

CAM Plug-in for TurboCAD 2026

The CAM Plug-in adds 2.5-axis milling, drilling and lathe toolpaths to TurboCAD 2026 Platinum and writes G-code for your CNC machine. This guide takes you from installation to your first G-code file in about ten minutes.

1. Check you can run it

You need	Details
TurboCAD 2026 Platinum, 64-bit	Build 33.0.65.1 or later (Help ▶ About). Update TurboCAD first if your build is older; the installer checks.
Windows 10 or 11, 64-bit	An administrator account for the installation, and an internet connection for activation.
Your license key	From your order email. It starts with the letters CM PEN .
Other TurboCAD editions	CAM commands only run in Platinum . If TurboCAD on this PC was last used as Professional, Deluxe or Designer, the installer warns you before activation, because activating uses your key on this PC.
Older CAM Plug-in (2019/2020, 32-bit)	Can stay installed. It belongs to the 32-bit TurboCAD 2019/2020 and keeps working there.

2. Install

1. Save your work and close TurboCAD (the installer closes it if it is still running).
2. Run `CAM_Plugin_for_TC2026_x64_65.1_r5.exe` and accept the Windows prompt. The installer is signed by **IMSI Design LLC**.
3. Confirm the TurboCAD 2026 folder (default `C:\Program Files\IMSIDesign\TCW2026`) and click **Next**.
4. **Activate**: the activation window opens in your Windows language. Enter the license key from your order email with your name and email address and click **Activate**. You need to be online (company proxies are fine). Without internet, email support@imsidesign.com for an offline activation code. Installing again or updating on the same computer does not ask for the key again.
5. Click **Install**, then **Finish**.

Your own tool library (`CAM\TOOLS\TOOLS.mdb`) and post-processor files (`CAM\CodeSystems`) are never overwritten. If you already have them, the installer keeps yours.

3. Start TurboCAD

Start TurboCAD 2026 Platinum. The CAM commands are on the **CAM** menu, and the CAM **palette**, which shows the job tree and the G-code, opens by itself a few seconds after the first start following the installation.

If you close the palette and want it back: **Options ▶ Toolbars and Menus, Palettes** tab, tick **CAM**, **Close**.

4. Your first job: a pocket

Set up the job

1. **CAM ▶ Start CAM Job...** opens the **CAM Setup Wizard**. Pick your machine type (Milling), units, material and post-processor, then finish the wizard. You can revisit each page later under **CAM ▶ Settings**.
2. **CAM ▶ Settings ▶ Tool Set Setup**: click **Add...** to open the **Tool Store**, choose a group (for example **Mill ▶ Flat End Mill**), select a cutter, add it to the tool set and close both dialogs with **OK**. The tool appears in the **Current Tools Setup** list.
3. **CAM ▶ Settings ▶ Part Setup**: set the stock size and confirm with **OK**.

Create the toolpath

4. Draw a closed shape, for example a 6 × 4 in rectangle.
5. Choose **CAM ▶ Functions ▶ Pocket Milling** and click the rectangle.
6. Right-click and choose **Setup** to check the cutting depth and stepover (the stepover must be smaller than the tool diameter), then right-click again and choose **Finish**. The toolpath is drawn and an operation appears in the CAM palette under **Part ▶ Tool**.

Write the G-code

7. Click the top item of the CAM palette tree. The palette's editor shows the G-code for the whole job.
8. **CAM ▶ G-code ▶ Save** writes it to a file for your machine.
9. Optional: **CAM ▶ Simulations ▶ Simulator** plays the cut back before you run it.

Toolpaths are linked to the geometry. Move or resize the shape and the operation regenerates.

5. What is on the CAM menu

Group	Commands
Milling	Pocket Milling, Side Milling, Path Milling, Chain-Offset Milling, Text Milling, Thread Milling, Offset Copying, Tool Moving
Holes	Drilling, Boring, Tapping, Tapping Head
Lathe	Lathe Turning, Lathe Groove, Lathe Drilling, Lathe Threading, Lathe Threading Tormach, Lathe PartOff
Contours	By Chain, By Pick, Join Polyline, Offset, Direction and Entry Point, Add Lead In/Lead Out, Lathe Polyline, Edit Lathe Polyline
Settings	Setup Wizard, Machine Setup, G-Code Generation Setup, Tool Set Setup, Part Setup, 3D Modeling Setup, CNC Setup
G-code	Save, Load, Send/Receive to and from the machine over RS-232

Post-processors included: Fanuc, Fanuc Lathe, Haas, Fadal, Mitsubishi, Servo and the generic base and Tormach formats (CAM\CodeