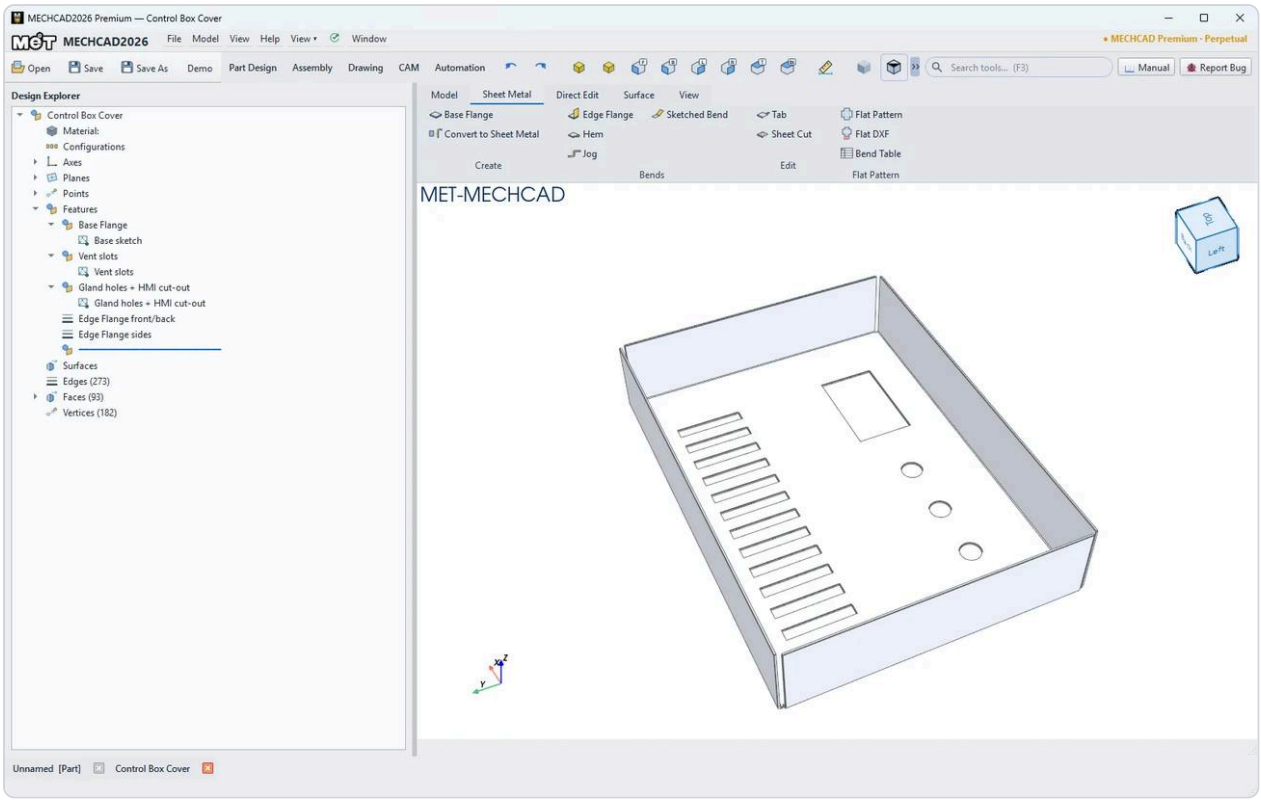
Design bent sheet metal parts (cabinets, covers, brackets). Set the material, thickness, bend radius and K-factor once in the Base Flange, then add flanges, hems and bends. When done, view the **Flat Pattern**, check the **Bend Table** and export a **Flat DXF** for laser cutting.

Every command opens the floating feature box with a live preview. To change a feature later, right-click › **Edit Feature**.

Where to find it

The **Sheet Metal** tab of the ribbon in the Part window:

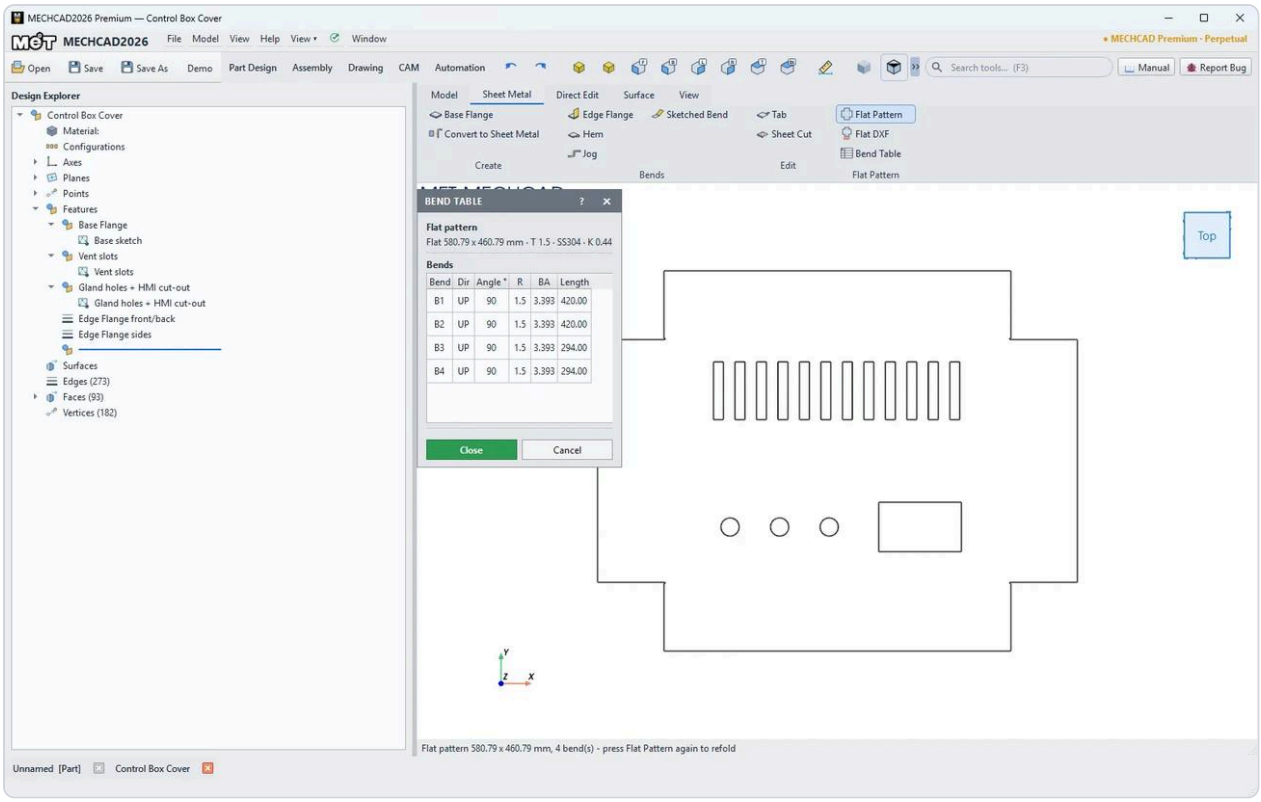
Group	Buttons
Create	Base Flange, Convert to Sheet Metal
Bends	Edge Flange, Hem, Jog, Sketched Bend
Edit	Tab, Sheet Cut
Flat Pattern	Flat Pattern, Flat DXF, Bend Table



Sheet Metal tab: an SS304 control box cover with vent slots, gland holes and an HMI cut-out, built from Base Flange and Edge Flange

Steps

1. Draw a sketch: a closed sketch = a flat plate, an open sketch (lines/arcs) = a bent profile.
2. Click **Base Flange**, pick the sketch, then choose the **Material** and **Gauge table thickness**. Thickness, bend radius and K-factor are filled in from the table (you can change them). Click **OK**.
3. Add flanges: click **Edge Flange** or **Hem**, then click straight edges of the sheet in the view (click again to remove). All picked edges become one feature.
4. To bend along a line, draw a sketch with one straight line on a panel face, then use **Jog** or **Sketched Bend**.
5. Add material with **Tab** or cut with **Sheet Cut** (a closed sketch on a panel face).
6. Click **Flat Pattern** to unfold (click again to refold), **Bend Table** to see the bends, and **Flat DXF** to save the DXF file.



Flat Pattern of the cover (580.79 × 460.79 mm, 4 bends) with the Bend Table

Base Flange

Option	Default	Description
Sketch	-	Closed = flat plate, open = bent profile
Material	Mild steel SPCC / SS400 cold rolled	SS304, SPCC/SS400, Aluminium 5052-H32 or Custom
Gauge table thickness	1.5 mm	0.8, 1.0, 1.2, 1.5, 2.0, 2.5, 3.0 mm (fills in T, R and K)
Thickness T	1.5 mm	Sheet thickness
Inside bend radius R	1.5 mm	Inside bend radius
K-factor	0.44	Neutral axis position used for the unfold
Bend allowance	K-factor	K-factor / Bend allowance table / Bend deduction table
Table rows	90° / 0 mm	Angle/value rows (used by the table modes)
Bend relief	Rectangular	Rectangular / Round (obround) / Tear (rip, no cut) / None
Relief width / Relief depth past bend	0 (= T)	Size of the relief at the bend
Corner relief	Rectangular	Rectangular / Round / Tear (none cut) / None
Corner relief size	0 (= T)	Size of the corner relief
Width (bent profile depth)	50 mm	Sheet width for an open sketch

Option	Default	Description
Flip material side	Off	Puts the material on the other side
Bent profile: mid-plane	Off	Centres the width on the sketch (open sketch)

Edge Flange

Option	Default	Description
Edges	-	Click straight edges of the sheet (top or bottom face)
Length	20 mm	Flange length
Length measured to	Outer virtual sharp	Outer virtual sharp / Inner virtual sharp / Tangent (straight leg)
Bend angle	90°	Bend angle (0.1–179°)
Flange position	Bend outside the edge	Bend outside the edge / Material inside (flush outer face) / Material outside (flush inner face)
Bend down (flip direction)	Off	Bends downward
Inside radius	0 (= sheet R)	Bend radius for this flange
Offset from edge start / end	0 mm	Sets the flange back from the edge ends
Bend relief	Sheet default	Uses the sheet setting, or pick one

Hem

Option	Default	Description
Hem type	Closed (flattened)	Closed (flattened) / Open (with gap) / Teardrop
Length	8 mm	Hem length (teardrop: 0 = to the panel)
Gap (open hem)	0 mm	Gap of an open hem
Teardrop radius	0 (= T)	Teardrop radius
Teardrop angle	230°	Teardrop angle (181–329°)
Position	Outside the edge	Outside the edge / Inside (folded back in)
Fold down (flip direction)	Off	Folds downward

Jog and Sketched Bend

Both commands use a sketch with one straight line drawn on a panel face.

Option	Default	Description
Offset (Jog)	10 mm	Step height of the jog
Offset measured (Jog)	Face to face (same side)	Face to face (same side) / Inside (gap between) / Overall (outside faces)
Jog angle / Bend angle	90°	Bend angle

Option	Default	Description
Inside radius	0 (= sheet R)	Bend radius
Fix projected length (Jog)	On	Keeps the projected length
Bend down / Jog down	Off	Bends downward
Bend the other side / Move the other side	Off	Swaps the side that moves

Other commands

Command	What it does
Convert to Sheet Metal	Turns a constant-thickness solid with round bends (for example an imported STEP part) into sheet metal. Thickness and radii are read from the solid; the largest flat wall stays fixed. Only Material and K-factor (0.44) are set.
Tab	Adds material from a closed sketch drawn on a panel face (it must touch the panel)
Sheet Cut	Cuts through the panel along a closed sketch. Tick *Cut the flat pattern* to cut across bends on the unfolded part, then refold
Flat Pattern	Toggles the unfolded/folded part. The status bar shows the flat size and the number of bends
Flat DXF	Saves a DXF in mm with layers OUTER, INNER, BEND_UP, BEND_DOWN (dashed) and BEND_NOTES
Bend Table	Shows the flat size, T, material, K and the bend list: Bend, Dir, Angle, R, BA, Length

> **Note:** Sheet Metal is available in MECHCAD **Expert** and **Premium**. On Standard the tab is visible, but clicking a command shows an upgrade notice. Create a Base Flange (or Convert to Sheet Metal) first; the other commands need it. If the part is already sheet metal, clicking Base Flange again opens Tab instead.

See also

- Feature Parameters
- Export STEP/STL
- MECHCAD editions

Chapter 4 quiz: Part

Choose the one best answer. Answers are in Appendix A

1. **Which Extrude Type pushes the sketch out symmetrically on both sides of the sketch plane?**

   A. To Depth
   B. Two Directions
   C. Mid Plane
   D. Up To Face / Plane

2. **Which Type is available only in Extrude Cut, not in Extrude Boss?**

   A. To Depth
   B. Mid Plane
   C. Up To Face
   D. Through All

3. **In the Fillet command, each edge you click is filleted. How do you finish?**

   A. Click Done, or press Esc, Enter or Space
   B. Double-click the last edge
   C. Click Regen
   D. Close the Design Explorer

4. **How do you drill holes at several positions in one go with Hole Wizard?**

   A. Ctrl+click every face before starting
   B. Draw the positions with the Point tool in a sketch, then choose the sketch's “all N points (Drill)” position
   C. Make one hole, then copy it with Ctrl+C / Ctrl+V
   D. Use Extrude Cut with Through All

5. **You rolled the part back to insert a feature. What must you do before saving or sending it to an assembly?**

   A. Regen
   B. Suppress the features below the bar
   C. Roll Back once more
   D. Roll to End

6. **How do you change the values of an existing feature later?**

   A. Delete it and create it again
   B. Use Undo until it is the last step
   C. Double-click it in the Design Explorer, or right-click › Edit Feature
   D. Open System Options › General

CHAPTER 5

Assembly

1. Assembly overview
   Join multiple parts with mates in the Assembly workbench, and the parts of the ribbon.
2. Insert Parts and Sub-Assemblies
   Bring parts, assemblies or CAD files into an assembly, and learn the sub-assembly rules.
3. Assembly Explorer and Properties panel
   The assembly tree, part states, right-click menu, and the part placement/colour panel.
4. Add Mates with the Assembly Mates Panel
   Select surfaces on two components, then choose a suggested mate type, with a preview before you confirm.
5. Mate Types
   What each of the 23 mate types does, and which surfaces you can use with it.
6. Edit, flip and delete mates
   Edit mate values from the Assembly Explorer or Properties panel, and deal with conflicting mates.
7. Auto Mate
   Automatically create mates at every contact point of the selected or visible parts.
8. Drag and move parts (Drag Part)
   Drag parts so they follow their mates, rotate or copy them, and set drag sensitivity.
9. Linear Pattern of parts
   Repeat a part or sub-assembly in rows, columns and layers.
10. Component state: Show/Hide, Transparent, Fix, Suppress
    Hide, show, make transparent, fix and suppress parts in an assembly.
11. Derived Part: Union, Subtract, Intersect, Scale and Mirror Part
    Create a new part from parts in an assembly without changing the originals.
12. Explode and Collapse
    Animate the parts apart into an exploded view, set the distance 0–100% with a slider, show center lines, and put them back together.
13. 3D Notes and Pictures in an Assembly
    Add 3D text labels that point to components, and place pictures on faces.
14. Pack and Go (.metpkg)

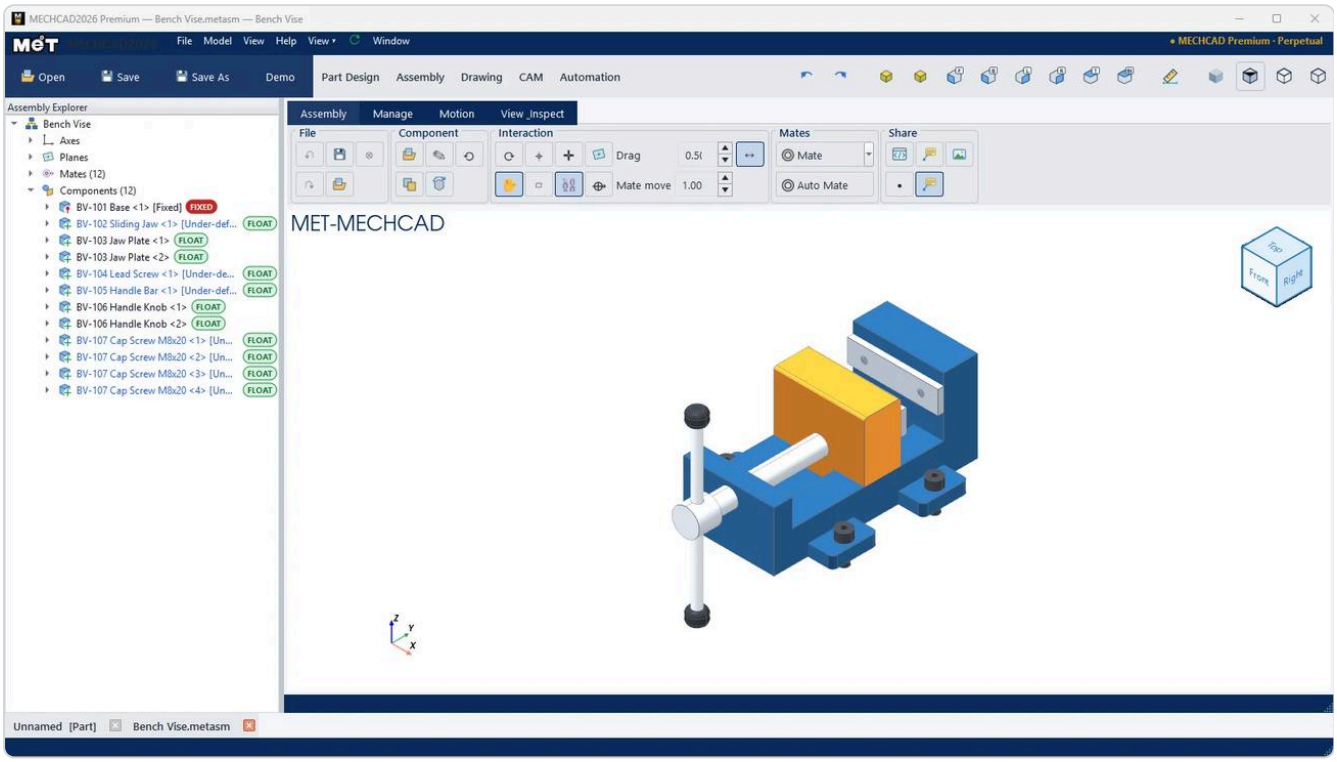
Bundle an assembly and all its components into one package to share or copy.

Assembly overview Standard

Join multiple parts with mates in the Assembly workbench, and the parts of the ribbon.

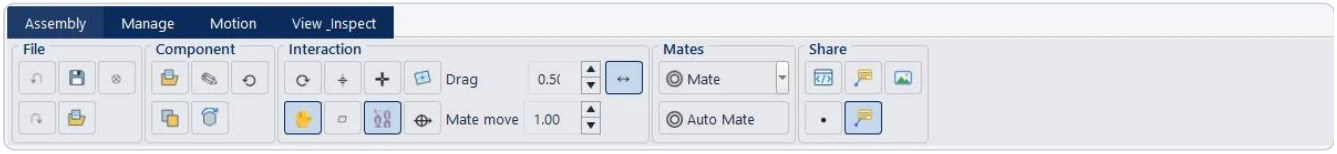
What it does

Use the **Assembly** tab to join parts and sub-assemblies with **mates**, drag parts so they move with their joints, explode the view, and pass the result on to Drawing / BOM. Assemblies are saved as `.metasm`.

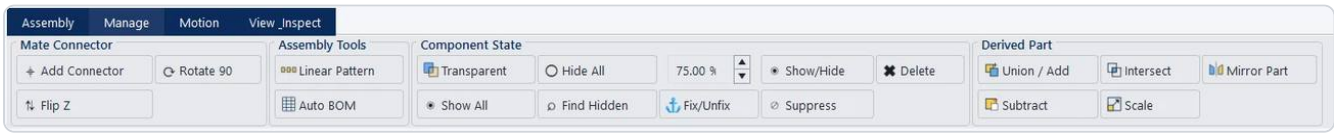


Bench vise assembly

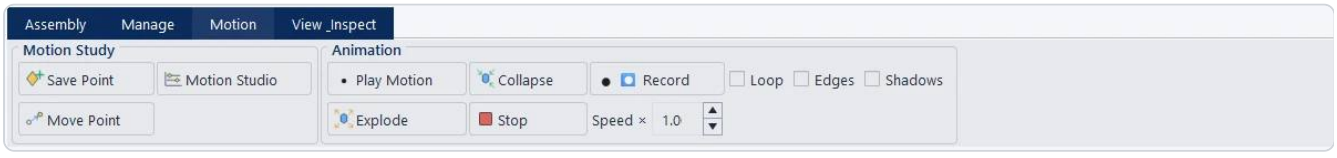
Assembly ribbon



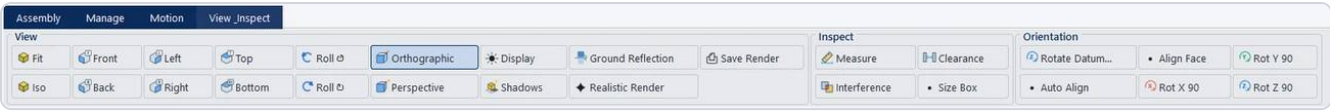
Assembly tab: File, Component, Interaction, Mates, Share



Manage tab: Mate Connector, Assembly Tools, Component State, Derived Part



Motion tab: Motion Study, Animation



View & Inspect tab: View, Inspect, Orientation

Tab › Group	Commands
Assembly › File	Undo, Redo, Save, Load, Close
Assembly › Component	Insert Part, Duplicate, Open Part, Reload Part, Update Sub-Asm
Assembly › Interaction	Regen, Drag Part, Axis Connector, Plane, Axis, Show Mates, Show Datums, Mate Axis, Drag (sensitivity), Mate move (speed), Min Motion
Assembly › Mates	**Mate** (split button with a list of mate types), **Auto Mate**
Assembly › Share	Export HTML, Export STL, 3D Notes, Show Notes, Picture
Manage › Mate Connector	Add Connector, Flip Z, Rotate 90
Manage › Assembly Tools	Linear Pattern, Auto BOM
Manage › Component State	Transparent, Show All, Hide All, Find Hidden, % transparency, Fix/Unfix, Show/Hide, Suppress, Delete
Manage › Derived Part	Union / Add, Subtract, Intersect, Scale, Mirror Part
Motion › Motion Study	Save Point, Move Point, Motion Studio
Motion › Animation	Play Motion, Explode, Collapse, ■ Stop, ● Record, Speed ×, Loop, Edges, Shadows
View & Inspect › View	Fit, Iso, standard views, Roll ↺/↻, Orthographic/Perspective, Display, Shadows, Ground Reflection, Realistic Render, Save Render
View & Inspect › Inspect	Measure, Interference, Clearance, Size Box
View & Inspect › Orientation	Rotate Datum..., Auto Align, Align Face, Rot X 90, Rot Y 90, Rot Z 90

Typical workflow

1. **Insert Part** the first part. It is fixed in place as the base.
2. Insert the next part, then add a Mate or use Auto Mate.
3. Drag parts to check how they move. See Drag and move parts.
4. Save as `.metasm`, then create a Drawing or Auto BOM.

See also

- Insert parts and sub-assemblies
- Assembly Explorer

Insert Parts and Sub-Assemblies Standard

Bring parts, assemblies or CAD files into an assembly, and learn the sub-assembly rules.

What it does

Brings components into an assembly. It supports MECHCAD files (`.metprt` , `.metasm`) and importable CAD files (STEP, IGES, SolidWorks, Alibre and others).

Where to find it

- **Assembly › Component › Insert Part**
- Right-click in the view or in Assembly Explorer › **Insert Part...**
- Drag a file from Windows Explorer and drop it in the view



Component group on the Assembly tab

Steps

1. Click **Insert Part** and select a file in the *Insert Part / Assembly* window.
2. The component appears in the view and in Assembly Explorer.
3. A STEP file that contains several solids asks how to import them:

Option	Meaning
Single Part	Fuses all solids into one rigid piece. Easiest to mate and lightest.
Sub-Assembly	Keeps each solid as a rigidly welded sub-assembly that you can open and edit later.

Related commands

Command	Meaning
Duplicate Part	Copies a component (if it is inside a sub-assembly, the whole sub-assembly is copied). You can also Ctrl+drag in the view to copy.
Duplicate Sub-Assembly	Right-click a sub-assembly row in Assembly Explorer. Copies only that sub-assembly and places the copy beside the original.
Open Part	Opens the component's source file in a new tab. After you edit it, the assembly updates.
Reload Part	Reloads the component file from disk.
Update Sub-Asm	Re-reads the linked sub-assembly and reports *+N / -M part(s)* and any mates that were dropped.

Sub-assembly rules

- An inserted sub-assembly is always **one rigid body**. Its inner parts are welded by hidden Fastened mates *[AutoFix]*.

- Drag, copy, delete, pattern, fix, hide, transparency and color all apply to the whole body.
- If you want the inner parts to move as joints, right-click › **Rigid Body › Ungroup Rigid Body**, then use Auto Mate.
- To combine several parts into one body, select two or more parts › **Rigid Body › Group Selection as Rigid Body**.

Copying a sub-assembly in a whole imported machine

When you open a whole machine (for example a STEP or other CAD file with hundreds of parts), the whole file is the main assembly. Copying never copies the whole machine:

- **Duplicate Part** or Ctrl+drag on a part copies the top-level sub-assembly that part belongs to (for example one leveling leg). If the part sits directly under the main assembly, only that part is copied.
- **Duplicate Sub-Assembly** on a sub-assembly row copies only that sub-assembly and names the copy after it.
- If you click **Duplicate Sub-Assembly** on the main assembly row, the program asks first: *This is the main assembly itself (N components). Duplicate the WHOLE assembly into itself?* The default answer is **No**.

An assembly cannot contain itself

An assembly cannot be inserted into itself. The program checks this before any file conversion starts and shows an *Insert* warning:

Case	Message
You pick the file of the open assembly	*[file name] is this assembly itself - an assembly can never contain itself.*
You pick the source file of the main assembly (the file imported as the whole machine)	*[file name] is the root of this assembly (it already holds all N components) - inserting it again would put the main assembly inside itself. To add a sub-assembly, insert its own file, or right-click the sub-assembly in the Explorer > Duplicate Sub-Assembly.*

To add another copy of a sub-assembly, insert that sub-assembly's own file, or use **Duplicate Sub-Assembly**.

See also

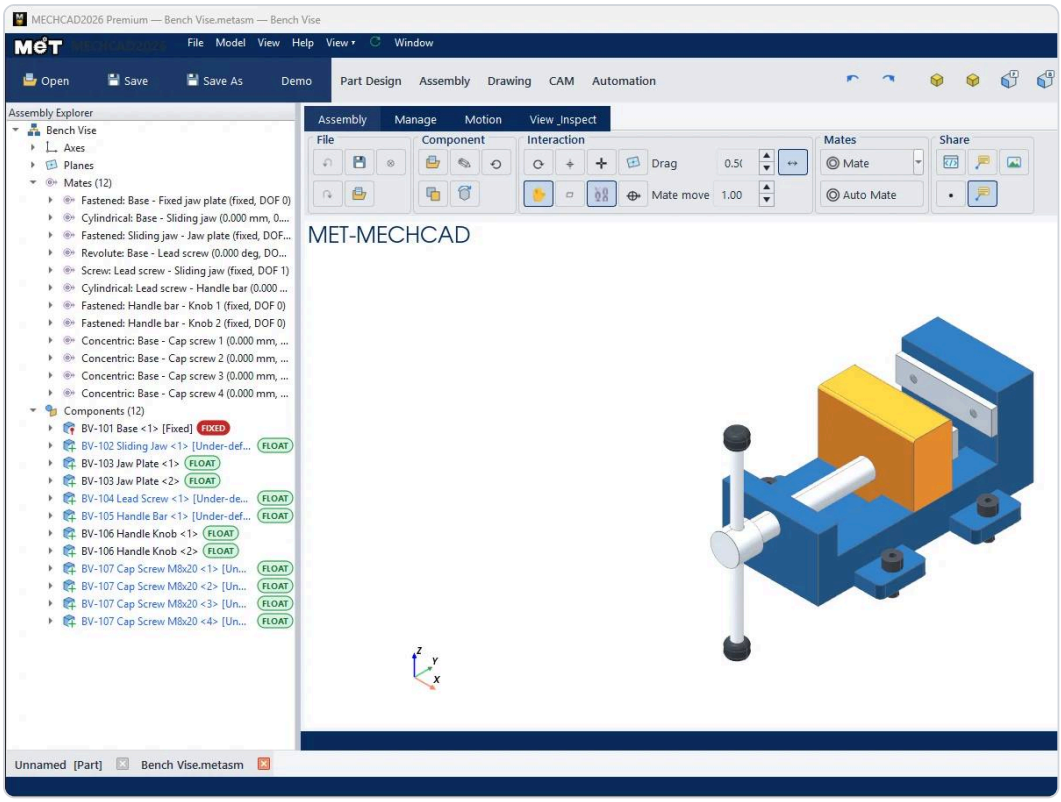
- Assembly Explorer
- Import CAD

Assembly Explorer and Properties panel Standard

The assembly tree, part states, right-click menu, and the part placement/colour panel.

What it does

The **Assembly Explorer** (left) shows axes, planes, the **Mates (n)** group and the **Components (n)** group. The **Properties / Info** panel (right) shows the placement and state of the selected part.

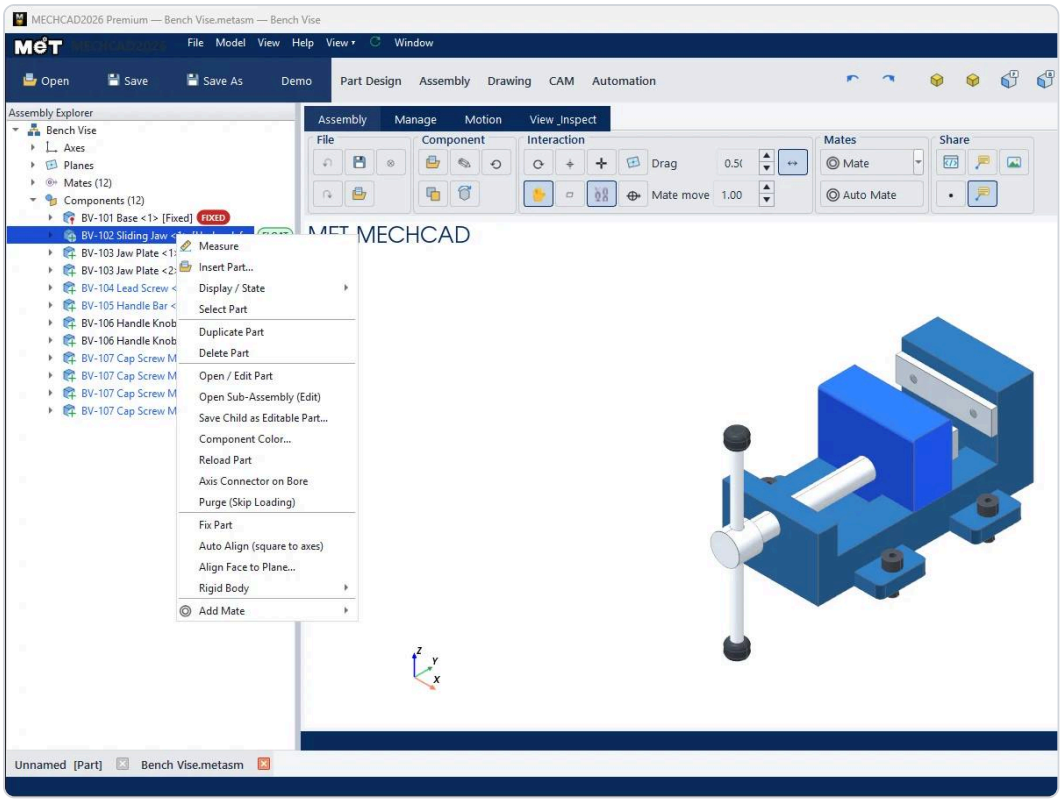


Assembly Explorer

Status labels

Label / colour	Meaning
FIXED (red)	The part is fixed.
FLOAT (green)	The part can move.
PART-FIXED	A sub-assembly that is a rigid body.
Black mate	Fully defined.
Blue mate *[Under-defined · N DOF]*	N degrees of freedom remain.
Red mate *[OVER-DEFINED]*	The mates conflict. Hover to see *Conflicting mates*.

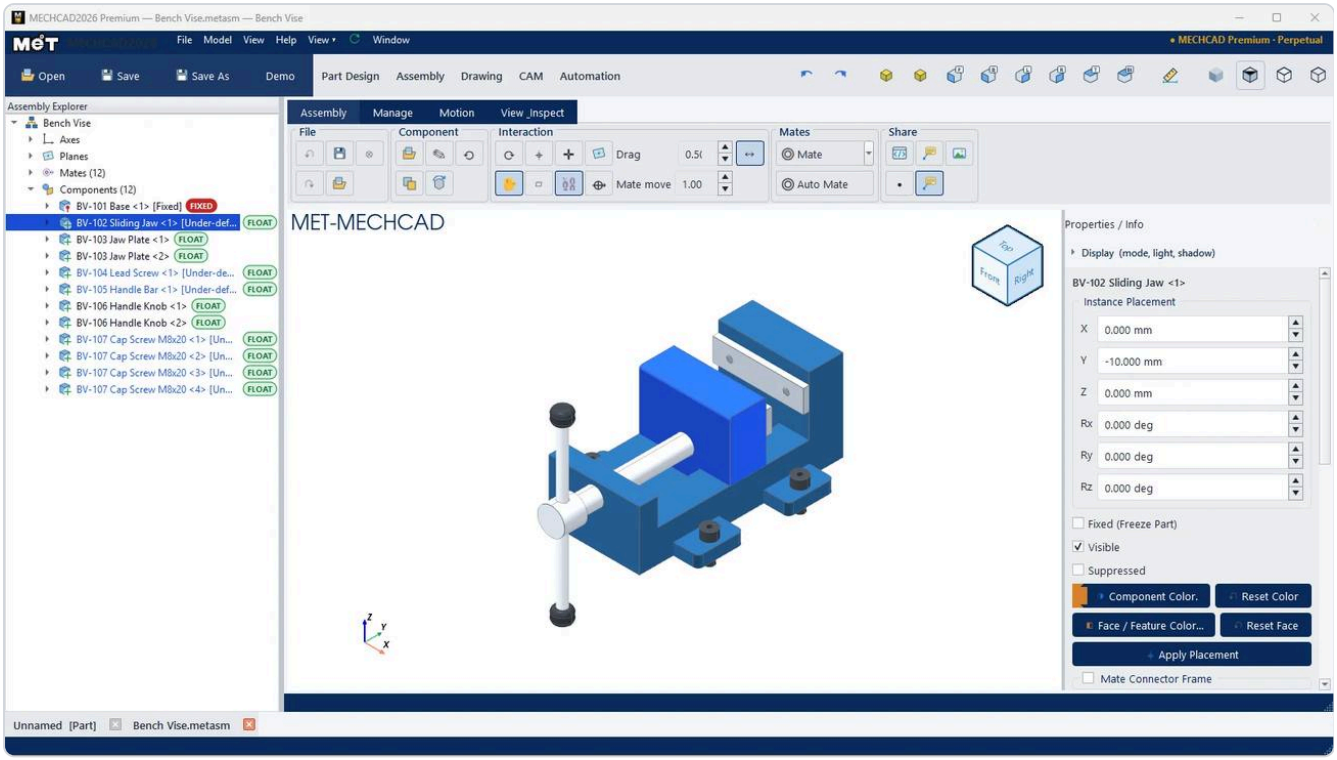
Right-click menu on a part



Right-click a part in the Assembly Explorer

Menu	Description
Measure / Insert Part…	Measure / add a part.
Display / State › Hide, Show, Transparent, Opaque, Fix, Float	Display and fixing state.
Select Part / Duplicate Part / Delete Part	Select, copy, delete.
Duplicate Sub-Assembly	(Right-click a sub-assembly row) Copies only that sub-assembly. See Insert Parts and Sub-Assemblies.
Open / Edit Part, Open Sub-Assembly (Edit)	Open for editing in a new tab.
Save Child as Editable Part...	Save an embedded part as an editable Part file.
Component Color... / Reset Component Color	Part colour.
Reload Part	Reload from file.
Axis Connector on Bore	Create an axis connector on a hole.
Purge (Skip Loading) / Reload (Unpurge)	Temporarily skip loading this part (helps large assemblies open faster).
Fix Part / Unfix Part	Fix / release.
Auto Align (square to axes) / Align Face to Plane…	Straighten a tilted imported part so it lines up with the axes.
Rigid Body ›	Group / Ungroup
Add Mate ›	All mate types

Properties / Info panel



Properties panel with a part selected

Section	Description
Instance Placement: X, Y, Z (mm) / Rx, Ry, Rz (deg)	Position and rotation. Click **Apply Placement**.
Fixed (Freeze Part) / Visible / Suppressed	State
Component Color... / Reset Color / Face / Feature Color... / Reset Face	Colours
Mate Connector Frame	Position of the mate connector
Mate	Values of the selected mate. See Edit and delete mates.

See also

- Component state
- Edit and delete mates

Add Mates with the Assembly Mates Panel Standard

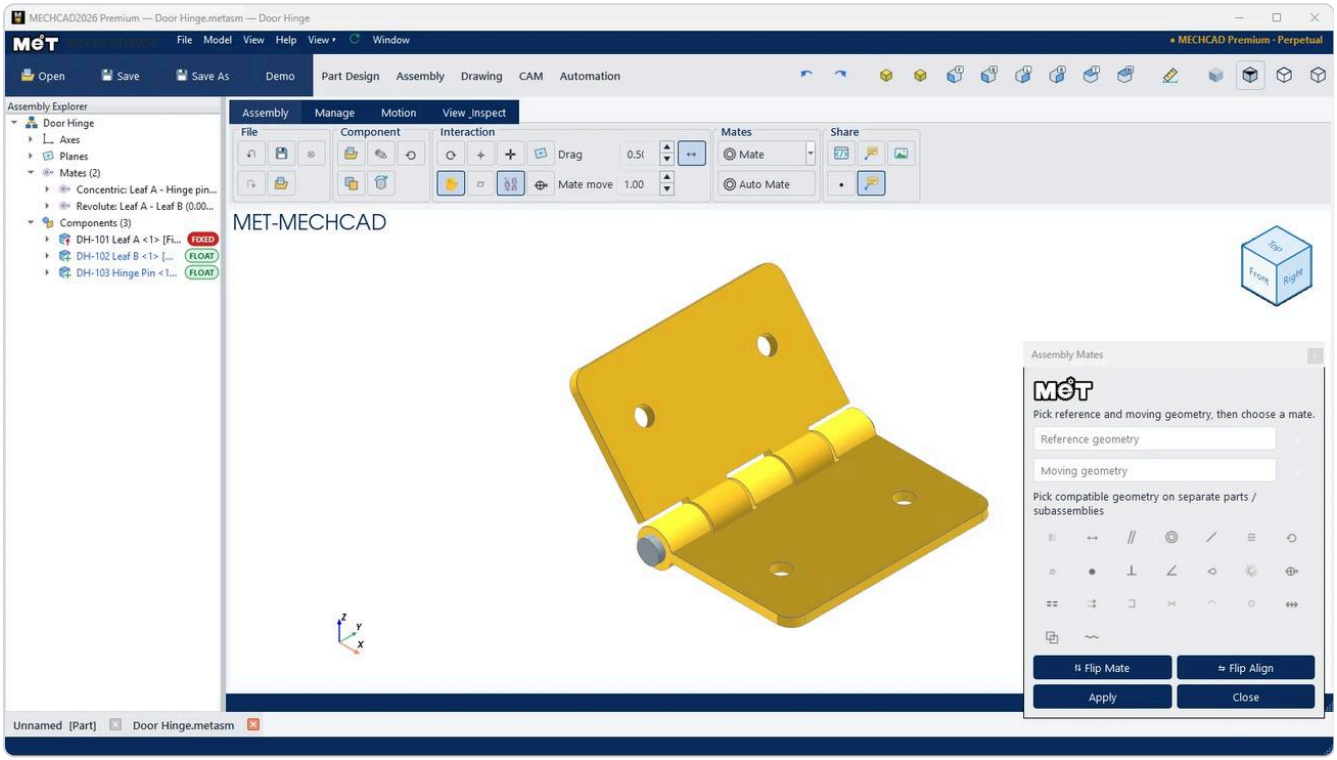
Select surfaces on two components, then choose a suggested mate type, with a preview before you confirm.

What it does

A **Mate** defines the relationship between components, such as concentric, face-to-face or distance. The **Assembly Mates** panel automatically suggests the mate types that fit the surfaces you select.

Where to find it

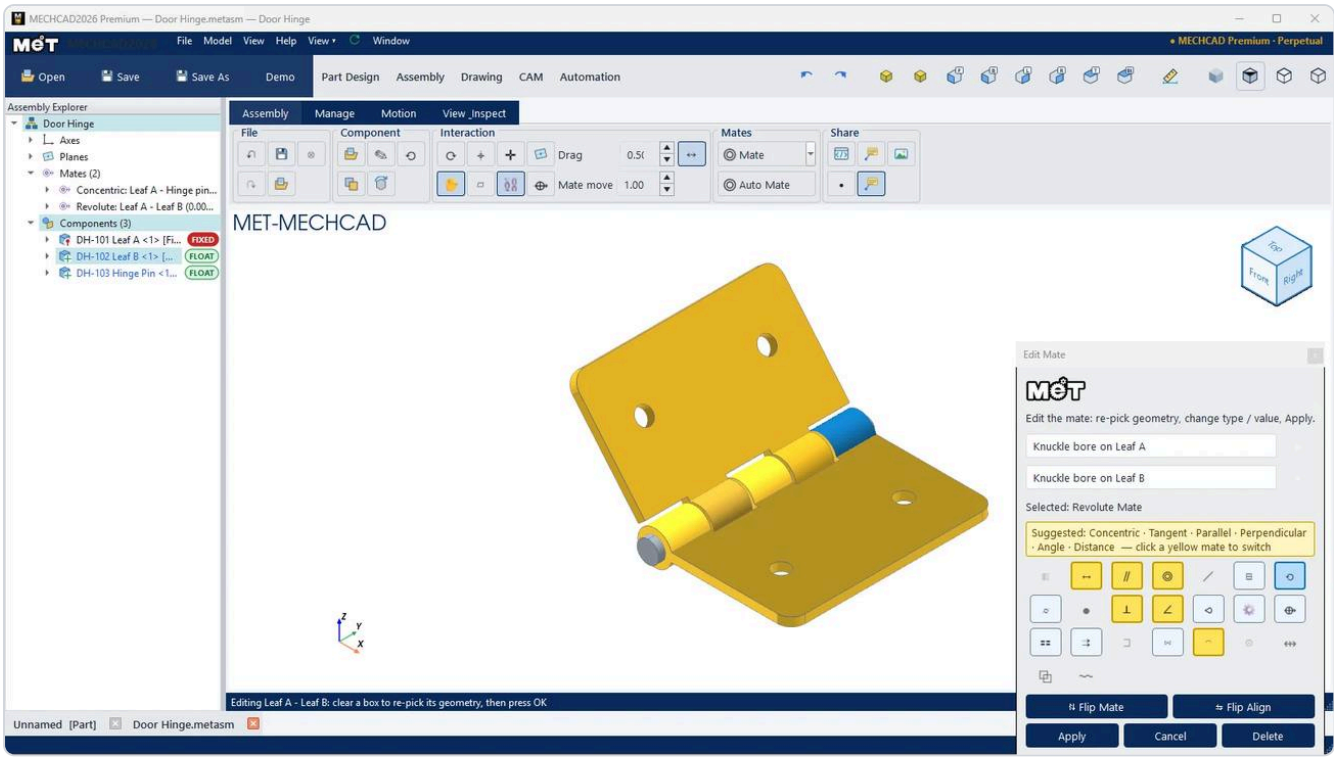
- **Assembly › Mates › Mate** (click the main button = Smart Mate), or choose a type from the button's arrow
- Right-click › **Add Mate ›** / **Constrain ›**



Assembly Mates panel before any surface is selected

Steps

1. Click **Mate**. The cursor becomes a selection box.
2. Click a surface, edge or hole on component 1 (Reference), then on component 2. **Component 2 is the one that moves.** If component 2 cannot move, the program swaps them for you.
3. If several faces are stacked under the cursor, press **Tab** to go to the next face (*Face n/N on — click to pick it, Tab for the next one.*).
4. The component moves to the preview position using the best mate type. The available buttons glow yellow (*Suggested: …*). Click another yellow button to change the type.
5. Use ⇅ **Flip Mate** or ⇆ **Flip Align** if the direction is wrong. Enter an Offset/Angle if needed.
6. Click **Apply** (*Mate applied — pick the next pair*), or press Esc to discard the preview.



Mate edit panel with suggested types (yellow)

Panel part	Meaning
Reference geometry / Moving geometry / Third geometry	What you selected (× = clear). Third is used for mates with 3 selections.
Value boxes	Offset (mm), Angle (deg), Distance (mm), Gear ratio, Pitch (mm/rev), Travel (mm/rev), Ratio, Radius (mm), Center offset (mm), Joint angle (deg), depending on the type
External gears / Internal gears	Gear Mate only
23 mate type buttons	Buttons that cannot be used with the selected surfaces are grey
⇅ Flip Mate	Mates to the other side of the face
⇆ Flip Align	Switches Aligned / Anti-aligned
Apply / Close	Confirm / close (edit mode: Apply / Cancel / Delete)

> **Note:** If a mate conflicts with an existing mate, you see ⚠ *Over-defined — this mate conflicts (likely a mis-picked face)* and a floating card with the buttons ↶ **Undo mate**, ↻ **Re-pick** and ✓ **Keep**.

Large imported assemblies

- A new mate solves only the parts it affects (the moving part and the parts mated to it). Fixed parts act as boundaries, so the whole assembly is not pulled into the solve. Imported machines with thousands of parts stay responsive.
- If a solve takes longer than about 0.3 seconds, a *Solving mates: N moving bodies, step n/N …* window appears with a **Cancel** button. Cancel discards the new mate and the parts go back to where they were.

Mates that cannot be added

Case	Status bar message	What to do
Both selected faces are in the same rigid sub-assembly	*Both faces are on the SAME rigid sub-assembly — it already moves as one body. Right-click → Rigid Body → Ungroup first, or open the sub-assembly in its own window to mate inside it.*	Right-click › **Rigid Body › Ungroup Rigid Body** first, or open the sub-assembly in its own window (**Open Sub-Assembly (Edit)**) and add the mate inside it.
Component 2 (the moving one) is in a sub-assembly that has a fixed member	*The moving part belongs to a sub-assembly with a FIXED member — Unfix it first, or pick this face as the REFERENCE (first pick) and the free part second.*	Unfix it first, or pick this face first (Reference) and the free part second.

See also

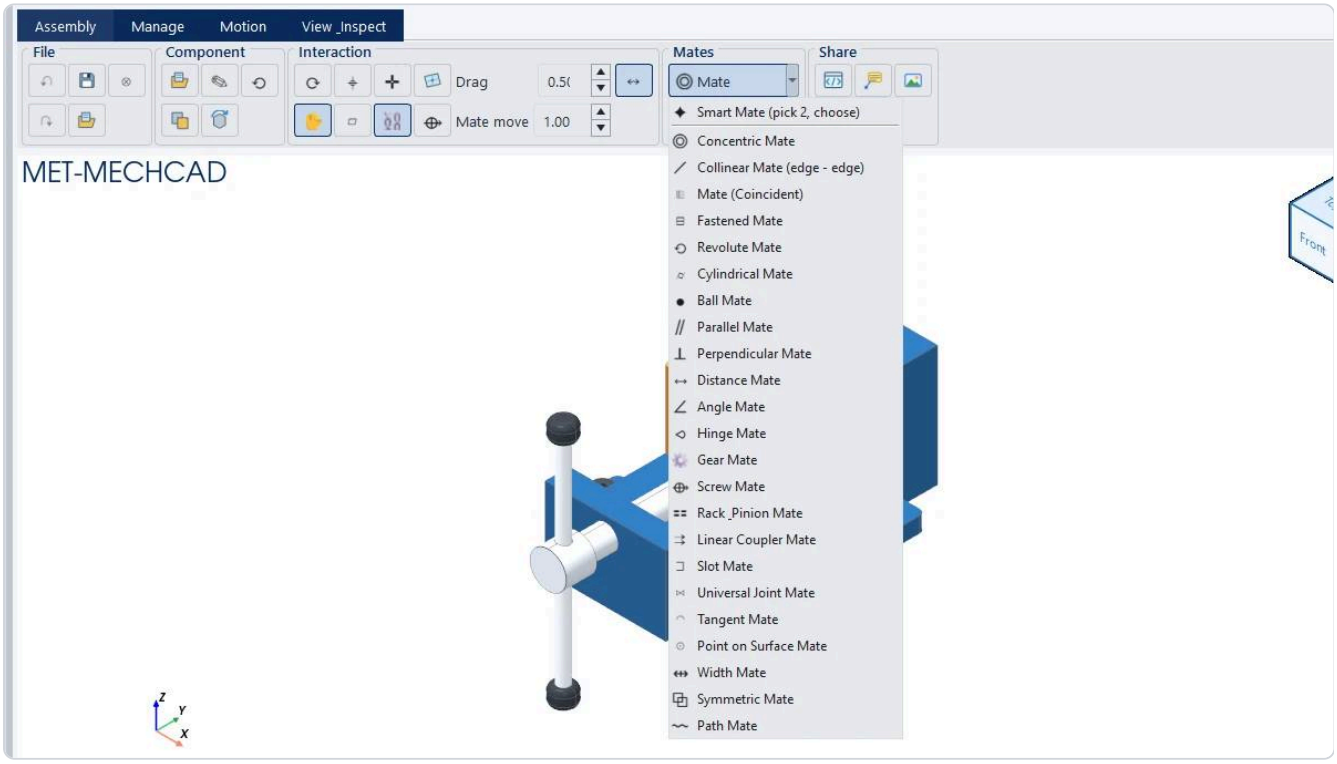
- Mate types
- Edit and delete mates
- Auto Mate

Mate Types Standard

What each of the 23 mate types does, and which surfaces you can use with it.

What it does

Lists every mate in the MECHCAD **Mate** button, with what it locks and the surfaces you must select.



Mate type list from the Mate button arrow

Mate	What it locks	What to select	Remaining freedom
Smart Mate	Picks the type for you from the surfaces	2 items	–
Concentric	Axes are straight and share one center. Slides/rotates ("Lock rotation" locks the rotation).	Hole / shaft / circular edge	2 (1 when locked)
Collinear (edge - edge)	Two edges (or an edge and an axis) lie on the same line	Straight edge / axis	–
Mate (Coincident)	Faces touch or are flush, with an offset	Flat face	3
Fastened	Locks all directions	Anything	0
Revolute / Hinge	Straight axis, fixed position along the axis, rotates	Axis	1
Cylindrical	Straight, shared axis; slides and rotates	Axis	2
Ball	Center points coincide	Anything	3
Parallel	Parallel (no distance set)	Face / axis / edge	4
Perpendicular	Perpendicular to each other	Face / axis / edge	5
Distance	Parallel and separated by a value (default is the current distance)	Face / axis / circle	3
Angle	Angle between faces = value	Face / axis / edge	5
Gear	Rotates relative to the other gear by the gear ratio (External / Internal)	Axis	1
Screw	One rotation = a move equal to the pitch	Axis	1
Rack & Pinion	One pinion rotation = rack moves by the travel	Axis	1
Linear Coupler	Moves relative by the ratio (negative = opposite direction)	Axis	1
Slot	Sits on a slot with its center on the slot path	Face / circle	2
Universal Joint	Universal joint with the joint angle	Axis	2
Tangent	Touches tangentially (axis is perpendicular to the face, separated by the radius)	Axis + face	4
Point on Surface	A point lies on a face	Anything + face	5
Width	The tab component is centered between two reference faces (3 selections)	2 faces + tab	3
Symmetric	Symmetric about a plane (3 selections: plane, moving component, opposite component)	Face	1
Path	A point slides along the edges of the path face	Face + follower	4

Types available for each pair of surfaces

Selected pair	Available mates
Axis + axis	Concentric, Tangent, Parallel, Perpendicular, Angle, Distance, Fastened (+ Revolute, Cylindrical, Hinge, Gear, Screw, Rack & Pinion, Linear Coupler, Universal)
Face + face	Coincident, Parallel, Perpendicular, Distance, Angle, Fastened (+ Width, Symmetric, Slot, Path)
Axis + face	Tangent, Perpendicular, Parallel, Point on Surface, Fastened
Circle + circle	Concentric, Coincident, Distance, Fastened
Edge + edge / axis + edge	Collinear, Parallel, Perpendicular, Angle, Fastened

> **Note:** For gears made with the Gear Wizard, the Gear Mate ratio is calculated from the real pitch radius. The values in Properties are **Ratio** (Gear / Linear coupler) and **Pitch / lead** (Screw / Rack & pinion).

See also

- Add mates
- Edit and delete mates

Edit, flip and delete mates Standard

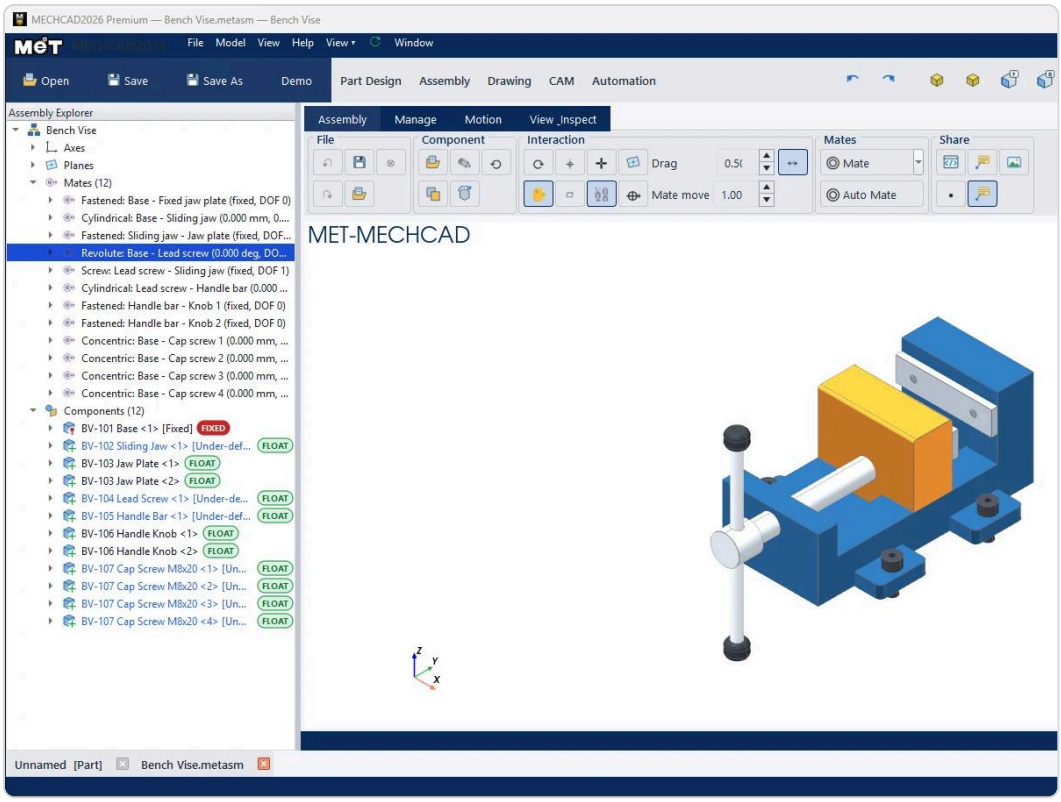
Edit mate values from the Assembly Explorer or Properties panel, and deal with conflicting mates.

What it does

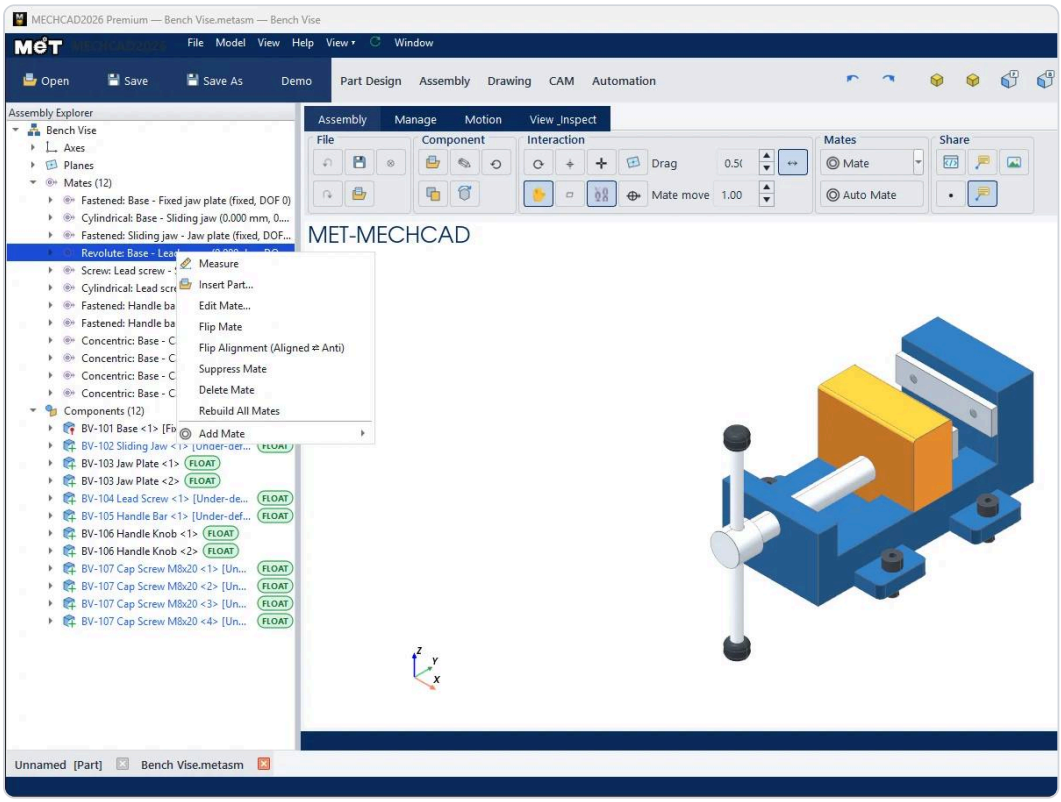
Changes the type, values or surfaces of an existing mate, flips its direction, suppresses it temporarily, or deletes it.

Where to find it

- Assembly Explorer › **Mates (n)** group › double-click a mate, or right-click › **Edit Mate...**
- The **Properties / Info › Mate** panel on the right when a mate is selected



Mate list in the Assembly Explorer



Right-click a mate

Right-click menu	Description
Edit Mate…	Opens the Mate panel in edit mode (pick new surfaces, change the type/values, then Apply).

Right-click menu	Description
Flip Mate	Makes the parts touch on the other side of the face.
Flip Alignment (Aligned ⇄ Anti)	Swaps the axis direction (Concentric, Coincident, Parallel, Distance, Fastened, Revolute, Cylindrical).
Suppress Mate / Unsuppress Mate	Turns the mate off/on temporarily.
Delete Mate	Deletes the mate.
Rebuild All Mates	Recalculates all mates.
Show / Hide Mate Details	Shows the Reference / Moving surfaces under each mate.

Properties panel › Mate

Field	Description
Axial offset / Minimum / Maximum (mm)	Distance and sliding limits
Angle / Minimum angle / Maximum angle (deg)	Angle and rotation limits
Ratio	Gear: r(reference)/r(mating) / Linear coupler: ratio b:a
Pitch / lead	Screw: distance per turn / Rack & pinion: rack distance per pinion turn
Lock rotation (key concentric frames)	Locks the rotation of a Concentric mate
Suppress mate	Turns the mate off temporarily
Apply	Applies all values and recalculates the assembly
⇄ Flip	Switches Aligned / Anti-Aligned

> **Note:** Editing and deleting a mate does not recalculate the whole assembly by itself. Click **Regen** (Interaction group) when you want every mate solved again.

See also

- Add mates
- Mate types

Auto Mate Standard

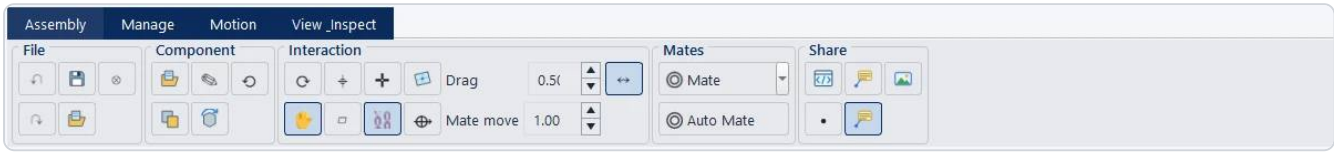
Automatically create mates at every contact point of the selected or visible parts.

What it does

Auto Mate finds where parts touch (face to face, hole to shaft) and creates all the mates for you. It suits assemblies imported from STEP, where the parts are already in the right positions but have no mates.

Where to find it

Assembly › Mates › Auto Mate



Auto Mate button in the Mates group

Steps

1. Select the parts you want. If you select none, all visible parts are used.
2. Click **Auto Mate**.
3. Check the list shown, then confirm. Click **Cancel** to undo everything.

> **Note:** A sub-assembly inserted as a rigid body must first be released with **Rigid Body › Ungroup Rigid Body**. Only then can Auto Mate create the joints inside it.

See also

- Add mates
- Insert parts and sub-assemblies

Drag and move parts (Drag Part) Standard

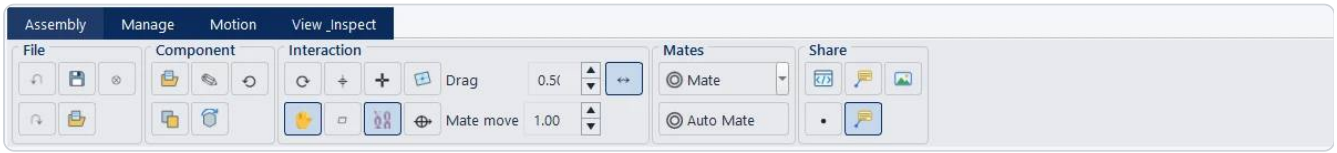
Drag parts so they follow their mates, rotate or copy them, and set drag sensitivity.

What it does

Checks how a mechanism moves by dragging parts in the view. Parts move according to their mates. Fixed parts do not move.

Where to find it

Assembly › Interaction › Drag Part (on by default), **Drag**, **Mate move**, **Min Motion**



Interaction group

Action (Drag Part on)	Result
Drag a part with the left button	Moves it according to its mates.
Shift + drag	Rotates the part (or the whole sub-assembly) about the mate axis or its centre.
Ctrl + drag	Copies the part, then drags the copy.

Action (Drag Part on)	Result
Drag on empty space	Draws a selection box.
Esc	Cancels the drag.

Option	Default	Description
Drag Part	On	Turn off to prevent accidental movement (left-drag then pans the view).
Drag	0.50	Drag sensitivity (0.05–20). A lower value gives finer control.
Mate move	1.00 ×	Speed at which a part slides into its mate position (0.01×–5×).
Min Motion	On	Keeps other parts moving as little as possible. Off = Max Motion, which lets the mechanism follow the cursor more freely.

> **Note:** A part locked with Fastened, or inside a fixed sub-assembly, cannot be rotated, and a message appears. You can enter exact positions in the Properties panel › **Instance Placement**.

See also

- Assembly Explorer and Properties panel

Linear Pattern of parts Standard

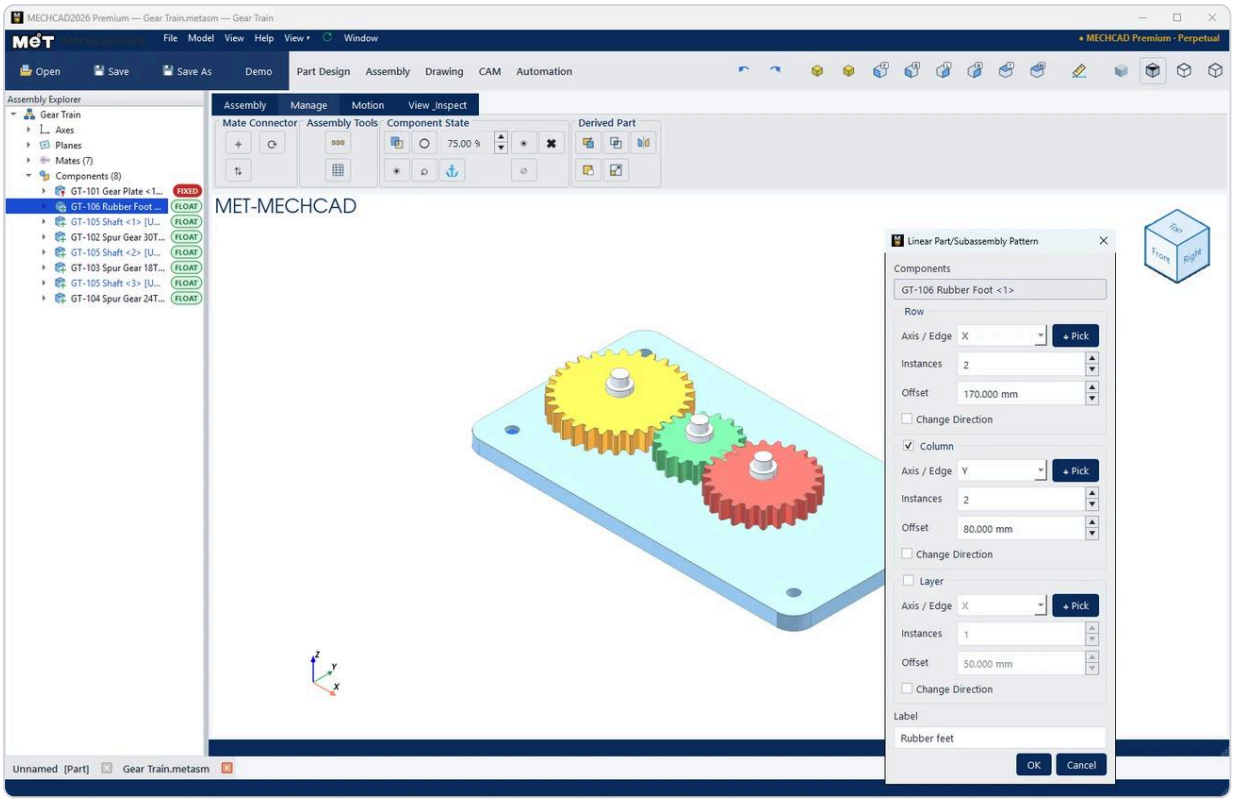
Repeat a part or sub-assembly in rows, columns and layers.

What it does

Creates copies of a part or sub-assembly in rows, columns and layers, for example bottles on a pallet or several screws.

Where to find it

Manage › Assembly Tools › Linear Pattern



Linear Part/Subassembly Pattern window

Steps

1. Select a part, then click **Linear Pattern**.
2. Under **Row**, choose **Axis / Edge** (X / Y / Z, or *Picked face…* and then click ⌖ Pick). Enter **Instances** and **Offset**.
3. Tick **Column** and/or **Layer** if you need more directions.
4. Set the **Label**, then click **OK**.

Option	Default	Description
Components	Selected part	The seed part
Axis / Edge	X	Direction of each set
Instances	–	Count (1–500)
Offset	50 mm	Spacing
Change Direction	Off	Reverses the direction
Label	Linear Pattern (1)	Name

See also

- Insert parts and sub-assemblies

Component state: Show/Hide, Transparent, Fix, Suppress Standard

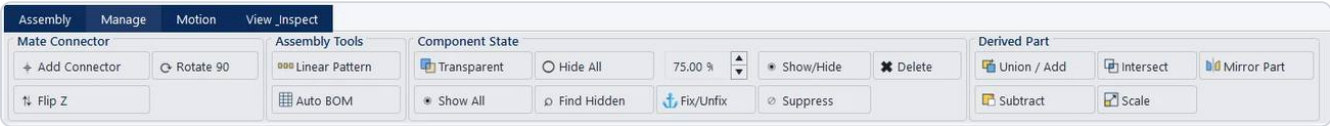
Hide, show, make transparent, fix and suppress parts in an assembly.

What it does

Controls the display and state of parts so that large assemblies are easier to work with.

Where to find it

Manage › Component State, or right-click a part › **Display / State ›**



Manage tab

Command	Description
Transparent	Makes the selected part transparent (click again to restore). The transparency follows the % field.
% (0–100)	0% = opaque, 100% = fully clear (default 75).
Show All	Shows every hidden part (suppressed parts stay off).
Hide All	Hides everything, so you can then Show only the parts you need.
Find Hidden	Selects and reveals every hidden part in the Explorer.
Fix/Unfix	Fixes / releases (works on a whole sub-assembly).
Show/Hide	Show / hide.
Suppress	Turns off the selected part or mate (it is not calculated).
Delete	Deletes.

> **Note:** In a very large assembly, use **Purge (Skip Loading)** (right-click a part) so that parts you do not need are not loaded.

See also

- Assembly Explorer
- Large assemblies run slowly

Derived Part: Union, Subtract, Intersect, Scale and Mirror Part Standard

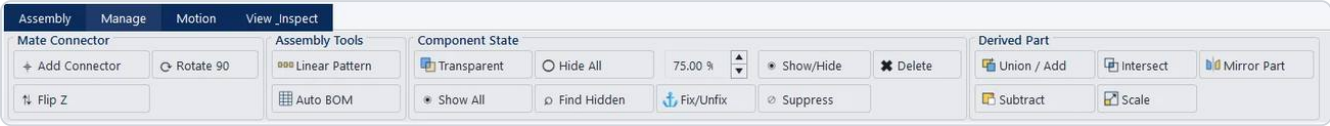
Create a new part from parts in an assembly without changing the originals.

What it does

Creates a new Part file from the selected parts and inserts it into the assembly. The original parts are not changed. For example, make a plate with holes that match the position of other parts (Subtract), or make a left/right pair (Mirror Part).

Where to find it

Manage › Derived Part



Derived Part group in the Manage tab

Command	Select	Result
Union / Add	2 parts	Merged into one part
Subtract	2 parts	The current part (target) minus the other part
Intersect	2 parts	The overlapping region
Scale	1 part	Scaled up/down
Mirror Part	1 part	A mirrored (opposite-hand) part

Steps

1. Select the number of parts shown in the table (the currently selected part is the target).
2. Click the command.
3. The new part is saved and inserted into the assembly.

See also

- Union, Subtract, Intersect in Part Design

Explode and Collapse Standard

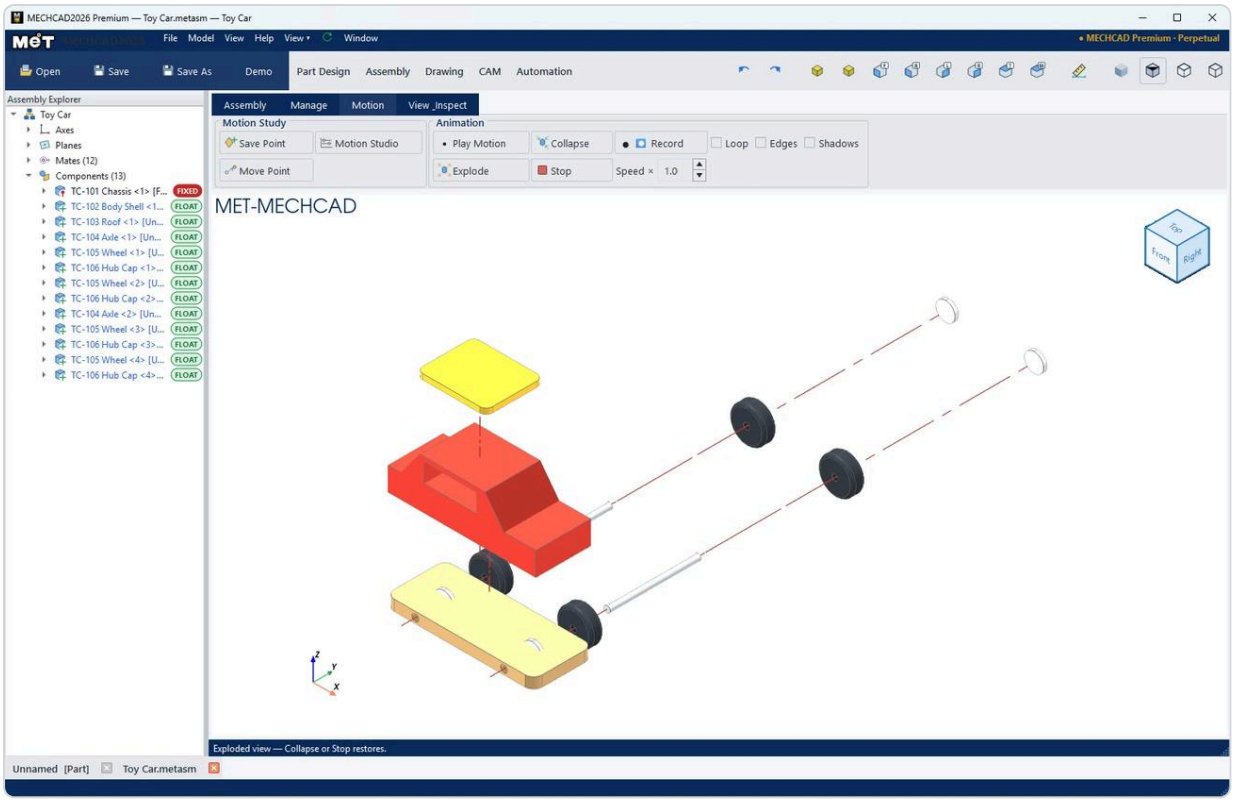
Animate the parts apart into an exploded view, set the distance 0–100% with a slider, show center lines, and put them back together.

What it does

Explode separates the parts along their mate directions as an animation. Use it for presentations or to check the assembly sequence. **Collapse** puts them back. The real positions and mates are not changed.

Where to find it

Motion › Animation › Explode / Collapse



Toy car exploded

Steps

1. Click **Explode** (the assembly needs at least 2 parts). The parts move fully apart (100%) and the floating **Explode** box opens beside the Explorer.
2. Drag the **Explode distance** slider, or type a % value, to set the distance yourself. The parts move live while you drag.
3. Tick or clear **Center lines** to show or hide the center lines.
4. Click **Collapse** to put the parts back, or ■ **Stop**. Click **Close** to close the box.
5. Adjust **Speed ×** if you want the animation slower or faster. Tick **Loop** to play back and forth continuously.

Explode box

Option	Default	Description
Explode distance (slider + %)	–	0% = assembled, 100% = fully exploded. The parts move live while you drag.
Explode	–	Plays the animation to 100%
Collapse	–	Plays the animation back to the assembled position
Center lines	On	A line from each part's assembled position to where it is now (the explode trail)
Close	–	Closes the box

The Explode / Collapse buttons in the box and on the ribbon drive the same value as the slider (the slider follows the animation). The explode distance and the Center lines setting are saved in the file and come back when you reopen it.

Ribbon options

Option	Default	Description
Speed ×	1.0	Speed (0.1–8)
Loop	Off	Plays Explode ⇆ Collapse continuously
Edges	–	Shows edge lines during playback
Shadows	–	Shows shadows during playback

Explode direction: along the axis of Concentric mate groups first. If there are none, it uses the normal of Coincident faces. If there are none of those either, it uses the nearest principal axis.

> **Note:** Dragging the slider does not re-solve mates or change the real component positions, so it stays smooth even on large assemblies with thousands of parts.

> **Note:** An exploded view for documents, with numbered balloons, can be made in Auto BOM (Edit Balloons).

See also

- Animation and video recording

3D Notes and Pictures in an Assembly Standard

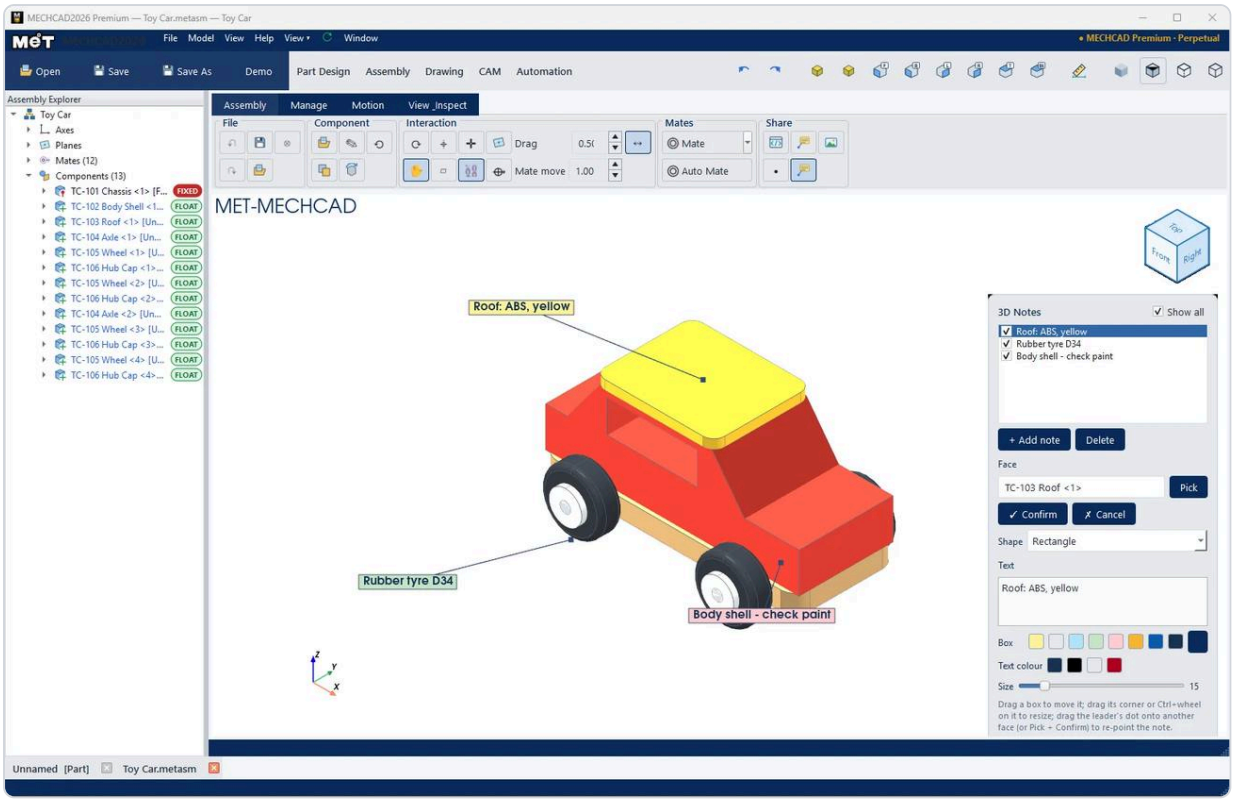
Add 3D text labels that point to components, and place pictures on faces.

What it does

- **3D Notes** are text labels in the 3D view that point to a face. Use them to mark sensors, motors or inspection points. They can appear in an HTML export.
- **Picture** places an image on the face of a component.

Where to find it

Assembly › Share › 3D Notes / Show Notes / Picture



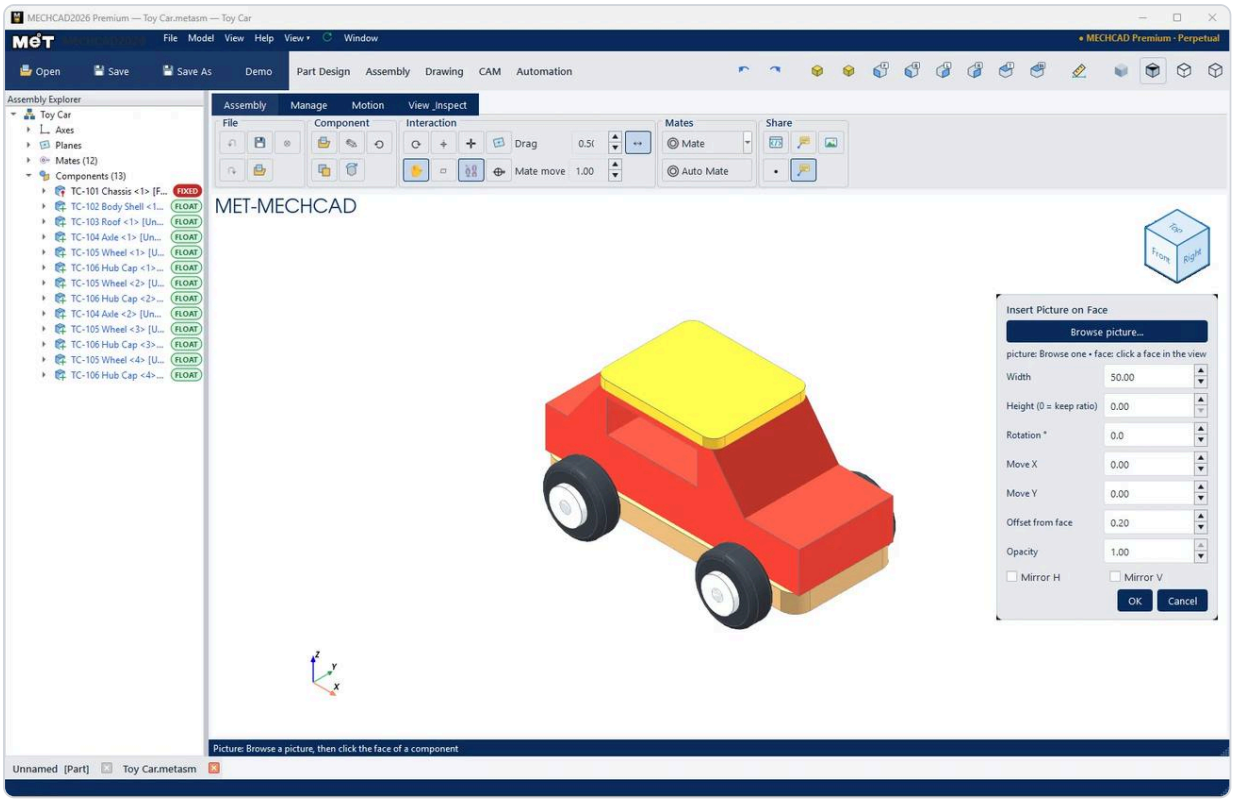
3D Notes panel

3D Notes

1. Click **3D Notes**, then click **+ Add note**.
2. Click **Pick**, then click the face the note should point to. Click **✓ Confirm**.
3. Type the **Text** and choose the **Shape**, **Box** colour, **Text colour** and **Size**.
4. Drag the note box to move it. Drag a corner, or use Ctrl+mouse wheel, to shrink or enlarge it.
5. **Show Notes** shows or hides all notes (the notes are kept).

Option	Meaning
Show all	Shows all notes in the list
Shape	Rectangle / Rounded rectangle / Circle
Box / Text colour	Box color / text color
Size	Text size

Picture



Insert Picture on Face panel

Click **Browse picture...**, then click a face. Adjust Width, Height (0 = keep ratio), Rotation °, Move X/Y, Offset from face, Opacity and Mirror H/V, then click **OK**.

See also

- Export HTML
- Pictures in Part Design

Pack and Go (.metpkg) Standard

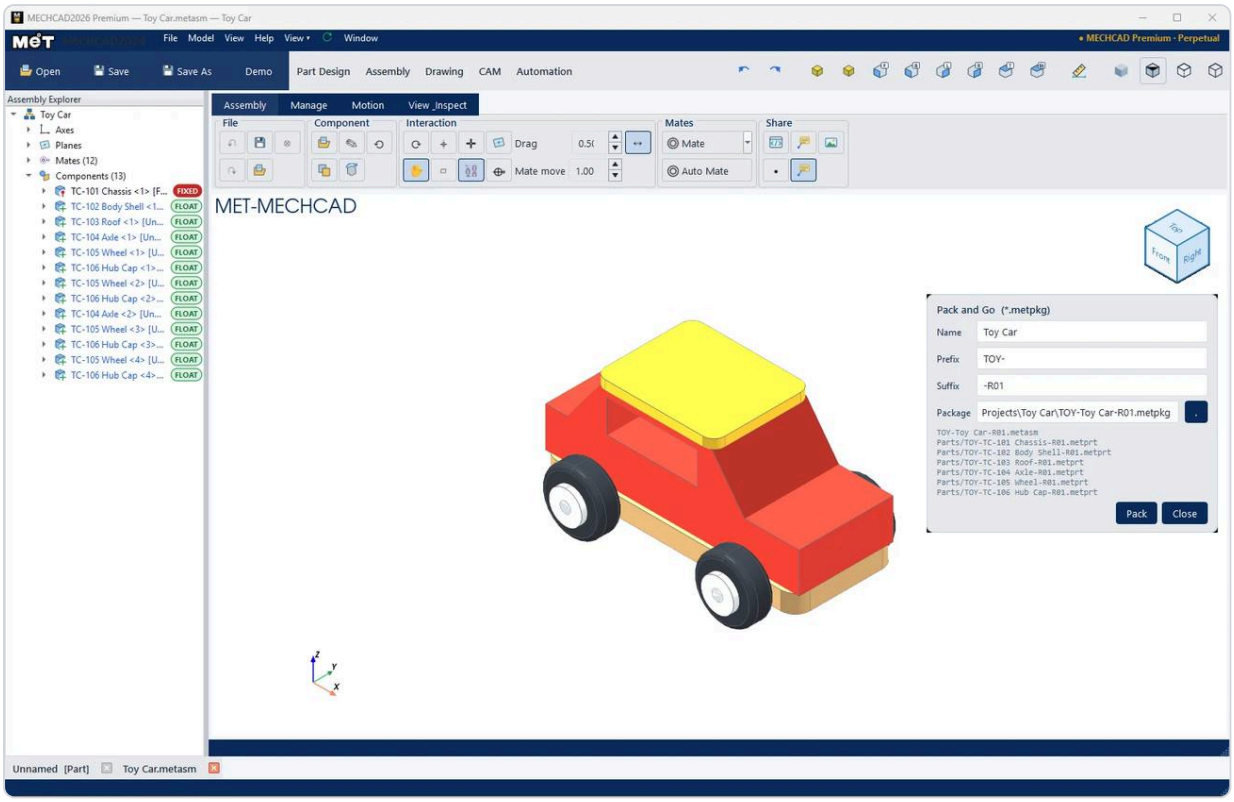
Bundle an assembly and all its components into one package to share or copy.

What it does

Pack and Go bundles the assembly file and every component it references into a single `.metpkg` file. Use it to send work to others, or to copy an assembly and rename its components with a prefix/suffix.

Where to find it

File › Pack and Go... (while an assembly is open)



Pack and Go card

Steps

1. Open an assembly and choose **File › Pack and Go…**.
2. Set the **Name**, and a **Prefix** / **Suffix** if you want. The resulting file names are listed below immediately.
3. Choose where to store the **Package** (the … button).
4. Click **Pack**.

Option	Meaning
Name	Assembly name in the package
Prefix	Text added before every file name
Suffix	Text added after the name, for example -R01
Package	Location of the `.metpkg` file

Inside the package: `manifest.json`, the `.metasm` file and `Parts/*.metprt` (components imported from STEP are converted to `.metprt` for you).

> **Note:** Double-click a `.metpkg` file to open it. The program unpacks it into a folder next to the package, then opens the assembly. If a file with the same name exists, it asks *Replace the existing package?*

See also

- MECHCAD file types

Chapter 5 quiz: Assembly

Choose the one best answer. Answers are in Appendix A

1. **When you add a Mate, which component moves?**

   A. The first component you pick (Reference)
   B. The second component you pick
   C. Both move halfway
   D. Always the larger component

2. **While picking a mate, several faces are stacked under the cursor. How do you reach the face behind?**

   A. Press Tab to go to the next face
   B. Hold Alt and click
   C. Hide the front part first
   D. Scroll the mouse wheel

3. **What happens to the first part you insert into a new assembly?**

   A. It floats freely until you add a mate
   B. It is converted to a sub-assembly
   C. It is placed at the cursor and hidden
   D. It is fixed in place as the base

4. **An inserted sub-assembly moves as one rigid body. What do you do to let its inner parts move as joints?**

   A. Double-click it
   B. Turn off Drag Part
   C. Right-click › Rigid Body › Ungroup Rigid Body, then add mates (e.g. Auto Mate)
   D. Use Explode

5. **Which command bundles an assembly and every component it references into one file to send to others?**

   A. Share › Export HTML
   B. File › Pack and Go… (creates a .metpkg file)
   C. File › Export STEP…
   D. Manage › Auto BOM

6. **After editing or deleting a mate, how do you make the program solve every mate again?**

   A. Save and reopen the file
   B. Click Show Mates
   C. Press Esc
   D. Click Regen in the Interaction group

CHAPTER 6

Drawing

1. Create a Drawing (Standard Views Creation)
   Create a 2D drawing from a part or assembly with standard views, dimensions and an automatic BOM.
2. Ribbon and Drawing Explorer
   All commands in the Drawing tab, and how to use the mouse on the sheet.
3. Views: Base, Projected, Section, Detail and Break View
   Add projected, section, detail and broken views to the sheet.
4. View Properties and the view right-click menu
   Set a view's scale, display style, tangent edges and orientation.
5. Drawing dimensions: Auto Dimension, Smart Dimension, Hole Callout and Ordinate
   Add automatic dimensions, click-to-dimension, hole callouts and ordinate dimensions.
6. Notes, Leader, Balloon and symbols (Center Line, Datum, GD&T)
   Add text, numbered balloons, centre lines, datums and Feature Control Frames.
7. BOM, balloons and tables (Hole, Revision, Cut List)
   Add a BOM table, automatic balloons, a hole table, a revision table and a cut list to the drawing.
8. Sheet Properties and Title Block
   Choose the paper size and fill in the title block.
9. Export PDF, DXF, DWG and Print
   Export the drawing as PDF, DXF or DWG, print it, and save .metdrw.

Create a Drawing (Standard Views Creation) Standard

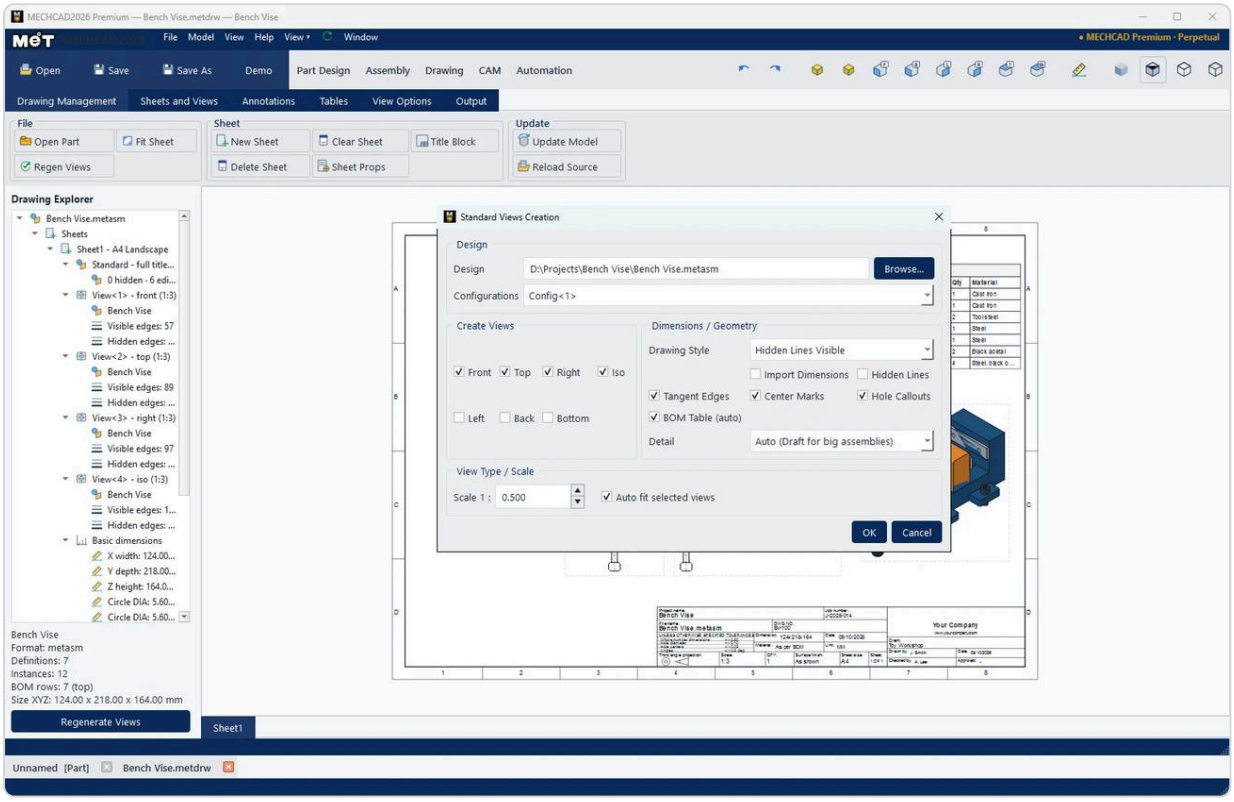
Create a 2D drawing from a part or assembly with standard views, dimensions and an automatic BOM.

What it does

The **Drawing** tab creates a 2D drawing from a Part file (`.metprt`) or an Assembly file (`.metasm`). It lays out the views in third-angle projection, sets the scale automatically, and leaves room for the title block and BOM. Drawings are saved as `.metdrw` .

Where to find it

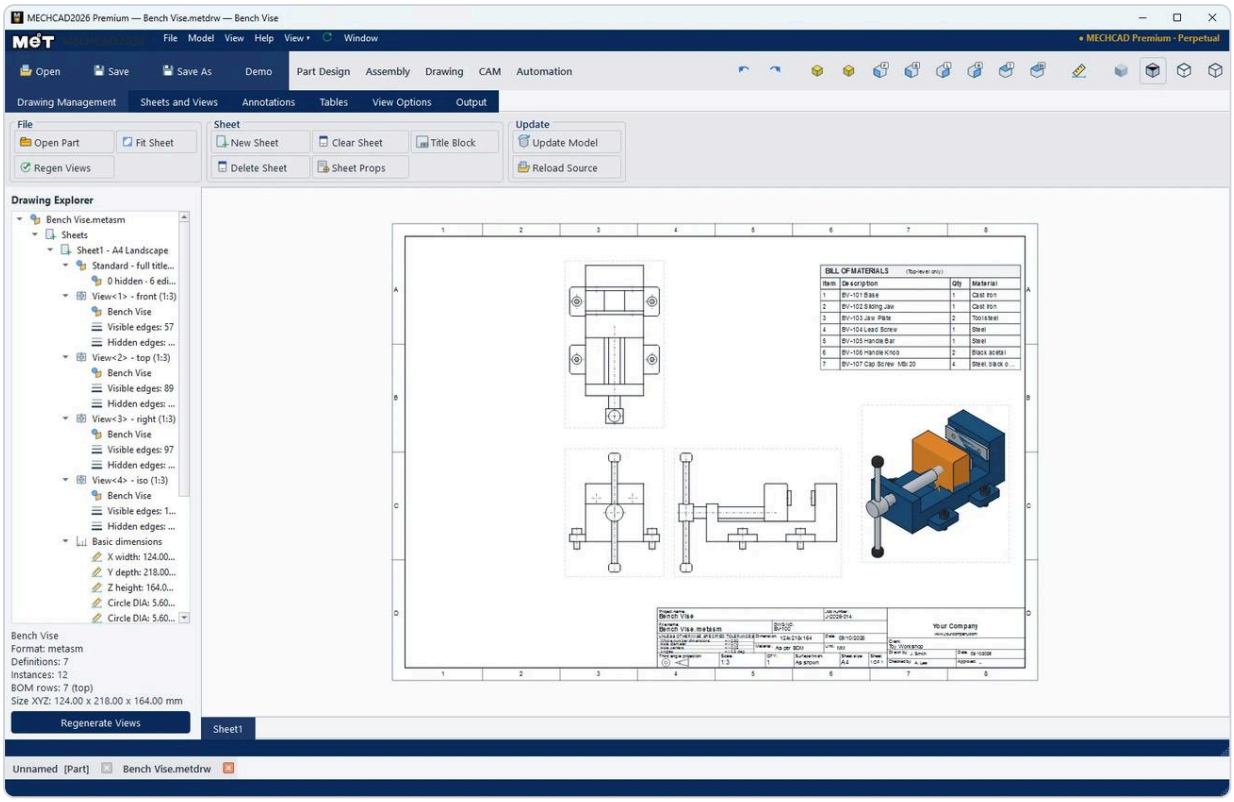
- **Drawing Management › File › Open Part**
- **File › New Drawing from Current** (uses the saved part/assembly that is currently open)
- **File › Open Drawing Source...**



Standard Views Creation window

Steps

1. Save the part or assembly first.
2. Go to the **Drawing** tab and click **Open Part** (or use **File › New Drawing from Current**).
3. In the **Standard Views Creation** window, choose the file under **Design** (Browse...), tick the views you want, and set the options in the table.
4. Click **OK**. The drawing is created with a title block.



Bench vise drawing: 3 views + isometric and a BOM table

Option	Default	Description
Design	–	A `.metprt` / `.metasm` file (choose `.metdrw` to open an existing drawing)
Configurations	Config<1>	Part configuration
Create Views	Front, Top, Right, Iso	Views to create (Left, Back, Bottom can be added)
Drawing Style	–	Fast Draft (all edges, instant) / Hidden Lines Visible / Hidden Lines Removed / Shaded with Visible Edges (assemblies use Hidden Lines Removed)
Detail	Auto	Auto (Draft for big assemblies) / Draft - least detail, fastest / Full - every part
Import Dimensions	–	Adds dimensions automatically
Hidden Lines / Tangent Edges	–	Hidden lines / tangent edges
Center Marks / Hole Callouts	–	Centre marks / hole callouts
BOM Table (auto)	–	A BOM table for assemblies
Scale 1 : / Auto fit selected views	–	Scale, or fit to the sheet automatically

> **Note:** Assemblies with 600 or more parts use Draft mode and turn tangent edges off automatically for speed. A `.metdrw` file refers to the source files, so keep them in the same place or in the same folder as the `.metdrw` (or use **Output › Pack Refs**).

See also

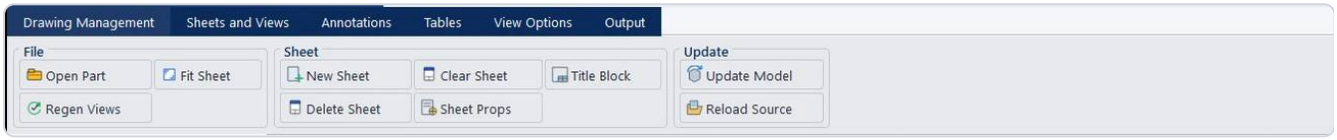
- Drawing ribbon
- Projected, Section, Detail and Break views

Ribbon and Drawing Explorer Standard

All commands in the Drawing tab, and how to use the mouse on the sheet.

What it does

A summary of the commands in the six Drawing ribbon tabs, and of the **Drawing Explorer** on the left, which shows the sheets, views, dimensions and tables.



Drawing Management



Sheets and Views



Annotations



Tables



View Options



Output

Tab › Group	Commands
Drawing Management › File	Open Part, Regen Views, Fit Sheet
Drawing Management › Sheet	New Sheet, Delete Sheet, Clear Sheet, Sheet Props, Title Block
Drawing Management › Update	Update Model (pulls in edits from the part), Reload Source
Sheets and Views › Views	Base View, Projected View, Detail View, Section View, Break View
Sheets and Views › Secondary Views	Auxiliary View, Aligned Section, Removed Section, Broken-Out Section, Partial View, Image
Sheets and Views › View Style	View Props, Scale 1:N, Reposition Dims
Annotations › Dimensions	Auto Dimension, Smart Dimension, Hole Callout, Ordinate Dim
Annotations › Notes	Note, Leader, Balloon
Annotations › Dim Style	Prec .00, Dim Color, Text Size (applies to new dimensions)
Annotations › Symbols	Center Line, Center Mark, Datum, Feature Control, Surface Finish, Weld, Hatch
Tables › BOM	Bill of Materials, Auto Balloons, Label Tags, Parts List
Tables › Tables	Hole Table, General Table, Revision Table, Cut List
View Options › Navigate	Pan, Zoom Fit, Actual Size
View Options › Display	Show Grid, Line Weights, Dark Sheet
Output › Export	Export PDF, Export DXF, Export DWG, Print Preview, Print
Output › Native	Save .metdrw, Pack Refs

Mouse and keys on the sheet

Action	Result
Mouse wheel	Zoom
Drag a view	Moves the view (a projected view can move only along the direction aligned with its parent view)
Double-click text	Edit Text: Text, Font, Size (pt), Bold/Italic, Align
Delete	Deletes the selection
Esc	Clears the dimension selection / leaves the tool
Ctrl+Z / Ctrl+Y	Undo / Redo (60 steps)

See also

- Create a Drawing
- Export PDF, DXF, DWG and Print

Views: Base, Projected, Section, Detail and Break View Standard

Add projected, section, detail and broken views to the sheet.

What it does

Creates additional views from an existing view.

Where to find it

Sheets and Views › Views



Views, Secondary Views and View Style groups

View	How to use it	Result
Base View	Click the button.	Creates a new standard view.
Projected View	Click the button, then click the left, right, top or bottom of an orthographic view.	A third-angle projected view at the same scale, linked to its parent view.
Section View	Click two points of the cutting line inside a view (the line is snapped horizontal/vertical).	*SECTION A-A* cut from the real solid, with 45° hatching, placed in line with the parent view. The cutting line and arrows are drawn on the parent view.
Detail View	Click the centre point, then a point on the circle.	*DETAIL B*, enlarged (standard scale near 2×), with a circle and a letter on the parent view.
Break View	Click two points inside the view.	Removes the middle of a long part. Dimensions still show the true length.

Steps (Section View)

1. Click **Section View**. The status bar shows *Section View: click two points of the cutting line on a view*.
2. Click the first and second points of the cutting line on the view.
3. Place the section view that is created.

Move and delete views

- Drag a view to move it (child views follow the parent view).
- Select it and press Delete, or right-click › **Delete View** (the view's dimensions are deleted too, and the layout is rearranged).

> **Note:** The **Secondary Views** group adds more views: Auxiliary View (click two points on an inclined edge), Aligned Section, Removed Section, Broken-Out Section, Partial View and Image (place a picture on the sheet).

See also

- View Properties

View Properties and the view right-click menu Standard

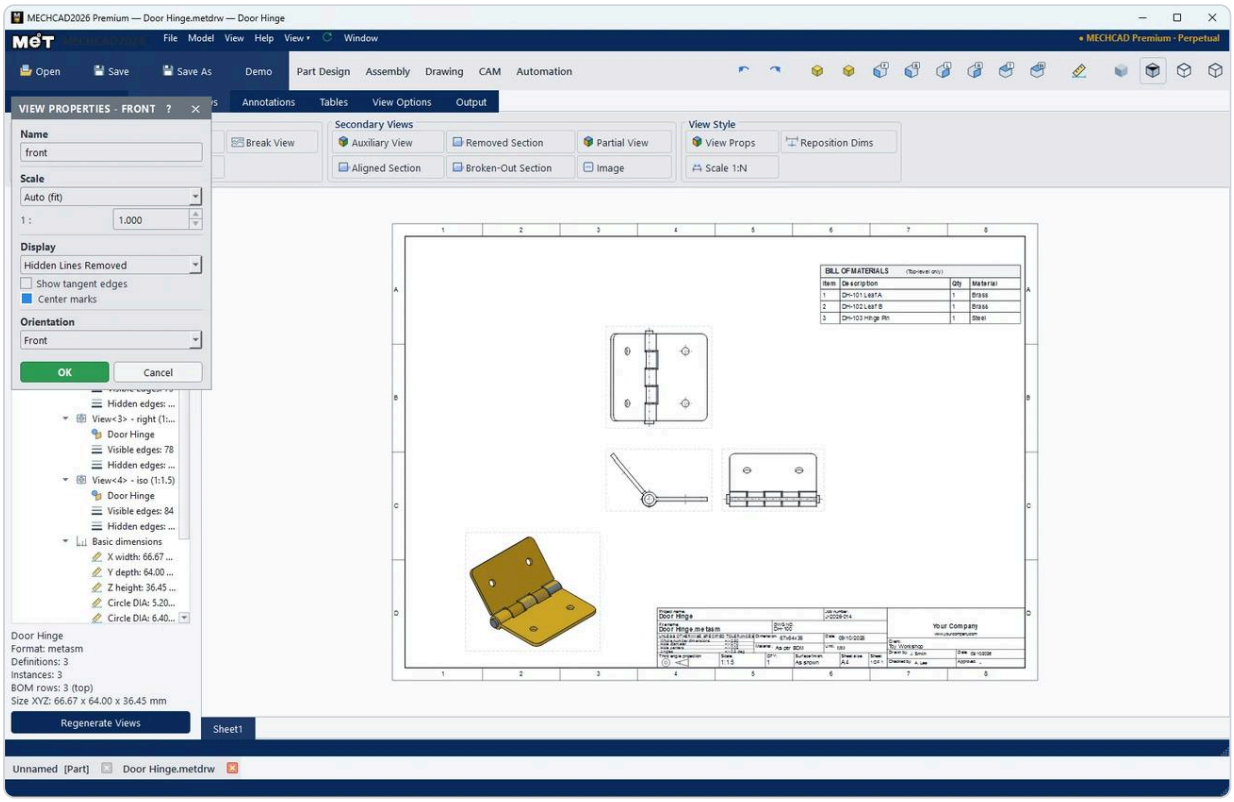
Set a view's scale, display style, tangent edges and orientation.

What it does

Adjusts the properties of each view.

Where to find it

- **Sheets and Views › View Style › View Props** (the selected view, or Front if none is selected)
- Right-click a view › **View Properties...**
- **Scale 1:N** sets the scale of the selected view (all views if none is selected)



VIEW PROPERTIES box

Option	Description
Name	View name
Scale	Auto (fit), 2:1, 1:1, 1:2, 1:5, 1:10, 1:20, 1:50, 1:100, Custom 1 : N
Display	Hidden Lines Removed / Hidden Lines Visible / Shaded / Shaded with Edges / Draft (all edges, no HLR)
Show tangent edges	Shows tangent edges
Center marks	Centre marks
Orientation	View direction (Front, Back, Top, Bottom, Left, Right, Isometric)

Right-click menu on a view

Menu	Description
View Properties... / Scale › / View Display ›	Properties, scale, display
Show Tangent Edges / Show Center Marks	Toggle display
Other Standard Views ›	Adds other standard views.
Hide '' in this View / Hide Parts in this View...	Hides parts in this view only (assemblies).
Rename... / Hide View / Regenerate View	Rename, hide, recalculate
Auto Balloons / Auto Label Tags / Remove Balloons / Tags	Balloons in this view
Lock View Position / Delete View / Regenerate All Views	Lock position, delete, recalculate all

> **Note:** To show a hidden view again, right-click an empty area of the sheet › **Show Hidden View**.

See also

- Projected, Section, Detail and Break views

Drawing dimensions: Auto Dimension, Smart Dimension, Hole Callout and Ordinate Standard

Add automatic dimensions, click-to-dimension, hole callouts and ordinate dimensions.

What it does

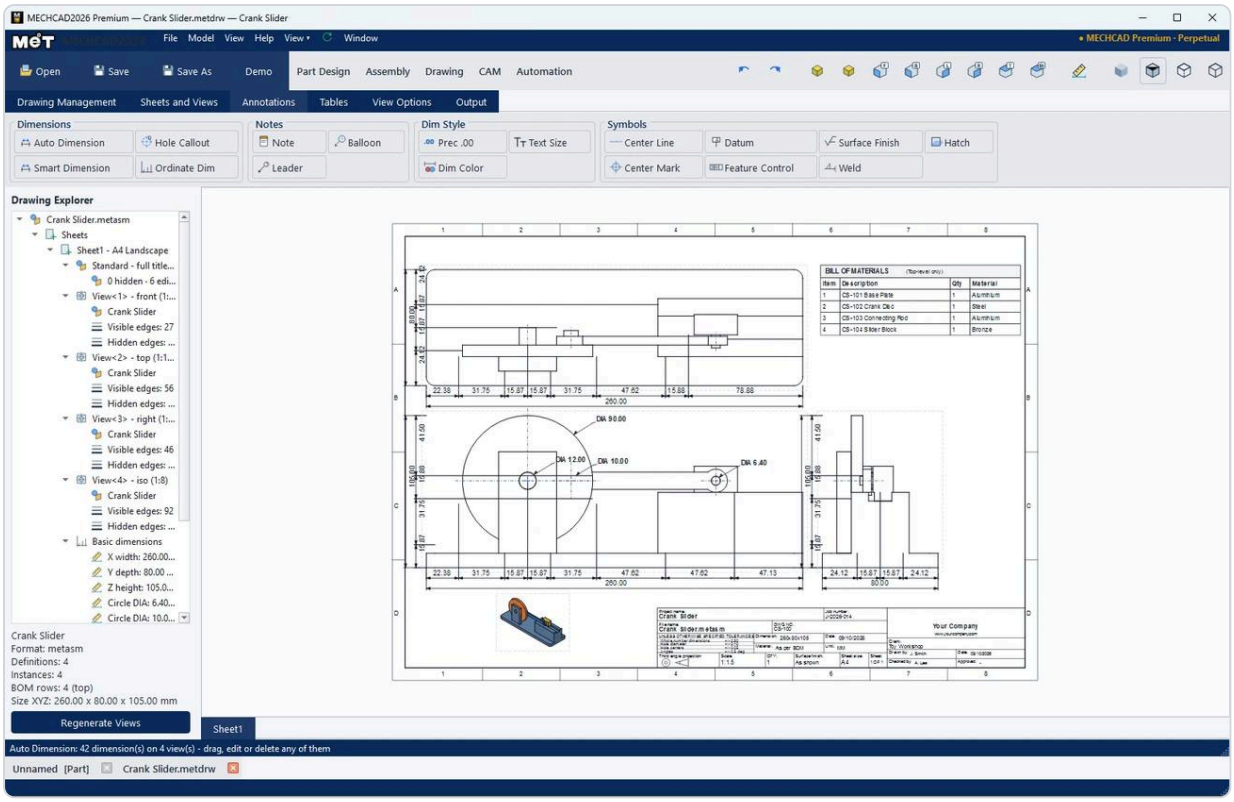
Adds dimensions to a 2D drawing.

Where to find it

Annotations › Dimensions and **Dim Style**



Annotations tab



Auto Dimension on the crank-slider drawing

Command	How to use it
Auto Dimension	Adds chain and overall dimensions in both directions, plus Ø, on every orthographic view (skips Section/Detail/Break views). Click again to replace the previous set.
Smart Dimension	Click a circle = Ø, an arc = R, two points = distance (horizontal/vertical/aligned), a point + an edge = perpendicular distance, an edge = length, two parallel edges = distance, two non-parallel edges = angle. Then click to place.
Hole Callout	Automatic hole text such as *4X Ø6.60 THRU*, depth ↧, counterbore ⌴, countersink ⌵ (on the selected view, or the view that shows the most holes).
Ordinate Dim	Click the base point, then the feature points. Shows *X … / Y …*.
Reposition Dims (View Style)	Arranges dimensions in rows outside the view so they do not overlap.

Dim Style	Description
Prec .00	Decimal places, 0–4
Dim Color	Colour of new dimensions
Text Size	Text size, 6–16 pt

Right-click menu on a dimension

Edit Text…, **Text Position** (Left / Center / Right), **Delete Dimension**

> **Note:** Holes smaller than 6 units on the sheet are skipped by Hole Callout. Use a Detail View to enlarge them first.

See also

- Notes, balloons and symbols

Notes, Leader, Balloon and symbols (Center Line, Datum, GD&T)

Standard

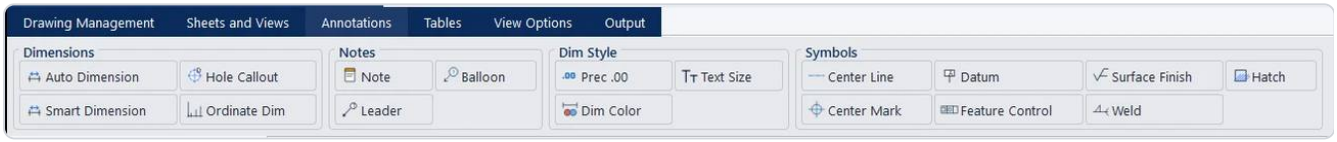
Add text, numbered balloons, centre lines, datums and Feature Control Frames.

What it does

Adds text and symbols to the drawing.

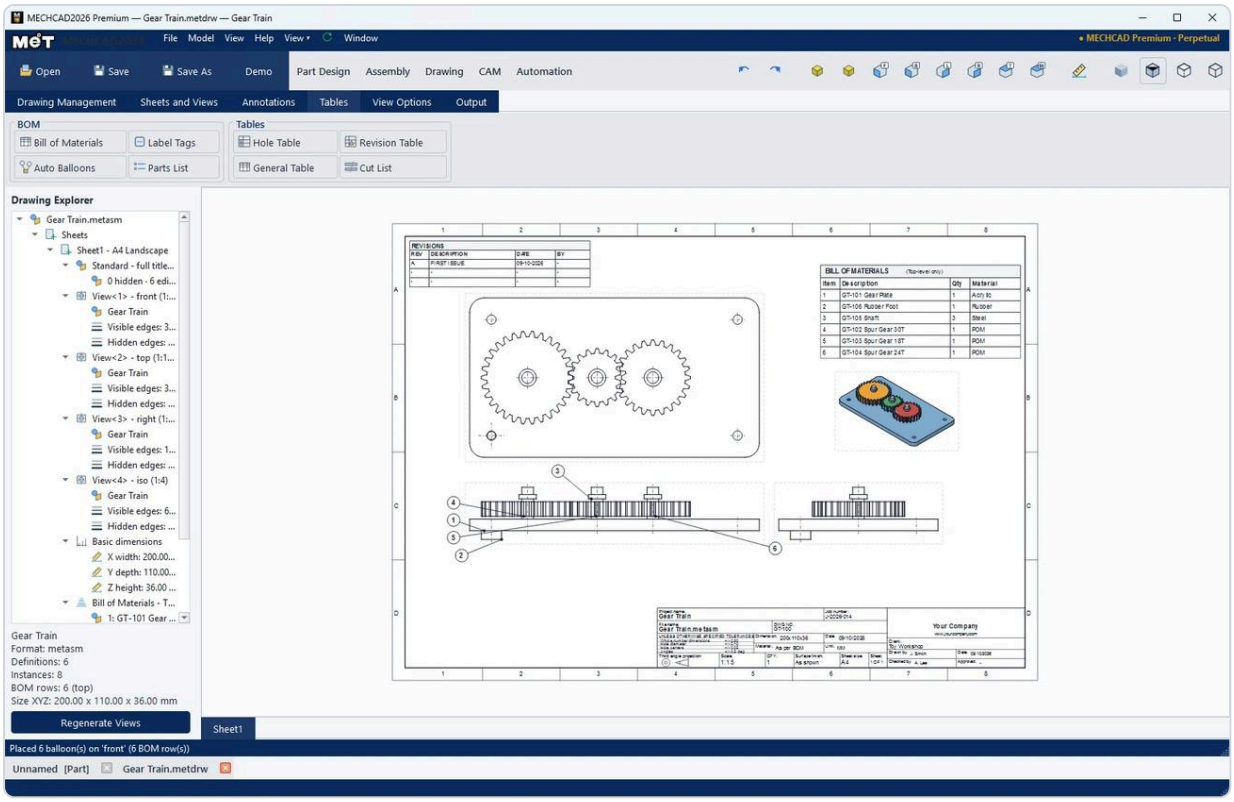
Where to find it

Annotations › Notes / Symbols, or right-click › **Annotation ›**



Annotations tab

Command	How to use it
Note	Type the *Note text:*, then click a position.
Leader	Type the *Leader text:*, click the anchor point, then click the text position.
Balloon	Click a point on the part, then click the balloon position (numbers increase automatically).
Center Line	Click two parallel edges (shaft/pipe) to get a centre line that Smart Dimension can snap to.
Center Mark	Adds a centre mark automatically (click again to remove).
Datum	Click a position to place a lettered frame A, B, …
Feature Control	Click a position, then enter the Characteristic (Position, Flatness, Straightness, Circularity, Cylindricity, Perpendicularity, Parallelism, Angularity, Concentricity, Symmetry, Profile, Circular Runout, Total Runout), Tolerance, ⌀ diametral zone, and Primary/Secondary/Tertiary datum.
Surface Finish	Places a surface-finish symbol (symbol only, no values).
Weld	Places a weld symbol (symbol only).
Hatch	Click two corners to get a 45° hatch pattern.



Revision and BOM tables on a drawing

> **Note:** Double-click any text to edit it (Text, Font, Size, Bold/Italic).

See also

- BOM and tables
- Dimensions in a Drawing

BOM, balloons and tables (Hole, Revision, Cut List) Standard

Add a BOM table, automatic balloons, a hole table, a revision table and a cut list to the drawing.

What it does

Creates tables on the drawing. The BOM table uses data from the assembly and from Part Properties.

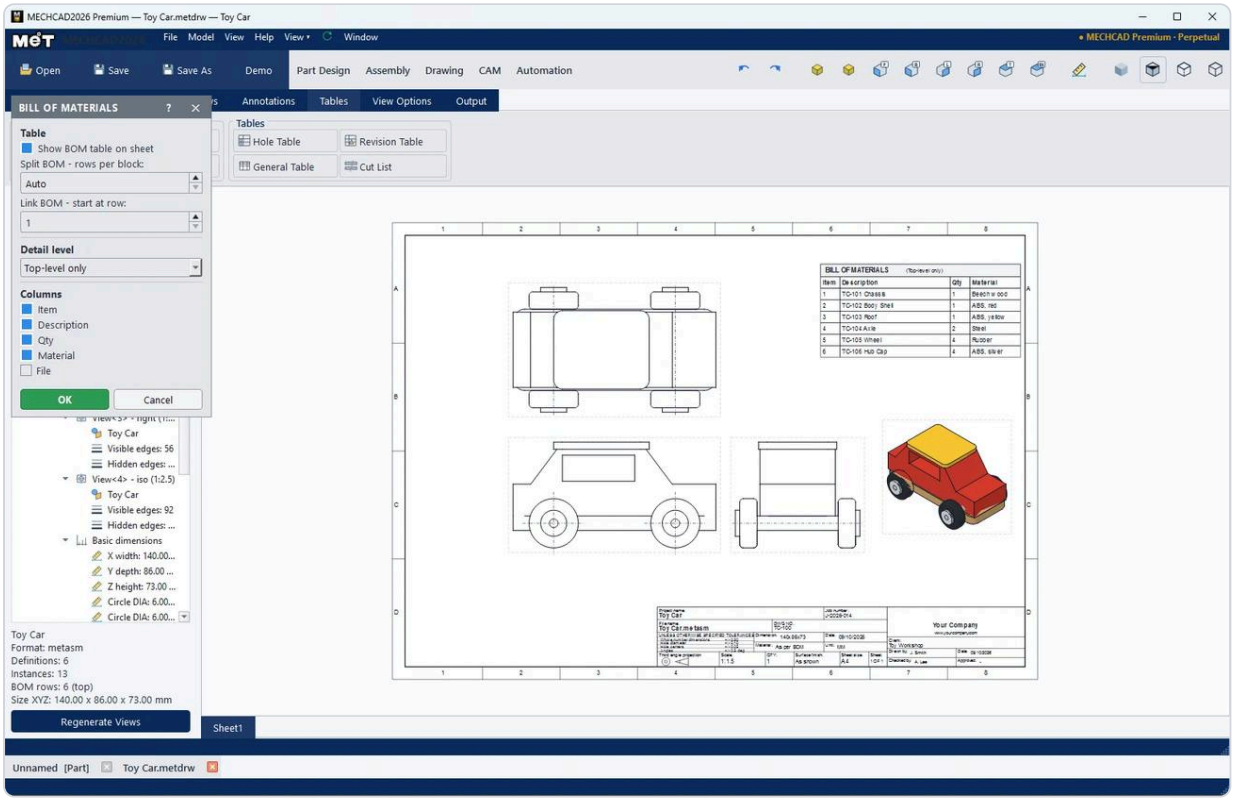
Where to find it

Tables › BOM / Tables



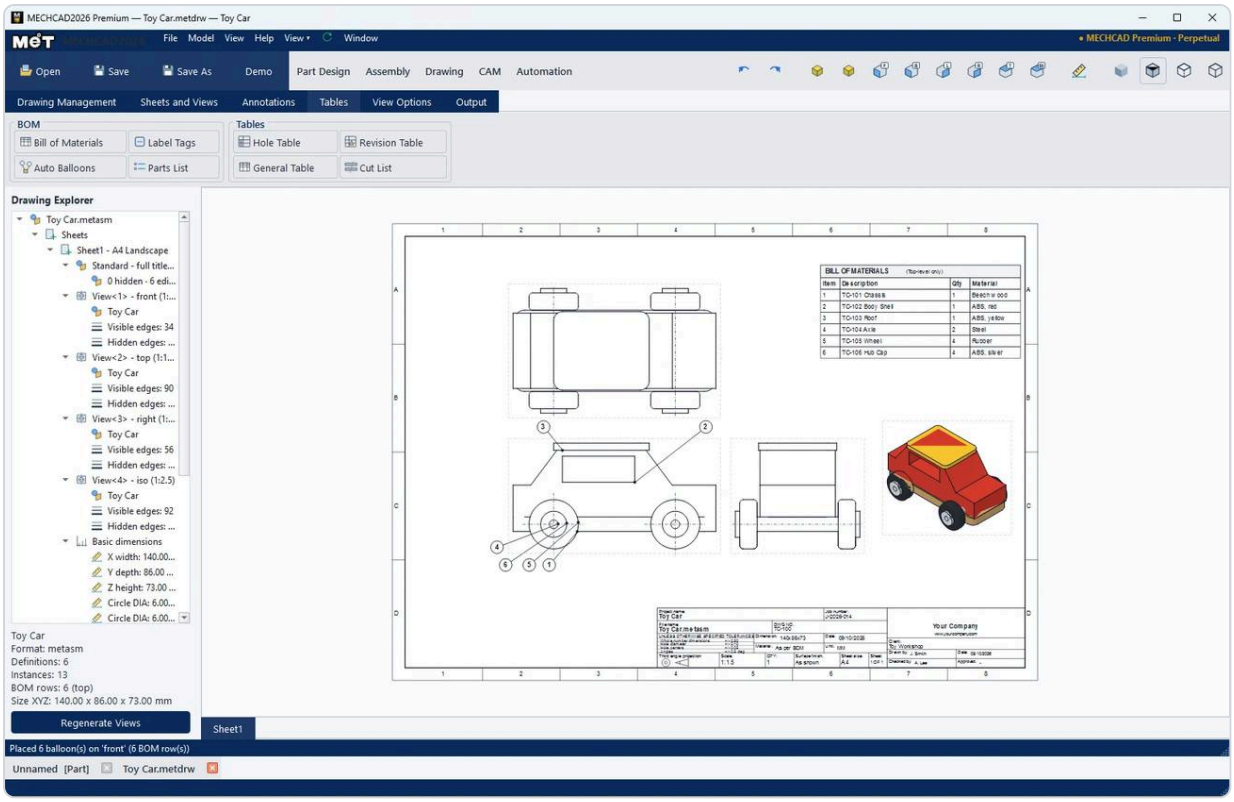
Tables tab

Command	Description
Bill of Materials	BOM settings: show/hide, detail level, columns.
Auto Balloons	Balloons for every BOM item on the Front view (right-click another view to place them on that view).
Label Tags	Part name labels with leader lines.
Parts List	Places/updates the BOM table on the sheet (*BILL OF MATERIALS* at the top right).
Hole Table	Detects holes and places a table of Tag / Ø / Qty / Depth / Pos X,Y.
General Table	An empty editable table.
Revision Table	A REV / DESCRIPTION / DATE / BY table (first row A / FIRST ISSUE / today).
Cut List	An ITEM / PROFILE / PART / LENGTH / QTY table.



BILL OF MATERIALS box

BOM option	Description
Show BOM table on sheet	Shows the table on the sheet.
Detail level	Top-level only / Parts only (flattened) / Indented (all levels)
Columns	Item, Description, Qty, Material, File



BOM table and balloons on the toy car drawing

> **Note:** Double-click a cell in a table to edit it. Right-click the BOM table › **BOM Properties...**, **Detail Level** ›, **Hide BOM Table**.

See also

- Auto BOM (Excel / PDF)

Sheet Properties and Title Block Standard

Choose the paper size and fill in the title block.

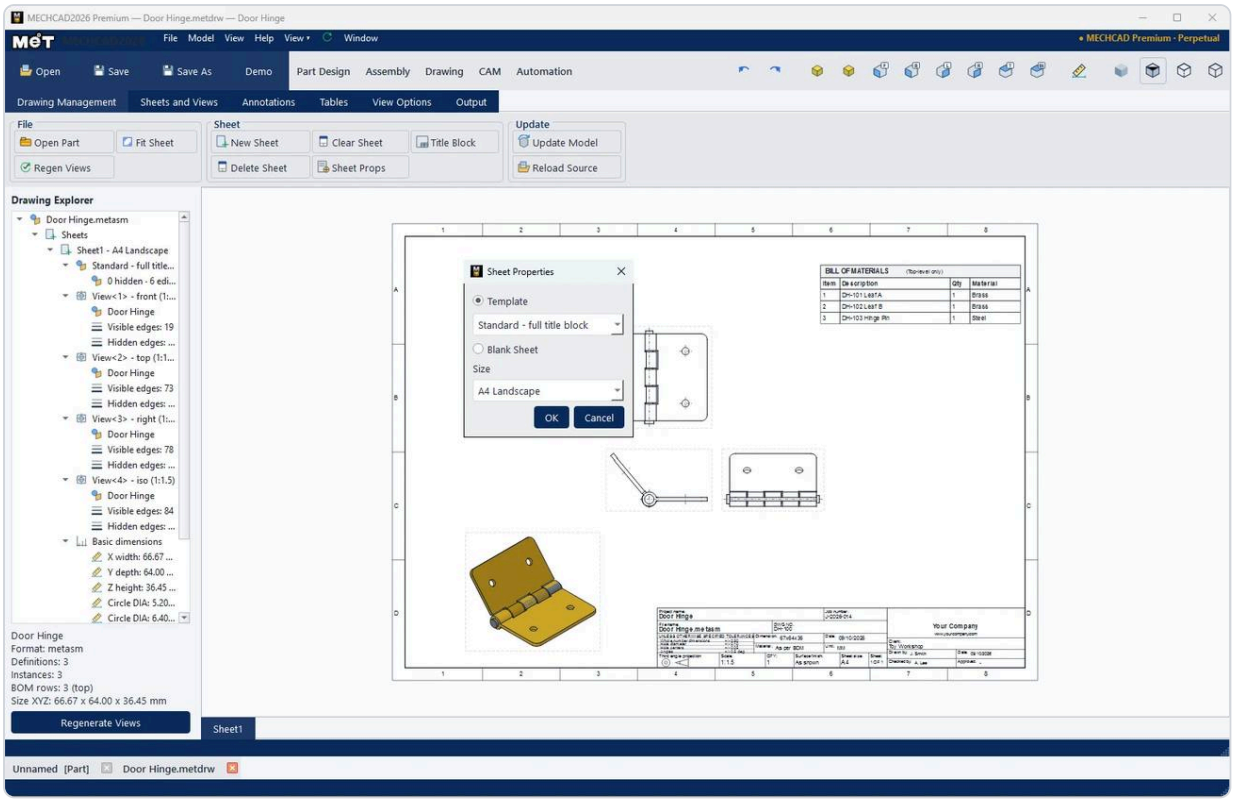
What it does

Sets the sheet template and size, and fills in the title block, which shows your own company name and logo.

Where to find it

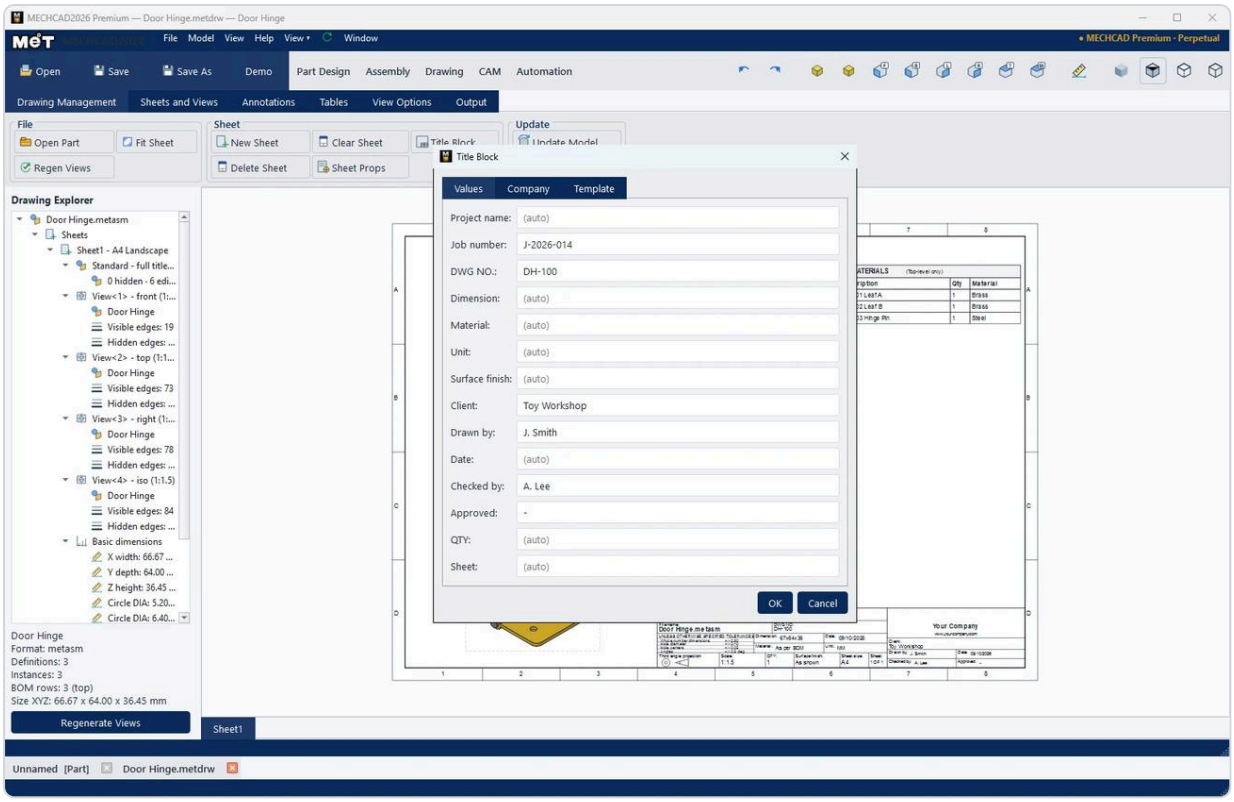
Drawing Management › Sheet › Sheet Props / Title Block / New Sheet

The Sheet Properties window lets you choose a **Template** (for example Standard - full title block) or a **Blank Sheet**, and the **Size** (ISO A0–A4 and ANSI A–E, landscape and portrait). Examples:



Sheet Properties

Sheet size	Size
A4 Landscape	297 × 210 mm
A3 Landscape	420 × 297 mm
A4 Portrait	210 × 297 mm



Title Block window

Field	Description
Project name	Project name (auto = part name)
Job number	Job number
DWG NO.	Drawing number
Dimension	Overall size (auto = X×Y×Z)
Material	Material (auto from the part; assemblies show As per BOM)
Unit / Surface finish	Unit / surface finish
Client / Drawn by / Date / Checked by / Approved	People involved (Drawn by, Checked by, Approved and Client are remembered for the next sheet)
QTY / Sheet	Quantity / sheet number

A blank field shows *(auto)*, meaning the automatic value is used. You can also edit a value by double-clicking the text in the title block.

> **Note:** The default tolerance box in the title block is: whole-number dimensions ±0.50, holes ±0.10, hole centre distance ±0.05, angles ±0.5°. **New Sheet** adds another sheet to the same drawing (choose its template and size); **Clear Sheet** removes every view and annotation from the shown sheet. Set your company name and logo on the **Company** tab of the Title Block window.

See also

- Part Properties

- Export PDF, DXF, DWG

Export PDF, DXF, DWG and Print Standard

Export the drawing as PDF, DXF or DWG, print it, and save .metdrw.

What it does

Exports or prints the current sheet.

Where to find it

Output › Export / Native



Output tab

Command	Result
Export PDF	A4/A3 PDF to match the sheet, in the sheet orientation, with no margin
Export DXF	DXF R2010 in mm (polylines, text)
Export DWG	DWG (AutoCAD 2018) through the DWG converter that comes with the program (nothing extra to install). An installed ODA File Converter is used as a fallback.
Print Preview	Preview before printing (landscape, fit to page)
Print	Prints to a printer
Save .metdrw	Saves the MECHCAD drawing
Pack Refs	Copies the drawing and the part files it refers to into one folder

Steps

1. Go to the **Output** tab.
2. Click an Export command and choose a file name.

> **Note:** If the conversion fails you see *DWG conversion failed* and the destination file is not changed. Try again, or use Export DXF instead.

See also

- File types

Chapter 6 quiz: Drawing

Choose the one best answer. Answers are in Appendix A

1. **Which projection does MECHCAD use to lay out the standard views of a new drawing?**

   A. First-angle
   B. Third-angle
   C. Isometric only
   D. It depends on the part size

2. **A .metdrw file refers to its source part/assembly files. Which command copies the drawing and its referenced files into one folder?**

   A. Save .metdrw
   B. Export PDF
   C. Output › Pack Refs
   D. New Sheet

3. **How do you create a Section View?**

   A. Click the left side of an orthographic view
   B. Click a centre point, then a point on a circle
   C. Select a view and press Delete
   D. Click two points of the cutting line inside a view, then place the section

4. **Which command adds chain and overall dimensions in both directions, plus Ø, to every orthographic view?**

   A. Auto Dimension
   B. Smart Dimension
   C. Ordinate Dim
   D. Hole Callout

5. **Which BOM Detail level lists the parts at all levels of the assembly structure?**

   A. Top-level only
   B. Parts only (flattened)
   C. Indented (all levels)
   D. Cut List

6. **A title block field shows “(auto)”. What does that mean?**

   A. The field is locked
   B. The automatic value is used (e.g. part name, material, overall size)
   C. The value is missing and printing is blocked
   D. The field is copied from the previous drawing

CHAPTER 7

Motion & Analysis

1. Motion Study and Motion Studio

   Record mechanism positions (points), build a motion sequence, play it, and export a video.

2. Play Motion, Spin, Linear Move and video/GIF recording

   Play joint motion, spin a part about an axis, and record the view as a video or GIF.

3. Interference and Clearance Check

   Find components that collide, and components closer together than a set distance, in an assembly.

4. Auto BOM (Excel / PDF with pictures and balloons)

   Create a parts list with pictures, sizes and quantities, and export Excel/PDF with exploded views and balloons.

5. Export HTML (3D web page for review)

   Export an assembly as a single HTML file that you can rotate, measure and explode in a web browser.

6. Automation Workbench (robot cells and machines)

   Lay out robots, cobots, delta robots, conveyors and workpieces, then simulate pick and place.

7. Robot Library

   Browse and insert Cobot, Delta and Palletizer robot models into a cell.

8. Pick & Place Wizard

   Set up robot pick-and-place with pick and place points, speed, and a row/column/layer placement pattern.

9. Motion Study Timeline (Automation)

   A keyframe timeline for motors, linear motion, and showing or hiding parts in a cell.

Motion Study and Motion Studio Expert

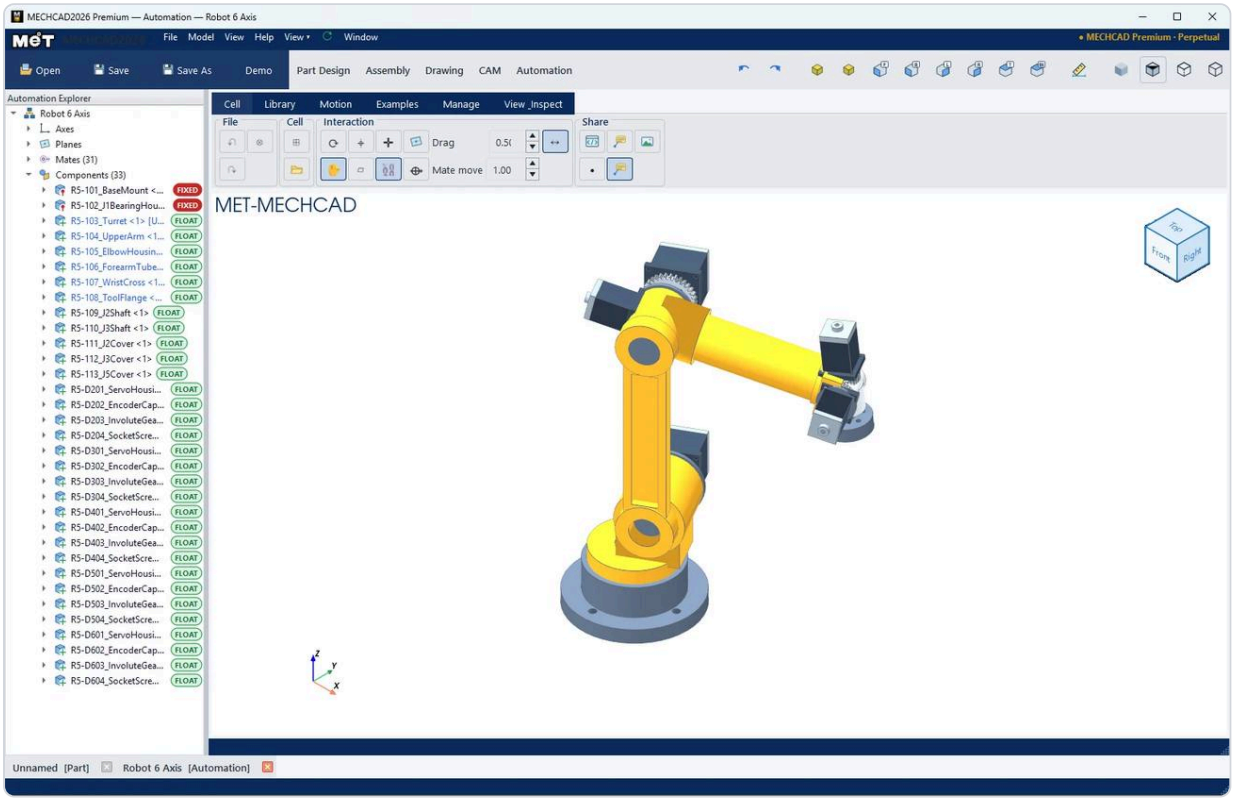
Record mechanism positions (points), build a motion sequence, play it, and export a video.

What it does

Motion Study records assembly poses (for example robot arm positions P1, P2, P3). It then builds a Move / Pick / Place sequence that you can play as an animation and export as an MP4 video.

Where to find it

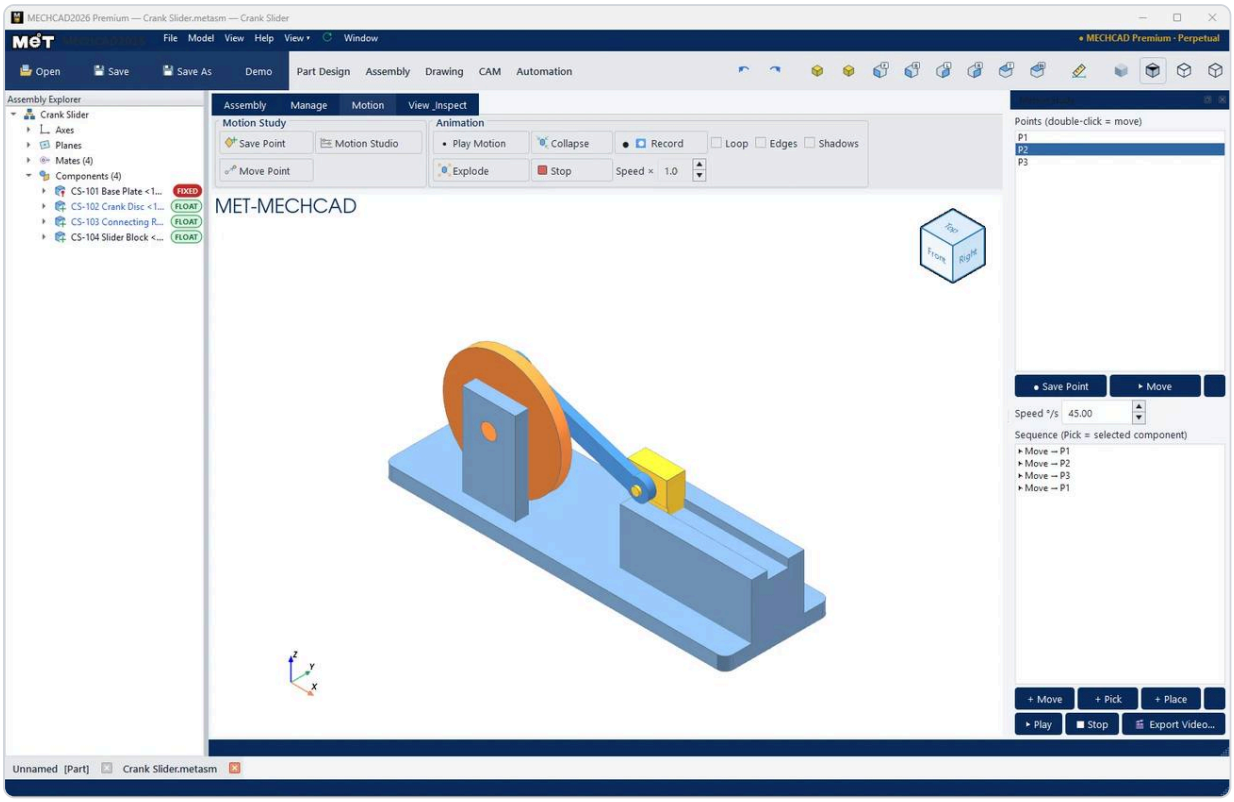
Motion › Motion Study › Save Point / Move Point / Motion Studio



Robot 6 Axis demo

Steps

1. Drag the components until the mechanism is in the pose you want, then click **Save Point** (or ● Save Point in the panel).
2. Repeat to record more points (P1, P2, …). Double-click a point in the list to move to that pose.
3. Add steps to the **Sequence**: **+ Move** (go to a point), **+ Pick** (pick the selected component), **+ Place** (place it).
4. Click ▶ **Play** to play, and ■ **Stop** to stop.
5. Click 🎬 **Export Video…** to save an MP4.



Motion Study panel

Part	Meaning
Points (double-click = move)	List of recorded poses. ✕ = delete.
Speed °/s	Joint speed (5–360, default 45)
Sequence (Pick = selected component)	The motion sequence
▶ Play / ■ Stop	Play / stop
🎬 Export Video...	Export an MP4 video

> **Note: Move Point** on the ribbon asks *Move robot to:*. If there are no points yet, it shows *No saved points yet — Save Point first.*

> **Note:** This command is in MECHCAD **Expert** and **Premium**. On Standard, the status bar shows a message suggesting an upgrade. See MECHCAD editions.

See also

- Animation and video recording
- Automation Workbench

Play Motion, Spin, Linear Move and video/GIF recording Standard

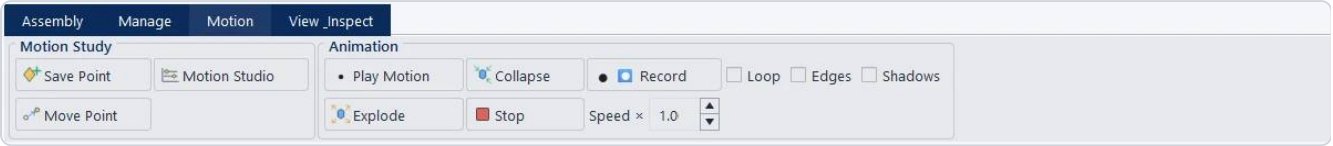
Play joint motion, spin a part about an axis, and record the view as a video or GIF.

What it does

Shows how a mechanism works as an animation.

Where to find it

- **Motion › Animation** › Play Motion, Explode, Collapse, ■ Stop, ● Record
- Right-click a part in the view › **Spin ›** / **Linear Move ›**



Motion tab

Command	Description	Edition
Play Motion	Plays the Motion Study if there is one. If not, it rotates Revolute/Hinge joints or slides parts that have a Distance mate between Minimum and Maximum.	Standard
Spin › Play / Stop	Spins a part about a hole/cylinder or a Mate axis (meshing gears spin at the gear ratio).	Standard
Spin › Reverse / Speed	Reverses direction / sets speed: Slow 30°/s, Medium 90°/s, Fast 180°/s, Very fast 360°/s.	Standard
Spin › Record GIF…	Records the spin as a GIF.	Expert
Linear Move › Play / Reverse / Speed	Slides back and forth between the Minimum and Maximum of a Distance mate (20–250 mm/s).	Standard
● Record	Counts down 3-2-1, then records the view (you can rotate and zoom while recording). Click ■ Stop to save an MP4.	Expert
Explode / Collapse	See Explode.	Standard

> **Note:** Spin does not work if the part has no axis (*no spin axis - needs a bore/cylindrical face or a concentric/revolute/cylindrical mate*) or if it is fixed (*part is Fixed - Unfix it first*). Exported videos and HTML pages carry the MET logo and copyright.

See also

- Motion Study
- Explode and Collapse

Interference and Clearance Check Expert

Find components that collide, and components closer together than a set distance, in an assembly.

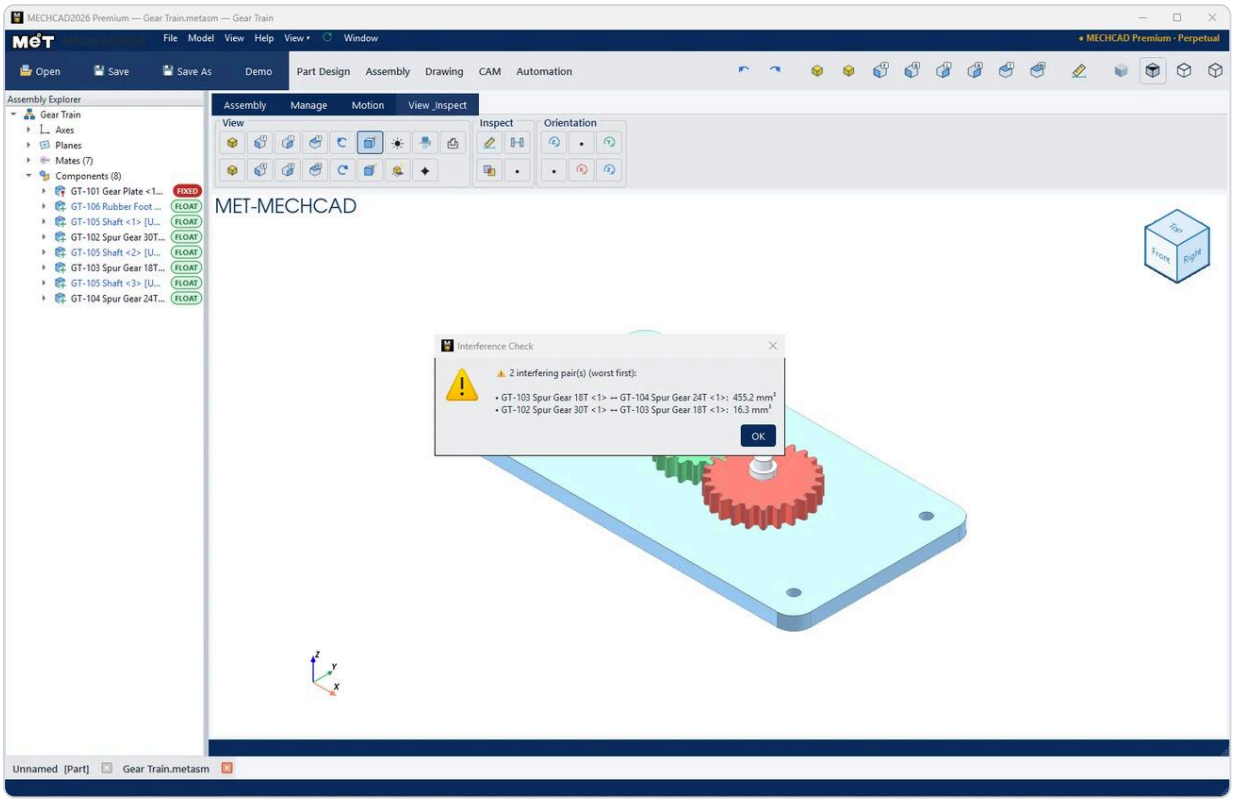
What it does

- **Interference** finds pairs of components that overlap, with the volume of the overlap.

- **Clearance** finds pairs of components that are closer together than a value you set.

Where to find it

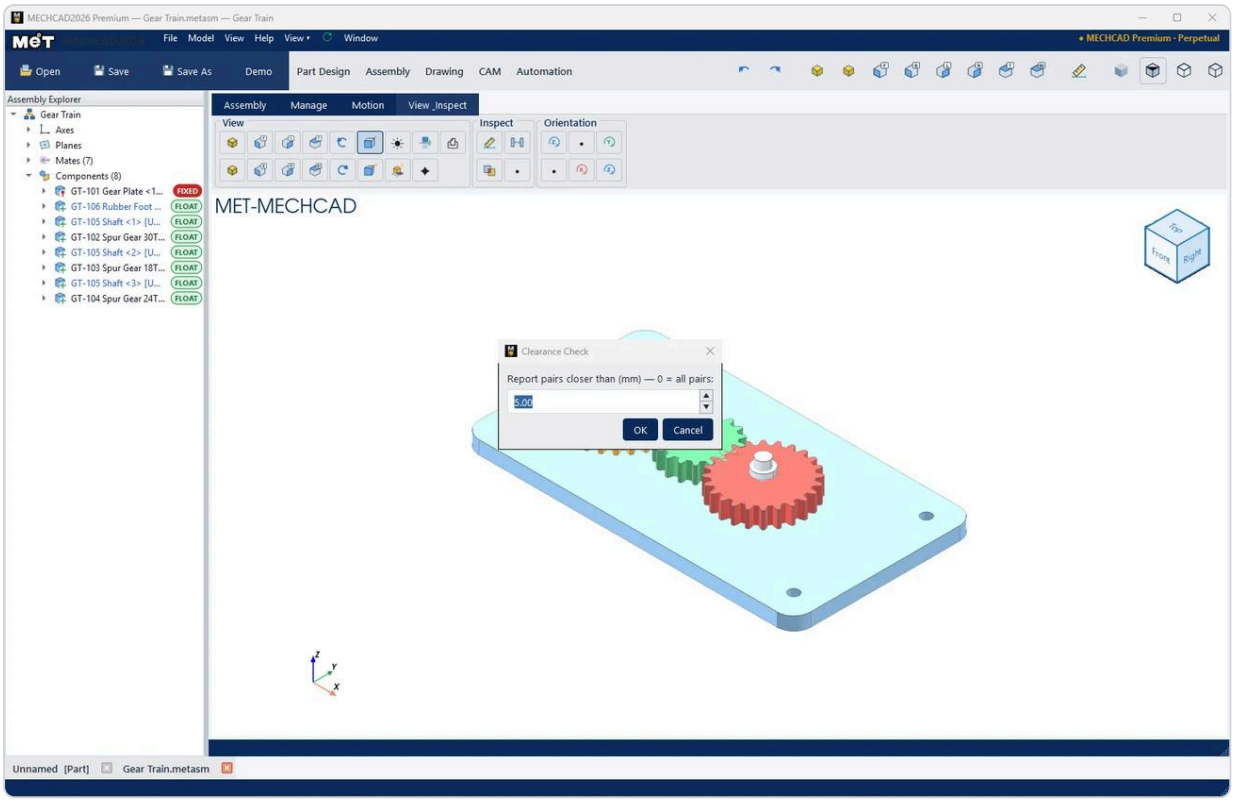
View & Inspect › Inspect › Interference / Clearance



Interference Check result

Steps

1. Click **Interference** and wait for the calculation.
2. The result shows *No interferences found between components.* or a list of *A ↔ B: V mm³*, sorted from the largest overlap (up to 30 pairs).
3. Click **Clearance** and enter the distance in *Report pairs closer than (mm) — 0 = all pairs* (default 5). Click OK to see the closest pairs.



Set the clearance distance

> **Note:** This command is in MECHCAD **Expert** and **Premium**. On Standard, the status bar shows a message suggesting an upgrade. See MECHCAD editions.

See also

- Measure

Auto BOM (Excel / PDF with pictures and balloons) Expert

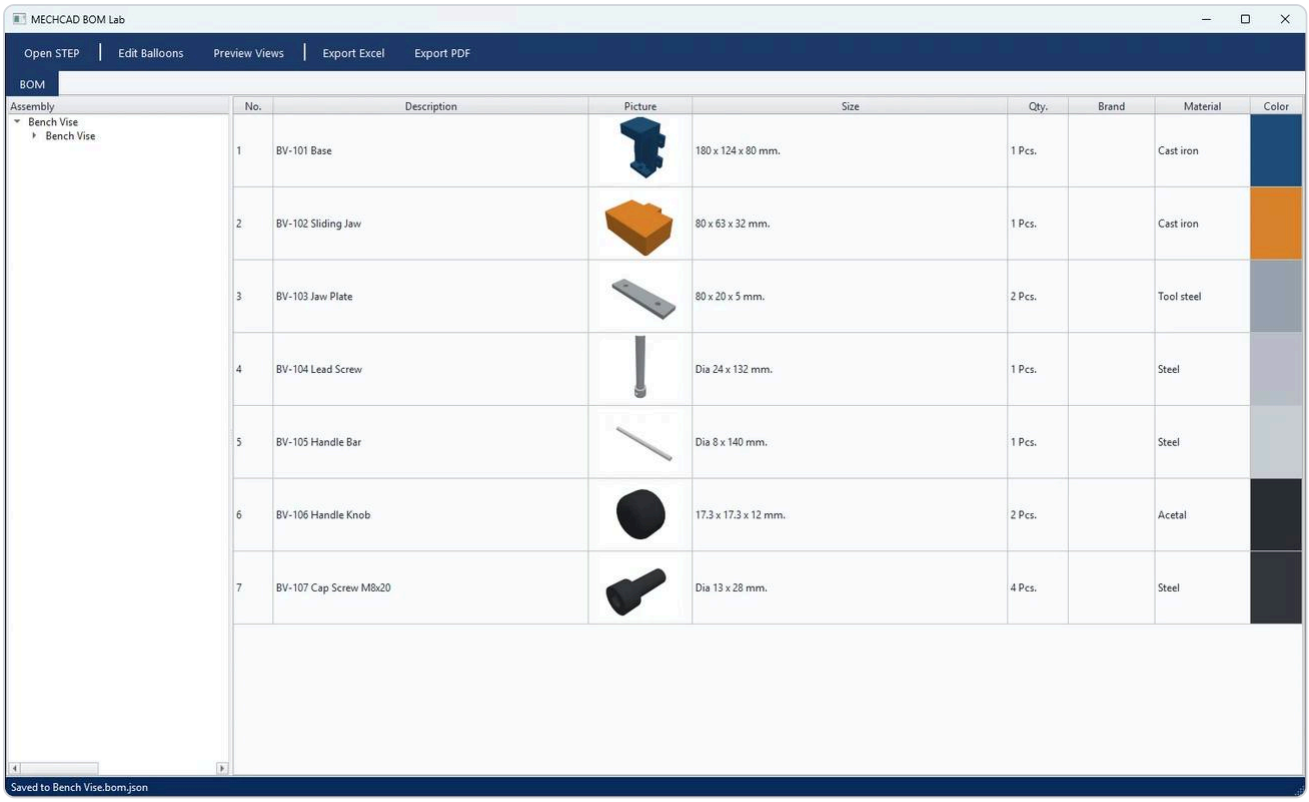
Create a parts list with pictures, sizes and quantities, and export Excel/PDF with exploded views and balloons.

What it does

Auto BOM opens the **MECHCAD BOM Lab** window. It builds a parts list from the open assembly, with pictures, sizes and quantities. You can add brand, material and colour, then export Excel or PDF with an exploded view and balloons.

Where to find it

Manage › Assembly Tools › Auto BOM



MECHCAD BOM Lab window

Steps

1. Open an assembly, then click **Auto BOM**.
2. Check the list. You can type **Brand** and **Material** directly in the table. Double-click **Color** to pick a colour.
3. Click **Edit Balloons** to arrange the exploded view and balloon positions, then **Preview Views**.
4. Click **Export Excel** (`.xlsx`) or **Export PDF** (`.pdf`).

Item	Description
Open STEP...	Build a BOM directly from a STEP file.
Assembly / Qty	Structure tree with quantities.
No., Description, Picture, Size, Qty., Brand, Material, Color	Table columns.
Edit Balloons	Exploded-view editor: Move Parts, Move Balloon, Fix View, Explode slider 0–300%, Reset View, Reset Balloons, Save / Cancel.
Status bar	Summary of the number of unique part sections and the total count.

> **Note:** This command is in MECHCAD **Expert** and **Premium**. On Standard, the status bar shows a message suggesting an upgrade. See MECHCAD Editions.

See also

- BOM and tables in a Drawing

Export HTML (3D web page for review) Expert

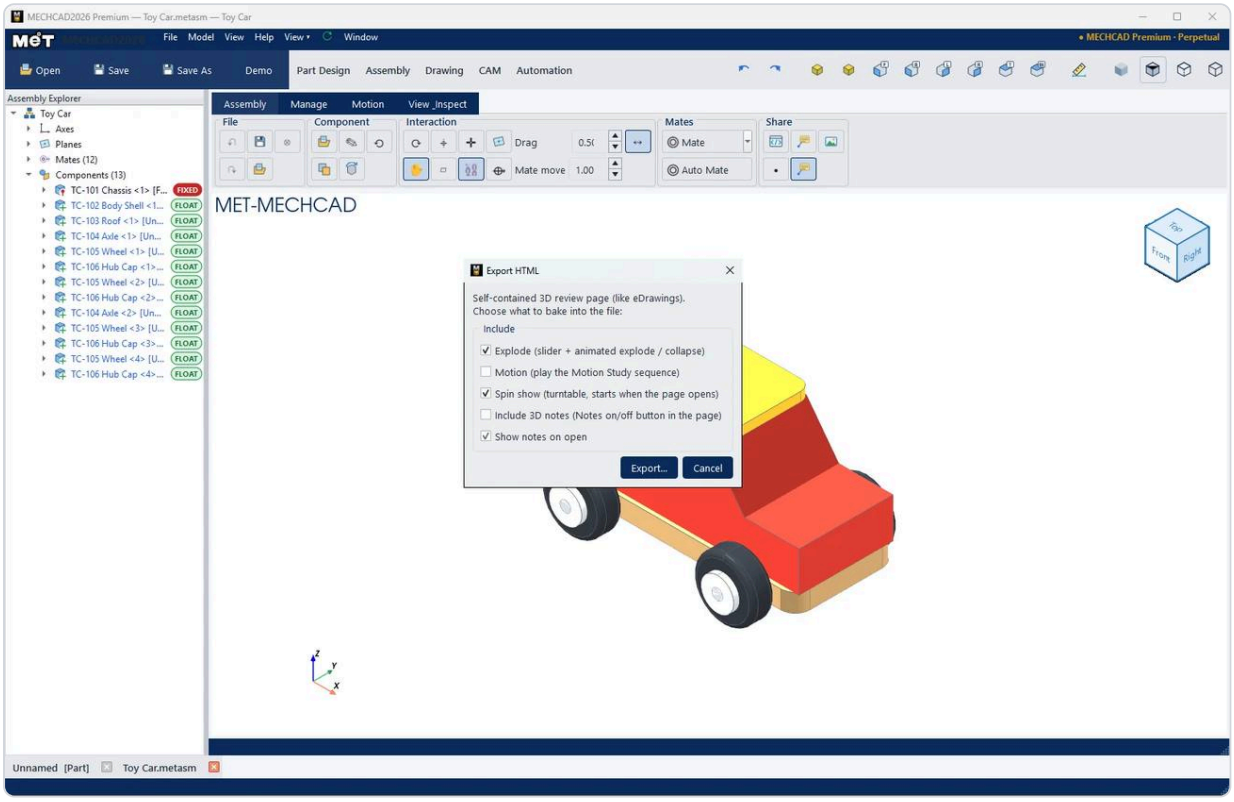
Export an assembly as a single HTML file that you can rotate, measure and explode in a web browser.

What it does

Creates a single HTML page (similar to eDrawings) so customers or your team can view the 3D model in a web browser without installing any software. It has measuring tools (edge snap, angle, midpoint, hole centre), an assembly tree, Undo/Redo and colour themes.

Where to find it

Assembly › Share › Export HTML



Export HTML window

Steps

1. Click **Export HTML**.
2. Choose what to include in the file (see the table).
3. Click **Export...** and choose a file name in the *Export 3D Assembly* window. Wait for the progress bar (you can click Cancel).

Option	Description
Explode (slider + animated explode / collapse)	A slider and an Explode button in the web page
Motion (play the Motion Study sequence)	Plays the Motion Study (available when motion data exists)
Spin show (turntable, starts when the page opens)	Spins the model automatically when the page opens

Option	Description
Include 3D notes (Notes on/off button in the page)	Includes 3D Notes with a button to turn them on/off
Show notes on open	Shows the notes as soon as the page opens

> **Note:** This command is in MECHCAD **Expert** and **Premium**. On Standard, the status bar shows a message suggesting an upgrade. See MECHCAD Editions.

See also

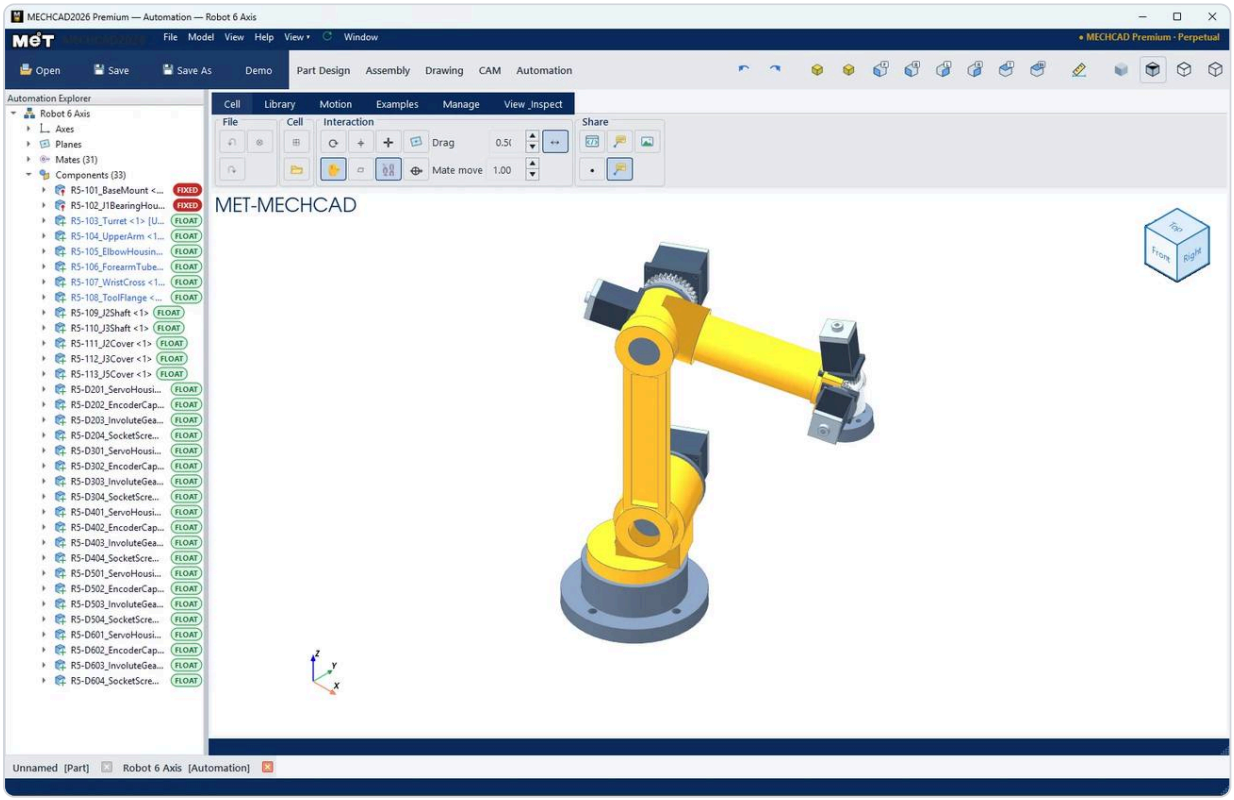
- 3D Notes and Picture
- Export STEP / STL

Automation Workbench (robot cells and machines) Expert

Lay out robots, cobots, delta robots, conveyors and workpieces, then simulate pick and place.

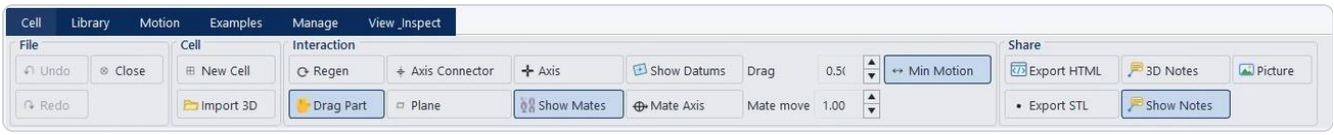
What it does

Use the **Automation** tab to build an automation cell: import robots, cobots, delta robots, equipment and workpieces, define pick and place points, and simulate the motion. Cells are saved as `.metauto`.



Robot 6 Axis demo

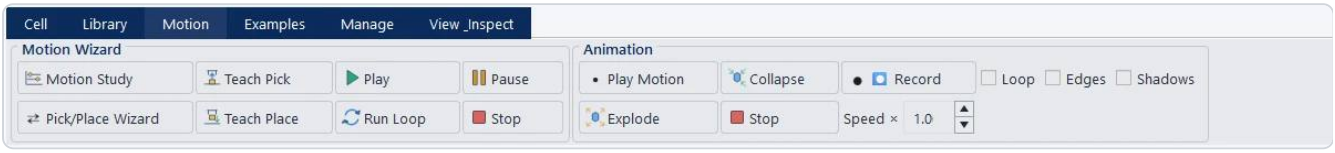
Ribbon



Cell tab

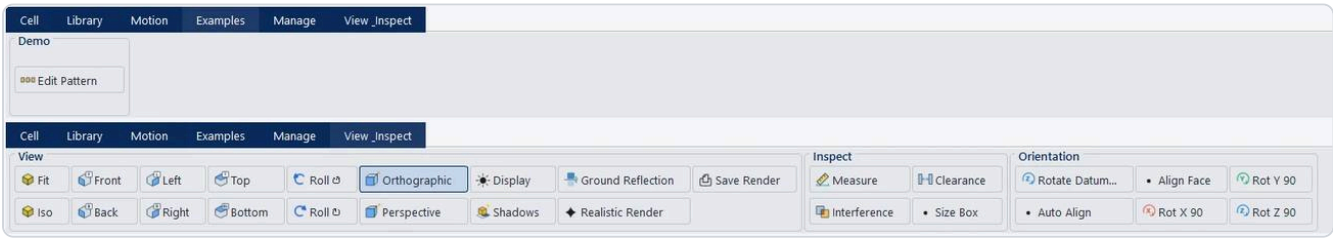


Library tab



Motion tab

Tab › Group	Commands
Cell › File	Undo, Redo, Close (save / open with Save / Open on the top bar)
Cell › Cell	**New Cell**, **Import 3D** (asks for the role: Robot / Cobot / Delta / Tool / Equipment / Workpiece)
Cell › Interaction	Same as Assembly: Regen, Drag Part, Axis Connector, Plane, Axis, Show Mates, Show Datums, Mate Axis, Drag, Mate move, Min Motion
Library › Robot	Robot Library, Cobot Library, Delta Library
Library › Equipment & Product	Conveyor, Actuator, Box, Bottle, Object
Motion › Motion Wizard	Motion Study, Pick/Place Wizard, Teach Pick, Teach Place, Play, Run Loop, Pause, Stop
Motion › Animation	Play Motion, Explode, Collapse, ■ Stop, ● Record
Examples › Demo	Edit Pattern
Manage / View & Inspect	Component State, Share (Export HTML/STL, 3D Notes, Picture), View, Inspect (Measure, Interference, Clearance)



Examples and View & Inspect tabs

Workflow

1. **New Cell**.
2. Import a robot from the Robot Library or use **Import 3D**.
3. Add conveyors, boxes and bottles from **Equipment & Product**.

4. Use the Pick/Place Wizard or **Teach Pick / Teach Place** (select a point, then click; the status bar shows *Pick taught at X… Y… Z… mm*).
5. Click **Play** or **Run Loop** (*Automation RUN LOOP active - click Stop to finish*), then **Stop**.

> **Note:** This workbench is in MECHCAD **Expert** and **Premium**. On Standard, the tab shows a card suggesting an upgrade. See MECHCAD Editions.

See also

- Robot Library
- Pick & Place Wizard
- Motion Study Timeline

Robot Library Expert

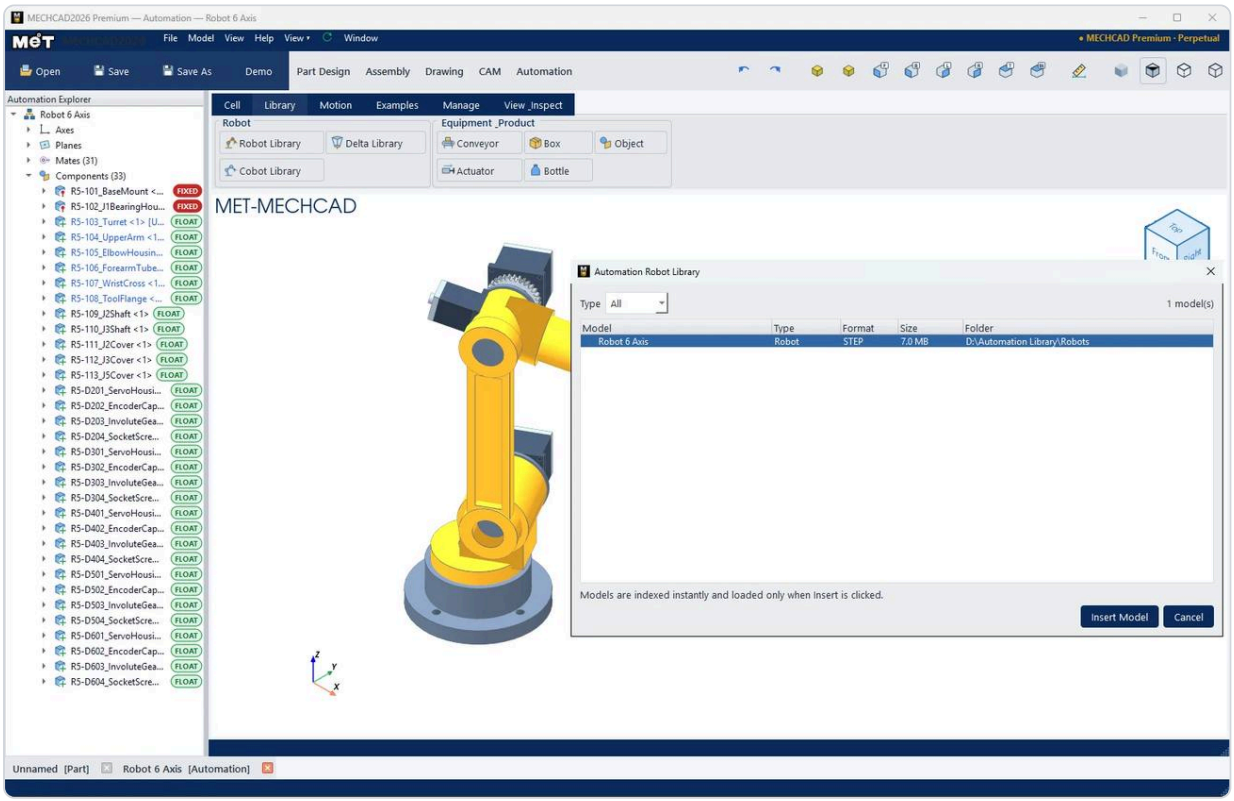
Browse and insert Cobot, Delta and Palletizer robot models into a cell.

What it does

The **Automation Robot Library** window lists the robot models on your computer. You can insert one into the cell right away (a model loads only when you click Insert).

Where to find it

Library › Robot › Robot Library / Cobot Library / Delta Library



Automation Robot Library window

Part	Meaning
Type	Filter: All / Robot / Cobot / Delta / Palletizer
Model, Type, Format, Size, Folder	List columns
Insert Model	Inserts the selected model

The program searches these folders:

- The folder in the `MECHCAD_ROBOT_LIBRARY` environment variable
- `<install folder>\Automation Library\Robots`
- The models that come with the program

Supported file types: STEP/STP, IGES/IGS, BREP, X_T/X_B, METASM, METPRT

> **Note:** If the list is empty (0 model(s)), put robot model files (for example STEP files from the manufacturer) in the folders above.

> **Note:** This command is in MECHCAD **Expert** and **Premium**. On Standard, the status bar shows a message suggesting an upgrade. See MECHCAD editions.

See also

- Automation Workbench

Pick & Place Wizard Expert

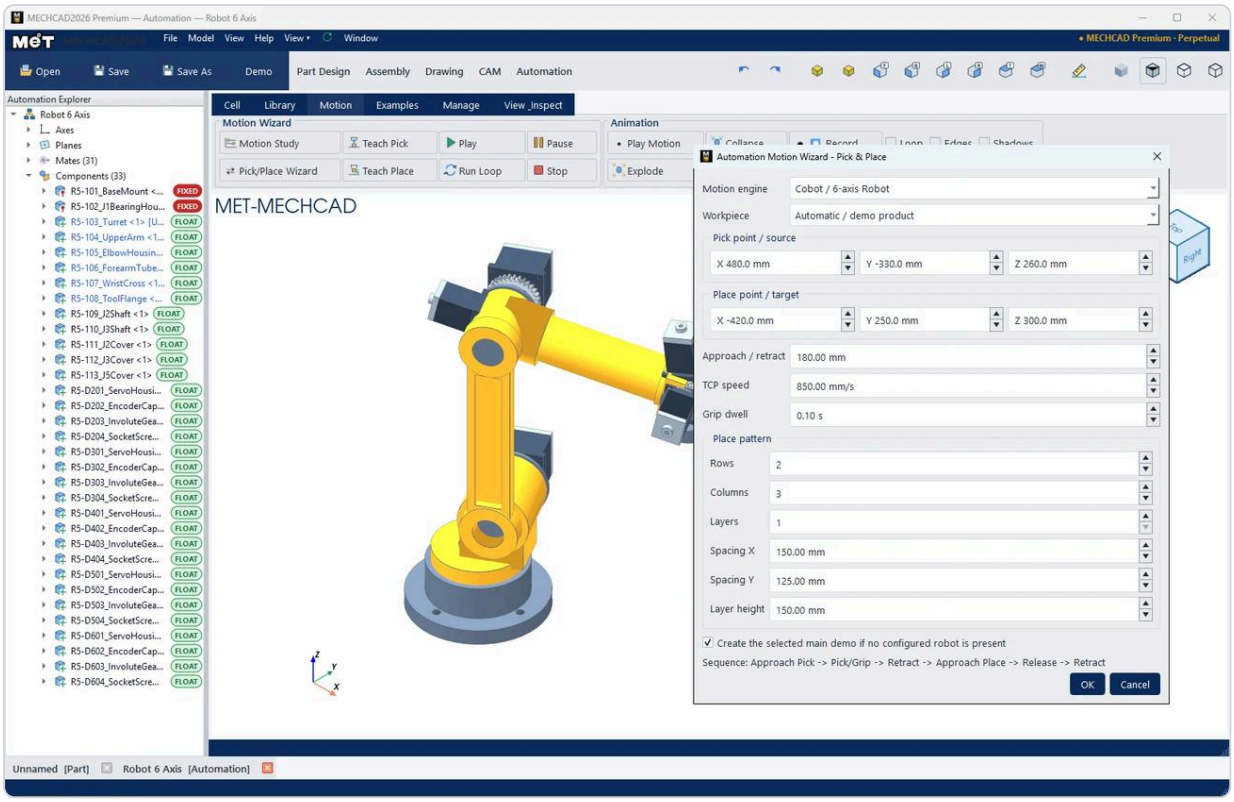
Set up robot pick-and-place with pick and place points, speed, and a row/column/layer placement pattern.

What it does

Creates an automatic pick-and-place sequence: *Approach Pick → Pick/Grip → Retract → Approach Place → Release → Retract*, with a placement pattern of rows, columns and layers.

Where to find it

Motion › Motion Wizard › Pick/Place Wizard



Automation Motion Wizard - Pick & Place window

Option	Meaning
Motion engine	Cobot / 6-axis Robot or Delta Robot
Workpiece	Automatic / demo product, or a component in the cell
Pick point / source (X, Y, Z)	Pick point (mm)
Place point / target (X, Y, Z)	Place point (mm)
Approach / retract	Approach and retract distance (mm)
TCP speed	Tool tip speed (mm/s)
Grip dwell	Pause time while gripping (s)
Place pattern: Rows, Columns, Layers, Spacing X, Spacing Y, Layer height	Placement pattern
Create the selected main demo if no configured robot is present	Creates a sample robot if the cell has none

Steps

1. Click **Pick/Place Wizard**.
2. Choose the Motion engine and Workpiece.
3. Enter the pick/place points (or use **Teach Pick** / **Teach Place** first).
4. Set the placement pattern, then click **OK**.
5. Click **Play** or **Run Loop**.

> **Note:** For a Delta Robot, if a point is outside the reach you see the warning *Motion Wizard - Reachability*.

> **Note:** This command is in MECHCAD **Expert** and **Premium**. On Standard, the status bar shows a message suggesting an upgrade. See MECHCAD editions.

See also

- Automation Workbench
- Motion Study Timeline

Motion Study Timeline (Automation) Expert

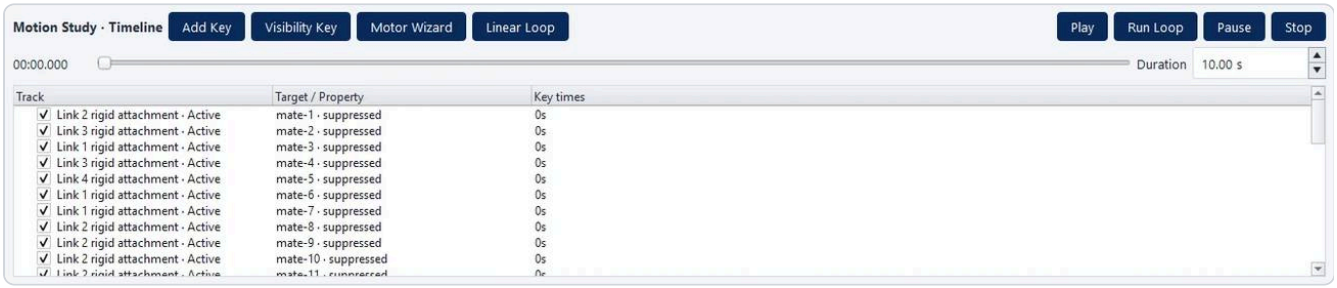
A keyframe timeline for motors, linear motion, and showing or hiding parts in a cell.

What it does

The **Motion Study · Timeline** panel on the Automation tab builds motion over time. You can add keyframes, rotary motors, back-and-forth moves, and show or hide parts.

Where to find it

Motion › Motion Wizard › Motion Study



Motion Study · Timeline panel

Part	Meaning
Add Key	Adds a keyframe from the current position
Visibility Key	A key that shows or hides a part
Motor Wizard	The *Rotary Motor Wizard* window creates continuous rotation
Linear Loop	The *Linear Loop Move Wizard* window creates back-and-forth moves
Play / Run Loop / Pause / Stop	Playback controls
Time 00:00.000 and the time bar	Current time position
Duration	Total length (seconds)
Track / Target / Property / Key times	List of tracks

> **Note:** This command is in MECHCAD **Expert** and **Premium**. On Standard, the status bar shows a message suggesting an upgrade. See MECHCAD editions.

See also

- Motion Study and Motion Studio

Chapter 7 quiz: Motion & Analysis

Choose the one best answer. Answers are in Appendix A

1. **In Motion Study, how do you record a pose of the mechanism?**

   A. Drag the components into the pose, then click Save Point
   B. Click ● Record
   C. Use Explode
   D. Add a mate with Distance

2. **You click ● Record, rotate the view, then click ■ Stop. What file is saved?**

   A. GIF
   B. AVI
   C. MP4
   D. HTML

3. **In Clearance, what does entering 0 in "Report pairs closer than (mm)" do?**

   A. Reports only overlapping pairs
   B. Reports all pairs
   C. Cancels the check
   D. Uses the default of 5 mm

4. **Which formats can Auto BOM (MECHCAD BOM Lab) export?**

   A. CSV and TXT
   B. DXF and DWG
   C. STEP and STL
   D. Excel (.xlsx) and PDF

5. **What is Export HTML for?**

   A. Publishing the assembly to the MET website
   B. Creating a 2D drawing
   C. A single web page so others can view and measure the 3D model in a browser without installing software
   D. Converting the assembly to STEP

6. **Which Editions include the Interference and Clearance checks?**

   A. Expert and Premium
   B. Standard only
   C. All editions
   D. Premium only

CHAPTER 8

CAM

1. CAM overview
   Create 2.5D / 3D milling and turning toolpaths from a part, simulate the cut, and generate G-code.
2. New Job, Template and Geometry
   Start a new CAM job from a milling or turning template and define the workpiece.
3. Stock, Frame (WCS) and Clearances
   Define the raw stock, the G54–G59 work zero and the safe planes.
4. Tool Library and tool settings
   Choose tools from the library and set holder size, spindle speed and default feeds.
5. Operation Parameters
   What the sections of the operation box and the General, Geometry, Tool, Levels, Passes, Linking and Stock pages mean, including Tool Compensation (D-offset).
6. Face and Pocket (2.5D)
   Face a surface and mill pockets with 2.5-axis operations.
7. Profile and Ramp Cut
   Mill along outer or inner edges with cutter compensation, and ramp down along an edge.
8. Drill and Hole
   Drill holes at multiple positions with Smart Hole (tick holes one by one, drill chosen automatically) and G81/G83 cycles.
9. Slot, Step, Chamfer, Corner Round, Engrave and Thread Mill
   More 2.5D operations: slots, steps, chamfers, corner rounds, engraving and internal thread milling.
10. 3D milling: Rough, Rest, Z-Level, Raster, Pencil and Mold Tactic
    Rough and finish 3D mold surfaces.
11. Turning: Face, Rough, Finish, Axial Drill and Part Off
    Create facing, G71 rough turning, G70 finish turning, axial drilling and part-off operations.
12. Turning: Groove and Thread (G75 / G76)
    Turn grooves and M3–M64 metric threads with G75 and G76 cycles.
13. Simulation (machining simulation and collision check)
    Simulate material removal, check collisions and cutting time, and step through the G-code line by line.
14. Machine Manager and post-processors

Choose the machine, post-processor and travel limits, and edit the G-code format.

15. Generate G-code and Shop PDF (setup sheet)

    Export G-code and print a setup sheet for the machine operator.

16. Sample CAM jobs (Mill Demo / Lathe Demo)

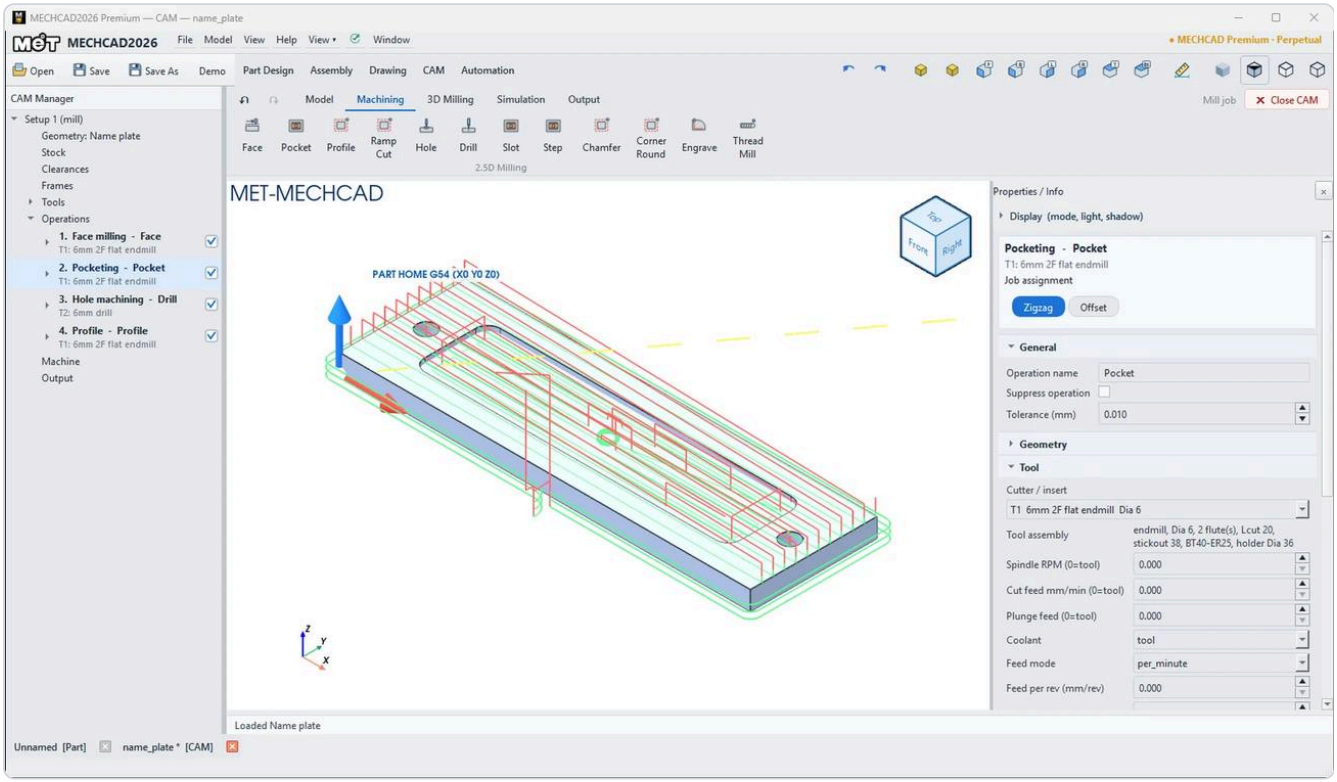
    Five sample milling jobs and five sample turning jobs that cover every operation.

CAM overview Premium

Create 2.5D / 3D milling and turning toolpaths from a part, simulate the cut, and generate G-code.

What it does

The **CAM** tab creates toolpaths for 3-axis CNC mills and lathes from a part in Part Design or from a STEP/IGES/STL file. It simulates material removal with collision checking, then generates G-code and a setup sheet. Jobs are saved as `.metcam`.



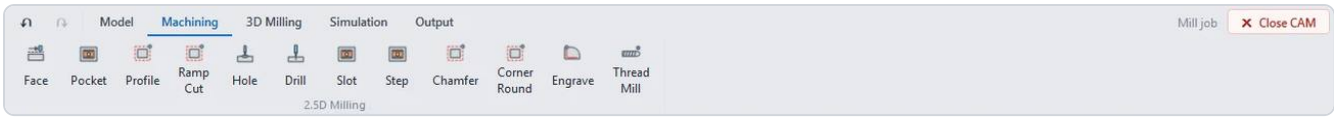
CAM tab with a four-operation name plate job open

Parts of the window

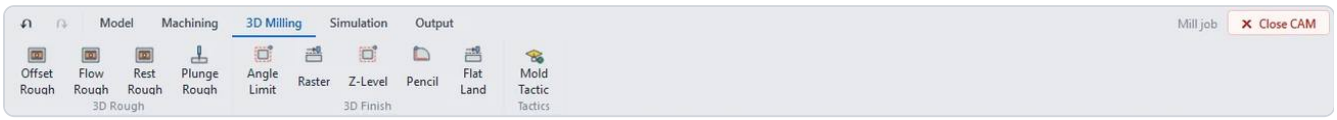
Part	Description
CAM Manager (left)	Setup 1 (mill/lathe): Geometry, Stock, Clearances, Frames, Tools, Operations, Machine, Output
Ribbon	Model, Machining, 3D Milling, Simulation, Output
Status badge (top right)	*Mill job* / *Lathe job* (sample jobs: *Demo (read-only) - use Save As*)
✕ Close CAM	Closes the CAM job (asks to save) and clears the toolpaths
Properties (right)	Settings for whatever is selected in the CAM Manager



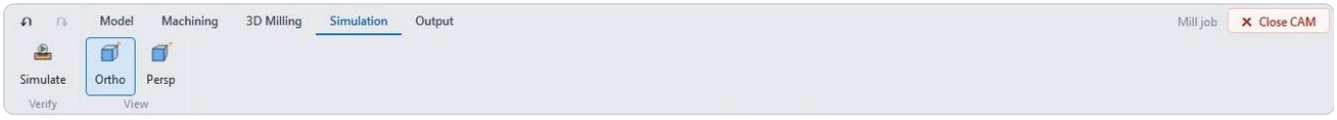
Model tab: Job, Part, Setup, Tools, Learn



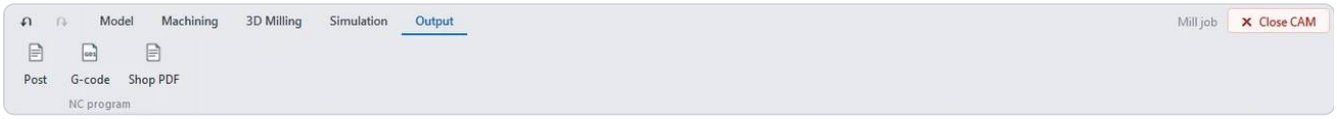
Machining tab: Turning (lathe) / 2.5D Milling (mill)



3D Milling tab: 3D Rough, 3D Finish, Tactics



Simulation tab: Simulate, Ortho, Persp



Output tab: Post, G-code, Shop PDF

Workflow

1. New Job. Choose a Mill or Turn template.
2. Check the Stock, Frame (WCS) and Clearances.
3. Add tools from the Tool Library.
4. Add operations: Face/Pocket, Profile, Drill, 3D, Turning.
5. Simulate.
6. Generate G-code and Shop PDF.

> **Note:** A lathe job hides the 3D Milling tab and the 2.5D group. A mill job hides the Turning group. If the window is narrow, buttons that do not fit move into the » menu. CAM Undo/Redo use Ctrl+Z / Ctrl+Y.

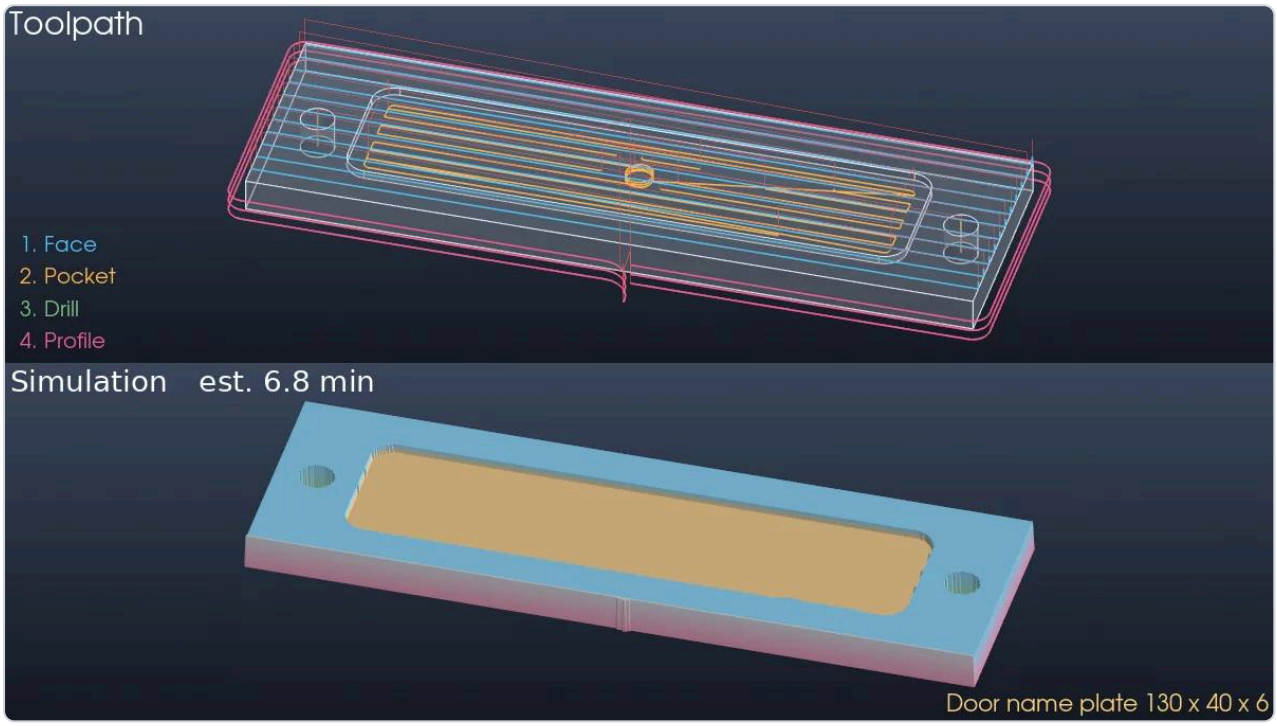
> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Try it: Mill a door name plate from start to finish

A 130 × 40 mm aluminium name plate with a recessed panel and two screw holes takes you through the whole CAM workflow, from model to G-code, in about ten minutes.

1. In **Part Design**, sketch a 130 × 40 rectangle on **XY** starting at the origin and **Extrude** it 6 mm.
2. Sketch on the top face: a 100 × 24 rectangle 15 mm from the left edge and 8 mm from the front edge, with R4 corners. **Extrude Cut** it 1.5 mm. Then add two Ø6 circles at X 7.5 and X 122.5 (Y 20) and **Extrude Cut** them **Through All**.
3. Open the **CAM** tab › **New Job** › **Mill Template** › **OK**. The stock becomes 132 × 42 × 6.5 mm.
4. Click **Face**, click the top face and press **Enter**.

5. Click **Pocket**, click the floor of the recessed panel and press **Enter**. Top Z 6 and Floor Z 4.5 are read from the model.
6. Click **Drill**, click both hole walls and press **Enter**. The T2 6 mm drill matches the holes.
7. Click **Profile**, click a bottom outer edge of the plate and press **Enter**. In **Properties › Levels**, set **Bottom / Floor Z** to 0 so the plate is cut free.
8. Click **Simulate**, then **Play**. The estimated time is about 6.8 min. Finish with **Output › G-code** and save `name_plate.nc` (FANUC_MET 3Axis post, about 240 lines).



Name plate: Face, Pocket, Drill and Profile toolpaths on the left, the simulated plate on the right

Challenge: Change the Face cutter to **T3 12mm 3F face mill** (**Properties › Tool › Cutter / insert**) and Simulate again. How much shorter is the estimated time?

See also

- Sample CAM jobs

New Job, Template and Geometry Premium

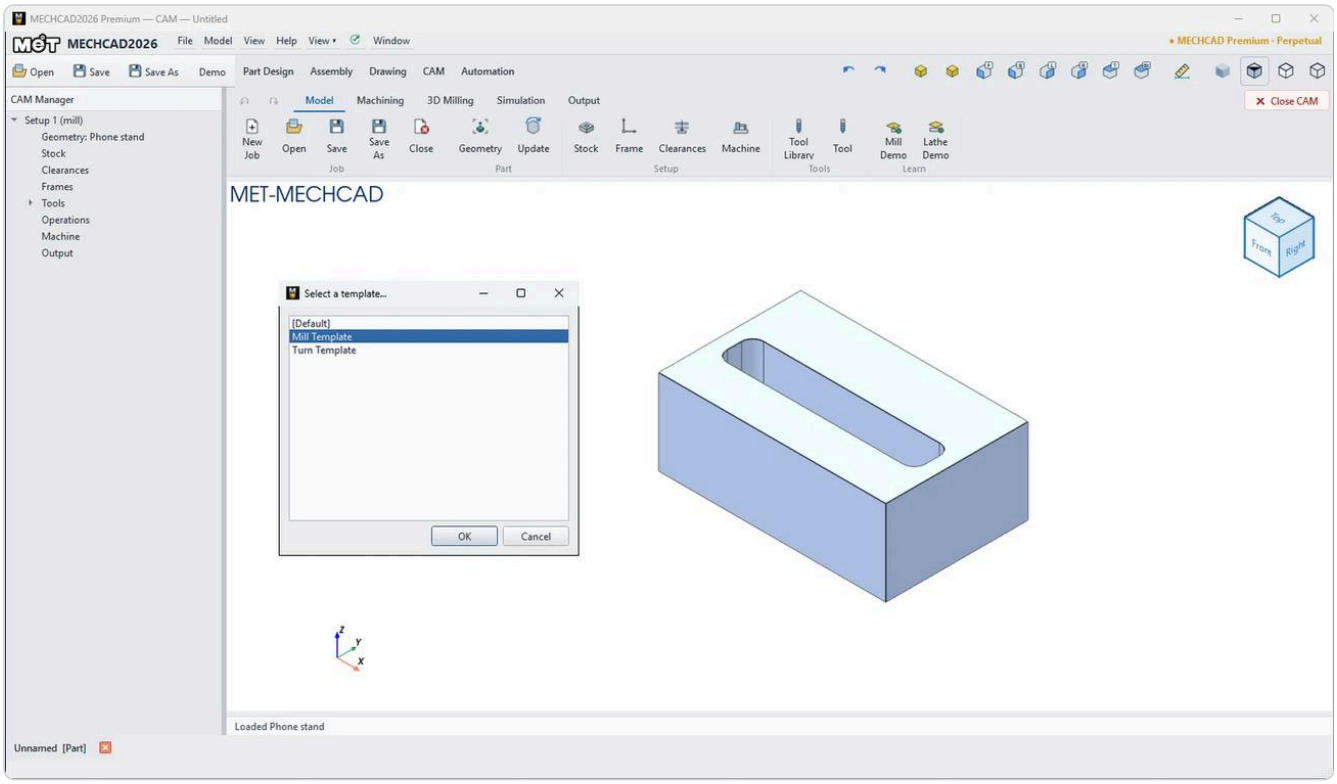
Start a new CAM job from a milling or turning template and define the workpiece.

What it does

Starts a new CAM job, chooses the machine type, and defines the workpiece to cut.

Where to find it

Model › Job › New Job / Open / Save / Save As / Close and **Model › Part › Geometry / Update**



Select a template... window

Steps

1. Click **New Job**. If a job is already open, you are asked *Discard the current operations and start a new job?*
2. Choose a template, then click **OK**.
3. The part from the open Part Design is used automatically. If there is none, click **Geometry** and choose a file (*CAD (*.step *.stp *.igs *.iges *.stl *.metprt)*).
4. Save with **Save** (`.metcam`).

Template	Settings applied
[Default] / Mill Template	Box stock exactly the size of the part bounding box, no allowance (**Part (default)**; the Stock node shows for example *Part box 60 x 40 x 20*), Clearance Z = top surface + 15, Frame G54, Post FANUC_MET 3Axis, machine 3-Axis M.C. Vertical, extension `.nc`
Turn Template	Round bar stock (**Standard bar (default)**) Ø = the next standard bar size that leaves at least 1 mm on the diameter (at most part Ø + 4), length = gripping length + part length + 2 (face allowance), machine Lathe Horizontal, Post fanuc_lathe, 12-station turret

Command	Description
Open	Opens a `.metcam` file (the original part file must exist).
Save / Save As	Saves a `.metcam` file (a sample job must be saved to another folder with Save As).
Close	Closes the job (asks Save / Discard / Cancel).
Geometry	Select/import the workpiece.
Update Geometry	Reloads the original part. Stock and operations are kept, but toolpaths must be recalculated.

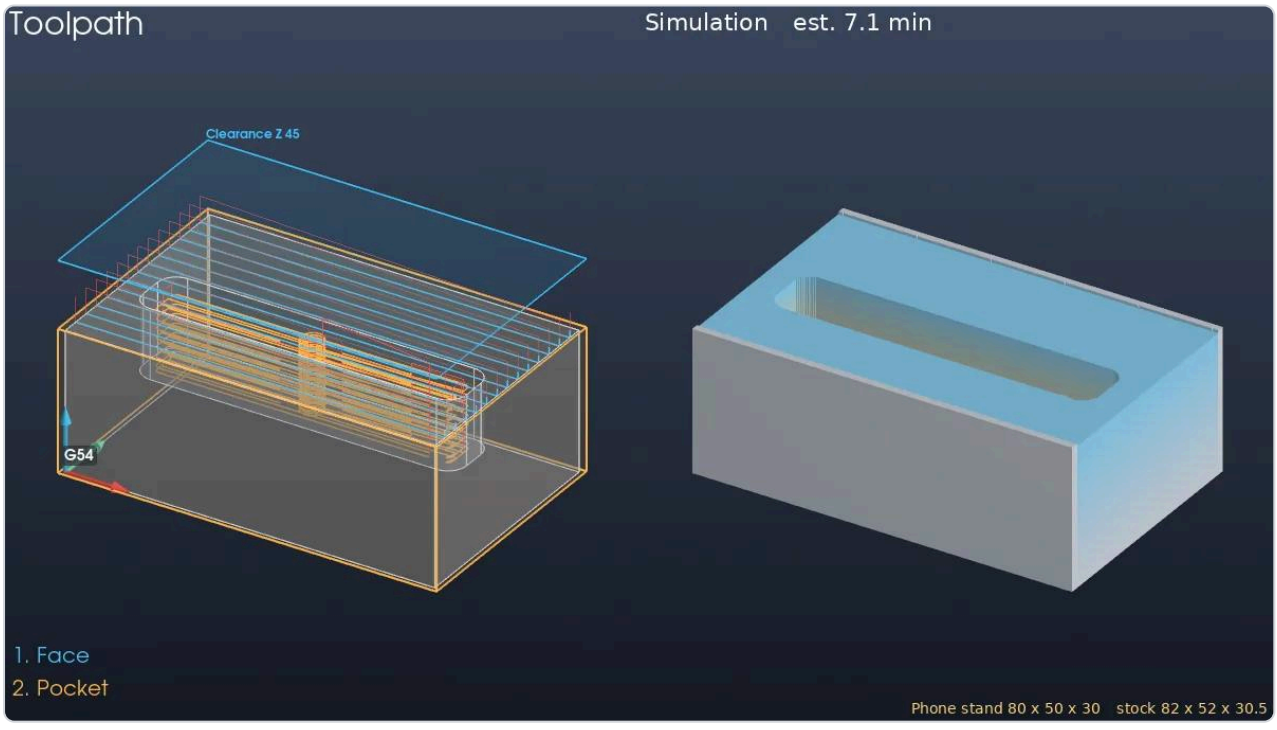
To add material around the part, type your own stock size, or set a D × L bar for turning, click the **Stock** node and choose a mode in its panel (see Stock, Frame (WCS) and Clearances).

> **Note:** If the original part was edited after the toolpaths were created, you see *Source geometry is out of date*. Click **Update Geometry**.

Try it: Start a job for an aluminium phone stand

An 80 × 50 × 30 block with a 13 mm cradle slot that holds your phone upright on the desk. This example is all about the job itself: template, stock, clearance, G54 and saving.

1. In **Part Design**, sketch an 80 × 50 rectangle on **XY** from the origin and **Extrude** it 30 mm.
2. Sketch on the top face: a 70 × 13 rectangle 5 mm from the left edge and 15 mm from the front edge, with R4 corners. **Extrude Cut** it 15 mm.
3. Open the **CAM** tab and click **New Job**. Choose **Mill Template** and click **OK**. The part is picked up from Part Design automatically.
4. Check what the template created. Click **Stock**: the stock is exactly the part box (*Part box: 80 x 50 x 30 mm*). Change **Stock** to **Part + allowance**. MECHCAD fills in Allowance -X, +X, -Y, +Y of 1 and Allowance +Z (top) of 0.5, which gives an 82 × 52 × 30.5 stock with material on top to face. Click **Clearances**: Clearance Z 45 (top + 15). Click **Machine**: Generic Fanuc 3-Axis Demo with the FANUC_MET 3Axis post. Click **Frame**: G54 at the model origin, so Z0 is the bottom of the part.
5. Add a **Face** on the top face and a **Pocket** on the floor of the slot, then **Simulate**. The estimated time is about 7.1 min.
6. Click **Save** and name the job `phone_stand.metcam`.



Phone stand: the orange stock box, Clearance Z 45 and G54 at the origin, with the Face and Pocket toolpaths

Challenge: Export the stand as a STEP file, then click **Geometry** and load that file instead of the Part Design body. Do the stock and clearance come out the same?

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Stock, Frame (WCS) and Clearances Premium

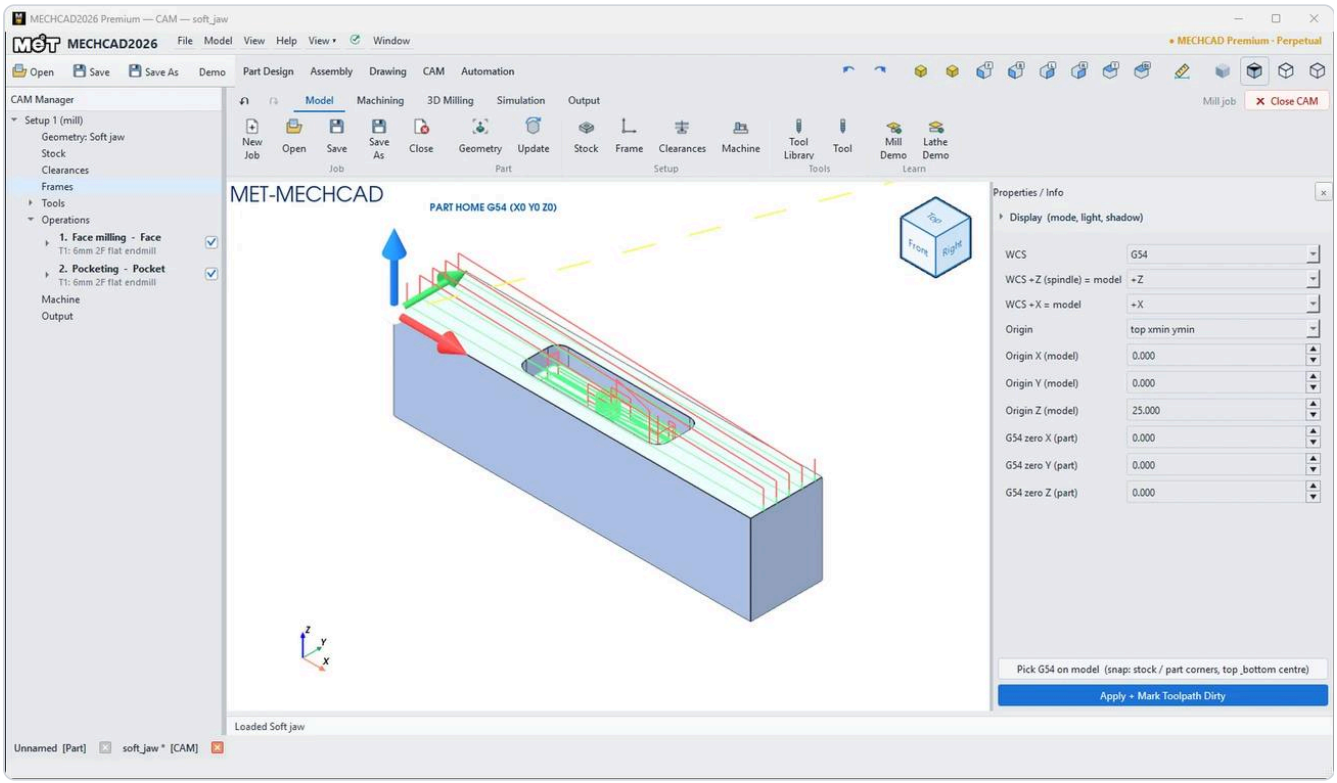
Define the raw stock, the G54–G59 work zero and the safe planes.

What it does

Sets up the job before you create operations: the stock size, the work coordinate system (WCS) and the tool clearance heights.

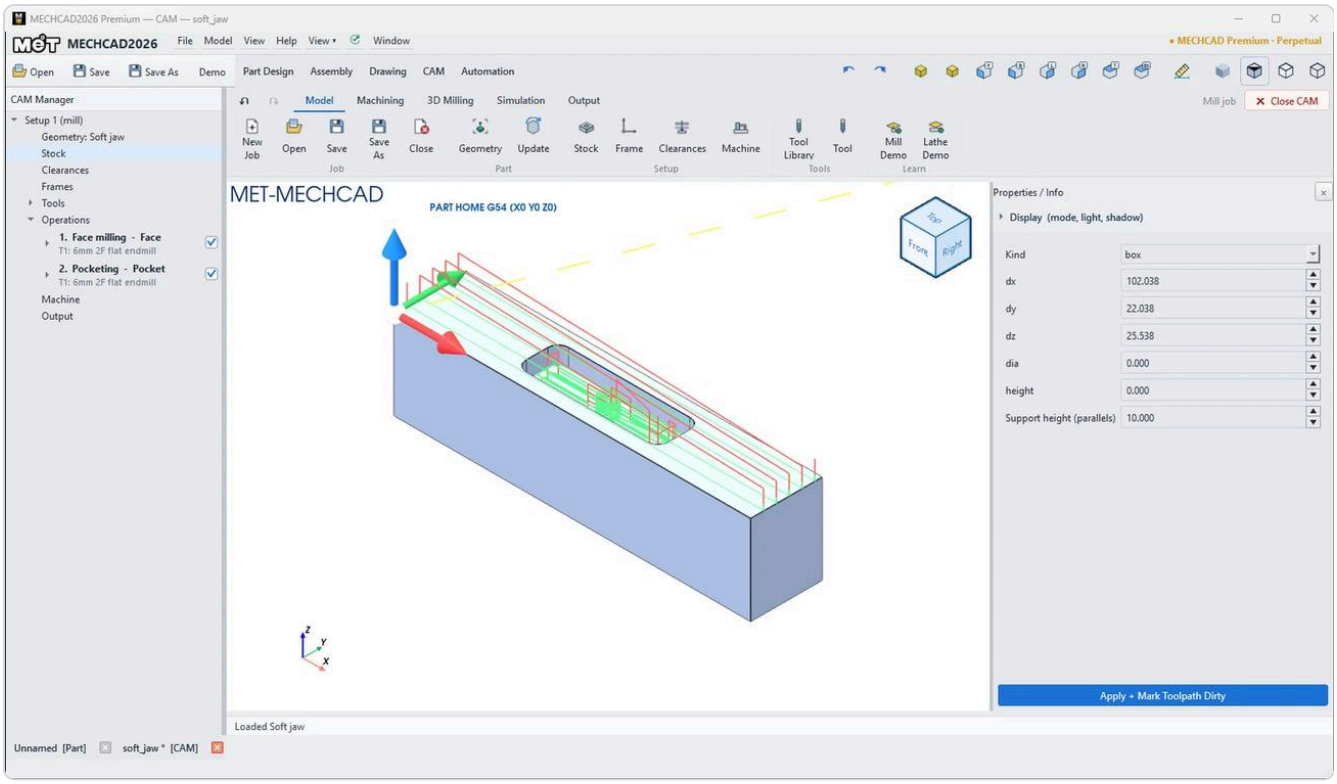
Where to find it

Model › Setup › Stock / Frame / Clearances, or click the node in the CAM Manager



Frame (WCS) in the Properties panel

Stock



Stock

A new job starts with a stock exactly the size of the part bounding box (no allowance). To add material or use your real raw size, choose a mode in the **Stock** box. The panel shows only the fields of that mode. A summary box at the top gives the part size (*Part box: … mm*) and the resulting stock (*Stock:* …, with its from … to … coordinates). Edits apply at once: the translucent stock in the viewport follows, every toolpath is marked for recalculation, and the old simulation is cleared. The Stock node in the CAM Manager shows the kind and size, for example *Part box 60 x 40 x 20*.

Milling

Stock	Description
Part (default)	A box exactly the size of the part bounding box, no allowance (the default of a new job).
Part + allowance	The part box plus a separate allowance on each side: **Allowance -X**, **+X**, **-Y**, **+Y**, **-Z (below)**, **+Z (top)**. When you first choose it, MECHCAD fills in 1 / 1 / 1 / 1 / 0 / 0.5.
Manual size	Type the size yourself (**Length X**, **Length Y**, **Height Z**) and choose **Placed**: *At position (min corner)* (enter **Position X / Y (min corner)** and **Position Z (bottom)**) or *Centred on part (XY)* (centred on the part in XY, bottom on the part bottom, the extra height on top).
Round bar (cylinder)	A **Bar diameter** × **Bar length** bar along **Bar axis** (Z / X / Y), centred on the part. A bar lying along X or Y is simulated as its enclosing box.
From CAD file	Uses the bounding box of the part in a CAD file as the stock. Click **Stock from CAD file…** to choose the file.
Support height (parallels)	Height of the parallels

Turning

Turning uses a round bar along the spindle axis (+Z). The front of the bar sits the face allowance in front of the part face; the rest of the length is behind the part, where the chuck grips it.

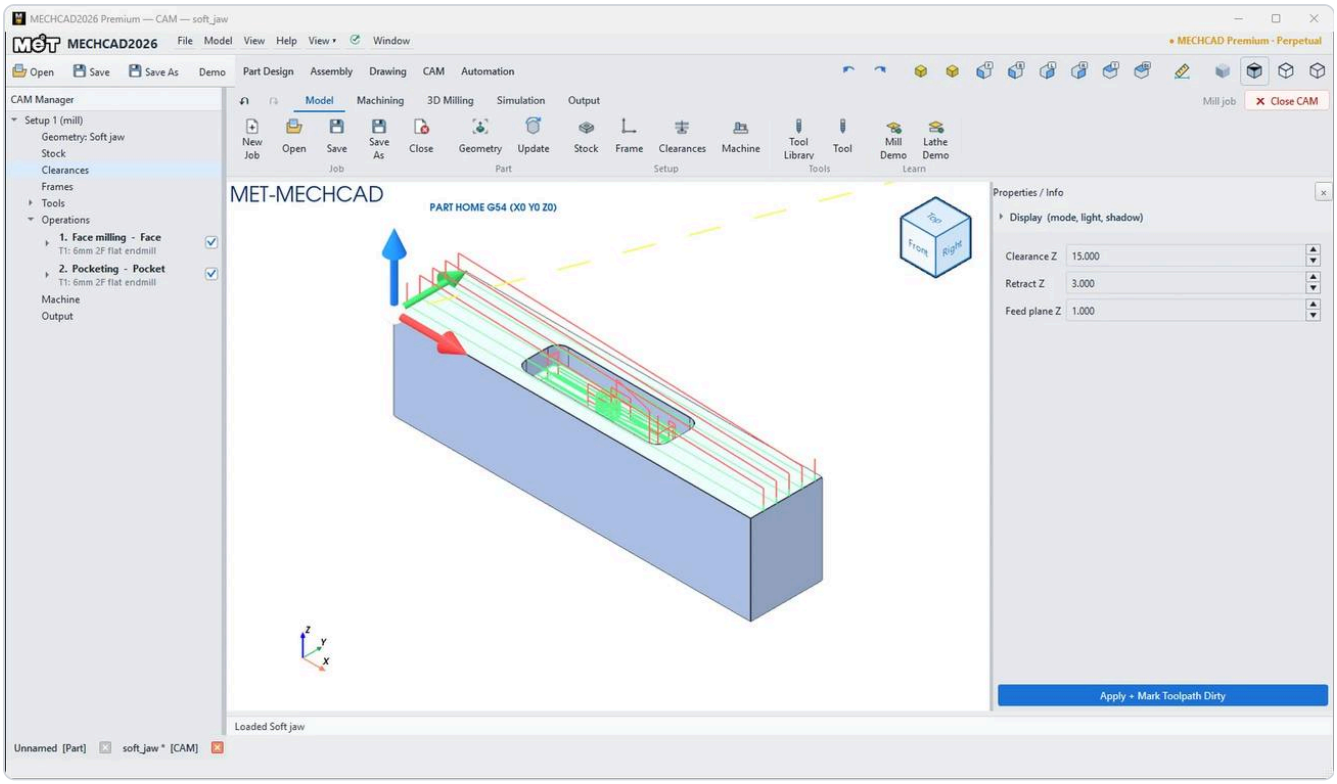
Option	Description
Bar stock	**Standard bar (default)** = the next standard bar size that leaves at least 1 mm on the part diameter (at most part Ø + 4), **Bar = part OD** = the largest part diameter, **Manual D x L** = your own size
Diameter D, Length L	Bar size (mm). Typing D or L switches the mode to Manual D x L automatically.
Face allowance (in front of part)	Material left in front of the part for the facing operation. Default 2 mm.

The summary box shows *Part: OD … x L … mm*, *Stock: … D… x L…* and the face allowance and grip length behind the part (*… mm to face, … mm behind the part (chuck grip)*). When you change the bar, turning operations that still use values from the old bar (bar radius, face stock, approach clearance) follow the new bar. Values you typed yourself are kept.

Frame (WCS)

Option	Description
WCS	G54–G59
WCS +Z (spindle) = model / WCS +X = model	Machine axis directions relative to the part (±X, ±Y, ±Z)
Origin	custom, top xmin ymin, top xmin ymax, top xmax ymin, top xmax ymax, top center, bottom xmin ymin
Origin X/Y/Z (model) / G54 zero X/Y/Z (part)	Zero point coordinates
Flip (turn the part over) / Rotate about Z	Turns the part over and rotates it about Z
Work zero (G54) on stock	Puts G54 on a point of the stock: a corner, an edge midpoint or the centre of the top (top …), bottom centre, bottom xmin ymin, or custom
Pick G54 on model	Click an orange point on a corner/centre of the stock or part (Esc = cancel)

Clearances



Clearances

Option	Default	Description
Clearance Z	25	Safe level for rapid moves
Retract Z	5	Retract level between operations
Feed plane Z	1	Level where feed cutting starts

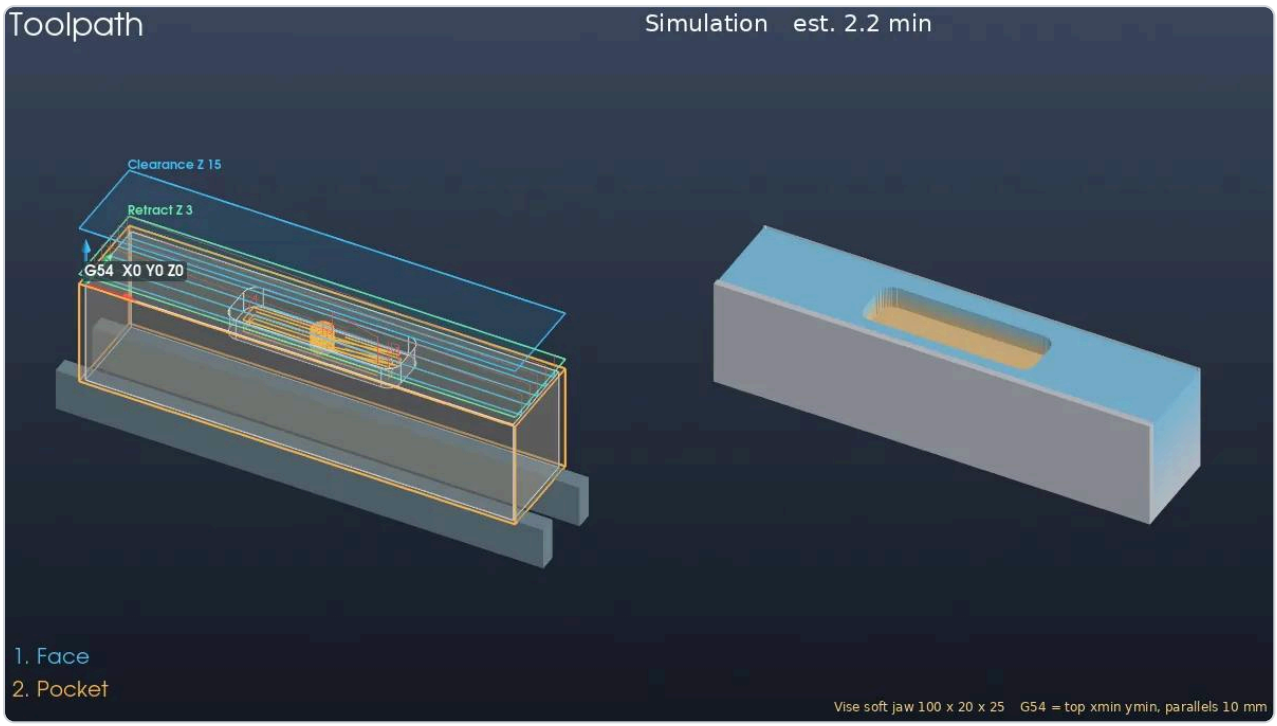
Each panel ends with the **Apply + Mark Toolpath Dirty** button (applies the values and marks the toolpaths for recalculation).

Try it: Set up a vise soft jaw on parallels

Soft jaws are usually the first thing a machinist makes for their own vise. Here it is a 100 × 20 × 25 aluminium jaw with a 40 × 12 nest pocket. You put G54 on the top front-left corner, sit the jaw on 10 mm parallels and set tight clearances.

1. Model the jaw: a 100 × 20 rectangle **Extruded** 25 mm. On the top face, sketch a 40 × 12 rectangle 30 mm from the left and 4 mm from the front, with R3 corners, and **Extrude Cut** it 5 mm.
2. **CAM** › **New Job** › **Mill Template**.
3. Do the **Frame** first. Set **Origin** to `top xmin ymin` and click **Apply + Mark Toolpath Dirty**. The top of the part is now Z0 and the front-left corner is X0 Y0. The stock is fitted to the part again.
4. Then the **Stock**: choose **Part + allowance** (MECHCAD fills in 0.5 mm on top, so there is material to face) and set **Support height (parallels)** to 10, then click Apply. (Do the stock after the frame, because applying the frame fits the stock again.)
5. **Clearances**: Clearance Z 15, Retract Z 3, Feed plane Z 1, then Apply.

6. Add a **Face** on the top face and a **Pocket** on the nest floor. The pocket now runs from Top Z 0 to Floor Z −5.
7. **Simulate** (about 2.2 min) and look at the G-code: the face pass cuts at `Z0` and the pocket stops at Z−5.



Soft jaw: G54 on the top front-left corner, 10 mm parallels, Clearance Z 15 and Retract Z 3

Challenge: Change **WCS** to G55 and generate the G-code again. The program header now calls G55 instead of G54.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Tool Library and tool settings Premium

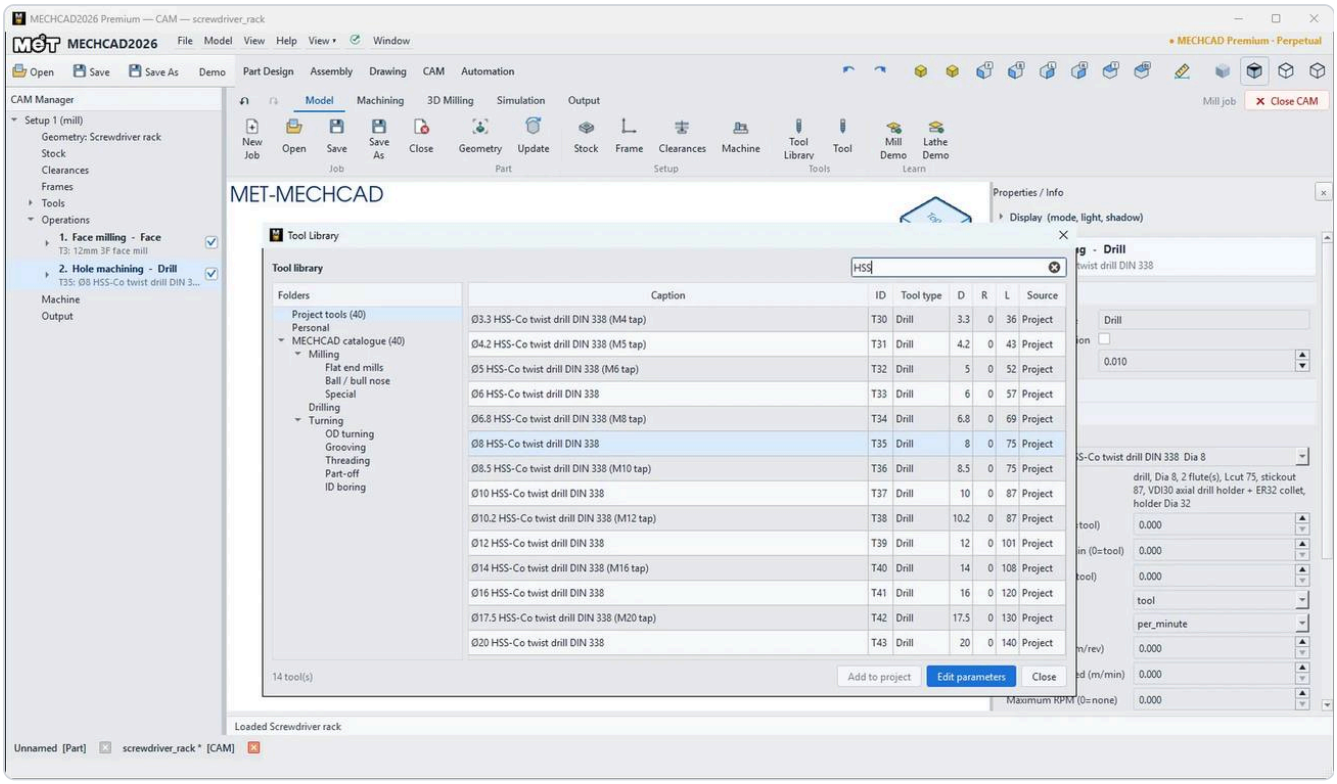
Choose tools from the library and set holder size, spindle speed and default feeds.

What it does

Manages the tools used in the job. The tools under **Project tools** are the tools of this job.

Where to find it

Model › Tools › Tool Library and the **Tools** node in the CAM Manager



Tool Library window

Item	Description
Search caption / ID / group / Ø...	Search for tools by name, ID, group or size (for example `drill 8`).
Folders	**Project tools**, **Personal** and **MECHCAD catalogue**, each split into tool groups (see the table below). The window opens on the group of the selected operation.
Caption, ID, Tool type, D, R, L, Source	Columns
New tool...	Creates a new tool in the Tool box.
Add to project	Copies a catalogue or Personal tool into the job.
Edit parameters	Edits the settings (or double-click).

Standard tool library by group

The **MECHCAD catalogue** holds standard metric tools you can buy off the shelf (IDs T100 and up) in addition to the original tool set. The S / F values of the standard tools are generic starting values; adjust them for your material and actual tool.

Group	Examples in the library
Face	Face mills D32–80 (R0.8)
Flat	2F / 3F / 4F end mills D1–20, plus long-reach
Bull nose	D4–20, corner R0.5 and up
Ball	D1–16

Group	Examples in the library
Drill	HSS drills D1–13 in 0.5 mm steps, tap drills and carbide drills
Spot/Center	90° spot drills
Tap	Taps M3–M12
Reamer	H7 reamers
Bore	Bore heads
Chamfer	90° and 60° chamfer mills with tip diameter
Thread mill	M4 and up
Slot/T-slot	T-slot and 45° / 60° dovetail
Corner round	R0.5–R5
Engrave	60° V engravers
Lathe	Turning tools, split into OD turning / ID boring / Grooving / Threading / Part-off

The same groups are used wherever you pick a tool: the **Tools** node in the CAM Manager lists the job tools by group (for example *Drill (5)*), and the tool selector in the **Tool** section of the operation box lists the tools by group, with the operation's own group first. Each row shows the Ø, and the *Filter tools: name, group, Ø ...* box narrows the list.

Main tool settings

Field	Description
Tool name / Tool type	Name / type (endmill, ball, bull, drill, center_drill, chamfer, corner_round, engraver, tap, reamer, boring, threadmill, tslot, dovetail, turning)
Included / point angle (deg), Tip diameter	Included angle of the tip and the flat tip diameter (chamfer mills, engravers, spot drills, and the drill point angle)
Diameter / insert width, Flutes, Corner / nose radius	Tool size
Cutting length (0 = 3×D), Stickout	Flute length and stickout
Holder diameter, Holder length, Arbor / holder style	Holder (for example BT40-ER32, HSK63A, PCLNR 2020), used for collision checking
Turning insert ISO code, shape, orientation, holder hand	Turning tool settings
Turret station, Geometry/wear offset, Tool-tip orientation code	Turret settings (turning)
Spindle direction, Default RPM, Default cut feed, Default plunge feed	Default spindle speed (6000) and feeds (300 / 100)
Default stepover / stepdown	Default stepover (0.4×D) / stepdown (1)

Field	Description
Tool material, Default coolant, CSS surface speed, maximum RPM	Tool material, coolant, surface speed

Chamfer tools

A chamfer mill uses **Included angle (A)** (for example 90° or 60°) and **Tip dia (d)** (the flat tip). MECHCAD uses these two values to put the cone exactly on the chamfer line. A **Chamfer** operation sets **Chamfer width** (default 0.5) and **Tip offset** (how far the tip goes below the bottom of the chamfer, default 0) in the **Path Setting** section of the operation box, and shows the computed position, for example *Tool axis 3.000 mm off the edge, tip at Z-3.000 (A90, tip d2)*. If the chamfer is too wide for the tool, you see *chamfer too wide for the tool*. If you click a chamfer face that is already in the model, MECHCAD takes the sharp edge and the width from that face.

The Tool box (create / edit a tool)

Click **New tool...**, or right-click a tool in the CAM Manager › **Edit Tool...**, to open the floating **Tool** box. It has the **Cutter** / **Holder** / **Speeds** tabs next to a drawing of the tool with its dimension letters (D, L, FL, SD, R, A ...).

Item	Description
Type / Subtype	Mill (End, Ball, Bull / Radius, Chamfer, Spot, Thread mill, T-slot, Dovetail, Engrave (V), Corner round, Bore head), Drill (Drill, Spot, Reamer), Tap, Lathe (Turning insert)
Cutter	Cutter dia (D), Tool length (L), Flute length (FL), Shank dia (SD), Radius (R), Included angle (A), Tip dia (d), Pitch (P), Flutes (F); only the fields of that type are shown
Holder	Holder, Holder dia, Holder / gauge length, Stickout (gauge to tip)
Speeds	Spindle S (rpm), Feed F (mm/min), Plunge (mm/min), Coolant, H offset, D offset (= *tool no.* means the tool number is used)
Add to Lib / Load Tool Shape / Save All / Load All	Stores the tool in your Personal library, and imports / exports tools as `.mettool` files
OK / Apply / Reset / Delete / Cancel	Apply / discard / delete the tool

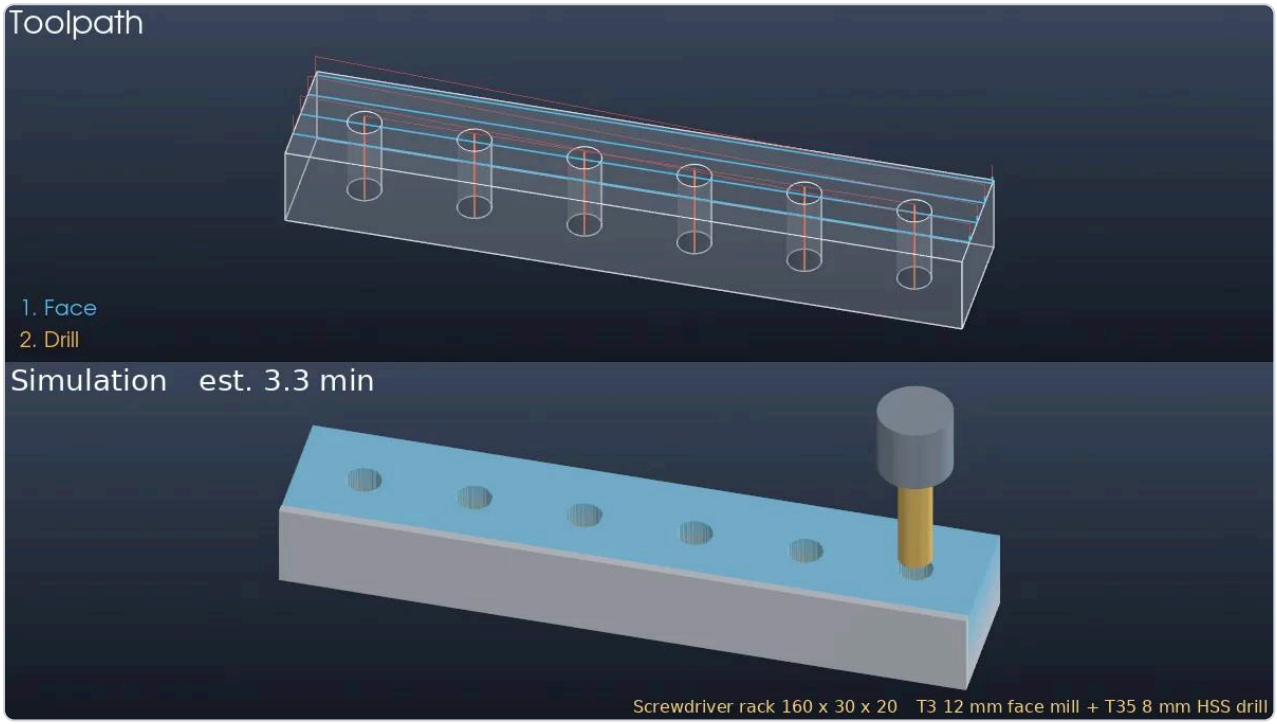
When you edit a tool that is in use, the toolpaths of the operations using it are marked for recalculation, and the cutter in the simulation changes with it (see Simulation).

Try it: Choose the right tools for a screwdriver rack

A 160 × 30 × 20 bench rack with six Ø8 holes keeps your screwdrivers standing. Out of the box, the operations would use a 6 mm end mill and a 6 mm drill. Here you pick the right cutters from the library.

1. Model a 160 × 30 × 20 block. On the top face, draw six Ø8 circles at Y 15 and X 15, 41, 67, 93, 119 and 145, then **Extrude Cut** them **Through All**.
2. **CAM** › **New Job** › **Mill Template**. In **Stock**, choose **Part + allowance** (0.5 mm on top to face) and set **Support height (parallels)** to 5 so the drill can break through.

3. Click **Model › Tools › Tool Library**. Type `HSS` in the search box and select **Ø8 HSS-Co twist drill DIN 338** (T35). It is already in **Project tools**. (Tools from the **MECHCAD catalogue** folder are copied in with **Add to project**.)
4. Click **Edit parameters** and set **Default RPM** to 1000.
5. Add a **Face** on the top face. In the **Tool** section of the operation box, choose **T3 12mm 3F face mill**. It needs far fewer passes than the 6 mm default.
6. Add a **Drill**: click all six hole walls and press **Enter**. Smart Hole picks the first Ø8 drill of the job for you, **T35**, and sets **Depth** to 22.903 (20 mm of plate + the 118° point + 0.5 mm to break through). Set **Peck depth** to 4.
7. **Simulate**. The estimated time is about 3.3 min, and the G-code changes tools with `T35 M06` and `S1000`.



Screwdriver rack: the face mill passes and six Ø8 holes, with the 8 mm drill on the last hole

Challenge: Add a **Chamfer** on the top edge of one hole with **T20 6mm 90deg Chamfer Mill** so the screwdrivers slide in easily.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Operation Parameters Premium

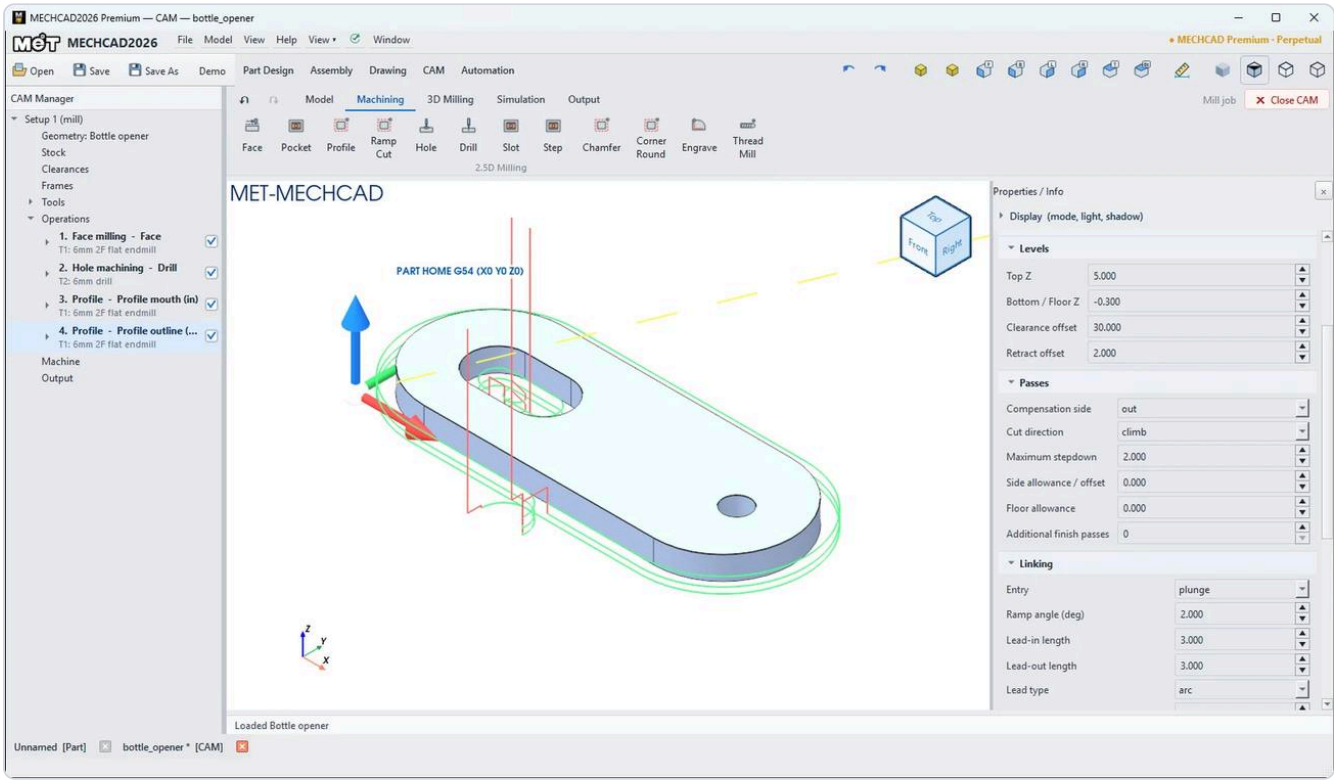
What the sections of the operation box and the General, Geometry, Tool, Levels, Passes, Linking and Stock pages mean, including Tool Compensation (D-offset).

What it does

A single click on an operation in the CAM Manager only selects it. Double-click it (or right-click › **Edit Parameters**) to open the operation box under the CAM Manager and show its toolpath. The box is split

into collapsible sections. Fields that do not apply to that operation are not shown.

Section in the box	Description
Features	Input type, the Add / Done / Remove / Reverse / Open/Close buttons for the picked edges/faces, Pick, Toolpath side
Tool	Tool selector (grouped by tool type, with a filter box) and Speed S / Feed F
Depths	Top Z, Bottom Z, Stepdown
Strategy and steps	Pocket pattern, Stepover, Cuts, *Moves outside the stock as rapids*
Path Setting	**Tool Compensation** and **D register** (Profile, Ramp Cut, Chamfer, Corner Round), and **Chamfer width** / **Tip offset** (Chamfer)
Lead in / Lead out	How the tool enters and leaves the cut
Limiting: Check surfaces	Surfaces the tool must not cut (Pick surfaces / Clear buttons)
Holes	The Smart Hole table of a drilling operation (see Drill and Hole)
Parameters	All the pages in the table below



Profile operation settings

Section	Key fields
General	Operation name, Suppress operation, Tolerance (mm) 0.01
Geometry	Source (model / faces / sketch / feature / profile), Edge chain, buttons Select Faces / Select Sketch/Chain / Select Feature

Section	Key fields
Tool	Cutter / insert, Spindle RPM (0=tool), Cut feed mm/min (0=tool), Plunge feed (0=tool), Coolant, Feed mode, CSS, Maximum RPM
Levels	Top Z, Bottom / Floor Z, Depth, Clearance offset (5), Retract offset (2)
Passes	Cutting tactic (zigzag / concentric), Compensation side (out / in / on), Cut direction (climb / conventional / both), Stepover, Maximum stepdown, Stock to leave, Finish allowance, Peck depth
Linking	Entry (plunge / ramp / helix / outside), Ramp angle 2°, Lead-in / Lead-out length, Lead type (arc / line / none), Tool Compensation (None / Computer, Offset (G41/G42 on contour), Wear (G41/G42 on centre path)), D register (0=tool no.), Output arcs (G2/G3), Transfer mode, Skip air cuts
Stock	Input stock (ignore / previous / reference), Reference operation, Publish result to downstream ops
Turning	Cut region (OD / ID / face / profile / axial), NC cycle output, Longitudinal direction, Thread infeed, Thread / Spring passes

Tool Compensation (D-offset)

Value	Description
None / Computer	MECHCAD compensates the cutter radius; the NC follows the cutter-centre path with no G41/G42 (default).
Offset (G41/G42 on contour)	The NC follows the part contour, with G41/G42 + D on the lead-in and G40 on the lead-out. The control offsets by the radius in D.
Wear (G41/G42 on centre path)	The NC follows the cutter-centre path, plus G41/G42 + D. D holds only the wear.
D register (0=tool no.)	The D number output with G41/G42. 0 = the tool's **D offset** (Speeds tab of the Tool box), or the tool number.

See Profile and Ramp Cut for details and an example.

Right-click menu on an operation

Menu	Description
Edit Parameters	Opens the operation box.
Simulate this operation	Simulates this operation only.
Recalculate	Recalculates the toolpath.
Duplicate	Copies the operation (*copy*).
Move Up / Move Down	Changes the operation order (Alt+Up / Alt+Down, or drag and drop).
Change type...	Changes the operation type and keeps the picked edges/faces, tool, depths and leads.
Change Tool ›	Changes the tool.
Rest machining from ›	(3D) Cuts only the material left by a previous operation.

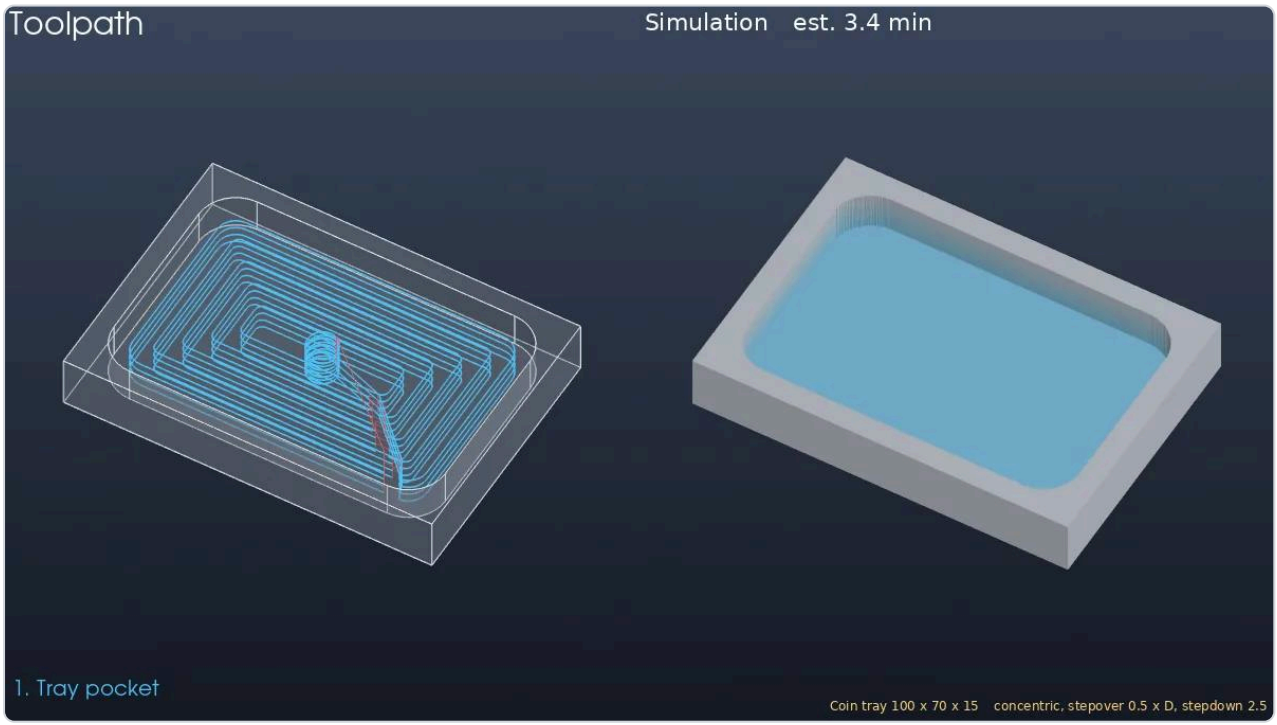
Menu	Description
Show / Hide Toolpath, Hide All Toolpaths	Shows or hides toolpaths.
Delete	Deletes the operation (asks for confirmation).

> **Note:** Clear the tick box in front of an operation to suppress it (its name then starts with *[Suppressed]*). Under each operation, the sub-rows Tool:, Geometry:, Tactic:, Levels:, Cutting: and Linking: jump straight to that section when you click them.

Try it: Tune the pocket of a coin tray

A 100 × 70 × 15 desk tray for coins and keys. You make one Pocket, then tune it field by field in the operation box.

1. Model a 100 × 70 block **Extruded** 15 mm. On the top face, sketch a 90 × 60 rectangle 5 mm in from every edge, with R10 corners, and **Extrude Cut** it 10 mm.
2. **CAM** › **New Job** › **Mill Template**. Click **Pocket**, click the tray floor and press **Enter**.
3. Double-click the operation in the CAM Manager. On the **General** page of the **Parameters** section, set **Operation name** to `Tray pocket`.
4. In **Tool**, set **Cutter / insert** to **T13** (10 mm), **Spindle RPM** to 4000 and **Cut feed** to 1200.
5. In **Passes**, set **Cutting tactic** to `concentric`, **Stepover** to 0.5, **Maximum stepdown** to 2.5 and **Corner smoothing radius** to 2.
6. In **Linking**, keep **Entry** `helix` and set **Ramp angle** to 3.
7. Right-click the operation › **Recalculate**, then **Simulate**. The estimated time is about 3.4 min, and the G-code shows `T13`, `S4000` and a helical G03 entry at F1200.



Coin tray: concentric rings with a helix entry, and the finished tray

Challenge: Right-click › **Duplicate**, clear the tick of the original to suppress it, and set the copy back to `zigzag`. Which tactic is faster for this tray?

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Face and Pocket (2.5D) Premium

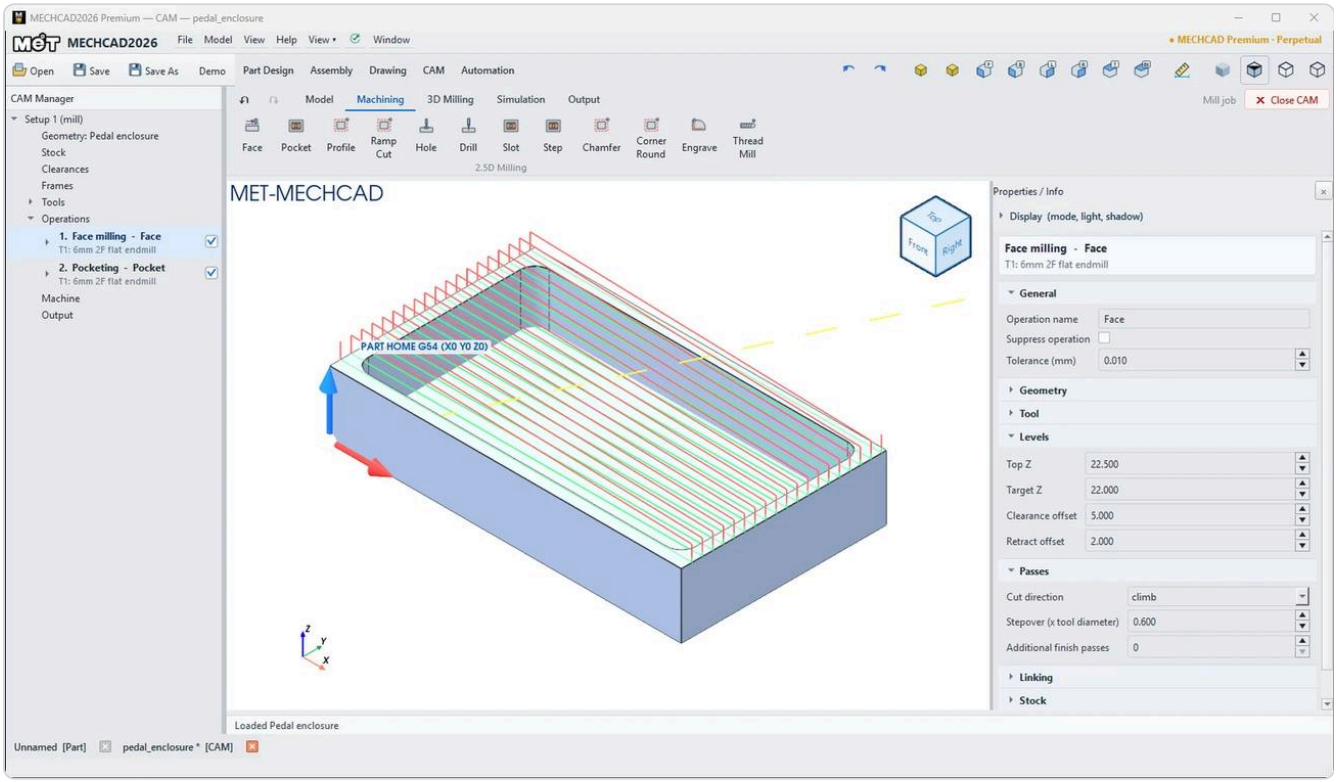
Face a surface and mill pockets with 2.5-axis operations.

What it does

- **Face** machines the top surface of the workpiece flat (stepover 0.6×D).
- **Pocket** mills a pocket in a zigzag or concentric pattern with helical entry (stepover 0.4×D, stepdown 2).

Where to find it

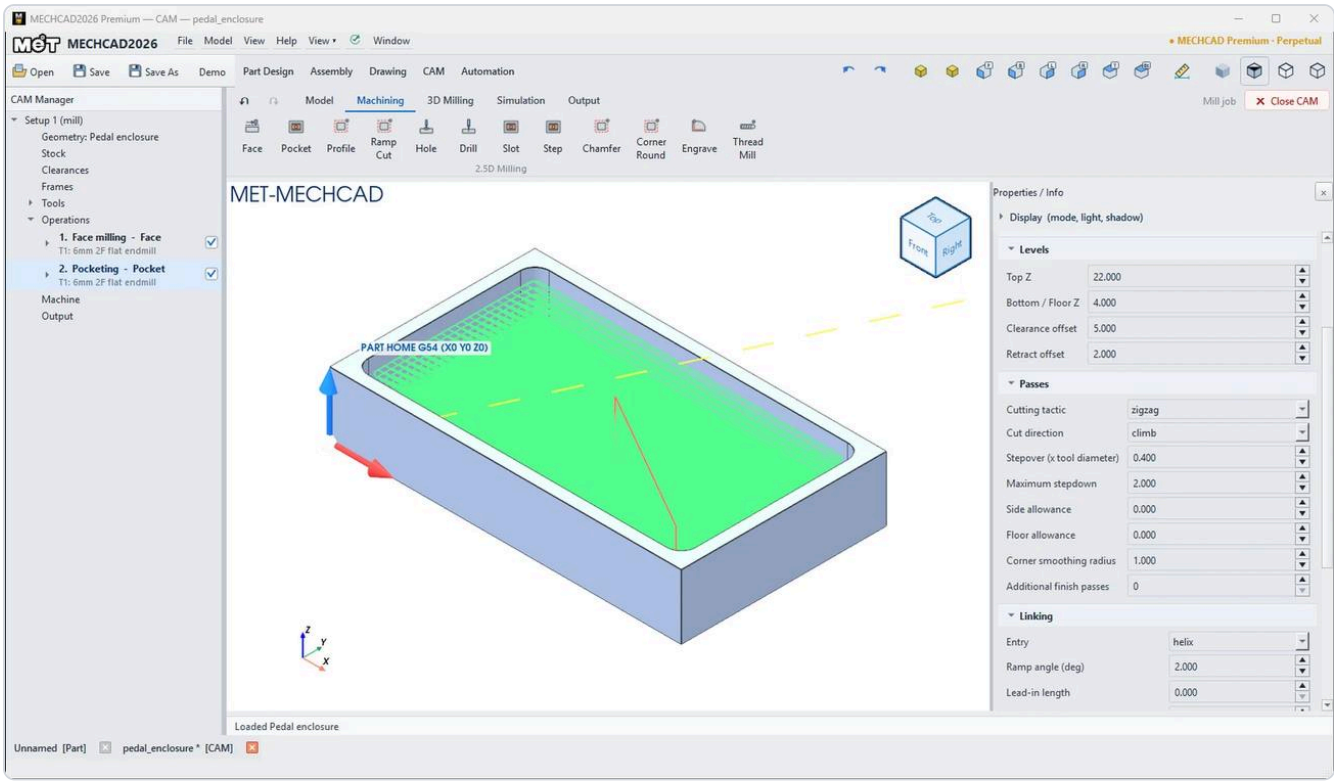
Machining › 2.5D Milling › Face / Pocket



Face operation

Steps

1. Click **Face** or **Pocket**.
2. Click a face in the view (status bar: *Pick face to create …, then Enter/Esc*).
3. Press **Enter** to create the operation.
4. Adjust the values in the Properties panel, then **Simulate**.



Pocket operation

Key defaults	Face	Pocket
Stepover	0.6 × D	0.4 × D
Maximum stepdown	–	2 mm
Cutting tactic	–	zigzag
Entry	–	helix
Corner smoothing radius	–	1 mm

Slot and **Step** use the same settings as Pocket (Slot = closed slot, Step = step or open pocket).

You can edit every operation in the **Properties** panel on the right when you click the operation in the CAM Manager. The panel sections are **General**, **Geometry**, **Tool**, **Levels**, **Passes**, **Linking** and **Stock** (see Operation Parameters for the meaning of each field).

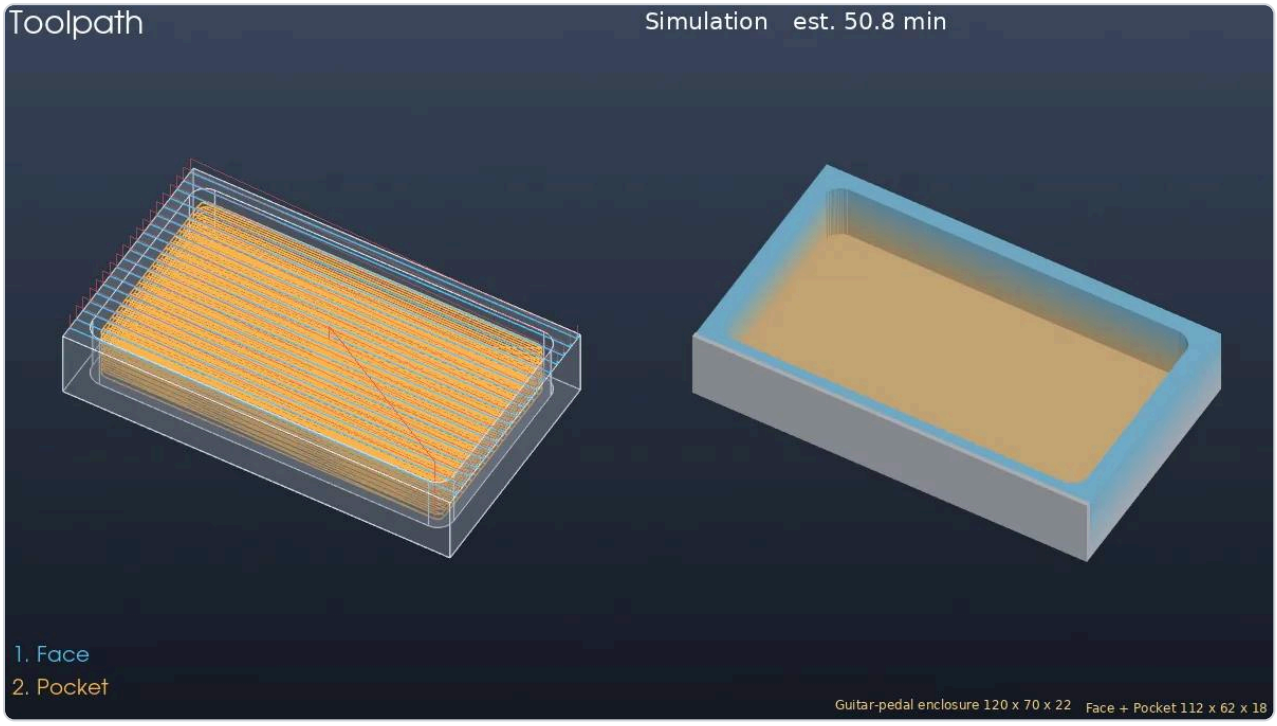
Try it: Hollow out a guitar-pedal enclosure

DIY pedal builders spend real money on good enclosures. Mill your own from a 120 × 70 × 22 block. Face the top, then pocket a 112 × 62 cavity 18 mm deep, which leaves 4 mm walls and a 4 mm floor.

1. Model a 120 × 70 rectangle **Extruded** 22 mm. On the top face, sketch a 112 × 62 rectangle 4 mm in from every edge, with R5 corners, and **Extrude Cut** it 18 mm.
2. **CAM** › **New Job** › **Mill Template**.
3. Click **Face**, click the top face and press **Enter**. Defaults: T1 6 mm, stepover 0.6 × D, Target Z 22.
4. Click **Pocket**, click the cavity floor and press **Enter**. MECHCAD reads Top Z 22 and Floor Z 4 from the walls. The pocket uses zigzag passes, a helix entry, stepover 0.4 × D and a 2 mm stepdown: nine levels

plus a wall pass.

5. In **Linking**, clear **Skip air cuts (G0 over cleared stock)**. In a pocket cut from a solid block, every pass is in material anyway.
6. **Simulate**. Expect about 48 min, because 18 mm is a deep pocket for a 6 mm cutter.



Pedal enclosure: Face and Pocket toolpaths and the hollowed-out block

Challenge: Change the pocket cutter to **T13** (10 mm, 40 mm flutes) and set **Maximum stepdown** to 4. How much faster is it?

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Profile and Ramp Cut Premium

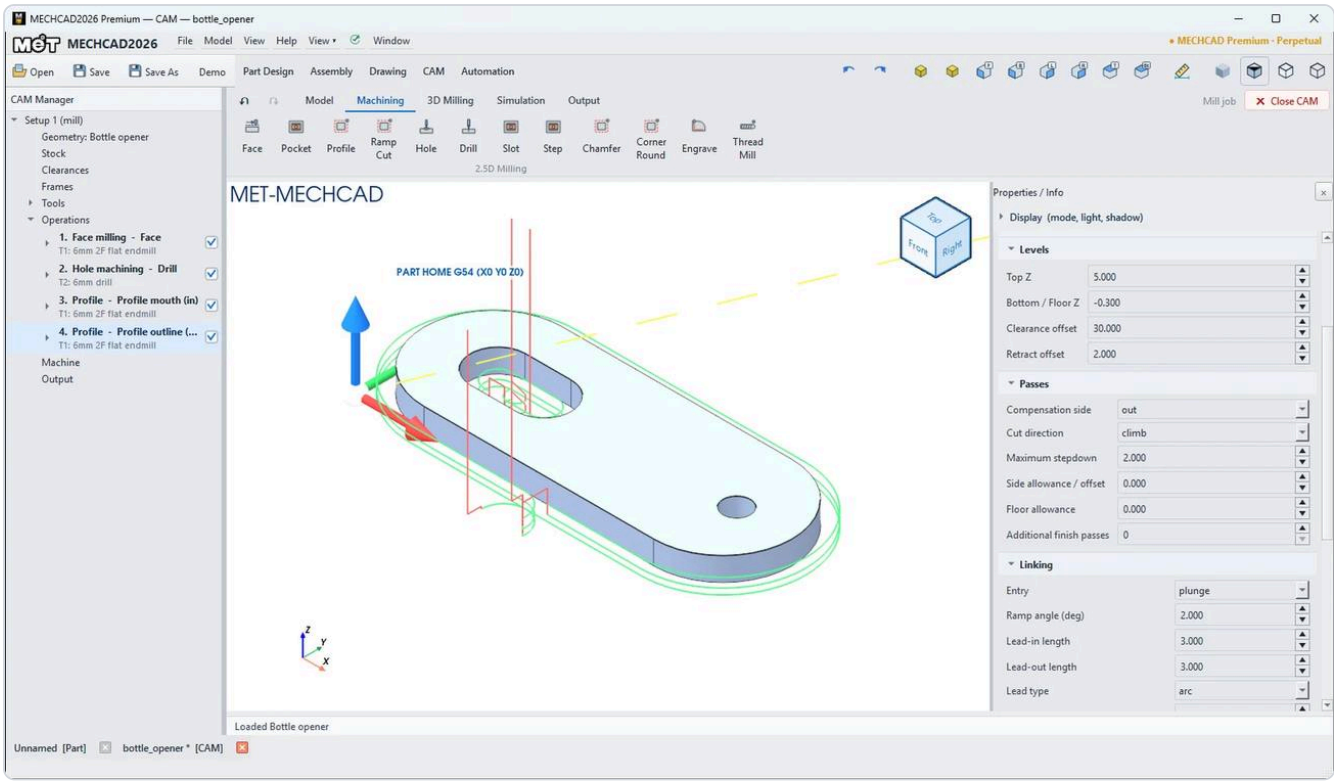
Mill along outer or inner edges with cutter compensation, and ramp down along an edge.

What it does

- **Profile** mills along an edge (outside, inside or on the line) with arc lead-in and lead-out. Cutter compensation is done by the computer, or G41/G42 + a D number is output so the control compensates (Tool Compensation).
- **Ramp Cut** ramps down continuously along the selected edge.

Where to find it

Machining › 2.5D Milling › Profile / Ramp Cut



Profile operation and its settings in the Properties panel

Steps

1. Click **Profile**.
2. Click an edge in the view (a continuous chain of edges is selected automatically), then press Enter.
3. Set **Compensation side**, **Maximum stepdown**, **Lead type** and **Tool Compensation**, then Simulate.

Key defaults	Value
Compensation side	out
Maximum stepdown	1 mm
Lead type	arc
Tool Compensation	None / Computer (see the next section)
D register	0 (= the tool number)
Edge chain	auto / loop / tangent / circle

Tool Compensation (D-offset)

Set it in the **Path Setting** section of the operation box for **Profile**, **Ramp Cut**, **Chamfer** and **Corner Round** (the **Linking** page under **Parameters** also has the **Tool Compensation** and **D register (0=tool no.)** fields).

Tool Compensation	Resulting G-code
None / Computer	MECHCAD computes the cutter-centre path; no G41/G42 in the NC (default).
Offset (G41/G42 on contour)	The NC follows the actual part contour, with G41/G42 + D on the lead-in and G40 on the lead-out. The control offsets the cutter by the radius in the D register.

Tool Compensation	Resulting G-code
Wear (G41/G42 on centre path)	The NC follows the cutter-centre path, plus G41/G42 + D. The D register holds only the wear, for size adjustment at the machine.

D register is the radius register output with G41/G42. It is active when Offset or Wear is chosen. A value of 0 (shown as *tool no.*) means the tool's **D offset** is used (the **Speeds** tab of the Tool box), and if the tool has none, the tool number.

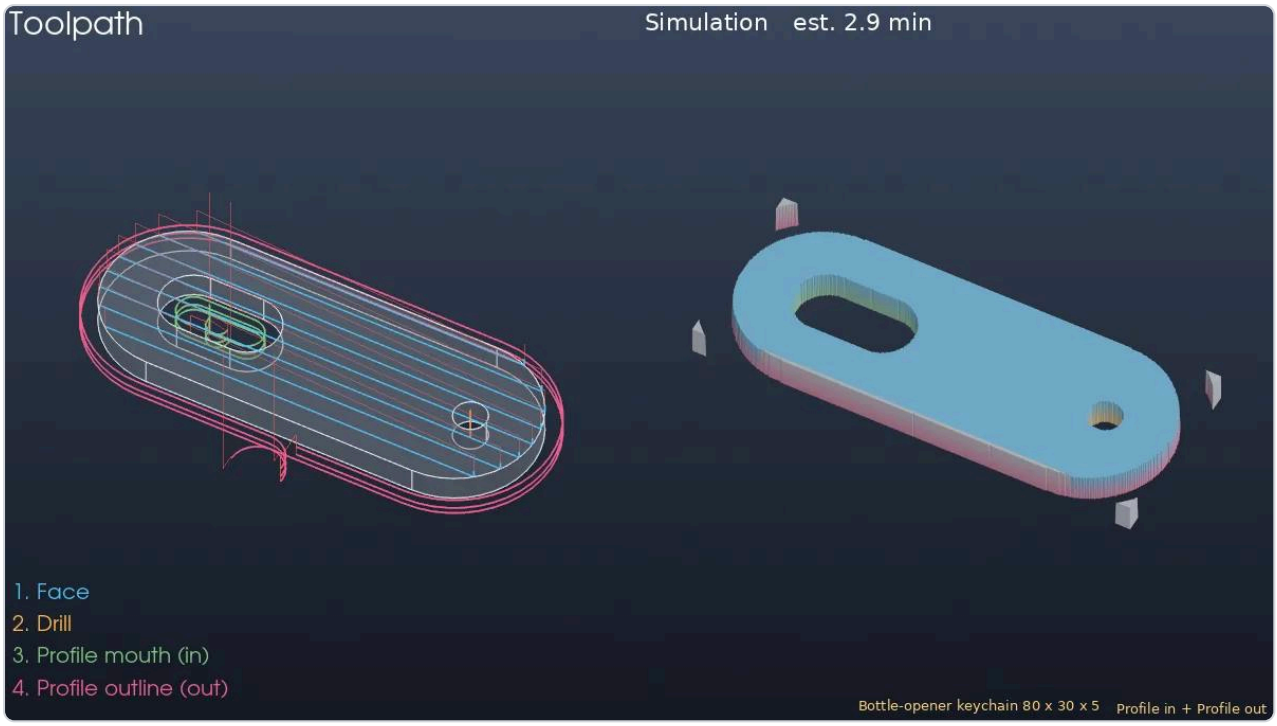
Chamfer (V-cutter chamfer, width 0.5 mm), **Corner Round** (R1 round) and **Engrave** (engrave along a line) use the same edge selection. See Other 2.5D operations.

Double-click an operation in the CAM Manager to open its operation box, where you can edit every value. Its **Parameters** section has the pages **General**, **Geometry**, **Tool**, **Levels**, **Passes**, **Linking** and **Stock** (see Operation Parameters for the meaning of each field).

Try it: Cut out a bottle-opener keychain

An 80 × 30 × 5 aluminium keychain with a slot that pops bottle caps. Two Profiles do the work: one inside the opener slot, one around the outline.

1. Model the outline as a slot shape 80 mm long and 30 mm wide (R15 ends) and **Extrude** it 5 mm. On the top face, draw a 22 × 12 slot whose ends are centred at X 16 and X 26 (Y 15), and a Ø6 circle at X 68 Y 15. **Extrude Cut** both **Through All**.
2. **CAM** › **New Job** › **Mill Template**. In **Stock**, choose **Part + allowance** (0.5 mm on top to face) and set **Support height (parallels)** to 3.
3. Add a **Face** on the top. Add a **Drill** on the Ø6 hole and set **Depth** to 5.5.
4. Click **Profile**, click the top edge of the opener slot and press **Enter**. Set **Compensation side** to `in` and **Bottom / Floor Z** to −0.3. In **Linking**, set **Lead arc radius** to 2, **Lead-in length** to 1 and **Lead-out length** to 1. The default lead (0.8 × D = 4.8 mm) is too big for a 12 mm wide opening and would cut into the opposite wall.
5. Click **Profile** again, click the top outer edge and press **Enter**. Keep **Compensation side** `out` and set **Bottom / Floor Z** to −0.3.
6. **Simulate** (about 2.9 min). The opener drops out of the blank, so hold the blank with screws or tape outside the outline.



Bottle opener: an inside Profile on the opener slot, an outside Profile on the outline, and the finished keychain

Challenge: Set the outline Profile's **Tool Compensation** to **Offset (G41/G42 on contour)** and look in the G-code for G41/G42 with the D number on the lead-in and G40 on the lead-out. Then enter another **D register** number and generate the G-code again.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Drill and Hole Premium

Drill holes at multiple positions with Smart Hole (tick holes one by one, drill chosen automatically) and G81/G83 cycles.

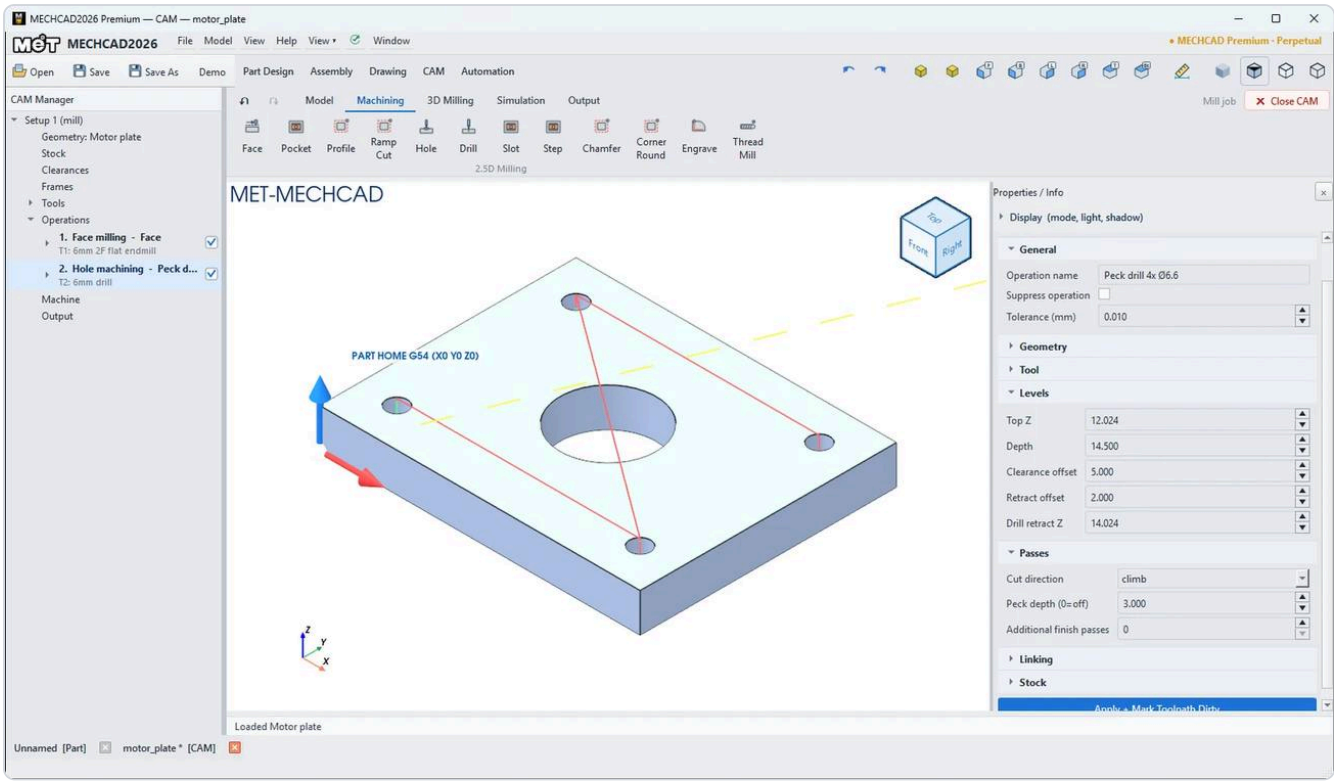
What it does

Drill, **Hole**, **Peck Drill** and **Center** drill the holes whose faces you select. All of them use **Smart Hole**: MECHCAD finds every hole in the faces you click, lists them in a table where you tick each hole, and chooses the drill for the hole size automatically. Enter a **Peck depth** to drill in steps.

Where to find it

Holes › Drilling › Center / Peck Drill / Drill / Hole

Command	Settings applied
Drill / Hole	Peck depth 3 (3 mm per peck)
Peck Drill	Peck depth 1
Center	Peck depth 0 (one pass), for spot / centre drills. After picking the holes, set a shallow **Depth** yourself.



Peck drill operation, 4x Ø6.6

Steps

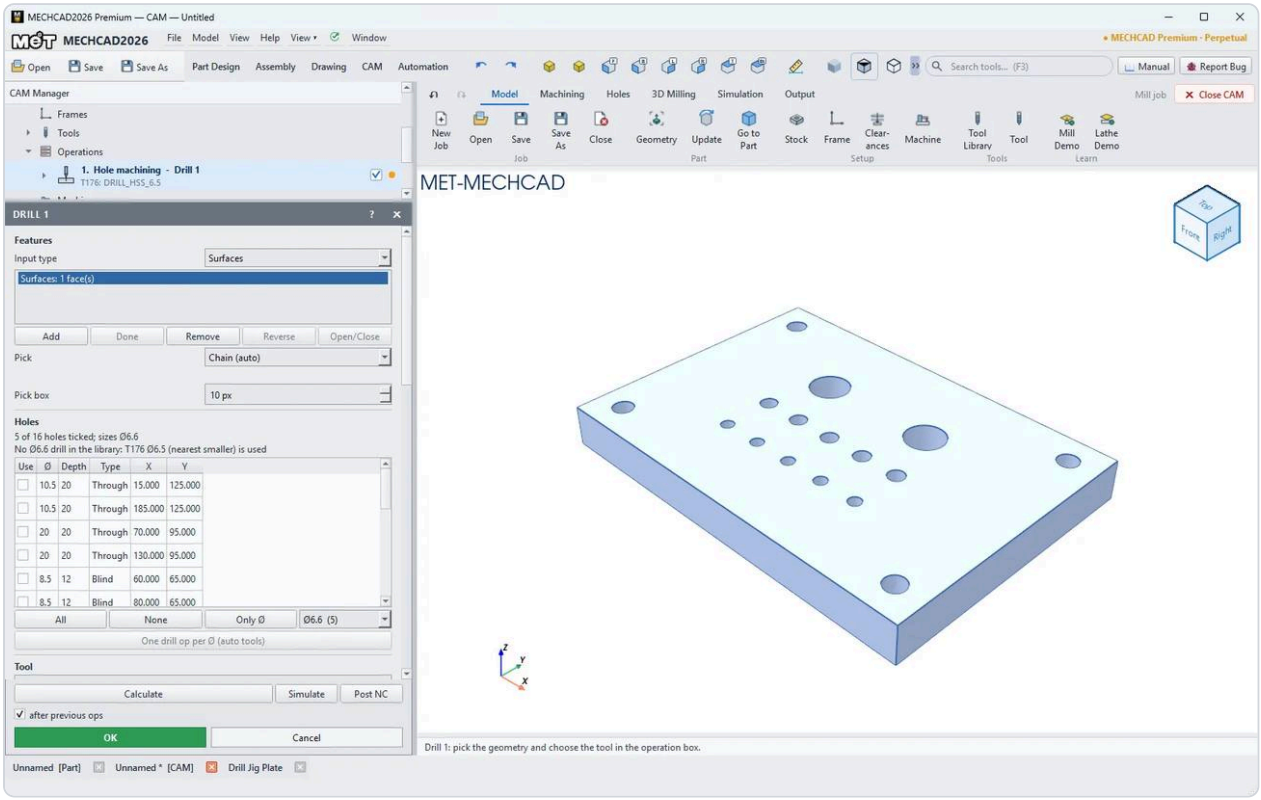
1. Click **Drill** (or **Hole**, **Peck Drill**, **Center**).
2. Click a flat face that has the holes in it (every hole whose rim lies on that face is found), or click the hole walls one by one (you can select several faces), then press Enter.
3. In the **Holes** section of the operation box, tick only the holes you want to drill.
4. Check the drill, **Peck depth** and **Drill retract Z**, then Simulate.

Smart Hole: the Holes table

Item	Description
Use, Ø, Depth, Type, X, Y	One row per hole: the tick, the size, the hole depth, the type (*Through* or *Blind*) and the position
All / None	Ticks every hole / clears every tick
Only Ø + size box	Ticks only the holes of the chosen size (the box shows each size with its hole count, for example *Ø6 (7)*)
One drill op per Ø (auto tools)	When several sizes are ticked, makes one operation per size, each with the drill that fits. The original operation keeps the smallest size, and the names get the size added, for example *Drill Ø8*
Summary line	For example *5 of 7 holes ticked; sizes Ø6*

- **Automatic drill:** when one hole size is ticked, MECHCAD chooses a drill of exactly that diameter, from the job tools first and then from the library. If there is no exact size, it uses the nearest smaller drill and tells you so (it never picks a drill larger than the hole). A library drill is copied into the job.

- **Depth:** a blind hole is drilled to its bottom. A through hole is drilled the hole length + the drill point + 0.5 mm. Top Z is the top of the hole, and **Drill retract Z** = Top Z + 2.
- Holes you leave unticked never reach the NC. With no hole ticked you see *Holes: tick at least one hole*.
- When you pick the faces again, holes that are found again keep their tick.



Smart Hole on a drill-jig plate: 16 holes found on the top face, only the Ø6.6 holes ticked, and the nearest smaller drill chosen

Drill cycle settings

Setting	Default	Description
Peck depth (0=off)	Per command (see the table above)	0 = drill in one pass, > 0 = peck drilling (the Q value)
Drill cycle (NC)	auto	auto = G83 with a Peck depth, G82 with a Dwell, otherwise G81. You can choose g81 / g82 / g83 / g73, or expanded (written as G00/G01 moves)
Return plane	g98	g98 = back to the initial level between holes, g99 = back to the R plane
Dwell (seconds)	0	Pause at the hole bottom (G82)
Drill retract Z	Top Z + 2	R plane
Depth / Bottom Z	From the hole	Depth

The default Fanuc mill post writes all the holes of an operation as one canned cycle, for example `G17 G98 G83 X.. Y.. Z.. R.. Q3 F..`, then the X / Y of each further hole, closed with `G80`. Other posts write G00/G01 moves.

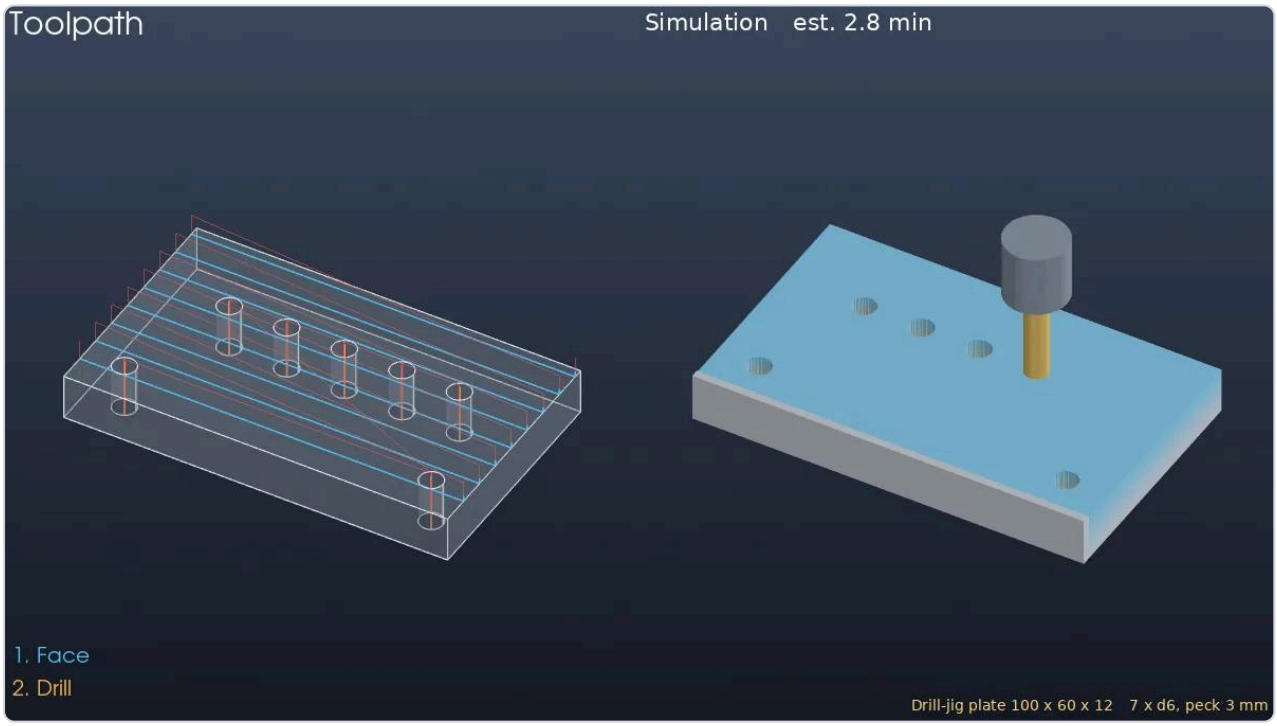
> **Note:** There is no tapping cycle yet. Use **Thread Mill** to cut internal threads by milling.

Double-click an operation in the CAM Manager to open its operation box, where you can edit every value. Its **Parameters** section has the **General**, **Geometry**, **Tool**, **Levels**, **Passes**, **Linking** and **Stock** pages (see Operation Parameters for the meaning of each field).

Try it: Drill a drill-jig plate

A drill jig puts every hole in a batch in exactly the same place. This 100 × 60 × 12 plate has five guide holes 15 mm apart and two dowel holes: one Drill operation, seven holes, peck drilling.

1. Model a 100 × 60 × 12 plate. Draw seven Ø6 circles: five at Y 40 (X 20, 35, 50, 65, 80) and two at (10, 10) and (90, 10). **Extrude Cut** them **Through All**.
2. **CAM** › **New Job** › **Mill Template**. In **Stock**, choose **Part + allowance** (0.5 mm on top to face) and set **Support height (parallels)** to 5.
3. Add a **Face** on the top face and change its cutter to **T3 12mm 3F face mill**.
4. Click **Drill**, click the top face of the plate (or all seven hole walls) and press **Enter**. The **Holes** table lists all 7 Ø6 holes as *Through*, all ticked. MECHCAD chooses the T2 6 mm drill and sets Top Z 12 and Depth 14.303 (12 mm of plate + the 118° point + 0.5 mm to break through).
5. **Peck depth (0=off)** is already 3 (the Drill default) and **Drill retract Z** is 14 (top + 2).
6. **Simulate**. The estimated time is about 2.8 min. Each hole is pecked in 3 mm steps, back up to Z14 between pecks. The G-code shows one `G83` cycle line with `Q3`, the remaining hole positions, and `G80`.



Drill-jig plate: seven drill positions on the left, the simulation paused at the fourth guide hole on the right

Challenge: Untick the two dowel holes in the **Holes** table and check that the G-code has only five positions left. Or set **Peck depth** to 0: the cycle becomes `G81`.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Slot, Step, Chamfer, Corner Round, Engrave and Thread Mill Premium

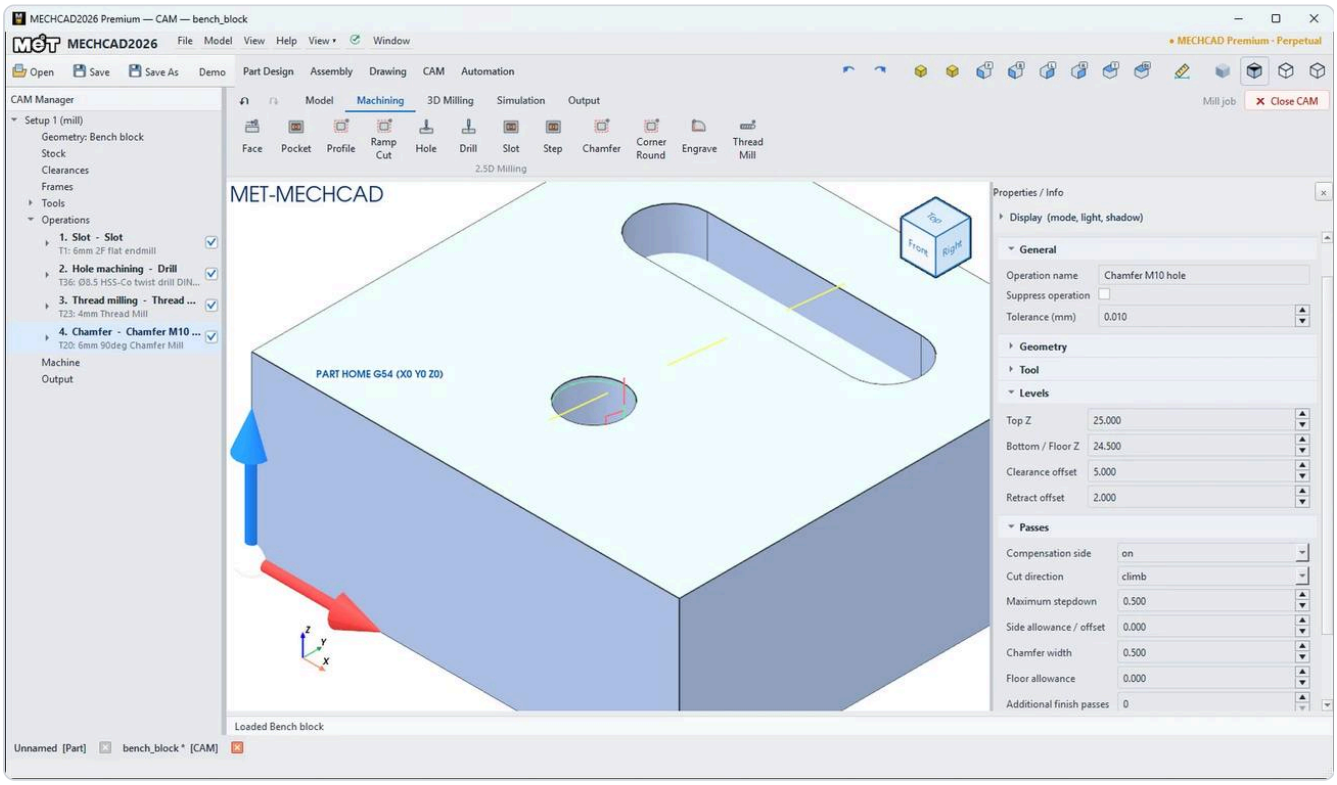
More 2.5D operations: slots, steps, chamfers, corner rounds, engraving and internal thread milling.

What it does

Operation	Select	Description / defaults
Slot	Face	Mills a closed slot. You can adjust stepdown/stepover.
Step	Face	Mills a step or open pocket.
Chamfer	Edge	Chamfers along the edge with a V/chamfer cutter (width 0.5 mm).
Corner Round	Edge	Rounds the edge with a corner-round cutter (R1).
Engrave	Edge	Engraves shallowly along a line.
Thread Mill	Hole (several holes)	Mills an internal thread with a helical path (created with depth 5, Ø = the picked hole Ø (at least 6) and pitch 1.5, so set the real thread size).

Where to find it

Machining › 2.5D Milling



Chamfer operation

Steps

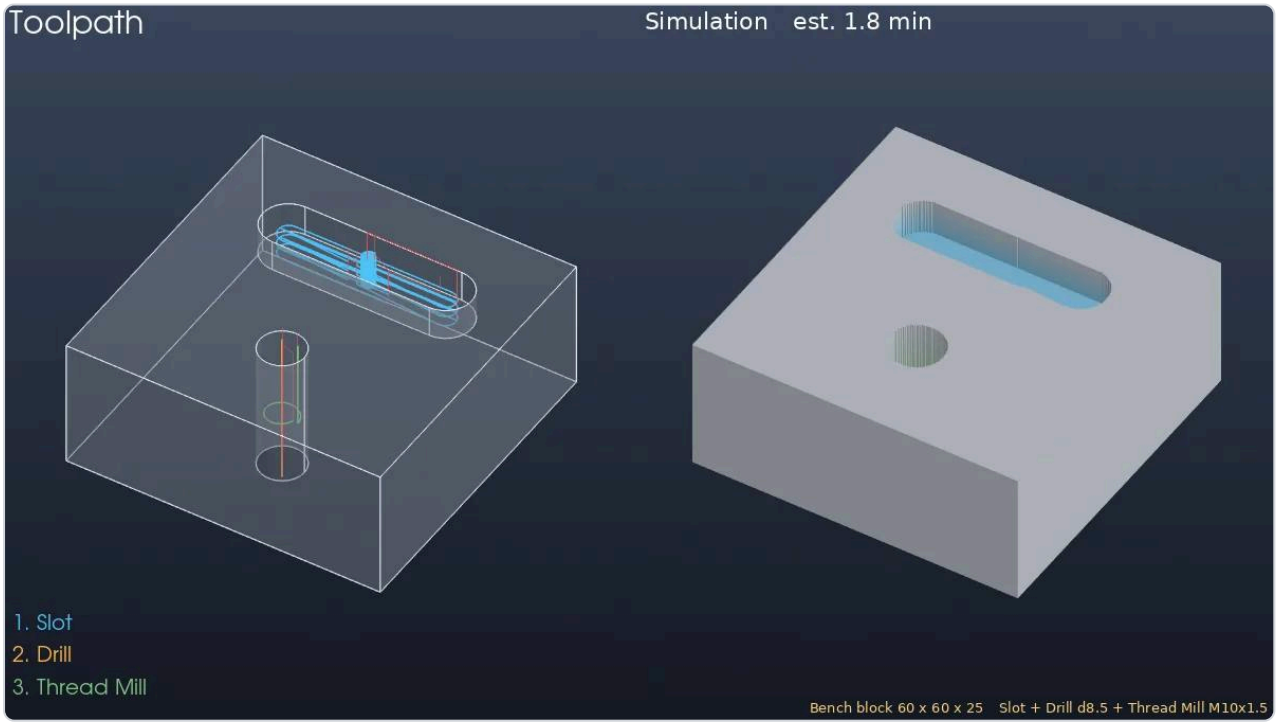
1. Click the command.
2. Click the face/edge/hole shown in the table, then press Enter.
3. Adjust the values in the Properties panel.

You can edit every operation in the **Properties** panel on the right when you click the operation in the CAM Manager. The panel sections are **General**, **Geometry**, **Tool**, **Levels**, **Passes**, **Linking** and **Stock** (see Operation Parameters for the meaning of each field).

Try it: Finish a machinist's bench block

A 60 × 60 × 25 bench block with a closed slot for pins and an M10 tapped hole for a clamp: a Slot, a Drill and a Thread Mill.

1. Model a 60 × 60 block **Extruded** 25 mm. On the top face, draw a 40 × 10 slot with its ends centred at X 15 and X 45 (Y 44) and **Extrude Cut** it 6 mm. Draw a Ø8.5 circle at X 30 Y 18 and **Extrude Cut** it **Through All**.
2. **CAM** › **New Job** › **Mill Template**. In **Stock**, set **Support height (parallels)** to 5.
3. Click **Slot**, click the slot floor and press **Enter**. You get Top Z 25 and Floor Z 19 with T1.
4. Click **Drill**, click the hole wall and press **Enter**. Choose cutter **T36 Ø8.5 HSS-Co twist drill (M10 tap)**, set **Peck depth** to 4 and **Depth** to 28.
5. Click **Thread Mill**, click the same hole wall and press **Enter**. Set **Thread major diameter** to 10, **Thread pitch** to 1.5 and **Depth** to 15. The tool is T23 4mm Thread Mill.
6. **Simulate** (about 1.8 min). Solid Verify shows **GOUGE** around the hole. That is expected: the model has a plain Ø8.5 hole, while the thread really cuts out to Ø10.



Bench block: Slot, Drill and Thread Mill toolpaths and the finished block

Challenge: Add a **Chamfer** on the top edge of the hole with T20 and **Chamfer width** 0.8, so the clamp screw starts easily.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

3D milling: Rough, Rest, Z-Level, Raster, Pencil and Mold Tactic Premium

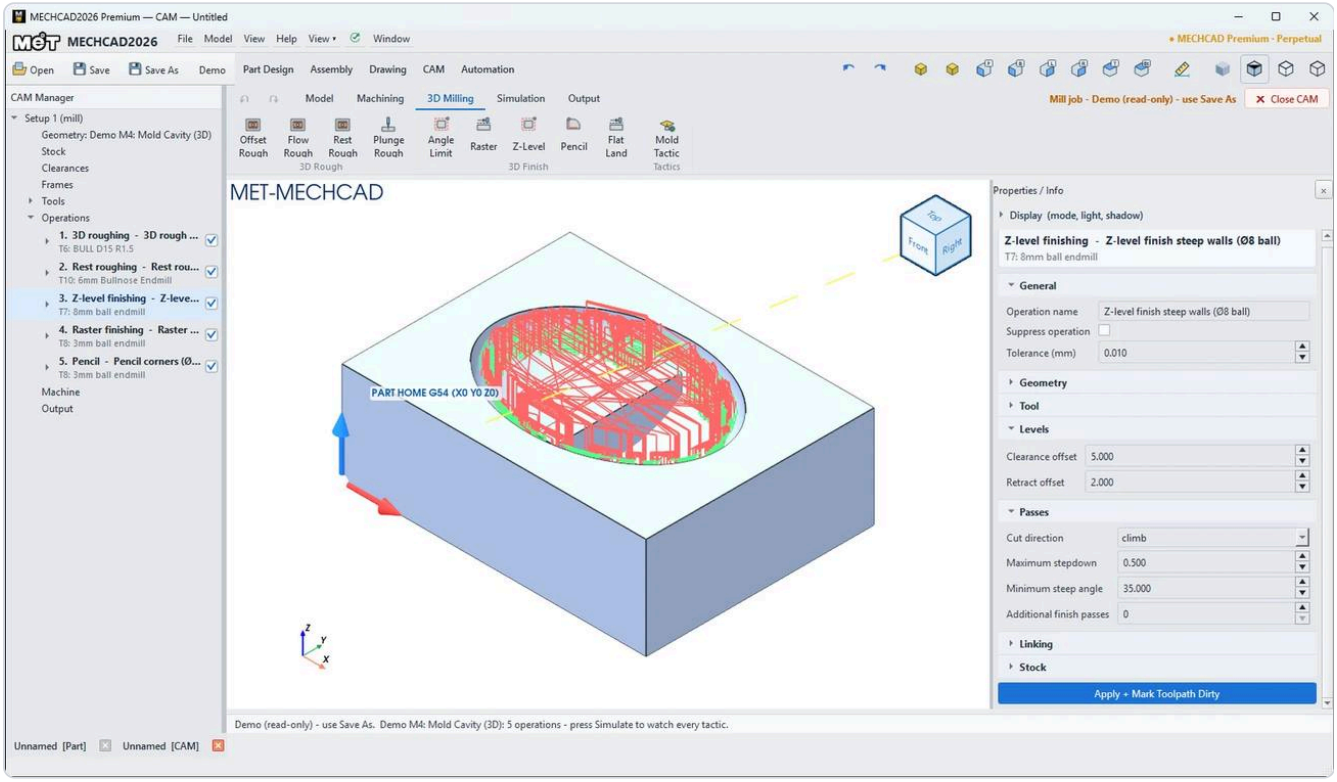
Rough and finish 3D mold surfaces.

What it does

Operations for 3D curved surfaces such as moulds (a workpiece is required).

Where to find it

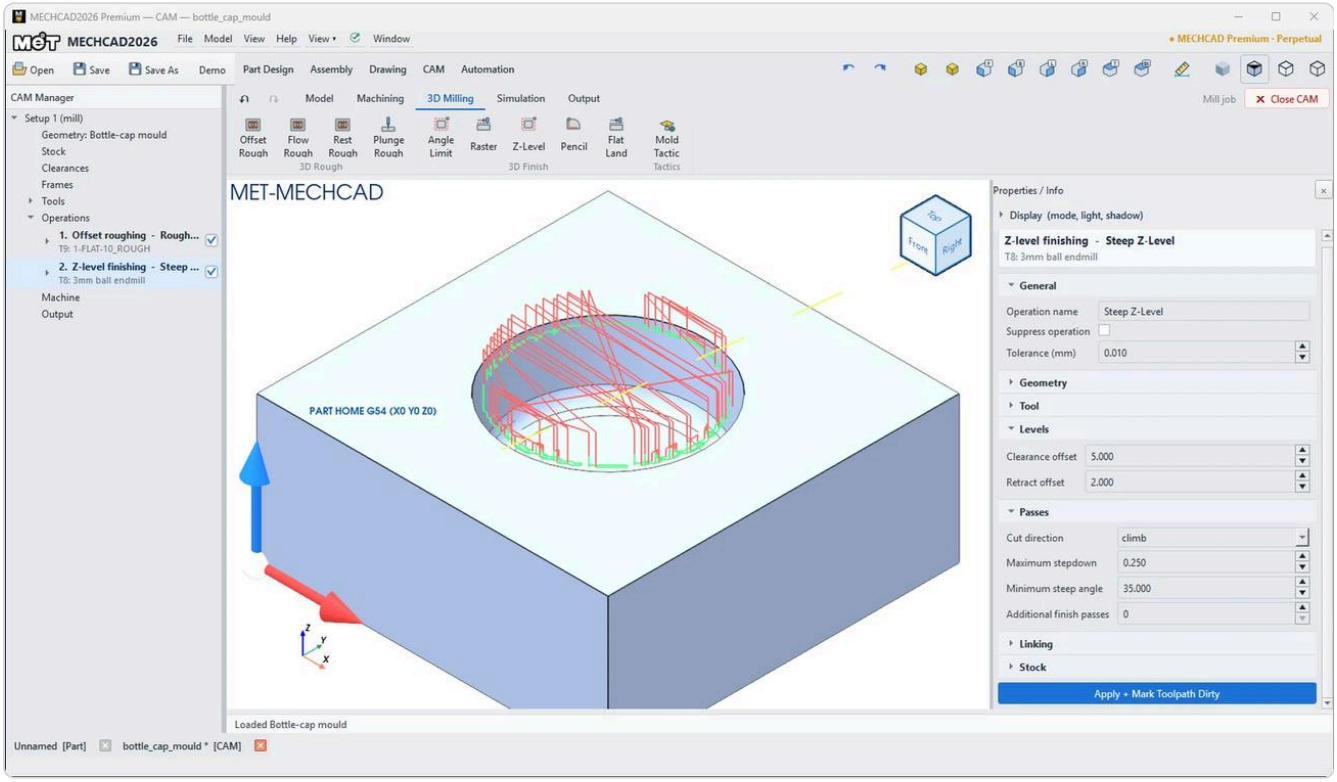
3D Milling › 3D Rough / 3D Finish / Tactics



Sample Mold Cavity (3D) job

Operation	Description	Defaults
Rough Offset 2D	Roughing with offset loops at constant Z levels.	stepdown 3, stepover 0.65×D, stock left 0.5
Smooth Flow Rough	Raster-style roughing.	stepdown 1.5, stepover 0.5×D, stock left 0.5
Rest Rough	Cuts only the material left by a reference operation (automatically references the previous operation).	stepdown 1, stock left 0.2

Operation	Description	Defaults
Plunge Rough	Roughing by axial plunging.	pitch 6 mm (0 = automatic 0.8×D), stock left 0.5
Angle Limiting	Mixed finishing: raster on shallow surfaces, Z-level on steep surfaces.	steep angle 40°, overlap 3°
Shallow Raster	Finishes shallow surfaces with a ball-nose cutter.	stepover 0.25 mm, maximum angle 40°
Steep Z-Level	Finishes steep surfaces with contour lines.	stepdown 0.25, minimum steep angle 35°
Pencil Cleanup	Cleans up concave corner grooves.	concavity 0.15
Flat Land	Finishes flat floors with a flat-end cutter.	stepover 0.4, maximum angle 5°
Mold Tactic	Creates a set of 5 linked operations: 01 Rough Offset 2D, 02 Rest Rough, 03–04 Angle Limiting semi/finish, 05 Pencil.	–



Z-level finish operation

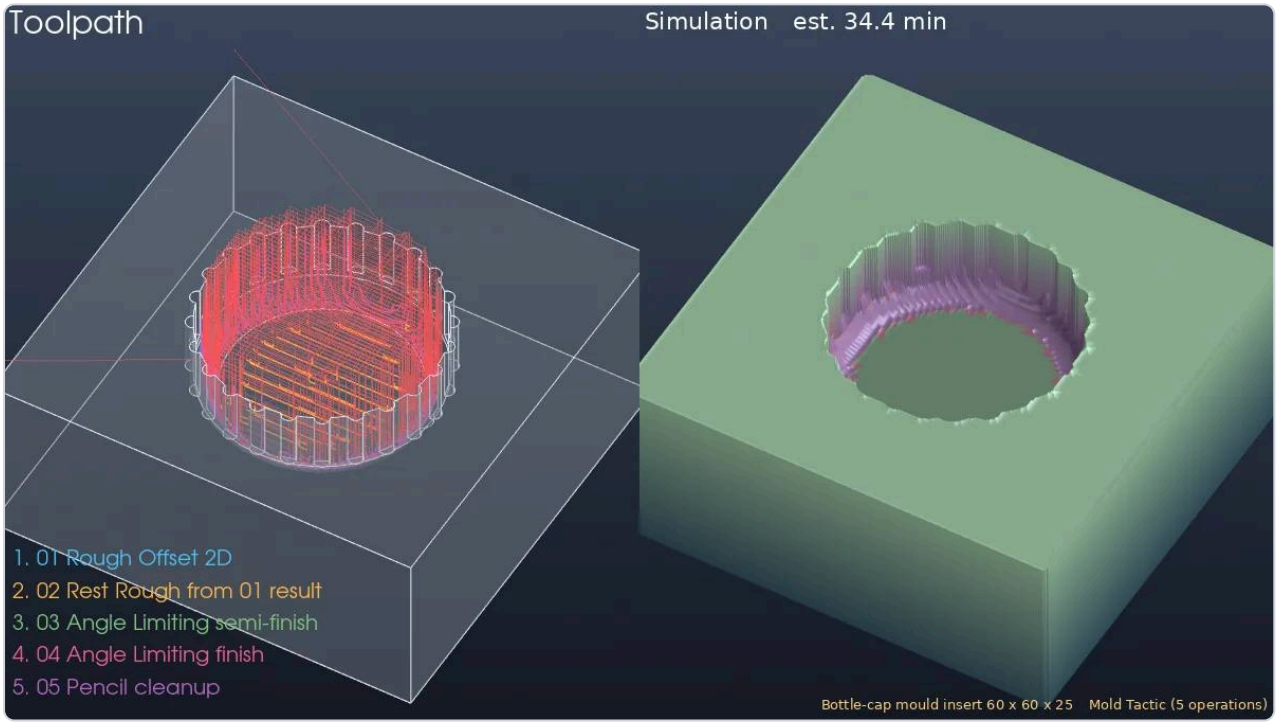
> **Note:** 3D toolpaths are calculated when you Simulate or right-click › **Show Toolpath** (*3-axis toolpath not calculated yet*).

You can edit every operation in the **Properties** panel on the right when you click the operation in the CAM Manager. The panel sections are **General**, **Geometry**, **Tool**, **Levels**, **Passes**, **Linking** and **Stock** (see Operation Parameters for the meaning of each field).

Try it: Mould a bottle cap

Ever wondered how a bottle-cap mould is cut? This is a 60 × 60 × 25 insert with a Ø32 × 12 cap cavity, an R3 radius at the bottom and 24 grip ribs. One click on Mold Tactic builds the whole five-step strategy.

1. Model a 60 × 60 block **Extruded** 25 mm. On the top face, draw a Ø32 circle at the centre and **Extrude Cut** it 12 mm. **Fillet** the bottom edge of the cavity R3.
2. Sketch on the top face again: a Ø2 circle centred on the cavity edge (X 46, Y 30). Make a circular pattern of 24 copies around the centre and **Extrude Cut** them 8 mm. These are the grip ribs.
3. **CAM** › **New Job** › **Mill Template**.
4. Click **3D Milling › Tactics › Mold Tactic**. Five operations appear: 01 Rough Offset 2D (T9, 10 mm), 02 Rest Rough from 01 (T1), 03 Angle Limiting semi-finish (T4, 6 mm ball), 04 Angle Limiting finish (T8, 3 mm ball) and 05 Pencil cleanup (T8).
5. Click **Simulate**. The 3D toolpaths are calculated now (about 25 s on our test PC). The estimated time is about 34 min.
6. Look at the ribs. The 3 mm ball cannot reach the bottom of the 2 mm grooves, so Solid Verify reports **RESIDUAL** there. Details this small need a smaller cutter or EDM.



Bottle-cap mould: the finishing toolpaths in the cavity and the simulated insert

Challenge: Add **3D Finish › Steep Z-Level** and compare its finish on the cavity wall with 04 Angle Limiting finish.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Turning: Face, Rough, Finish, Axial Drill and Part Off Premium

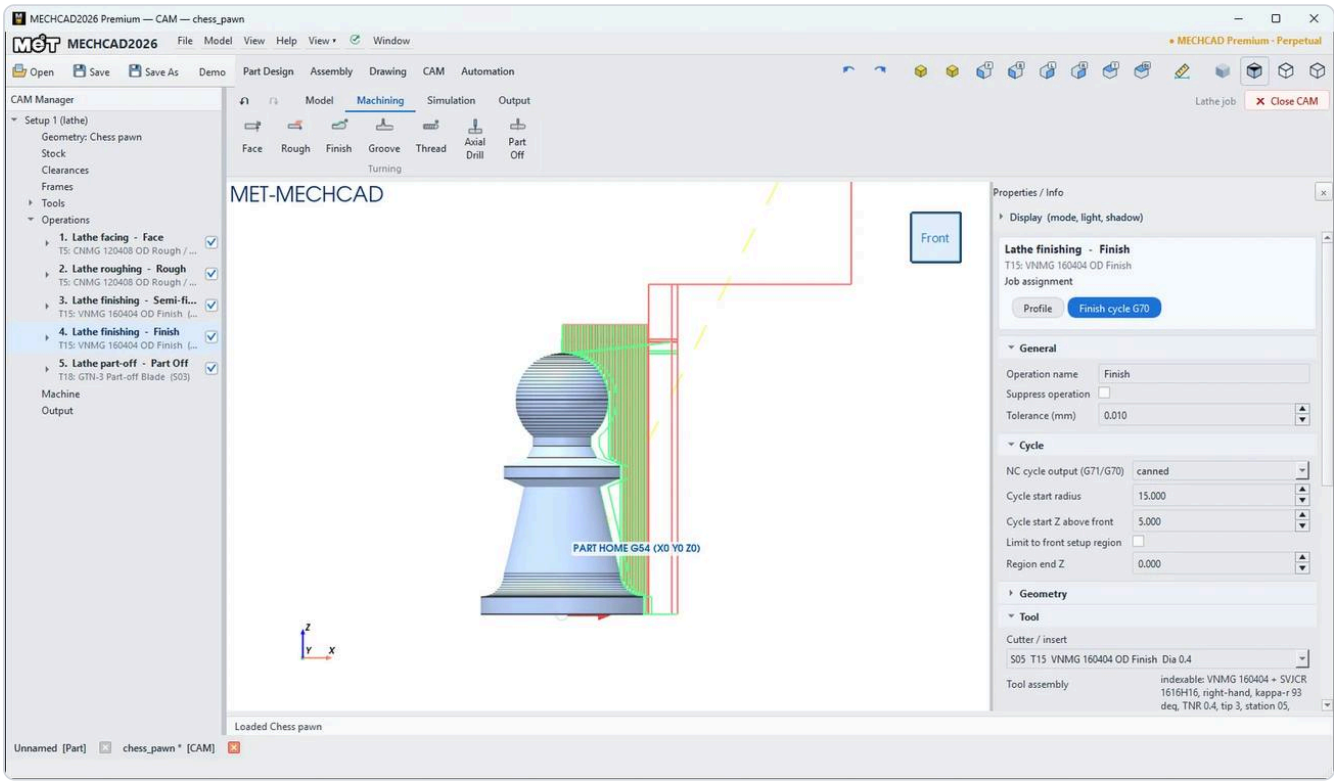
Create facing, G71 rough turning, G70 finish turning, axial drilling and part-off operations.

What it does

Lathe operations (choose the **Turn Template** in New Job). They work from the turning profile of the part, so you do not need to select surfaces.

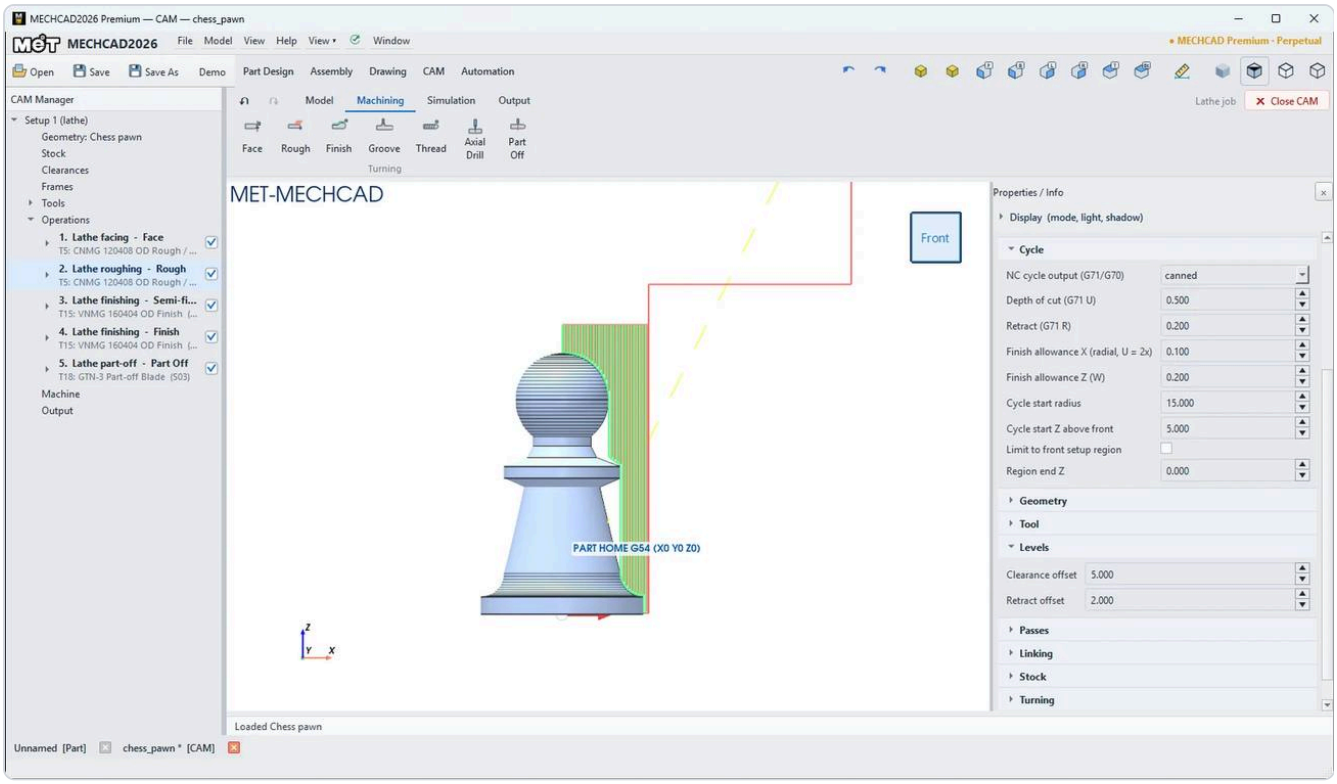
Where to find it

Machining › Turning › Face, Rough, Finish, Groove, Thread, Part Off, Axial Drill



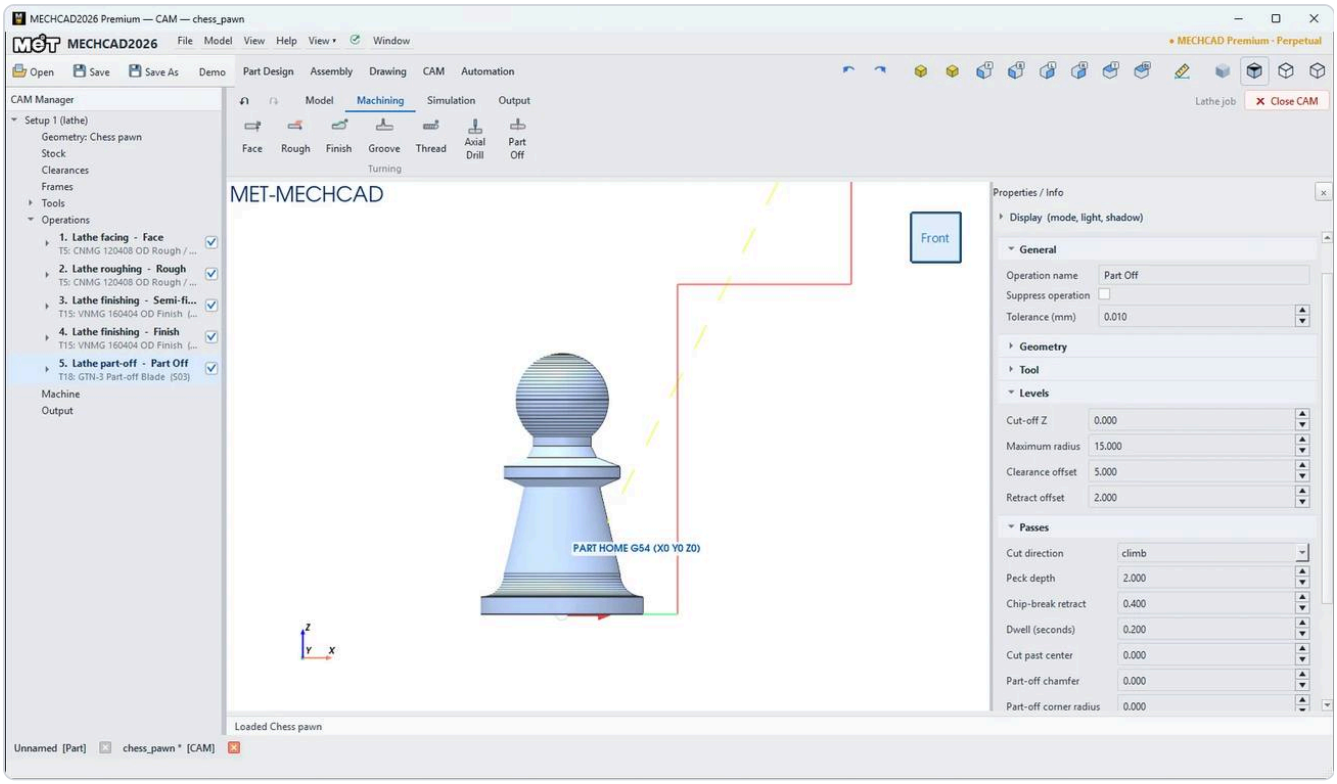
Turning job for a chess pawn: Face, Rough, Finish and Part Off toolpaths

Operation	Defaults	Default tool
Face	allowance 1, depth per pass 0.5, finish 0.1, from outside to centre	T5
Rough (G71)	filled from the part profile: depth per pass 0.5 (G71 U), retract 0.2 (G71 R), finish allowance X 0.1 / Z 0.2, 0.3 mm/rev, towards the chuck	T5
Finish (G70)	Follows the profile, excluding shoulder grooves	T15
Axial Drill	peck 4, retract 1, expanded or G74 cycle	T19
Part Off	peck 2, chip-break retract 0.4, dwell 0.2 s	T18



OD Rough G71 settings (Cycle section)

Fields in the Cycle section (Rough)	Description
NC cycle output (G71/G70)	canned (G71/G70) or expanded
Depth of cut (G71 U) / Retract (G71 R)	Depth per pass / retract distance
Finish allowance X (radial, U = 2x) / Finish allowance Z (W)	Finish allowance
Cycle start radius / Cycle start Z above front	Cycle start point

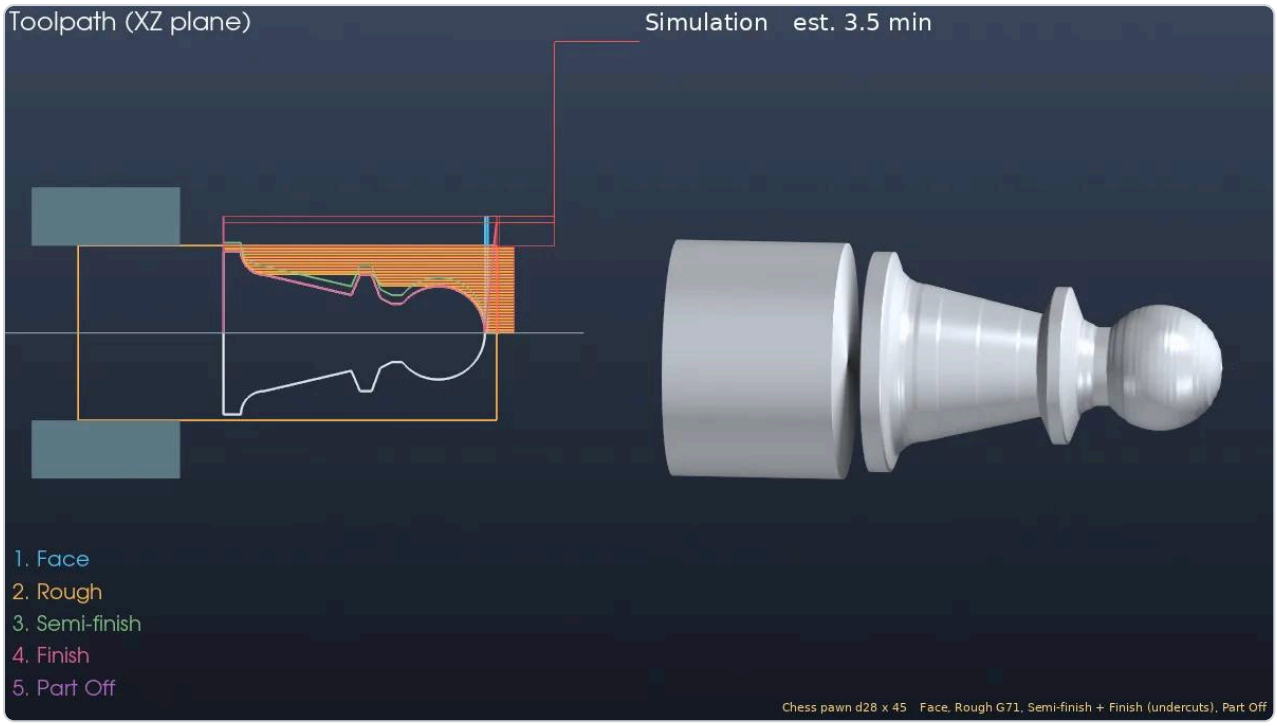


Part off operation

Try it: Turn a chess pawn

A Ø28 × 45 aluminium pawn with a base, a cove, a tapered waist, a collar and a ball head is a classic first lathe project. You use Face, Rough, two Finish passes and Part Off.

1. In **Part Design**, sketch on the **XZ** plane the half outline on the right of the Z axis: base radius 14 (Z 0 to 3), an R4 cove in to radius 10 at Z 7, a straight taper to radius 6.5 at Z 22, a collar of radius 10 (Z 23.5 to 25.5), a neck of radius 5 at Z 29, and a ball of R8 centred at Z 37 that ends on the axis at Z 45. **Revolve** it 360° about Z.
2. **CAM** › **New Job** › **Turn Template**. The stock is a Ø30 × 72 bar with 25 mm in the chuck, and the post is fanuc_lathe.
3. Click **Face** and **Rough**. The rough is filled in from the profile: G71 U0.5 R0.2, finish allowance 0.1 (X) and 0.2 (Z), 0.3 mm/rev.
4. Click **Finish** and rename it `Semi-finish`. In **Turning**, set **Undercut / groove handling** to `include`, and in **Passes** set **Profile offset** to 1.5.
5. Click **Finish** again and set **Undercut / groove handling** to `include`. This pass follows the neck and waist with the 35° VNMG insert (T15). With `exclude` it would stop at the ball.
6. Click **Part Off** (T18, cut at Z0), then **Simulate**. The estimated time is about 3.5 min, and the pawn drops off the bar.



Chess pawn: lathe toolpaths in the XZ plane on the left, the parted-off pawn on the right

Challenge: Make the ball R9 in Part Design and run the job again. Does the semi-finish still leave enough for the final pass?

See also

- Groove and Thread

You can edit every operation in the **Properties** panel on the right when you click the operation in the CAM Manager. The panel sections are **General**, **Geometry**, **Tool**, **Levels**, **Passes**, **Linking** and **Stock** (see Operation Parameters for the meaning of each field).

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Turning: Groove and Thread (G75 / G76) Premium

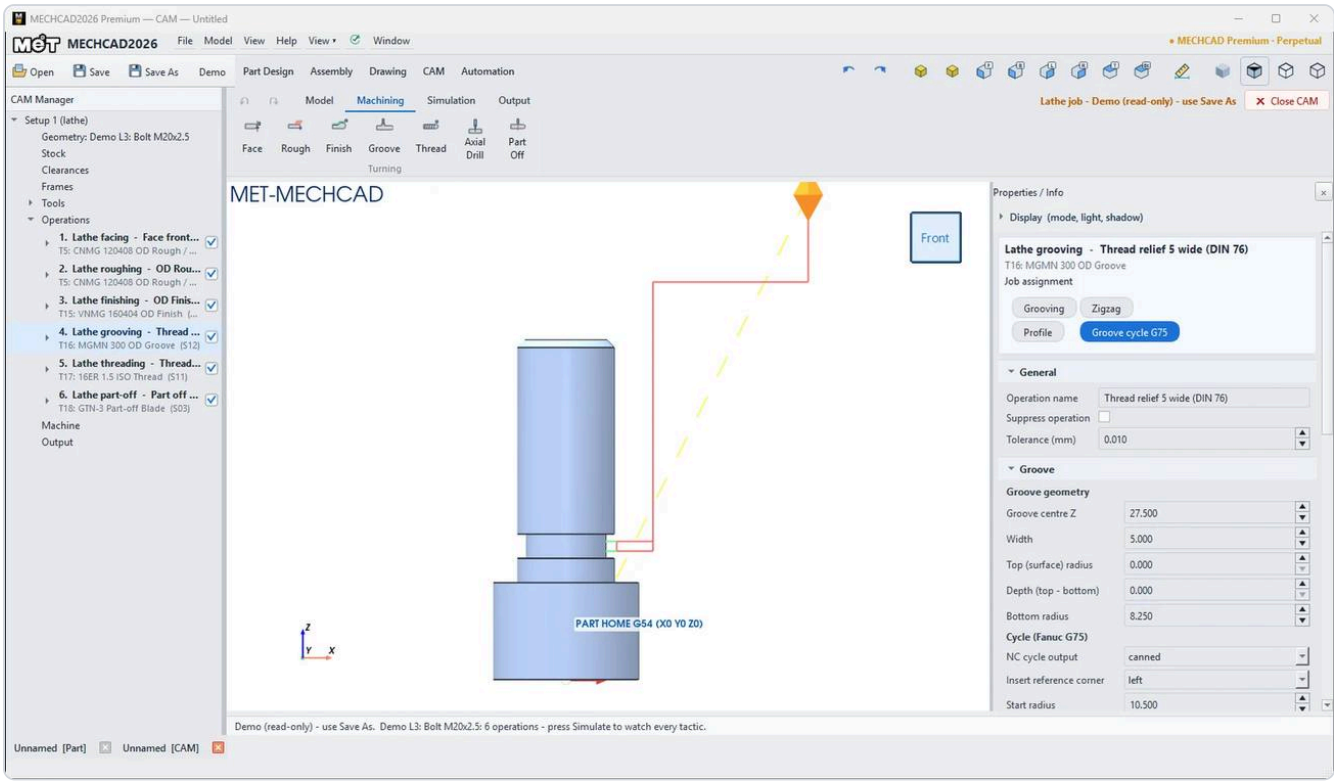
Turn grooves and M3–M64 metric threads with G75 and G76 cycles.

What it does

- **Groove** turns a groove (single pass, peck or multi-pass) with the G75 cycle.
- **Thread** turns external/internal threads with the G76 cycle (or G92 / G32).

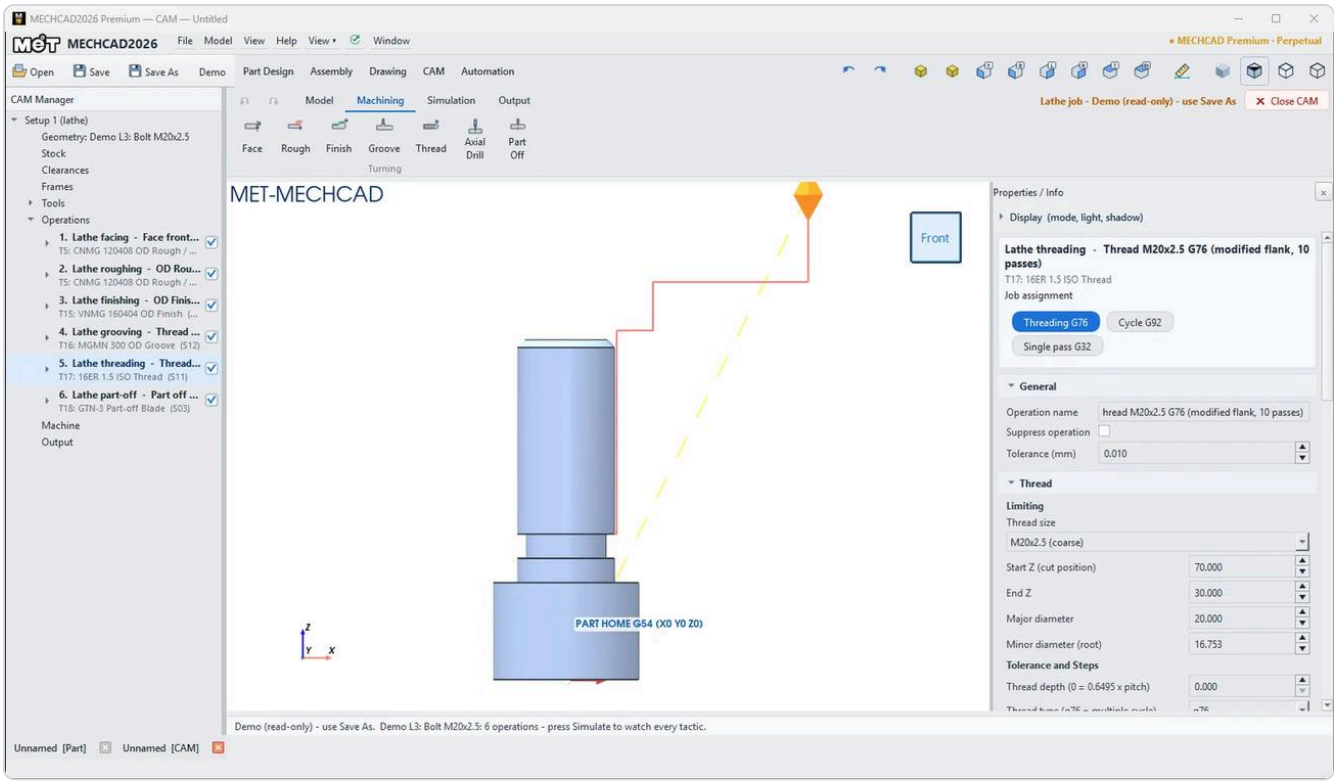
Where to find it

Machining › Turning › Groove / Thread



Groove operation (thread relief)

Groove	Default / description
Groove centre Z, Width, Top (surface) radius, Depth, Bottom radius	Groove position and size (width 4)
NC cycle output	canned (G75) / expanded
Insert reference corner	left / right / center
Peck depth (P) / Shift (Q) / Retract e (G75 R)	2 / 0 = insert width / retract distance
Groove finish pass	Finishes the groove surface (on)



Thread operation M20x2.5 G76

Thread	Default / description
Thread size	Custom or M3–M64 (coarse/fine). Fills in the pitch and Ø.
Start Z (cut position) / End Z	Thread range
Major diameter / Minor diameter (root)	Major Ø / root Ø
Thread depth (0 = 0.6495 x pitch)	Thread depth
Thread type	g76 (multiple cycle) / g92 / g32
First pass depth / Min depth	0.25 / 0.05
Hand of thread / Pitch / Thread angle	Right / 1.5 / 60°
Infeed	modified_flank / radial / alternate_flank
Pullout length (G76 chamfer) / Finish depth (G76 R)	1 / 0.1
Number of finish / Number of spring	Number of finish passes / spring (dwell) passes (1)

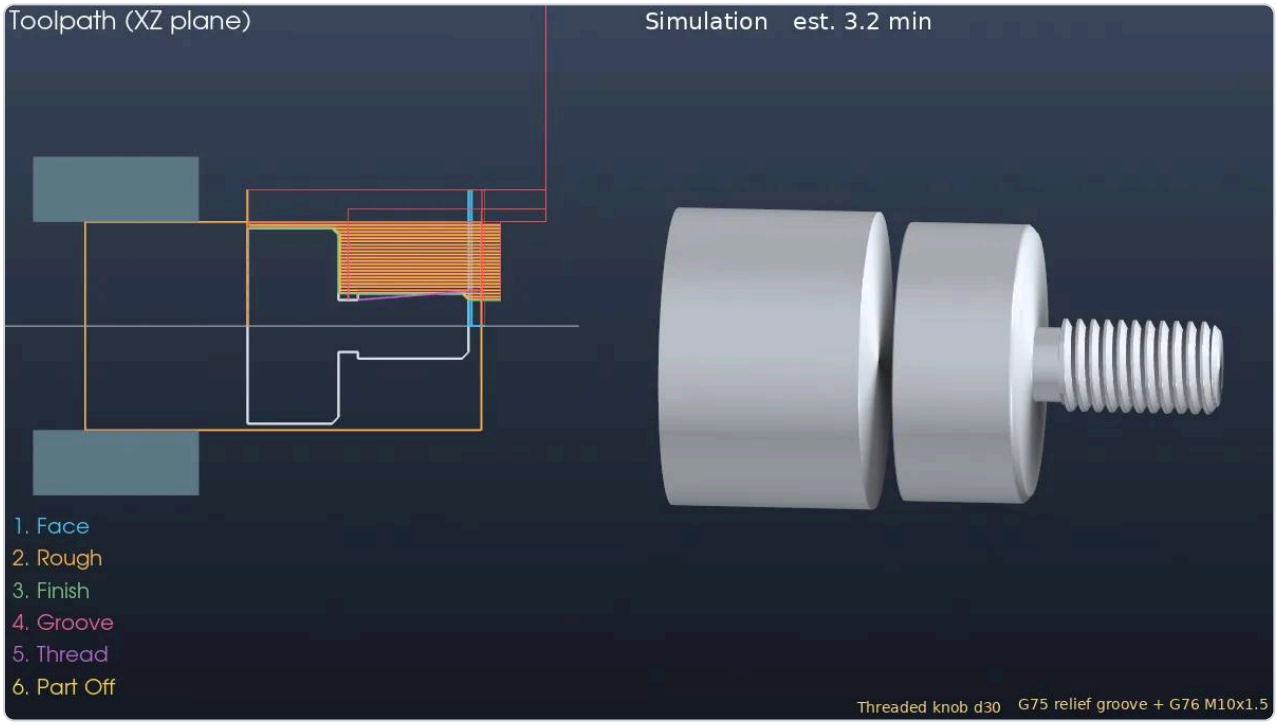
> **Note:** When MECHCAD finds the groove or the thread in the part profile, it fills in these values instead of the defaults above: Groove peck depth 1 with the finish pass off; Thread first pass depth 0.5, min depth 0, pull-out 0 and a size proposed from the diameter. Check the pitch: a Ø10 stem is proposed as M10x1.

You can edit every operation in the **Properties** panel on the right when you click the operation in the CAM Manager. The panel sections are **General**, **Geometry**, **Tool**, **Levels**, **Passes**, **Linking** and **Stock** (see Operation Parameters for the meaning of each field).

Try it: Turn a threaded knob

A Ø30 knob with an M10 × 1.5 stud is handy for jigs and fixtures. After Face, Rough and Finish come the two stars of this page: a thread-relief Groove and a G76 Thread.

1. In **Part Design**, sketch on **XZ** the half outline (radius, Z): (0, 0) › (15, 0) › (15, 13) › (14, 14) › (4, 14) › (4, 17) › (5, 17) › (5, 33) › (4, 34) › (0, 34). **Revolve** it about Z. That gives a Ø30 head, a 3 mm wide Ø8 relief groove and a Ø10 stem with a 1 mm chamfer.
2. **CAM** › **New Job** › **Turn Template** (Ø32 bar). Add **Face**, **Rough** and **Finish**.
3. Click **Groove**. MECHCAD finds the relief by itself: Groove centre Z 15.5, Width 3, Top radius 5, Bottom radius 4, Peck depth (P) 1, canned G75 with T16 MGMN 300 (a 3 mm insert, exactly the groove width).
4. Click **Thread**. MECHCAD proposes M10x1 for the Ø10 stem. Change **Thread size** to `M10x1.5 (coarse)`. **Start Z** 34 and **End Z** 17 end the thread in the relief groove. The tool is T17 16ER 1.5 ISO.
5. Add **Part Off**, then **Simulate**. The estimated time is about 3.2 min and Solid Verify shows PASS. The G-code contains `G75 X8. Z14. P1000 Q3000 F0.1` and `G76 X8.0515 Z17. P974 Q500 F1.5`.



Threaded knob: lathe toolpaths on the left, the M10 thread and relief groove after simulation on the right

Challenge: Change **Infeed** to `radial` or **Hand of thread** to `left`, then compare the G76 lines.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Simulation (machining simulation and collision check) Premium

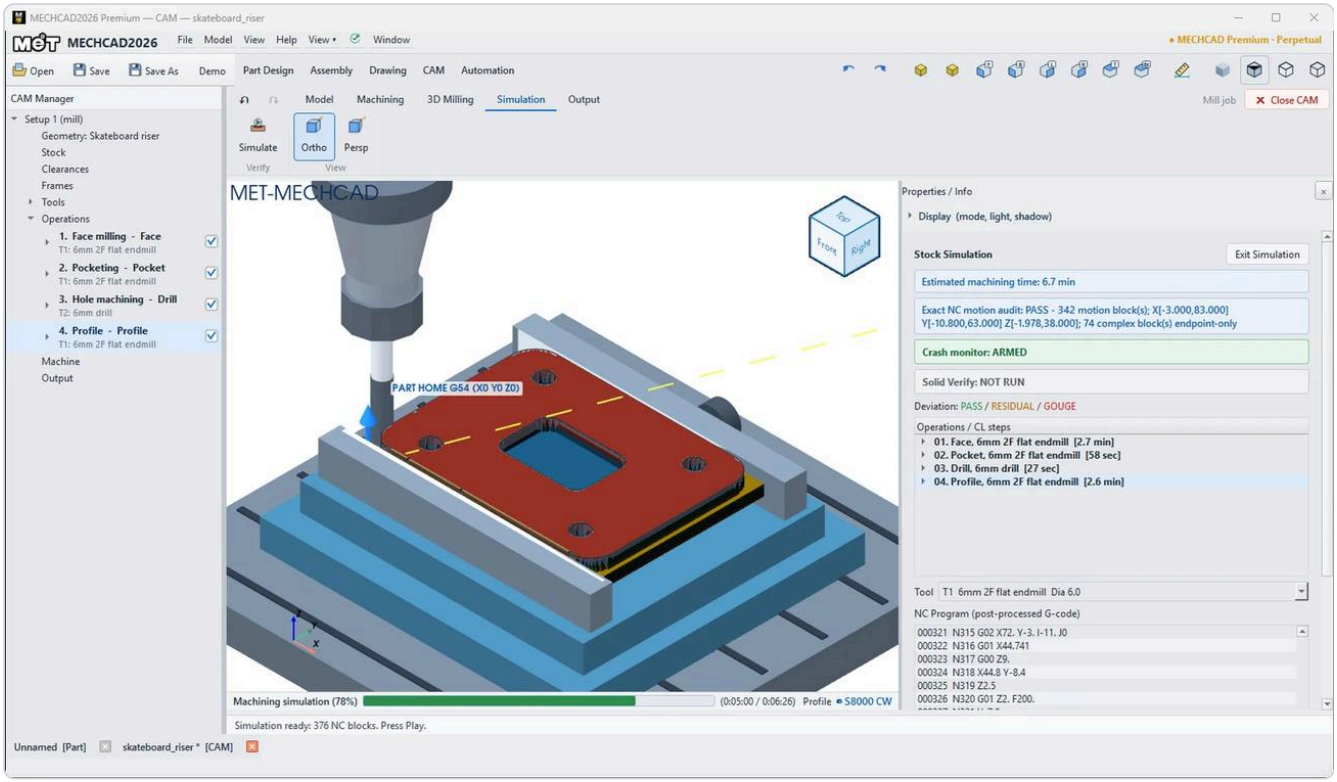
Simulate material removal, check collisions and cutting time, and step through the G-code line by line.

What it does

Simulates material removal on the raw stock using the post-processed G-code. It checks the cutter and holder for collisions with the vise/chuck, and checks the surface result (Solid Verify).

Where to find it

Simulation › Verify › Simulate (and **Output › Simulate**)



Stock Simulation screen

Steps

1. Click **Simulate**. The program checks the setup first. If there are errors, *Setup errors — … blocked* opens and you must fix them first (warnings ask *Continue anyway?*).
2. Wait for *Preparing toolpaths…* until *Simulation ready: N NC blocks. Press Play.* appears.
3. Click **Play** / **Pause** / **Step** / **Next Operation >>** / **Reset**. Adjust **Speed** and **Loop**.
4. After playback, check the **Solid Verify** result (PASS / RESIDUAL / GOUGE).
5. Click **Exit Simulation** to go back to editing.

Item	Description
Estimated machining time	Estimated cutting time
Exact NC motion audit	Checks the G-code motion against the travel limits
Crash monitor: ARMED	Checks for collisions. If one occurs, it stops with *CRASH ALARM* before the crash.
Solid Verify / Deviation	PASS / RESIDUAL (material left) / GOUGE (overcut)
Operations / CL steps	Double-click to jump to a step.

Item	Description
Tool	Changes the tool of the selected operation.
NC Program (post-processed G-code)	The current line is highlighted.
Part ghost / Stock transparent / Tool holder / Vise / Chuck	Display options
Status bar	*Machining simulation (NN%)*, time, operation, spindle speed

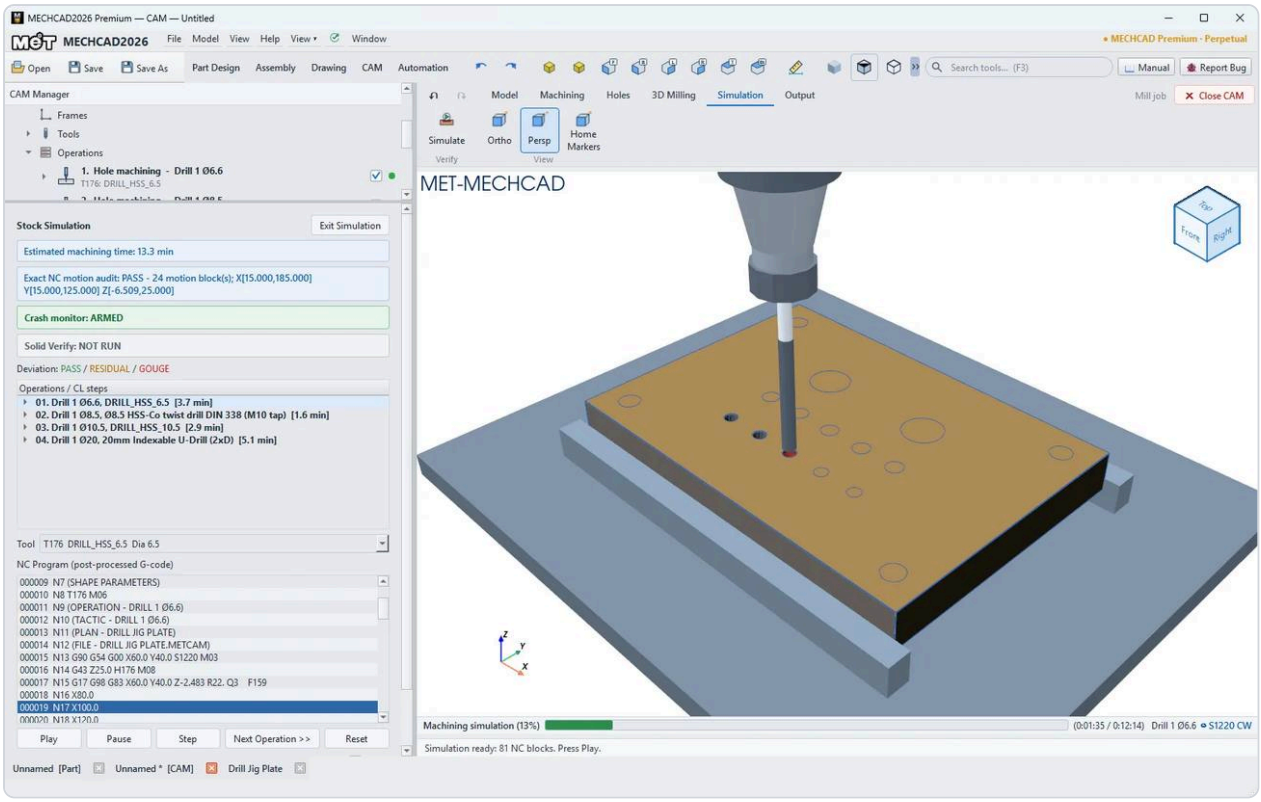
Cutter shape in the simulation

The cutter in the simulation is drawn from the actual tool, and material is removed with the same shape, so the result matches the tool geometry.

Tool type	Simulated tip
Ball	Half sphere
Bull nose (with a Corner radius)	Flat with rounded corners of that radius
Chamfer, Engraver	A cone of the **Included angle**, with a flat tip of the **Tip diameter**
Drill, Spot/Center	A point of the drill's point angle (for example 118°); an angle of 0 gives a flat-bottom drill
Corner round	A concave rounded tip of that radius
Other tools (Flat)	Flat

When you edit a tool (for example the angle or tip of a chamfer mill, a corner radius, the diameter or the type), the cutter is redrawn at once, the toolpaths that use the tool are marked for recalculation, and the next simulation uses the new shape.

> **Note:** The setup check covers, for example: a missing tool or a tool value of 0, holes with no position, a floor above the top surface, cutting into the table, a depth greater than the flute length, a stepdown of 0, threads and grooves in the wrong order, a post that does not match the machine type, and a WCS that is not G54–G59.

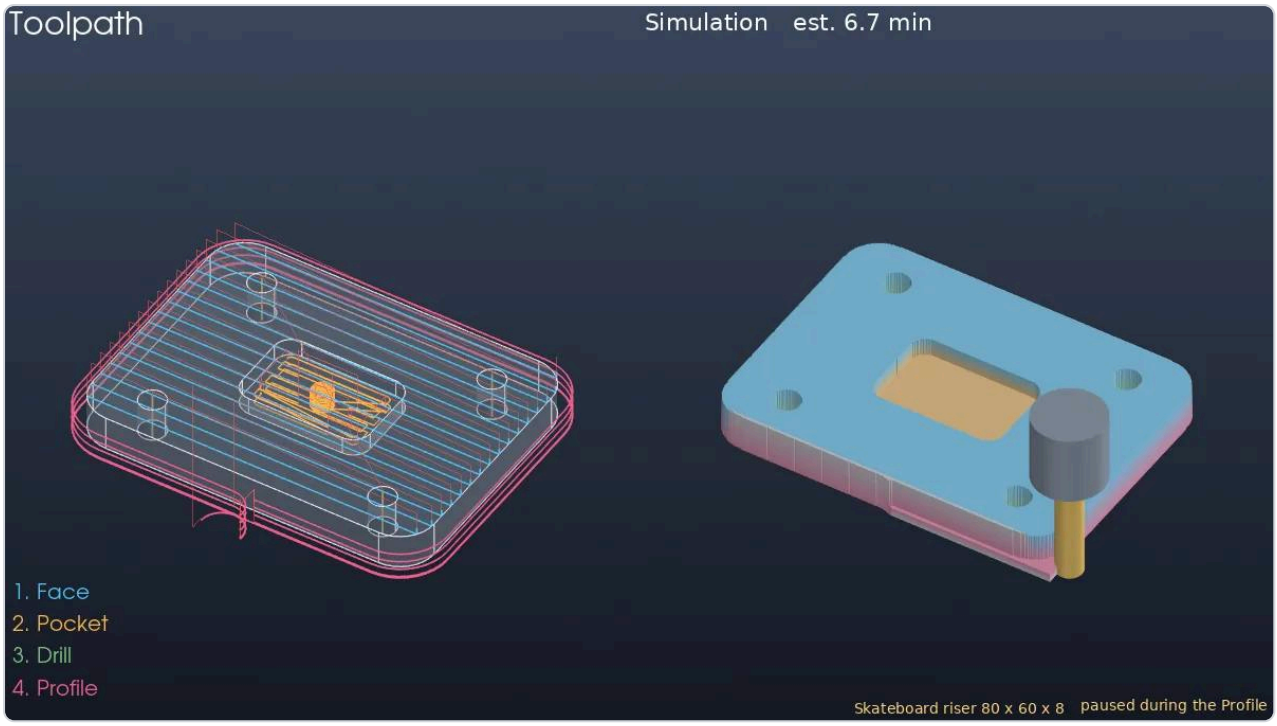


Drill-jig plate in Stock Simulation: the Ø6.5 drill, drawn with its point angle, running a G83 peck cycle

Try it: Watch a skateboard riser being cut

An 80 × 60 × 8 aluminium riser pad with four truck holes (54 × 41 pattern) and a lightening pocket. You build four operations, then use the simulator to check each one before you trust the G-code.

1. Model an 80 × 60 plate with R8 corners, **Extruded** 8 mm. Add four Ø6 holes at X 13 / 67 and Y 9.5 / 50.5, **Through All**. Add a 30 × 20 pocket with R4 corners in the centre, 4 mm deep.
2. **CAM** › **New Job** › **Mill Template**. In **Stock**, choose **Part + allowance** (0.5 mm on top to face) and set **Support height (parallels)** to 3. Add a **Face** (top), a **Pocket** (pocket floor), a **Drill** (four hole walls, **Depth** 10) and a **Profile** (top outer edge, **Bottom / Floor Z** −0.3).
3. Click **Simulate** and wait for *Simulation ready: N NC blocks. Press Play.*
4. Click **Play** and drag **Speed**. Use **Next Operation >>** to jump from one operation to the next. Click **Pause** during the Profile: the current line is highlighted in **NC Program**.
5. Turn on **Stock transparent** or **Tool holder** to look at the cutter from any side.
6. After playback, read **Solid Verify**. The check compares on a grid (about 0.6 mm on this part), so a few single cells exactly on a vertical wall can be flagged. A real problem shows up as a coloured patch.
7. Click **Exit Simulation** to go back to editing.



Skateboard riser: all four toolpaths on the left, the simulation paused during the Profile on the right

Challenge: Set the pocket's **Maximum stepdown** to 4 (one level only), simulate again and compare the estimated time with the 6.7 min you had before.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Machine Manager and post-processors Premium

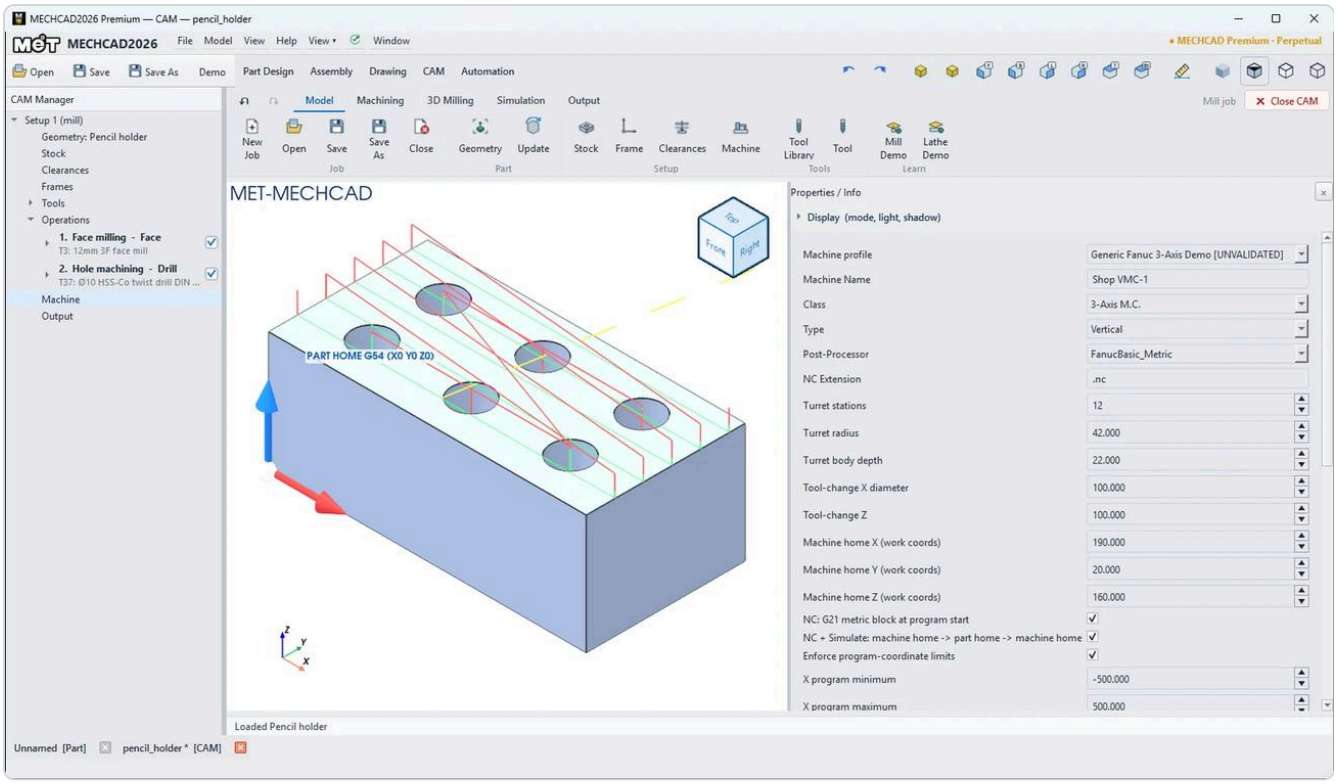
Choose the machine, post-processor and travel limits, and edit the G-code format.

What it does

Sets up the CNC machine and the post-processor used to generate G-code. You can edit the G-code format yourself.

Where to find it

The **Machine** node in the CAM Manager, or **Model › Setup › Machine** / **Output › Post**



Machine Manager in the Properties panel

Option	Description
Machine profile	Custom / legacy (no envelope), Generic Fanuc 3-Axis Demo [UNVALIDATED], Generic Fanuc 2-Axis Lathe Demo [UNVALIDATED]
Machine Name / Class / Type	Name / 3-Axis M.C. or Lathe / Vertical or Horizontal
Post-Processor	FANUC_MET 3Axis, FanucBasic_Metric, FanucBasic_Inch, Fanuc15i_3axis, BASIC_ISO, GRBL, LinuxCNC, fanuc_lathe, fanuc_mill, grbl
NC Extension	G-code file extension (`.nc`)
Turret stations / radius / body depth, Tool-change X / Z	Turret settings (turning)
Machine home X/Y/Z	Machine home position
NC: G21 metric block at program start	Adds G21 at the start of the program
NC + Simulate: machine home -> part home -> machine home	Moves through home
Enforce program-coordinate limits / X/Y/Z minimum / maximum	Allowed coordinate limits
Commissioning status	Status of calibration against the real machine
G-code Pattern (editable)	JSON G-code format of the post

Edit the G-code format (pattern)

The **G-code Pattern** field has the keys `name`, `unit` (mm/inch), `prog`, `header[]`, `tool_change[]`, `rapid`, `feed`, `arc_cw`, `arc_ccw`, `footer[]` and the placeholders `{PROG} {WCS} {TOOL} {RPM} {SAFE_Z} {X} {Y} {Z}`

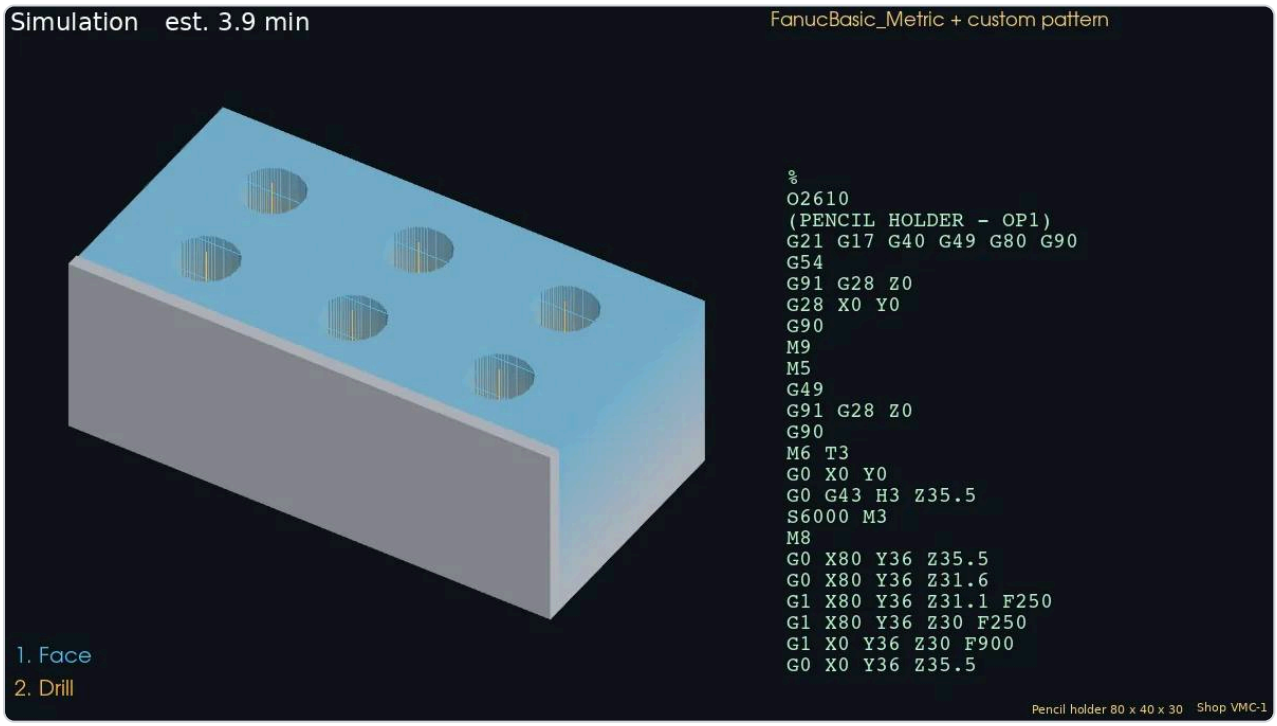
`{I} {J} {FEED}`. Your edits are saved per post in that job. If the JSON is invalid, you see *Pattern JSON error: ...*.

> **Note:** fanuc_lathe, fanuc_mill and grbl are programmed posts (*class-based post — no editable pattern*). Profiles marked **[UNVALIDATED]** have not been calibrated against a real machine. Always test the first program on the machine with care.

Try it: Put your own header on a pencil holder program

An 80 × 40 × 30 desk pencil holder with six Ø10 holes, 25 mm deep. You name your machine, choose FanucBasic_Metric, and add your own program number and comment to the program.

1. Model an 80 × 40 × 30 block. Draw six Ø10 circles at X 15 / 40 / 65 and Y 11 / 29 and **Extrude Cut** them 25 mm.
2. **CAM** › **New Job** › **Mill Template**. Add a **Face** with T3. Add a **Drill** on the six hole walls with cutter **T37 Ø10 HSS-Co twist drill**, **Peck depth** 5 and **Depth** 25. (For blind holes the default depth goes down to Z0, so you must set it.)
3. Click **Machine** in the CAM Manager. Set **Machine Name** to `Shop VMC-1` and **Post-Processor** to `FanucBasic_Metric`.
4. In **G-code Pattern (editable)**, change `"prog": 1000` to `"prog": 2610`. In `"header"`, add `"(PENCIL HOLDER - OP1)"` after `"O{PROG}"`. Apply.
5. **Output › G-code**. The program now starts with `%`, `O2610`, `(PENCIL HOLDER - OP1)` and `G21 G17 G40 G49 G80 G90`.



Pencil holder: the simulated block and the first lines of the custom FanucBasic_Metric program

Challenge: Clear **NC + Simulate: machine home -> part home -> machine home** and generate again. The `G28 X0 Y0` line after the header disappears.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Generate G-code and Shop PDF (setup sheet) Premium

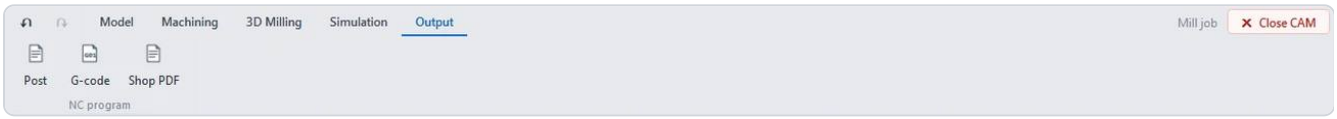
Export G-code and print a setup sheet for the machine operator.

What it does

- **Generate G-code** creates the NC program file for the selected post-processor.
- **Print Shop PDF** creates an A4 landscape setup sheet: machine data, WCS, stock, tool sequence, operation sequence, estimated time, and a pre-run checklist.

Where to find it

Output › NC program › Post / G-code / Shop PDF



Output tab

Steps

1. Check the job with Simulate first.
2. Click **G-code** and choose a file name in the *Export G-code* window (default `part.nc`). When it finishes you see *Wrote N lines to …*.
3. Click **Shop PDF** and choose a file name (default `<job name>_Shop_Setup.pdf`).

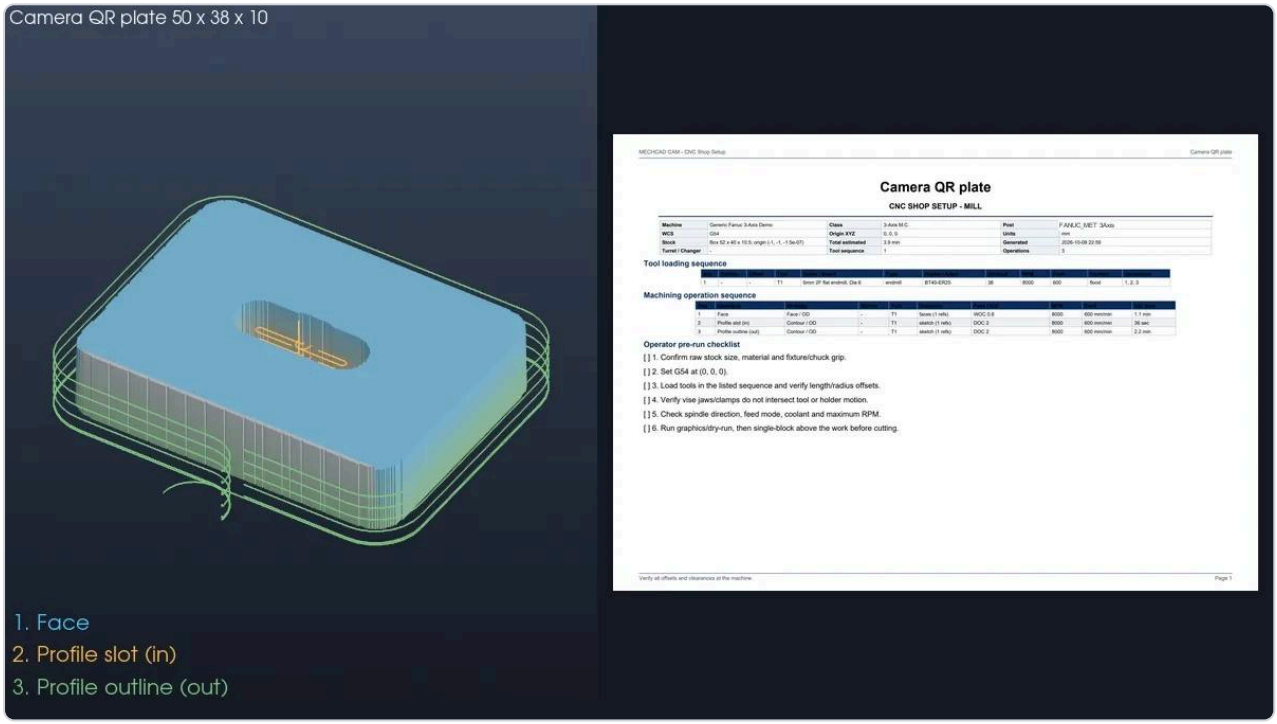
Shop PDF contents	Details
Setup table	Machine, Class, Post, WCS, Origin XYZ, Units, Stock, Total time
Tool loading sequence	Seq, Station, Offset, Tool, Cutter / Insert, Type, Holder / Arbor, Stickout, RPM, Feed, Coolant
Machining operation sequence	Seq, Operation, Strategy, Station, Tool, Geometry, Pass / Cut, RPM, Feed, Est. time
Operator pre-run checklist	Checks to make before running the machine

> **Note:** G-code and Shop PDF generation run the same setup checks as Simulate. Always verify offsets and clearances at the machine (*Verify all offsets and clearances at the machine.*).

Try it: Post a camera quick-release plate

A 50 × 38 × 10 quick-release plate with a 7 mm slot for a 1/4" camera screw. Three operations, then the payoff: an NC file for the machine and a one-page setup sheet for the operator.

1. Model a 50 × 38 plate with R4 corners, **Extruded** 10 mm. Draw a 20 × 7 slot in the middle (ends centred at X 18.5 and X 31.5, Y 19) and **Extrude Cut** it **Through All**.
2. **CAM** › **New Job** › **Mill Template**. Set **Support height (parallels)** to 3. Add a **Face**. Add a **Profile** on the slot's top edge with **Bottom / Floor Z** −0.3, **Lead arc radius** 1, **Lead-in length** 0.5 and **Lead-out length** 0.5. Add a **Profile** on the outer top edge with **Bottom / Floor Z** −0.3.
3. **Simulate** first. The estimated time is about 3.9 min.
4. Click **Output › G-code**. The setup check warns that the demo machine profile is UNVALIDATED. Read the warning, then click **Yes**. Save as `camera_qr_plate.nc` . You see *Wrote 293 lines to …*.
5. Click **Shop PDF** and save it (default name `<part name>_Shop_Setup.pdf`). The page lists the machine (Generic Fanuc 3-Axis Demo), the FANUC_MET 3Axis post, the stock (52 × 40 × 10.5), the tool T1 and three operations: Face 1.1 min, slot Profile 36 s and outline Profile 2.2 min. It ends with the pre-run checklist.



Camera plate: the simulated plate on the left, the CNC Shop Setup PDF on the right

Challenge: Change the Face cutter to T3 and print the PDF again. The tool loading table now has two tools.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Sample CAM jobs (Mill Demo / Lathe Demo) Premium

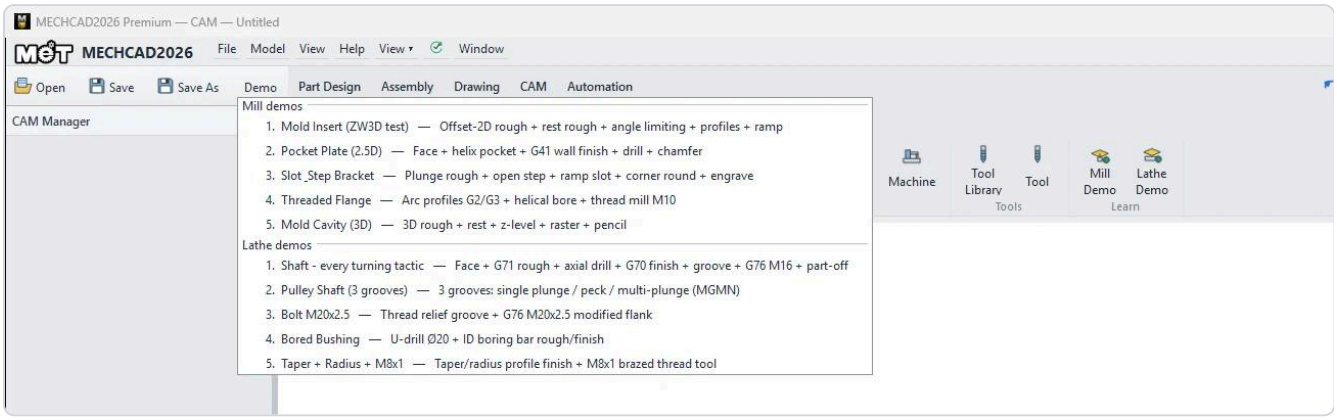
Five sample milling jobs and five sample turning jobs that cover every operation.

What it does

Ready-made jobs with the workpiece, tools and all operations, ready to Simulate. They are good for learning.

Where to find it

Model › Learn › Mill Demo / Lathe Demo, or the **Demo** button on the top bar while you are on the CAM tab



Sample CAM jobs menu

Mill demos	What you learn
1. Mold Insert	Offset-2D rough, Rest rough, Angle limiting, Profile, Ramp
2. Pocket Plate (2.5D)	Face, helical-entry Pocket, G41 wall finishing, Drill, Chamfer
3. Slot & Step Bracket	Plunge rough, Open step, Ramp slot, Corner round, Engrave
4. Threaded Flange	G2/G3 curved Profile, Helical bore, M10 Thread mill
5. Mold Cavity (3D)	3D rough, Rest, Z-level, Raster, Pencil

Lathe demos	What you learn
1. Shaft - every turning tactic	Face, G71 rough, Axial drill, G70 finish, Groove, G76 M16, Part-off
2. Pulley Shaft (3 grooves)	Single-pass / peck / multi-pass grooves
3. Bolt M20x2.5	Thread relief groove, G76 M20x2.5 modified flank
4. Bored Bushing	U-drill Ø20, ID boring rough/finish
5. Taper + Radius + M8x1	Taper/curved profile, brazed threading tool

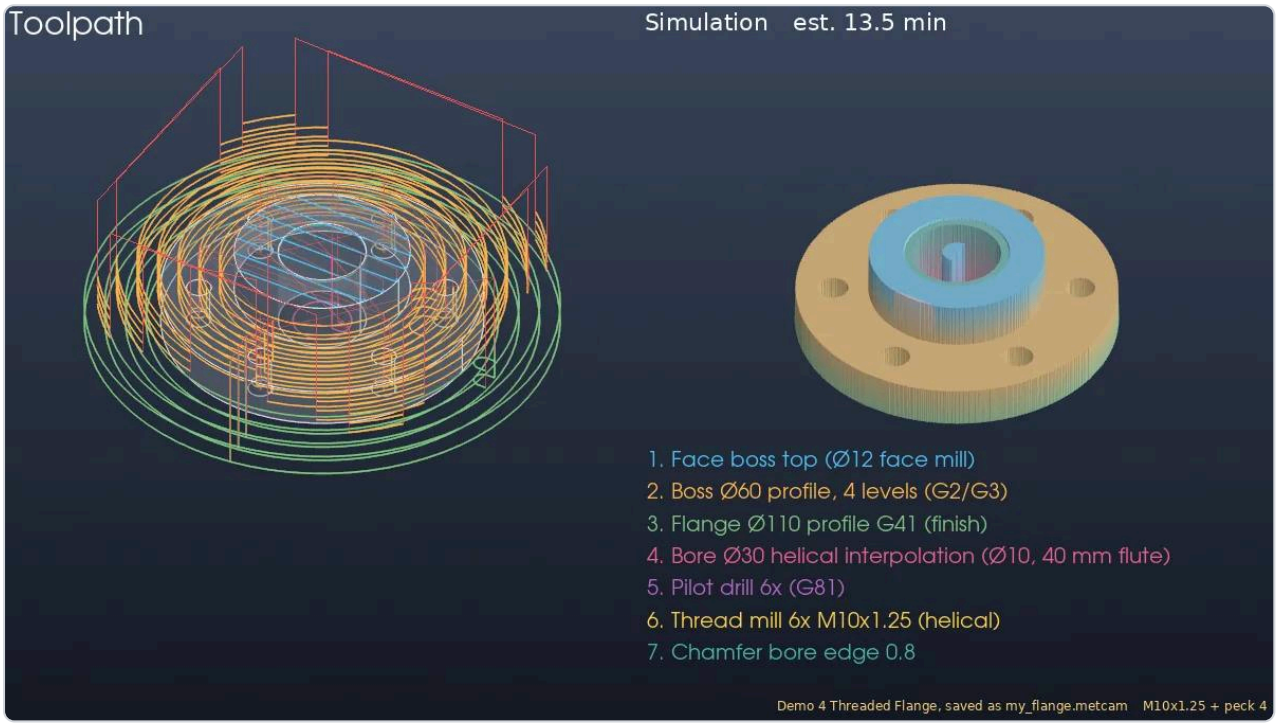
> **Note:** The demo jobs are read-only. Use **Save CAM As** to save one as your own job (you cannot save into the demo folder).

Try it: Make the Threaded Flange demo your own

Demo 4 is a Ø110 flange with a boss, a Ø30 bore and six M10 holes on a bolt circle, with seven ready operations. You turn it into your own job and change the threads to a fine pitch.

1. On the **CAM** tab, click **Model › Learn › Mill Demo › 4. Threaded Flange**. The badge shows *Demo (read-only) - use Save As*.
2. Click **Simulate** to see all seven operations. The estimated time is about 13.2 min.
3. Click **Output › Save CAM As** and save `my_flange.metcam` in your own folder.

4. Click **Thread mill 6x M10x1.5 (helical)**. Set **Thread pitch** to 1.25 and rename it `Thread mill 6x M10x1.25`.
5. Click **Pilot drill 6x (G81)** and set **Peck depth** to 4.
6. **Simulate** again. The estimated time is now about 13.5 min, because the pecking and the finer pitch add about 20 s.



Threaded Flange demo after the changes: toolpaths of all seven operations and the simulated flange

Challenge: Open **Lathe Demo › 3. Bolt M20x2.5**, save it as your own job, change **Thread size** to M20x1.5 and compare the G76 lines in the G-code.

> **Note:** CAM is available only in MECHCAD **Premium** (and during the trial).

Chapter 8 quiz: CAM

Choose the one best answer. Answers are in Appendix A

1. **A new job from the Mill Template uses which work frame and post-processor?**

   A. G59 and GRBL
   B. G54 and FANUC_MET 3Axis
   C. G54 and fanuc_lathe
   D. G55 and LinuxCNC

2. **You see "Source geometry is out of date" after editing the part. What do you do?**

   A. Start a New Job
   B. Delete all operations
   C. Click Update Geometry, then recalculate the toolpaths
   D. Change the post-processor

3. **After simulation, Solid Verify shows GOUGE. What does it mean?**

   A. Material is left on the part
   B. The tool hit the vise
   C. A toolpath was not calculated
   D. The part was overcut

4. **A machine profile is marked [UNVALIDATED]. What does that mean?**

   A. It has not been calibrated against a real machine; test the first program with care
   B. It cannot generate G-code
   C. It is only for lathes
   D. It needs a Premium licence upgrade

5. **In a Drill operation, what does Peck depth = 0 mean?**

   A. Drilling is disabled
   B. Drill in one pass
   C. Use a G83 cycle
   D. Drill to the default depth of 10 mm

6. **What does Print Shop PDF create?**

   A. A PDF of the part drawing with dimensions
   B. The G-code printed as PDF
   C. An A4 landscape setup sheet with machine data, WCS, stock, tool and operation sequence, estimated time and a pre-run checklist
   D. An invoice for the machining job

CHAPTER 9

Import / Export

1. MECHCAD file types and files you can open
   All MECHCAD native file types, and the CAD formats you can open, import and export.
2. Import CAD (STEP, IGES, STL) and CAD Import Options
   Import CAD files as a part or assembly, and what the import options mean.
3. Open SolidWorks and Alibre files in MECHCAD
   Read .sldprt/.sldasm and .ad_prt/.ad_asm directly, and what you get after import.
4. Export STEP and STL
   Export a part as STEP for other CAD programs, and as STL for 3D printers.

MECHCAD file types and files you can open Standard

All MECHCAD native file types, and the CAD formats you can open, import and export.

What it does

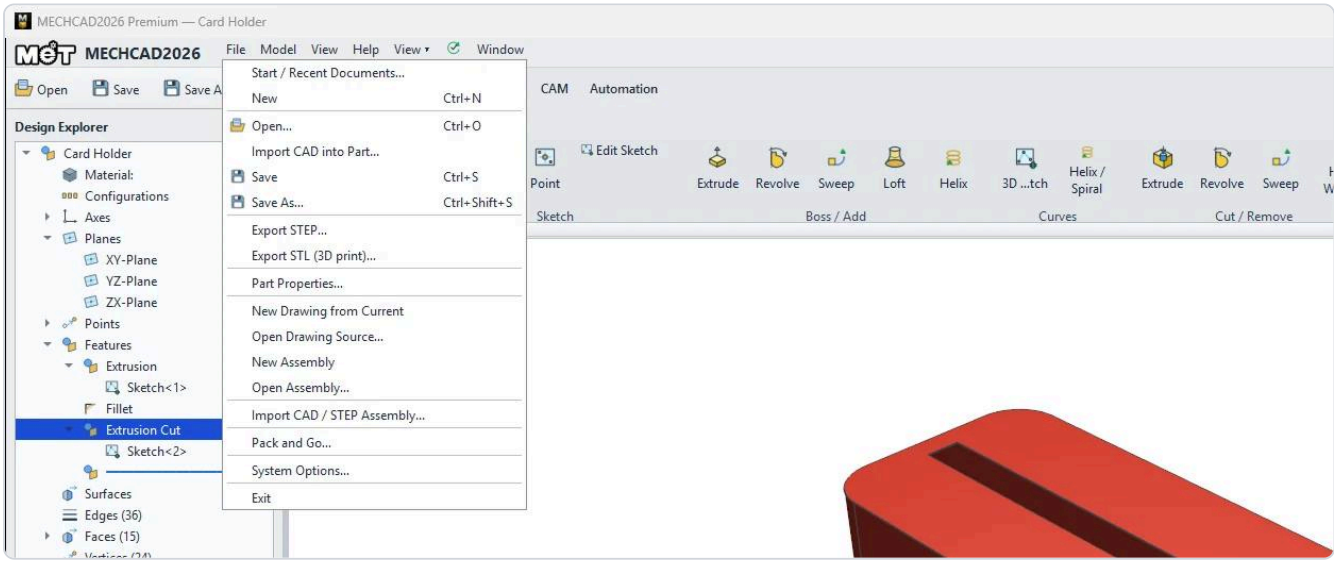
Summarises the file types that MECHCAD saves, opens, imports and exports.

Native files

Extension	Type	Opens in tab
`.metprt` (and legacy `.mcd_prt`)	MECHCAD Part	Part Design
`.metasm` (and legacy `.mcd_assm`)	MECHCAD Assembly	Assembly
`.metdrw`	MECHCAD Drawing	Drawing
`.metcam`	MECHCAD CAM	CAM
`.metauto`	MECHCAD Automation Cell	Automation
`.metpkg`	MECHCAD Package (Pack and Go)	Unpacks, then opens the assembly

Every file type has a preview image in Windows Explorer and opens with a double-click (direct installer). If the file is already open, MECHCAD switches to its tab.

Files you can open/import (File › Open...)



File menu

Format	Extension	Result
STEP	`.step .stp`	Part or Assembly (structure and positions are kept)
IGES	`.iges .igs`	Part
Mesh	`.stl`	Part (can be converted to a solid)
BREP	`.brep`	Part

Format	Extension	Result
SolidWorks	`.sldprt` `.sldasm`	Part as editable sketches + feature tree / Assembly converted to MECHCAD files, without SolidWorks
Alibre	`.ad_prt` `.ad_asm`	Part as editable sketches + features / Assembly converted to MECHCAD files (for sharing work with Alibre users)
Siemens NX	`.prt`	Part (solid)
Parasolid	`.x_t` `.x_b`	Part
MET feature JSON	`.mfj`	Part

Export

From	Format	Command
Part	STEP, STL	File › Export STEP… / Export STL (3D print)…
Assembly	STL, HTML 3D	Share › Export STL / Export HTML
Drawing	PDF, DXF, DWG	Output › Export PDF / DXF / DWG
CAM	G-code (`.nc`), PDF setup sheet	Output › G-code / Shop PDF
Motion	MP4, GIF	● Record, Export Video…, Record GIF…

> **Note:** MECHCAD does not open DWG files directly (DWG is available only as an export from Drawing). Files from other CAD systems not listed here must be exported to STEP first.

Try it: Follow a drawer-knob project through its files

Scenario: you design a drawer knob, put it on a drawer front, make a drawing and send everything to a workshop. Watch which file each step creates.

1. Model the knob in **Part Design** and save it: `drawer-knob.metprt`.
2. In **Assembly**, insert the knob and a drawer-front panel and save: `drawer-front.metasm`.
3. **File › New Drawing from Current** and save the drawing: `drawer-front.metdrw`.
4. **Pack and Go** the assembly into one file, `drawer-front.metpkg`, and e-mail it. The workshop double-clicks it and MECHCAD unpacks and opens the assembly.
5. The workshop wants a 3D file for their own CAD: **File › Export STEP…** from the knob part.

Challenge: Look at the folder in Windows Explorer with large icons: every MECHCAD file shows its own preview.

See also

- Import CAD
- SolidWorks and Alibre
- Export STEP / STL

Import CAD (STEP, IGES, STL) and CAD Import Options Standard

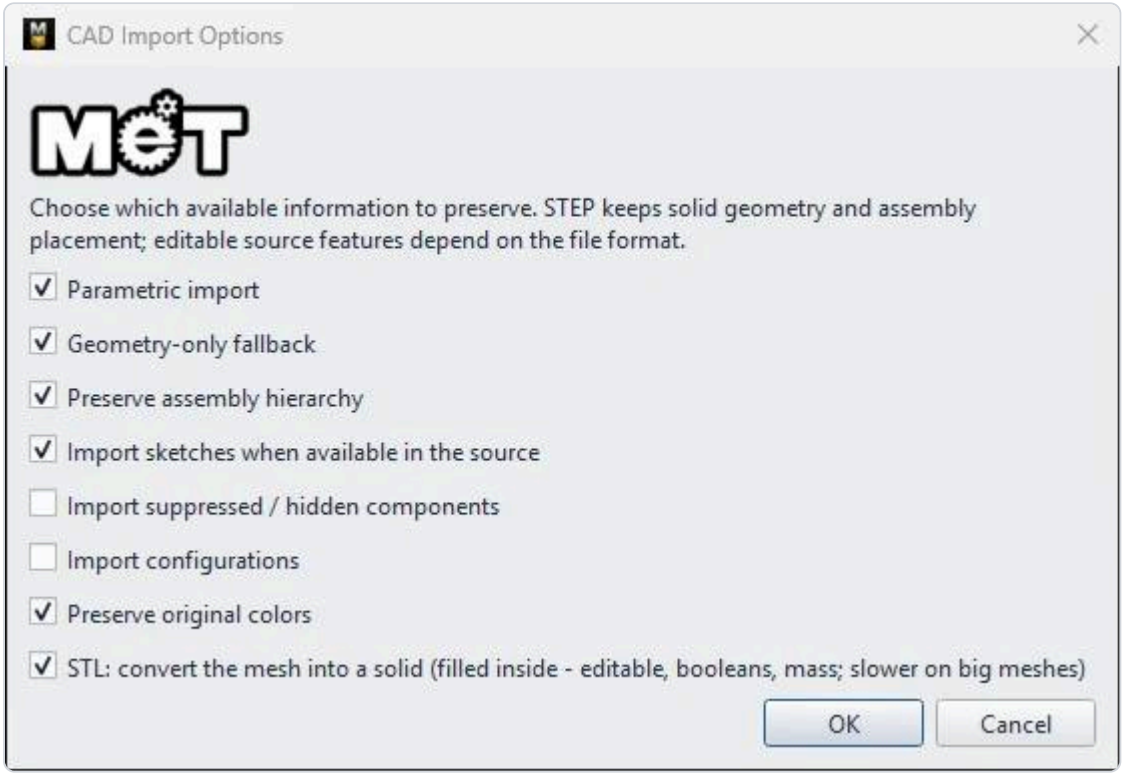
Import CAD files as a part or assembly, and what the import options mean.

What it does

Imports CAD files from other programs into MECHCAD.

Where to find it

Command	Result
File › Open...	Opens a CAD file in a new tab according to its type.
File › Import CAD into Part...	Opens it in Part Design. Simple parts that pass validation become editable features (*Import CAD into Part Design* window).
File › Import CAD / STEP Assembly...	Imports an assembly as parts with their positions (*Import CAD* window).
Assembly › Insert Part	Inserts a CAD file into the open assembly.



CAD Import Options window

CAD Import Options

Option	Default	Description
Parametric import	On	Creates editable features when the source data has them.
Geometry-only fallback	On	If features cannot be created, imports a solid.
Preserve assembly hierarchy	On	Keeps the sub-assembly structure.

Option	Default	Description
Import sketches when available in the source	On	Imports sketches when they exist.
Import suppressed / hidden components	Off	Imports hidden/suppressed parts.
Import configurations	Off	Imports configurations.
Preserve original colors	On	Keeps the original colours.
STL: convert the mesh into a solid	On (STL only)	Converts the mesh to a solid (allows Boolean edits and mass calculation, but is slow for large meshes).

At least one of the first two options must be on (*Enable Parametric import or Geometry-only fallback.*).

Steps

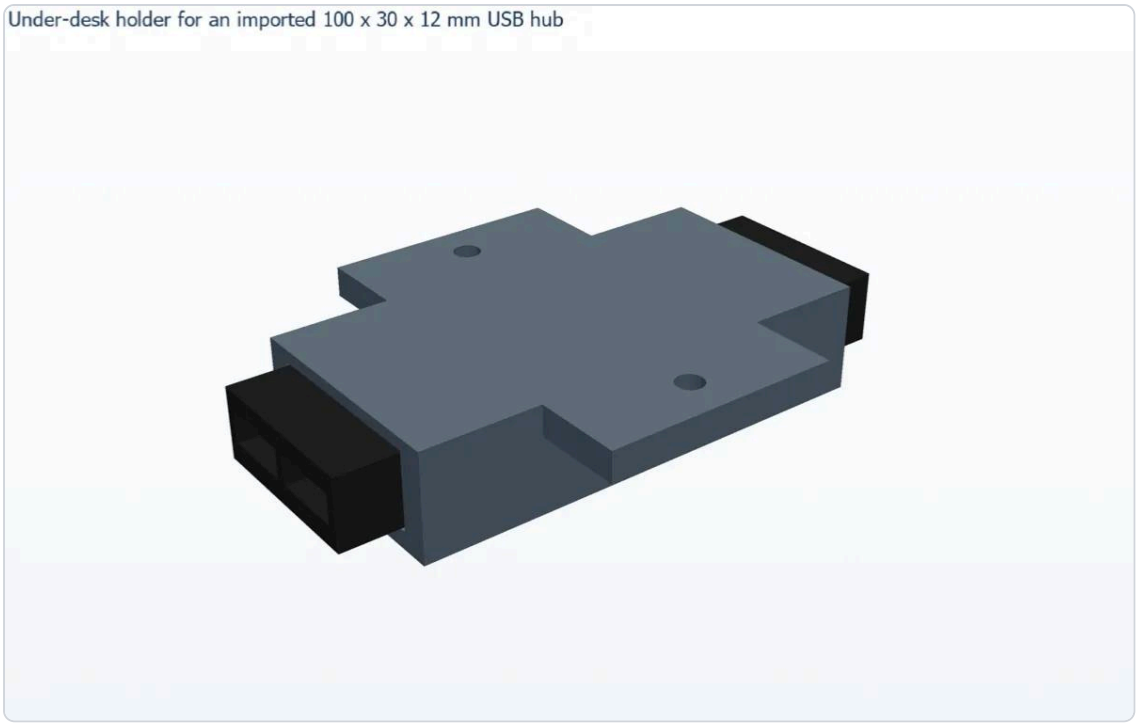
1. Choose an import command, then choose the file.
2. Set the options in the **CAD Import Options** window, then click OK.
3. Wait for the progress bar (*Reading CAD product structure…*, *Meshing part n / total*). Click Cancel to cancel.
4. When it finishes, a report shows *Native parametric import* / *Neutral assembly import* / *Geometry-only import* with the number of parts.

> **Note:** A part imported without history can be edited with Move Face, or converted to features with Recognize Features. An imported part that is tilted can be straightened with **Auto Align** in Assembly.

Try it: Under-desk holder for a USB hub

Many manufacturers publish STEP models of their products. Import one, measure it, and design a holder that clips the hub under your desk.

1. No STEP file at hand? Make a stand-in: a new part, rectangle **100 × 30**, **Extrude 12**, then **File › Export STEP…** as `usb-hub.step`. Close the part.
2. Choose **File › Import CAD into Part…** and pick the STEP file. Keep the default **CAD Import Options** and click **OK**. The report tells you which kind of import was used.
3. Right-click in the view › **Size Box**: **100 × 30 × 12** mm. Plan for 0.5 mm of clearance on every side.
4. New part: rectangle **80 × 40**, **Extrude 18**. Right-click an end face (40 × 18) › **New Sketch on this face**: rectangle **31 × 13**, **4.5** mm from the side and **3** mm from the bottom. Cut it **Through All**: this is the tunnel for the hub.
5. Right-click the top face › **New Sketch on this face**. Draw two **40 × 15** tabs, one along each long edge, outside the body. **Extrude 5** mm with ⇄ so they grow downward and stay flush with the top.
6. Sketch a **Ø4.5** hole in the middle of each tab and cut **Through All**. Screw the holder under the desk and slide the hub in.



USB hub (black) sitting in the under-desk holder

Challenge: Import the hub with **File › Import CAD / STEP Assembly...** instead, and place it in the holder in the Assembly tab to check the fit.

See also

- File types
- SolidWorks and Alibre

Open SolidWorks and Alibre files in MECHCAD Standard

Read .sldprt/.sldasm and .ad_prt/.ad_asm directly, and what you get after import.

What it does

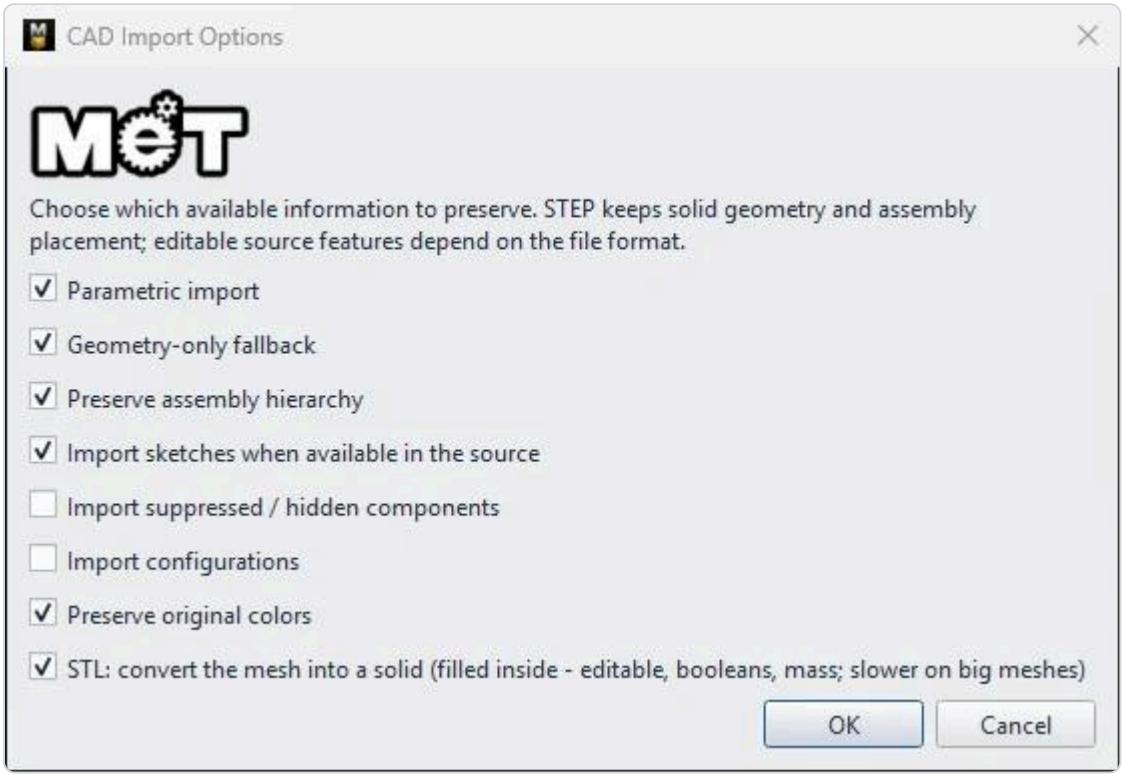
MECHCAD reads SolidWorks and Alibre files by itself, so you can open SolidWorks files without SolidWorks installed and share work with colleagues, customers and suppliers who use Alibre Design.

File	Result
SolidWorks Part (`.sldprt`)	Rebuilt as editable sketches + feature tree. Read in a separate process for stability.
SolidWorks Assembly (`.sldasm`)	Converted to parts + `.metasm`, saved in `Documents\MECHCAD Imports\<name>`
Alibre Part (`.ad_prt`)	Rebuilt as editable sketches + features (*...as editable sketches + features - Save As .metprt to keep it*)
Alibre Assembly (`.ad_asm`)	Converted to parts + `.metasm`, saved in `Documents\MECHCAD Imports\<name>`
Parasolid (`.x_t` , `.x_b`)	Part

File	Result
Siemens NX Part (`.prt`)	Part (solid)

Steps

1. Choose **File › Open...** and select a `.sldprt`, `.sldasm`, `.ad_prt` or `.ad_asm` file.
2. For an assembly, a progress panel *Importing … assembly* - … appears (you can click Cancel).
3. Check the result, then **Save As** a MECHCAD file.



CAD Import Options window

> **Note:** Steps in a SolidWorks or Alibre part that cannot be rebuilt as features yet are kept as exact geometry and listed for you to check. An Alibre file whose features cannot be created at all is imported as a solid instead. If you see *Native CAD Reader Required*, export a STEP file from the original program and import the STEP.

Try it: Open a colleague's speaker stand and monitor-arm clamp

Scenario: a colleague sends you a SolidWorks speaker-stand assembly (`speaker-stand.sldasm`) and an Alibre monitor-arm clamp (`arm-clamp.ad_prt`). You want to work on both files in MECHCAD.

1. **File › Open...**, choose `speaker-stand.sldasm`. The panel *Importing … assembly* shows the progress.
2. When it finishes, the converted parts and `.metasm` are in `Documents\MECHCAD Imports\speaker-stand`.
3. **File › Open...**, choose `arm-clamp.ad_prt`. It opens as editable sketches and features.
4. Double-click a feature of the clamp, change a value (for example the hole size) and **Save As** `arm-clamp.metprt` to keep it.

5. If a file shows *Native CAD Reader Required*, ask your colleague for a STEP export and import that instead.

Challenge: Compare the speaker stand's part list in the Assembly Explorer with the one your colleague sees in SolidWorks.

See also

- Import CAD
- Troubleshooting file import

Export STEP and STL Standard

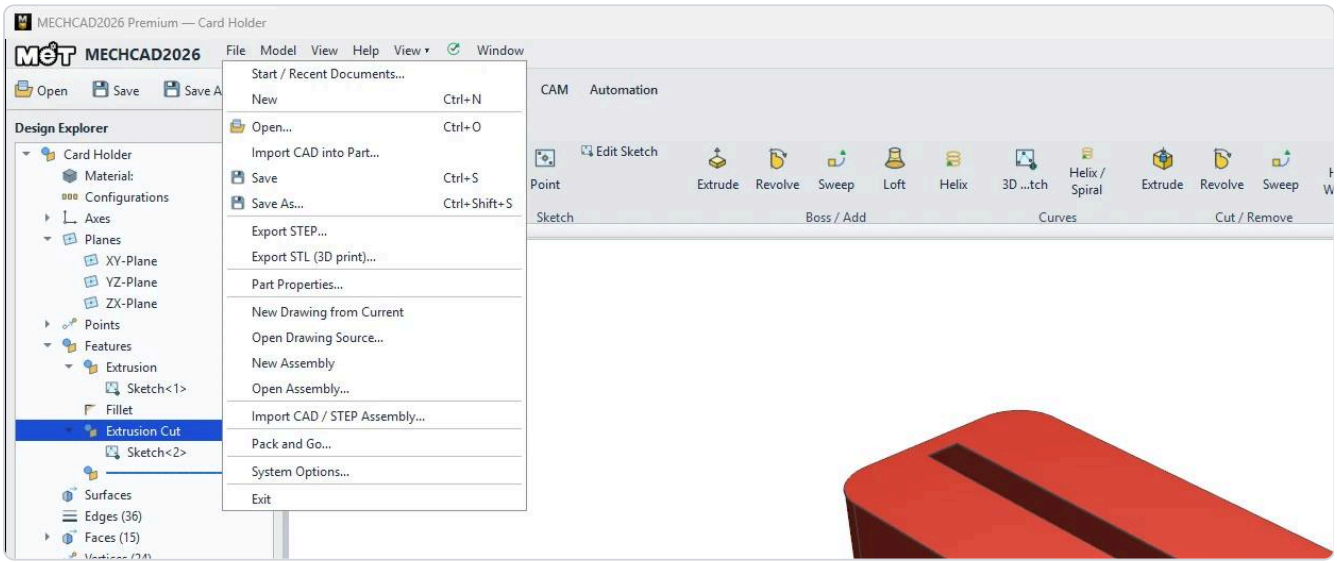
Export a part as STEP for other CAD programs, and as STL for 3D printers.

What it does

- **Export STEP…** exports a part as STEP for other CAD programs.
- **Export STL (3D print)…** exports a binary STL in mm for 3D printers and slicers.
- Assembly: **Share › Export STL** exports the selected parts, or all visible parts if none are selected.

Where to find it

File › Export STEP… / Export STL (3D print)…



File menu

Steps

1. Open the part (use Roll to End first if you have rolled back).
2. Choose **File › Export STEP…** or **Export STL (3D print)…**.
3. Enter a file name. For STL, choose the quality in the file type field.

STL quality	Description
STL for 3D printing - Fine	Detailed, large file
… - Standard	General use
… - Draft / small file	Coarse, small file

Try it: Earphone cable winder, ready for the 3D printer

Wrap your earphone cable around this H-shaped winder and plug the jack into the centre hole: no more knots in your bag.

1. Sketch on **XY** a rectangle **60 × 30** and **Extrude 3** mm.
2. Right-click the top face › **New Sketch on this face**. Draw two **10 × 10** squares, one at each short end, **10** mm from the front edge and touching the end edge. Add a **Ø6** circle in the centre (30, 15).
3. **Finish Sketch**, then **Cut / Remove › Extrude** with **Through All**.
4. **Fillet** with **Radius 1.5**: click the 4 outside vertical corners and the 8 vertical corners of the notches, then **Done**.
5. Choose **File › Export STL (3D print)…**. In the file type list, choose **STL for 3D printing - Fine**, and save `cable-winder.stl`.
6. Choose **File › Export STEP…** and save `cable-winder.step` for a friend who uses another CAD program.
7. Open the STL in your slicer. The size is 60 × 30 × 3 mm, already in millimetres.



Earphone cable winder, 60 x 30 x 3 mm

Challenge: Export the same part again with **Draft / small file** and compare the two STL file sizes.

See also

- File types

- Export HTML

Chapter 9 quiz: Import / Export

Choose the one best answer. Answers are in Appendix A

1. **Can MECHCAD open DWG files directly?**

   A. No; DWG is available only as an export from Drawing
   B. Yes, with File › Open…
   C. Yes, with File › Import CAD into Part…
   D. Only DWG files saved as AutoCAD 2018

2. **Do you need SolidWorks installed to open a .sldprt file in MECHCAD?**

   A. Yes, the same version as the file
   B. No, MECHCAD reads it by itself
   C. Yes, but only for assemblies
   D. No, but you must convert it to IGES first

3. **You see "Native CAD Reader Required" when opening a file. What should you do?**

   A. Reinstall MECHCAD
   B. Rename the file extension to .stp
   C. Export a STEP file from the original program and import the STEP
   D. Open it with Pack and Go

4. **What is the trade-off of the import option "STL: convert the mesh into a solid"?**

   A. It removes colours but is faster
   B. It keeps the feature history but needs Alibre
   C. It halves the file size but loses holes
   D. It allows Boolean edits and mass calculation, but is slow for large meshes

5. **In an assembly, you click Share › Export STL with nothing selected. What is exported?**

   A. Nothing; you must select parts first
   B. Only the base (fixed) part
   C. All visible parts
   D. All parts, including hidden ones

6. **In CAD Import Options, which two options cannot both be turned off?**

   A. Preserve original colors and Import configurations
   B. Parametric import and Geometry-only fallback
   C. Preserve assembly hierarchy and Import sketches
   D. Import suppressed / hidden components and STL conversion

CHAPTER 10

License & Editions

1. License Management: Edition Badge, Deactivate and License Certificate
   Check the licence status, move the licence to another computer, and print a licence certificate.
2. Updates (Check for Updates)
   Check for and install new versions: in-app updates for MA customers, and the direct download for everyone else.

License Management: Edition Badge, Deactivate and License Certificate Standard

Check the licence status, move the licence to another computer, and print a licence certificate.

What it does

Shows and manages the licence on this computer.

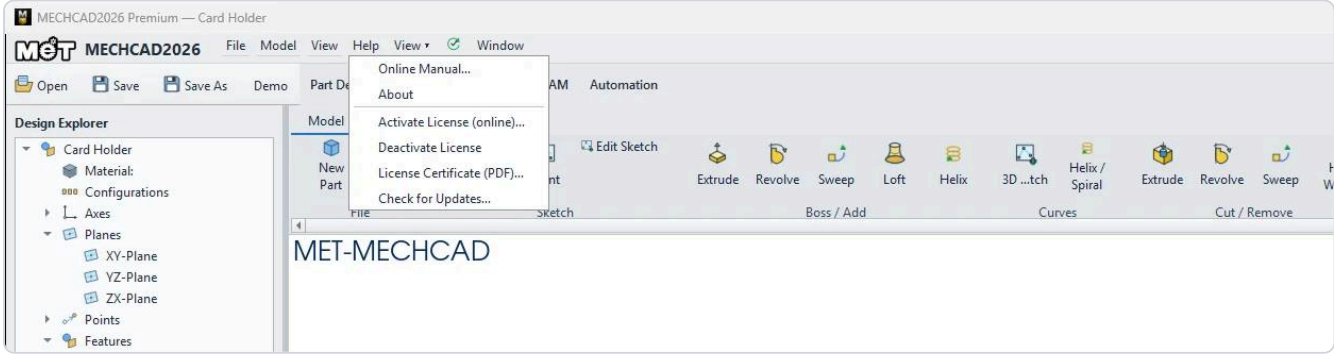
Edition badge (top right)

Shows the edition and its term, for example ● *MECHCAD Premium · Perpetual*, ● *MECHCAD Trial · 12 days left*, ● *MECHCAD Expert · until 2027-01-31* or ● *UNLICENSED* (red). The badge turns orange when 7 days or fewer remain. Hover over it to see the details (Product, Edition, Server, Machine, Expiry). Click it to open the Welcome window.



About MECHCAD2026 showing version, edition and licence

Help menu



Help menu

Command	Meaning
About	Version (Release version / Build), edition, licence status and install channel.
Activate License (online)…	Opens the Welcome window to enter a key or change the edition.
Deactivate License	Releases the seat on this computer (*Release this machine's seat on the license server?*). Use it before you move to a new computer.
License Certificate (PDF)…	Creates a PDF licence certificate (asks for the licence holder and company name if they are not set yet).
Check for Updates…	See Updates.

Move the licence to a new computer

1. On the old computer: **Help › Deactivate License** › Yes (*Seat released.*).
2. On the new computer: install the program, then enter the same key in the Welcome window.

> **Note:** If the old computer is offline, you see *Server unreachable. Activation kept; retry to release the seat.* Connect to the internet and try again, or contact MET.

Try it: Move your licence from the office PC to a home laptop

Scenario: you will work from home this month on a cable-management box, so the licence has to move to your laptop.

1. On the office PC, hover over the badge and note the edition and expiry.
2. Choose **Help › Deactivate License** › **Yes** and wait for *Seat released.*
3. On the laptop, install MECHCAD and enter the same key and e-mail in the Welcome window.
4. Check the badge: same edition, same expiry.
5. Choose **Help › License Certificate (PDF)…** and save the certificate for your IT records.

Challenge: If you forgot step 2 and the office PC is offline, connect it to the internet and try **Deactivate License** again.

See also

- Activate with a license key
- MECHCAD editions

Updates (Check for Updates) Standard

Check for and install new versions: in-app updates for MA customers, and the direct download for everyone else.

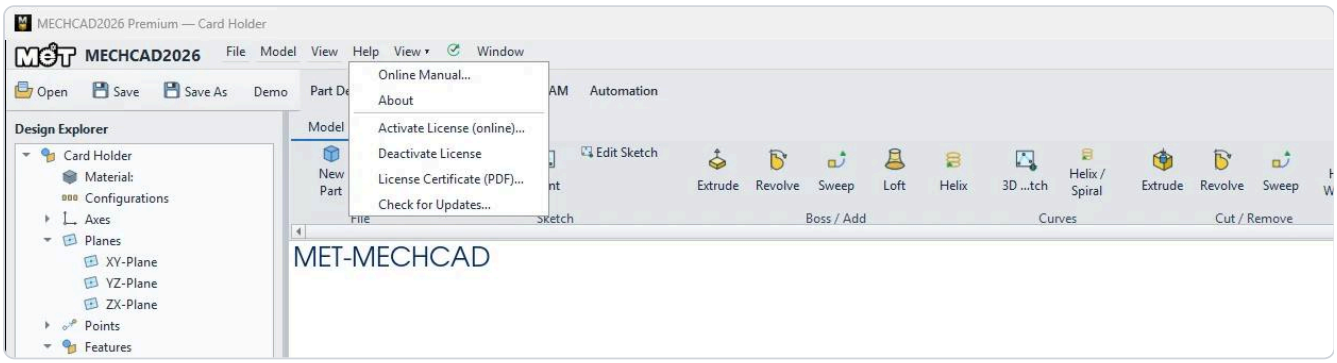
What it does

Customers who installed with the direct installer and have a Maintenance Agreement (MA) receive new versions directly through the program.

Without MA (for example during the 30-day trial), download the latest installer from https://license.mechdesigned.com/download/MECHCAD and install it over the current version. A copy installed from the Microsoft Store updates through the Store.

Where to find it

Help › Check for Updates... (it also checks automatically 6 seconds after the program starts)



Help › Check for Updates... menu

Steps

1. Choose **Help › Check for Updates...**.
2. If a new version is available, the **MECHCAD Update Available** window shows the version number.
3. Click **Download & Install**. The program downloads with a progress bar, then closes itself to install. Or click **Later** (the automatic notice does not ask again for that version).

Message	Meaning
You are up to date.	You have the latest version.
Activate an online license to check MA updates.	You must activate an online licence first.
This license does not currently include MA update coverage.	The licence has no active MA. Contact MET.
This copy was installed from Microsoft Store - updates arrive through the Store.	Updates come through the Microsoft Store.
Could not verify updates with the license server.	Check your internet connection.

Try it: Install an update in the middle of a project

Scenario: you are an MA customer working on a keyboard wrist rest, and a new version comes out.

1. Start MECHCAD. About 6 seconds later, **MECHCAD Update Available** appears with the new version number.
2. You are busy, so click **Later**. The automatic notice does not ask again for this version.
3. When you have finished, **Save** the wrist rest, then choose **Help › Check for Updates...**.

4. Click **Download & Install**. Watch the progress bar. MECHCAD closes and the installer runs.
5. Open the wrist rest again and check the version in **Help › About**.

Challenge: Choose **Help › Check for Updates…** right after the update: you should see *You are up to date*.

See also

- Install MECHCAD 2026

Chapter 10 quiz: License & Editions

Choose the one best answer. Answers are in Appendix A

1. **You are moving your licence to a new computer. What do you do first on the old computer?**

   A. Uninstall MECHCAD
   B. Delete the %APPDATA% folder
   C. Help › Deactivate License
   D. Help › Check for Updates…

2. **What happens when you click the Edition badge at the top right?**

   A. The Welcome window opens
   B. The licence is deactivated
   C. The website opens
   D. Nothing; it only shows a tooltip

3. **Where do you create a PDF licence certificate?**

   A. File › Export PDF
   B. Drawing › Output › Export PDF
   C. Help › About
   D. Help › License Certificate (PDF)…

4. **Who receives new versions directly through Help › Check for Updates…?**

   A. Every user, including the trial
   B. Direct-installer customers with a Maintenance Agreement (MA)
   C. Microsoft Store users only
   D. Only computers without internet

5. **Without MA (for example during the trial), how do you update MECHCAD?**

   A. You cannot update until you buy MA
   B. Uninstall first, then reinstall from the Store
   C. Download the latest installer and install it over the current version
   D. Use Help › Deactivate License

6. **In the MECHCAD Update Available window you click Later. What happens?**

   A. The automatic notice does not ask again for that version
   B. The update installs at the next restart
   C. It asks again in 6 seconds
   D. The licence is paused

CHAPTER 11

Troubleshooting

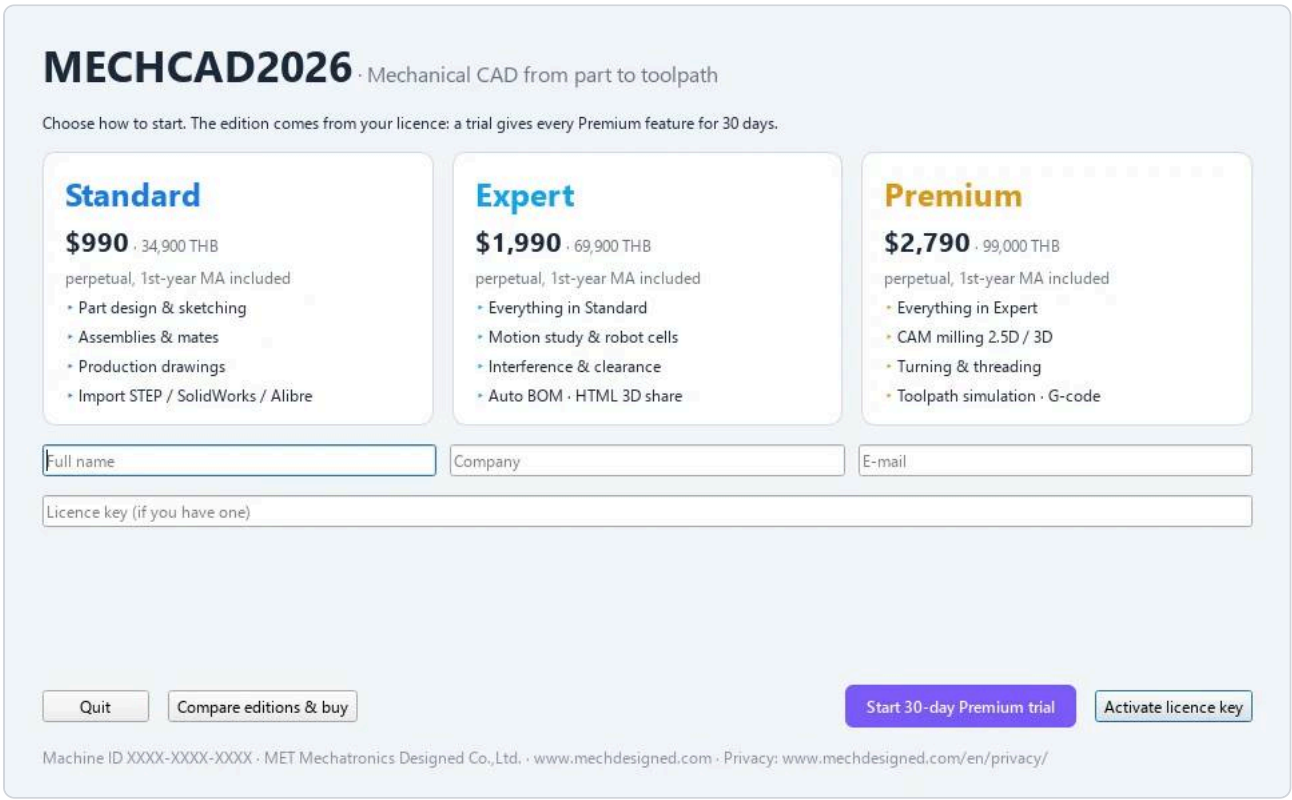
1. Troubleshooting Licence Problems
   Causes and fixes when you cannot activate, the licence has expired, or the computer is offline.
2. Troubleshooting Display and Graphics Card Problems
   The 3D view is blank, choppy or distorted, or clicks pick the wrong object on a high-resolution screen.
3. Large Assemblies Run Slowly
   Tips to open, rotate and edit assemblies with thousands of components faster.
4. Auto-Recover and crash reports
   Automatic assembly backups, recovery, and the log files written when the program crashes.
5. Troubleshooting CAD Import Problems
   What to check when importing STEP, SolidWorks, Alibre or STL fails or the result is incomplete.
6. Report a Bug
   Send a problem report with a screenshot straight to the MET team from inside the program. You choose every item that is sent.

Troubleshooting Licence Problems Standard

Causes and fixes when you cannot activate, the licence has expired, or the computer is offline.

Symptoms and fixes

Symptom / message	Cause	Fix
The Welcome window appears every time you start	There is no licence, or it has expired	Enter a license key or start a trial. See Activate.
A free trial was already used on this computer or with this e-mail.	The trial has already been used	Contact MET to buy or extend.
This license key has expired.	The key has expired	Renew with MET.
All N seats are in use.	All seats are in use	Run **Help › Deactivate License** on another computer first.
Offline grace elapsed — reconnect to the license server.	Offline for more than 7 days	Connect to the internet and restart the program.
The badge shows ● *UNLICENSED*	The licence check failed	Click the badge to open Welcome, then activate again.
A button/tab says … *is part of MECHCAD Expert and Premium*	Your edition does not include it	See Editions. You can upgrade with the same key.



Welcome window

Information to have ready when you contact MET

- The **Machine ID** (at the bottom of the Welcome window, or hover over the Edition badge)

- The program version from **Help › About**
- The e-mail used to activate

> **Note:** The activation data is stored in `%APPDATA%\MET-MECHCAD`. Do not delete this folder unless necessary.

Try it: "All seats are in use" on a new home-office laptop

Scenario: you install MECHCAD on a new laptop to finish a desk-clock design at home, and activation says *All N seats are in use*.

1. On the old office PC, choose **Help › Deactivate License › Yes**. Wait for *Seat released*.
2. On the laptop, click the badge to open the Welcome window and enter the same key and e-mail again.
3. If the old PC is broken, write down the laptop's **Machine ID** (at the bottom of the Welcome window) and the version from **Help › About**, and send both to MET with the e-mail you used to activate.
4. When you work offline (on a trip, for example), you have 7 days before *Offline grace elapsed*. Connect and restart MECHCAD before that.

Challenge: Hover over the badge and find the expiry date and the server name.

See also

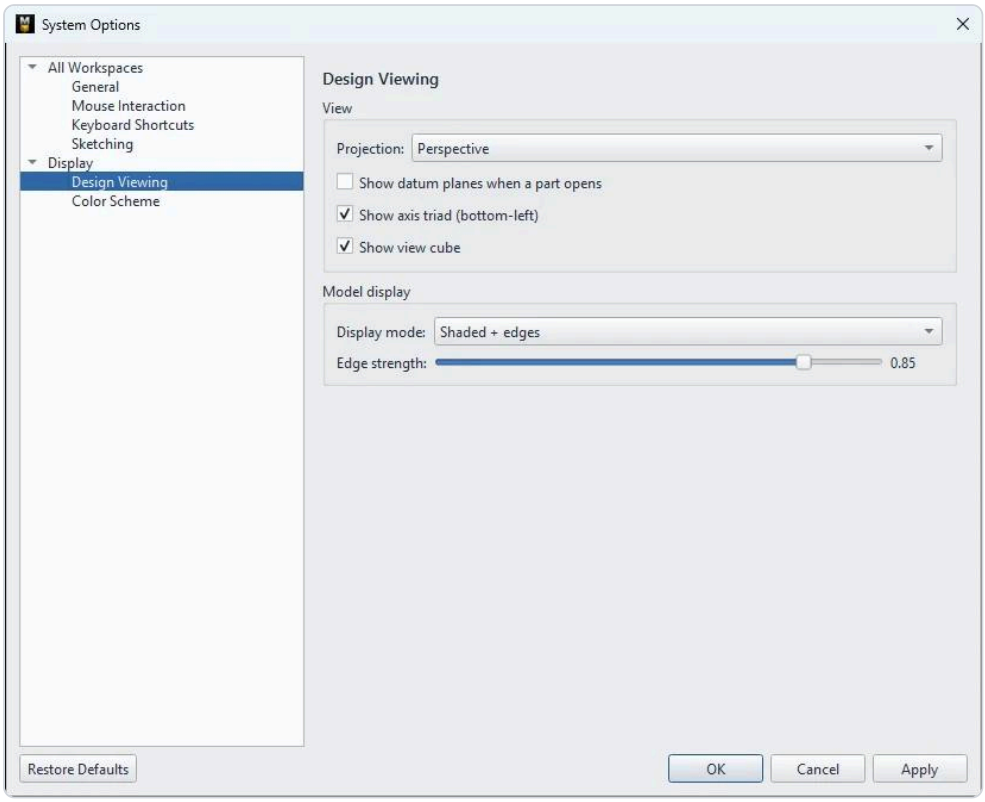
- License management

Troubleshooting Display and Graphics Card Problems Standard

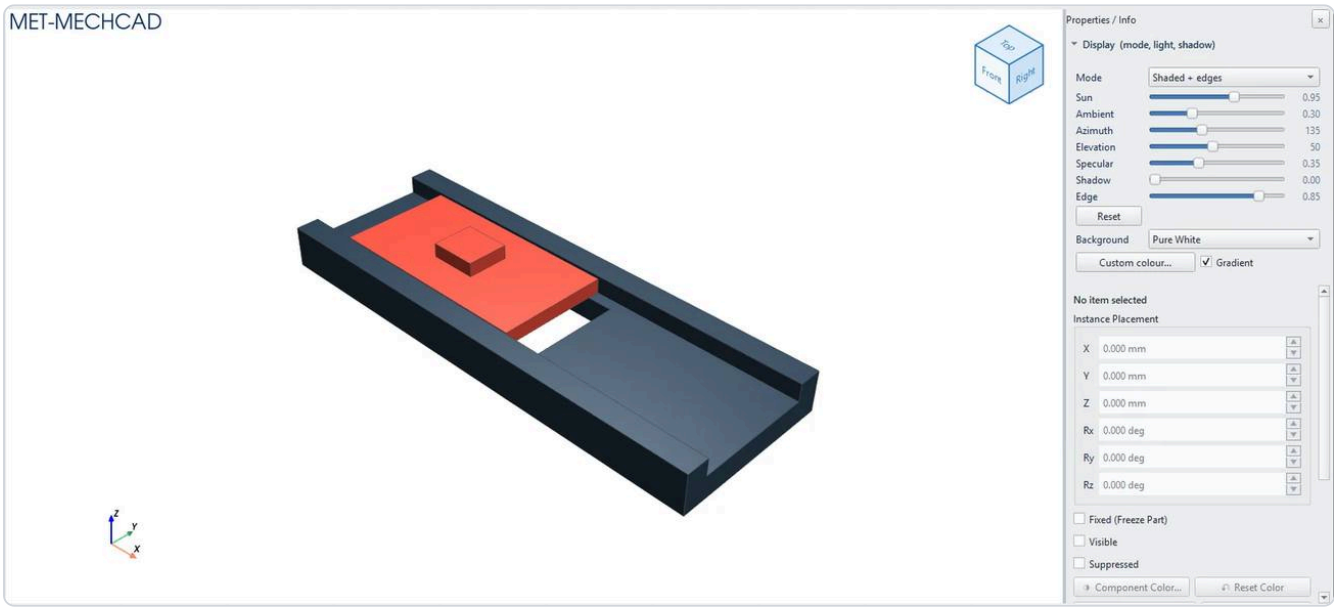
The 3D view is blank, choppy or distorted, or clicks pick the wrong object on a high-resolution screen.

Symptoms and fixes

Symptom	Fix
The 3D view is blank, or the program closes at startup	Update the graphics card driver. On a laptop with two GPUs, use the discrete GPU (the direct installer has the option *Prefer high-performance graphics for MECHCAD*, or set it in Windows › Settings › Display › Graphics).
Rotating a large part is choppy	Switch to **Shaded** mode (no edges). Turn off Shadows / Ambient Occl. / Realistic Render. See Large assemblies.
Overlapping surfaces flicker	Switch between Orthographic / Perspective, or Zoom to fit.
Clicks pick the wrong object on a 150% screen	Use the latest version (fixed in 2026.x) and Check for Updates.
The mouse rotates/pans awkwardly	Set the preset in **System Options › Mouse Interaction**. See Mouse use.
The background is too dark/bright	**System Options › Color Scheme**, or the **Display › Background** panel.



System Options › Design Viewing



Display panel

> **Note:** If the program freezes or closes by itself, its logs are in `%LOCALAPPDATA%\MECHCAD\logs` (fault.log, freeze.log) and `%LOCALAPPDATA%\MECHCAD\mechcad_crash.log` . Attach these files when you report the problem to MET.

Try it: A choppy standing-desk frame on a laptop

Scenario: you open a standing-desk frame assembly on a laptop with two GPUs, and rotating it is jerky.

1. Check that Windows uses the discrete GPU for MECHCAD (**Windows › Settings › Display › Graphics**, or the installer option *Prefer high-performance graphics for MECHCAD*). Restart MECHCAD.
2. Click the **Shaded** display button (no edges).
3. Turn off **Shadows**, **Ambient Occl.** and **Realistic Render** if they are on.
4. Rotate again. If the frame is still slow, follow Large assemblies.
5. If MECHCAD closes by itself, send MET the files from `%LOCALAPPDATA%\MECHCAD\logs` and `mechcad_crash.log`.

Challenge: Compare the rotation speed in **Shaded + Edges** and **Shaded** on the same model.

See also

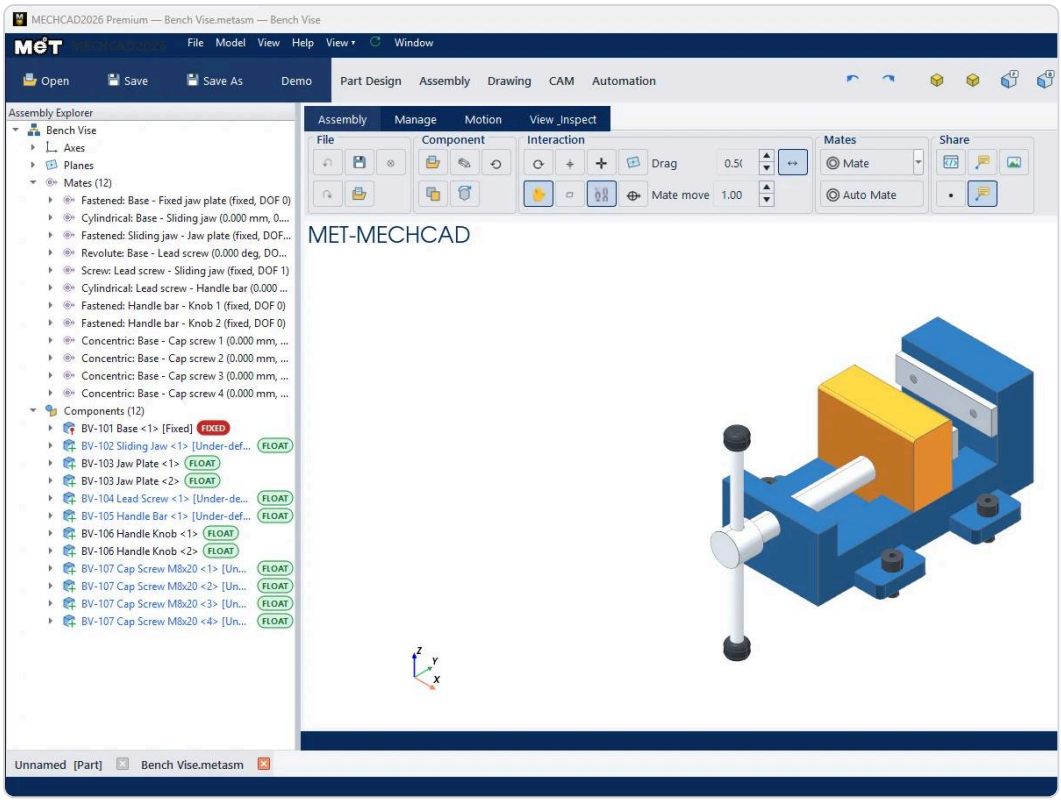
- Display modes
- Auto-Recover and error reports

Large Assemblies Run Slowly Standard

Tips to open, rotate and edit assemblies with thousands of components faster.

Tips

Method	Result
Use **Shaded** mode instead of Shaded + Edges	Less edge drawing
Turn off Shadows, Ground Reflection and Realistic Render	Faster drawing (large assemblies turn shadows off automatically)
Purge (Skip Loading) components you do not need (right-click a component)	The component is not loaded
Hide All, then Show only the part you work on	Fewer components displayed
Import a multi-solid STEP file as a **Single Part** when you do not need inner joints	Lighter part, easier to mate
Use sub-assemblies instead of placing every component in the main assembly	A sub-assembly is a rigid body, so fewer mates to solve
For drawings of a large assembly, use **Detail: Auto / Draft**	Faster drawing creation (600 or more components use Draft automatically)
Delete or Suppress conflicting mates (red)	The solver runs faster
Fix the main frame (for example the machine frame) before adding mates	A new mate solves only the parts it affects; fixed parts act as boundaries, so the whole assembly is not pulled in



Assembly Explorer: FIXED / FLOAT status badges

> **Note:** Editing or deleting a mate does not recalculate the whole assembly by itself. Click **Regen** when you want to re-solve every mate.

> **Note:** If a mate takes a while to solve, a *Solving mates* window with a **Cancel** button appears and the window stays responsive. Cancel discards the new mate. See Add mates.

Try it: Speed up a four-seat office desk pod

Scenario: a desk pod for four people with thousands of screws and cable trays rotates slowly, and you only need to work on one drawer unit.

1. Switch to **Shaded** and turn off Shadows and Ground Reflection.
2. Right-click the cable trays you do not need today › **Purge (Skip Loading)**.
3. **Hide All**, then **Show** only the drawer unit you are working on.
4. Group each desk into a sub-assembly, so the main assembly solves 4 rigid bodies instead of hundreds of mates.
5. Delete or suppress mates shown in red, then click **Regen** once.
6. For the drawing of the whole pod, use **Detail: Auto / Draft**.

Challenge: Time how long a full rotation takes before step 1 and after step 5.

See also

- Component states
- Troubleshooting display problems

Auto-Recover and crash reports Standard

Automatic assembly backups, recovery, and the log files written when the program crashes.

What it does

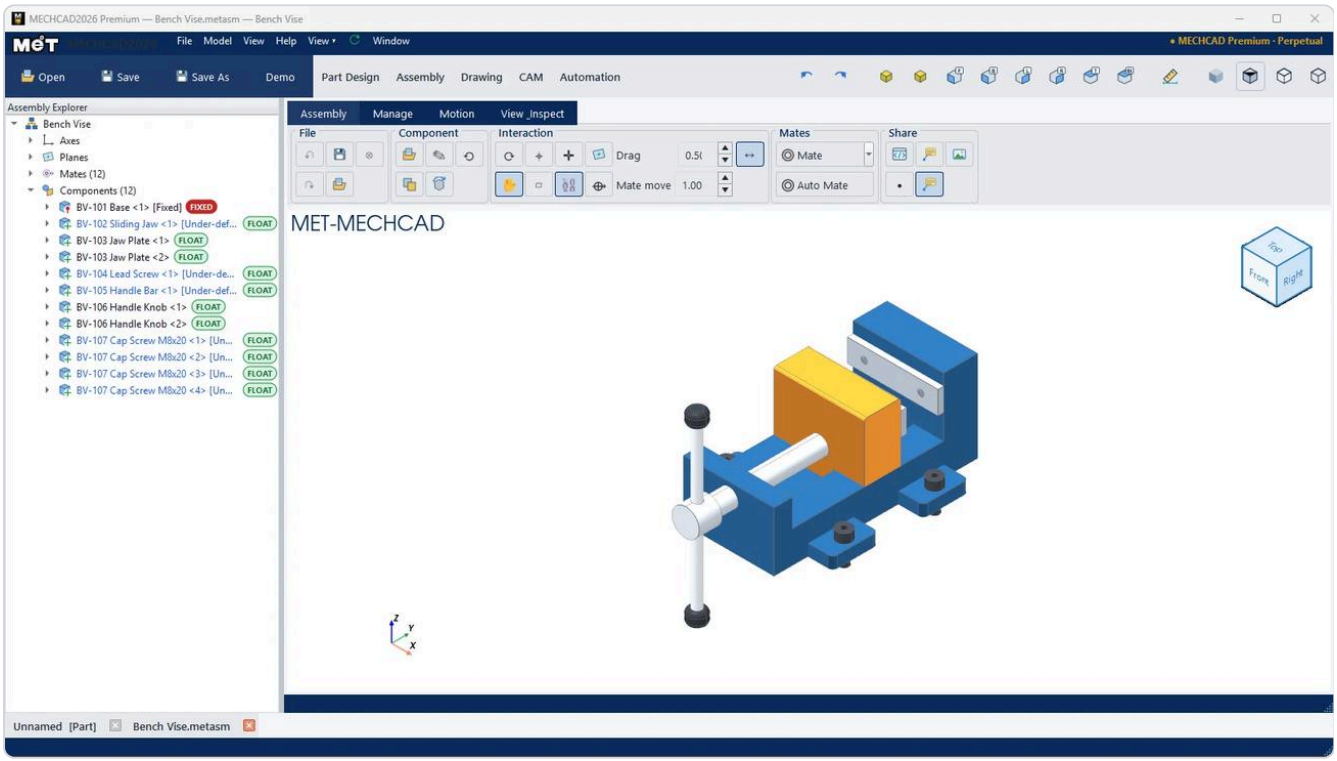
Protects your work if the program or the computer shuts down unexpectedly.

Auto-Recover (Assembly)

- A backup is saved every 10 minutes, only when there are unsaved changes and you have stopped working for at least 3 seconds. Nothing is saved while you are adding a mate, dragging, or importing a file.
- Backup files are in `%LOCALAPPDATA%\MECHCAD\AutoRecover`. Your real working file is not touched.
- The status bar shows *Auto-Recover saved (…)*.

Recover a file

1. Open the original assembly.
2. If a newer backup exists, you are asked *Open the recovered version? (No = open the saved file)*.
3. Answer **Yes** to open the recovered version (*Recovered version opened - Save to keep it*), then **Save**.
4. A normal Save deletes the backup file.



Assembly window

When the program crashes or freezes

File	Contents
`%LOCALAPPDATA%\MECHCAD\logs\fault.log`	Fatal errors

File	Contents
`%LOCALAPPDATA%\MECHCAD\logs\freeze.log`	Where the program froze
`%LOCALAPPDATA%\MECHCAD\mechcad_crash.log`	Crash log

The installed version automatically sends an error report (with no model data) to MET so the problem can be fixed.

> **Note:** Auto-Recover works for assemblies. For parts, drawings and CAM, press **Save** (Ctrl+S) regularly.

Try it: A recovery drill with a two-tier desk shelf

Scenario: you are building a two-tier desk shelf assembly when the laptop battery dies. Practise the recovery now, so you know what to do when it really happens.

1. In the **Assembly** tab, insert a few parts (a top board and two side panels, for example) and do not save.
2. Keep working, then take your hands off the mouse. After about 10 minutes, the status bar shows *Auto-Recover saved (...)*.
3. Simulate the power cut: close MECHCAD from **Task Manager › End task**.
4. Start MECHCAD and open the same assembly. Answer **Yes** to *Open the recovered version?*.
5. You see *Recovered version opened - Save to keep it*. Click **Save**: the backup file is deleted, and your work is safe.

Challenge: Open `%LOCALAPPDATA%\MECHCAD\AutoRecover` in Explorer during step 2 to see the backup appear.

See also

- Display troubleshooting

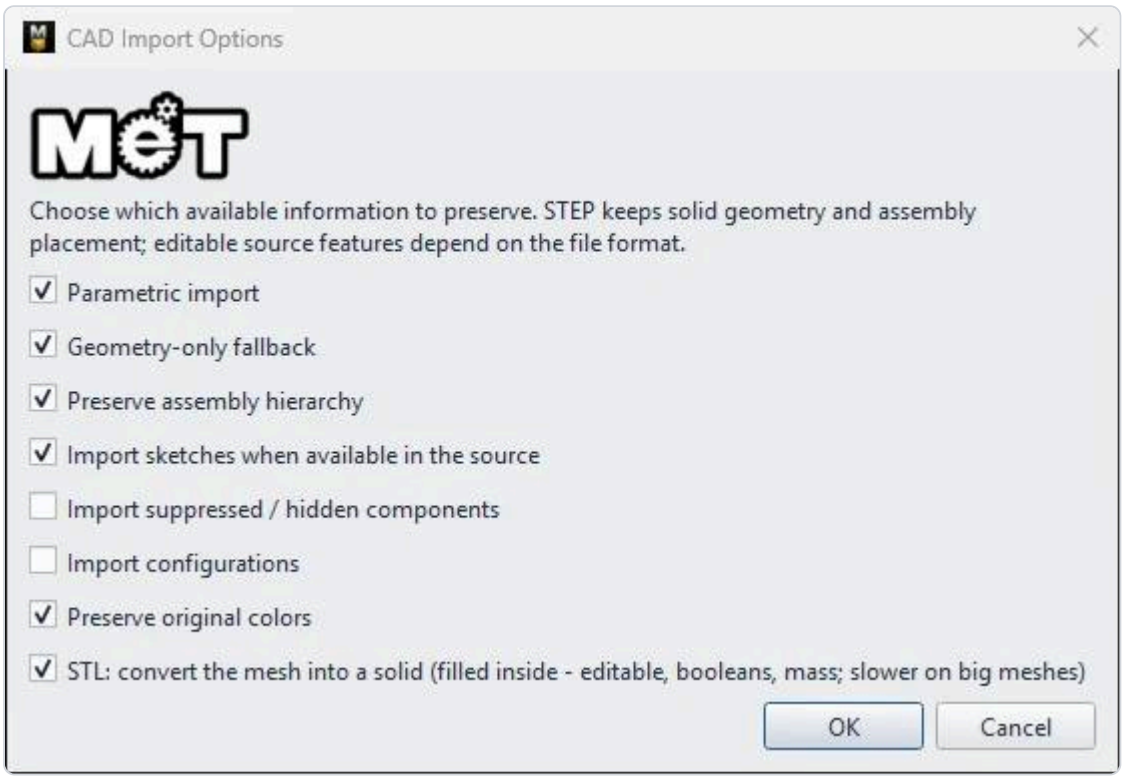
Troubleshooting CAD Import Problems Standard

What to check when importing STEP, SolidWorks, Alibre or STL fails or the result is incomplete.

Symptoms and fixes

Symptom	Fix
Native CAD Reader Required	Export from the source program as STEP (AP214/AP242), then import the STEP file.
The imported part has no feature history	This is normal for STEP/IGES. Use Recognize Features or Move Face.
Gaps/leaks in the surfaces after import	Use **Simplify** (Heal) in Direct Edit.
STL takes very long to convert to a solid	Turn off *STL: convert the mesh into a solid* in CAD Import Options.
Components are tilted after importing an assembly	Right-click › **Auto Align (square to axes)** or **Align Face to Plane...**

Symptom	Fix
A large assembly opens slowly	Import as a **Single Part** if you do not need joints, and see Large assemblies.
Colors are missing	Turn on *Preserve original colors*.
Components hidden in the source are not visible	Turn on *Import suppressed / hidden components*.



CAD Import Options window

Try it: Fix a tilted USB desk fan and a slow STL

Scenario: you download a USB desk fan as STEP from the manufacturer and a pencil sharpener as STL from a 3D-print site.

1. Import the fan with **File › Import CAD / STEP Assembly...**. It comes in without colours: import it again with *Preserve original colors* on.
2. The fan arrives tilted. In the Assembly tab, right-click it › **Auto Align (square to axes)**.
3. You only need the fan as a reference, not its moving blade: import it as a **Single Part**. It is lighter and easier to mate.
4. The sharpener STL takes a long time: in **CAD Import Options**, turn off *STL: convert the mesh into a solid*. You can still see it and place it.
5. You need to edit a hole in the fan's mount: use Move Face on the imported solid.

Challenge: Try Recognize Features on the fan's mount to turn it into editable features.

See also

- Import CAD

- SolidWorks and Alibre

Report a Bug Standard

Send a problem report with a screenshot straight to the MET team from inside the program. You choose every item that is sent.

What it does

Reports a problem to the MET team from inside the program, with a screenshot and the information that helps fix it quickly. The report goes straight to MET through the MET licence server. It is never posted on a public website.

Where to find it

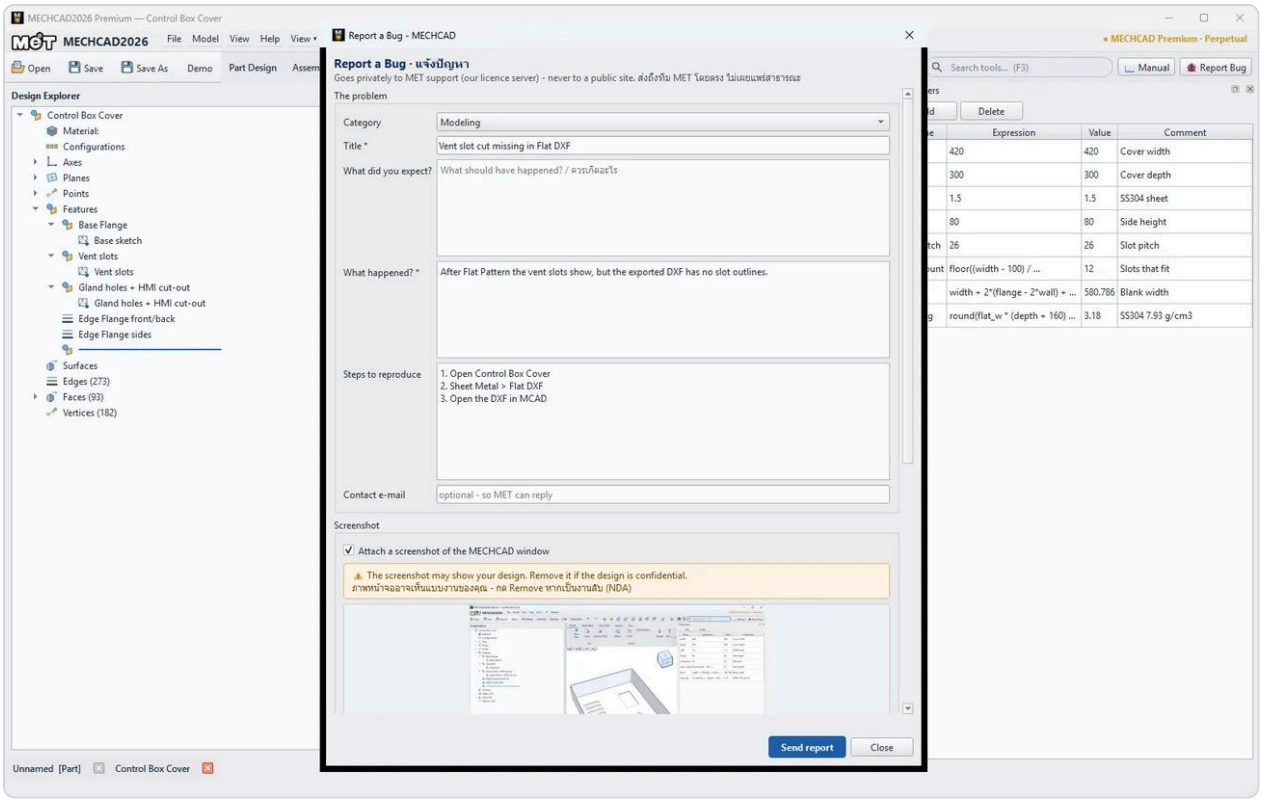
- **Help › Report a Bug...**
- Shortcut **Ctrl+Shift+B**
- The 🐞 **Report Bug** button on the right of the workbench bar, next to 📖 **Manual** (in a narrow window only the 🐞 icon is shown)

The *Report a Bug - MECHCAD* window does not block the program: you can keep working while you fill it in. Only one report window is open at a time.

Steps

1. Get the problem on screen first, then press **Ctrl+Shift+B**. The program captures the MECHCAD window before the form appears.
2. Under **The problem**, choose a **Category** (preset from the workbench you are using), then fill in **Title** and **What happened?** Both are required.
3. Fill in **What did you expect?** and **Steps to reproduce** if you can. The more detail, the faster the fix.
4. Check the image under **Screenshot**. Click **Retake** to capture again (the form hides for a moment), **Remove** to leave the image out, or **Save copy...** to keep the image yourself.
5. Under **Attached information**, untick anything you do not want to send.
6. Click **Send report** and note the ticket number that appears. Quote it when you contact MET.

Field	Meaning
Category	Modeling, Sketch, Assembly, Drawing, CAM, Import-Export, UI, Licence, Other
Title *	A short summary, for example *Fillet fails on a slot edge*
What did you expect?	What should have happened
What happened? *	What happened instead, and any error text
Steps to reproduce	The steps that cause the problem, one by one
Contact e-mail	An address MET can reply to (optional; filled in from the licence when available)



Report a Bug form: the problem, contact e-mail and screenshot sections

What is sent

Item	Default	Content
Attach a screenshot of the MECHCAD window	On	An image of the MECHCAD window only (scaled down, JPEG)
App version, build, edition, channel, OS	On	Program version, edition and operating system
Current workbench and command	On	The workbench and the last command used
Selection (names only)	On	Only the names of the items selected in the Design Explorer
Recent crash / freeze report IDs	On	IDs of recent crash or freeze reports on this PC
Last … log lines	On	The most recent log lines. Click **Show log lines** to read them before sending.
Attach current file	**Off**	The open document. Sent only when you tick it.

- The report also carries the machine ID and licence key, so MET knows which PC it came from.
- File paths, user names, computer names and e-mail addresses in the logs and in the text you type are removed before sending.
- **Your model file is never sent automatically. Attach current file** works only for a saved MECHCAD document (such as .metprt, .metasm, .metdrw, .metcam, .metauto) of up to 0.7 MB, and only the file extension is sent, not the file name.
- The whole report must be 3 MB or less. If it is larger, the program shortens the log lines first.

> **Note:** The screenshot may show your design. If the design is confidential (NDA), click **Remove** and do not tick **Attach current file**.

When there is no internet connection

If the report cannot be sent, it is saved on the PC and sent automatically the next time MECHCAD starts. The status bar then shows *Bug report sent from the offline queue:* … with the ticket number. Reports waiting longer than 30 days are dropped.

See also

- Auto-Recover and crash reports
- Troubleshooting licence problems
- Main menus

Chapter 11 quiz: Troubleshooting

Choose the one best answer. Answers are in Appendix A

1. **How often does Auto-Recover save a backup of an assembly?**

   A. Every minute
   B. Every 10 minutes, when there are unsaved changes
   C. Every 30 minutes
   D. Only when you close the program

2. **Auto-Recover protects assemblies. What should you do for parts, drawings and CAM jobs?**

   A. Nothing; they are protected too
   B. Turn on Auto-Recover in System Options
   C. Use Pack and Go every hour
   D. Press Save (Ctrl+S) regularly

3. **Rotating a large part is choppy. What is the first fix?**

   A. Switch to Shaded mode (no edges) and turn off Shadows / Ambient Occl. / Realistic Render
   B. Increase the Undo history limit
   C. Switch the mouse preset to SolidWorks
   D. Re-activate the licence

4. **Components are tilted after importing an assembly. How do you square them up?**

   A. Re-import with Preserve original colors on
   B. Use Explode, then Collapse
   C. Right-click › Auto Align (square to axes) or Align Face to Plane…
   D. Delete their mates

5. **In a large assembly, how do you stop a component you do not need from being loaded?**

   A. Set its transparency to 100%
   B. Right-click the component › Purge (Skip Loading)
   C. Rename the file
   D. Add a Fastened mate

6. **What information should you have ready when you contact MET about a licence problem?**

   A. Your Windows password
   B. The installer file
   C. The Machine ID, the version from Help › About, and the e-mail used to activate
   D. A screenshot of the Recycle Bin

Appendix A · Quiz answers

Chapter 1 · Getting started

No.	Answer	Review topic
1	**B** Prefer high-performance graphics for MECHCAD	Install MECHCAD 2026 (Direct Download / Microsoft Store) (p. 9)
2	**C** Once per computer and per e-mail	Welcome Window and 30-Day Premium Trial (p. 11)
3	**D** Within the 7-day offline grace period	Activate a licence key (p. 13)
4	**A** Removes the entry from the list; the actual file is not deleted	Start Center: Recent Documents (Start / Recent Documents) (p. 18)
5	**B** It becomes a hole automatically	Create your first part in 5 minutes (p. 20)
6	**C** Premium only (and the trial)	MECHCAD Editions: Standard, Expert and Premium (p. 15)

Chapter 2 · Interface & settings

No.	Answer	Review topic
1	**A** F3	Ribbon and Command Search (F3) (p. 37)
2	**B** Drag with the right button	Mouse Use and View Rotation (p. 44)
3	**C** 64 steps	System Options (Program Settings) (p. 55)
4	**D** Ctrl+Shift+P	Keyboard Shortcuts (p. 61)
5	**A** The ΔX/ΔY/ΔZ distance and the angle	Measure (Distance, Angle and Size) (p. 64)
6	**B** Shaded + Edges	Display modes and the Display panel (p. 49)

Chapter 3 · Sketch

No.	Answer	Review topic
1	**C** The sketch is fully defined	Sketch Overview: Start, Finish and the Sketch Ribbon (p. 71)
2	**D** Returns to Select	Sketch Overview: Start, Finish and the Sketch Ribbon (p. 71)
3	**A** Hold Alt	Snaps and Auto H/V (p. 101)
4	**B** Type R or Ø/D before the number	Smart Dimension and Smart Angle (p. 90)
5	**A** A Reference (driven) dimension that only shows the value	Smart Dimension and Smart Angle (p. 90)
6	**B** Right-click the face › Insert Sketch on Surface	Sketch on a Face of the Part (p. 74)

Chapter 4 · Part

No.	Answer	Review topic
1	**C** Mid Plane	Extrude (Boss) (p. 119)
2	**D** Through All	Extrude Cut (p. 122)
3	**A** Click Done, or press Esc, Enter or Space	Fillet and Chamfer (p. 130)
4	**B** Draw the positions with the Point tool in a sketch, then choose the sketch's “all N points (Drill)” position	Hole Wizard (standard holes) (p. 128)
5	**D** Roll to End	Rollback, Regen and Suppress (p. 152)
6	**C** Double-click it in the Design Explorer, or right-click › Edit Feature	Feature Parameters: the feature edit box (p. 117)

Chapter 5 · Assembly

No.	Answer	Review topic
1	**B** The second component you pick	Add Mates with the Assembly Mates Panel (p. 174)
2	**A** Press Tab to go to the next face	Add Mates with the Assembly Mates Panel (p. 174)
3	**D** It is fixed in place as the base	Assembly overview (p. 168)
4	**C** Right-click › Rigid Body › Ungroup Rigid Body, then add mates (e.g. Auto Mate)	Insert Parts and Sub-Assemblies (p. 170)
5	**B** File › Pack and Go… (creates a .metpkg file)	Pack and Go (.metpkg) (p. 190)
6	**D** Click Regen in the Interaction group	Edit, flip and delete mates (p. 179)

Chapter 6 · Drawing

No.	Answer	Review topic
1	**B** Third-angle	Create a Drawing (Standard Views Creation) (p. 194)
2	**C** Output › Pack Refs	Create a Drawing (Standard Views Creation) (p. 194)
3	**D** Click two points of the cutting line inside a view, then place the section	Views: Base, Projected, Section, Detail and Break View (p. 197)
4	**A** Auto Dimension	Drawing dimensions: Auto Dimension, Smart Dimension, Hole Callout and Ordinate (p. 200)
5	**C** Indented (all levels)	BOM, balloons and tables (Hole, Revision, Cut List) (p. 203)
6	**B** The automatic value is used (e.g. part name, material, overall size)	Sheet Properties and Title Block (p. 205)

Chapter 7 · Motion & Analysis

No.	Answer	Review topic
1	**A** Drag the components into the pose, then click Save Point	Motion Study and Motion Studio (p. 211)
2	**C** MP4	Play Motion, Spin, Linear Move and video/GIF recording (p. 212)
3	**B** Reports all pairs	Interference and Clearance Check (p. 213)
4	**D** Excel (.xlsx) and PDF	Auto BOM (Excel / PDF with pictures and balloons) (p. 215)
5	**C** A single web page so others can view and measure the 3D model in a browser without installing software	Export HTML (3D web page for review) (p. 217)
6	**A** Expert and Premium	Interference and Clearance Check (p. 213)

Chapter 8 · CAM

No.	Answer	Review topic
1	**B** G54 and FANUC_MET 3Axis	New Job, Template and Geometry (p. 230)
2	**C** Click Update Geometry, then recalculate the toolpaths	New Job, Template and Geometry (p. 230)
3	**D** The part was overcut	Simulation (machining simulation and collision check) (p. 265)
4	**A** It has not been calibrated against a real machine; test the first program with care	Machine Manager and post-processors (p. 269)
5	**B** Drill in one pass	Drill and Hole (p. 250)
6	**C** An A4 landscape setup sheet with machine data, WCS, stock, tool and operation sequence, estimated time and a pre-run checklist	Generate G-code and Shop PDF (setup sheet) (p. 272)

Chapter 9 · Import / Export

No.	Answer	Review topic
1	**A** No; DWG is available only as an export from Drawing	MECHCAD file types and files you can open (p. 278)
2	**B** No, MECHCAD reads it by itself	Open SolidWorks and Alibre files in MECHCAD (p. 282)
3	**C** Export a STEP file from the original program and import the STEP	Open SolidWorks and Alibre files in MECHCAD (p. 282)
4	**D** It allows Boolean edits and mass calculation, but is slow for large meshes	Import CAD (STEP, IGES, STL) and CAD Import Options (p. 280)
5	**C** All visible parts	Export STEP and STL (p. 284)
6	**B** Parametric import and Geometry-only fallback	Import CAD (STEP, IGES, STL) and CAD Import Options (p. 280)

Chapter 10 · License & Editions

No.	Answer	Review topic
1	**C** Help › Deactivate License	License Management: Edition Badge, Deactivate and License Certificate (p. 289)
2	**A** The Welcome window opens	License Management: Edition Badge, Deactivate and License Certificate (p. 289)
3	**D** Help › License Certificate (PDF)…	License Management: Edition Badge, Deactivate and License Certificate (p. 289)
4	**B** Direct-installer customers with a Maintenance Agreement (MA)	Updates (Check for Updates) (p. 290)
5	**C** Download the latest installer and install it over the current version	Updates (Check for Updates) (p. 290)
6	**A** The automatic notice does not ask again for that version	Updates (Check for Updates) (p. 290)

Chapter 11 · Troubleshooting

No.	Answer	Review topic
1	**B** Every 10 minutes, when there are unsaved changes	Auto-Recover and crash reports (p. 300)
2	**D** Press Save (Ctrl+S) regularly	Auto-Recover and crash reports (p. 300)
3	**A** Switch to Shaded mode (no edges) and turn off Shadows / Ambient Occl. / Realistic Render	Troubleshooting Display and Graphics Card Problems (p. 296)
4	**C** Right-click › Auto Align (square to axes) or Align Face to Plane…	Troubleshooting CAD Import Problems (p. 301)
5	**B** Right-click the component › Purge (Skip Loading)	Large Assemblies Run Slowly (p. 298)
6	**C** The Machine ID, the version from Help › About, and the e-mail used to activate	Troubleshooting Licence Problems (p. 295)

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Appendix C · System requirements and support

System requirements

Operating system	Windows 10 / 11 64-bit
Processor	x64, 4 cores or more
Memory	8 GB (16 GB recommended for large assemblies)
Graphics	OpenGL 3.2 or later
Disk space	3 GB
Display	1920×1080 or larger, 100-200% scaling

Installing and updating

- Download the installer from www.mechdesigned.com/en/software/mechcad or the Microsoft Store (search "MECHCAD")
- 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 MECHCAD2026

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

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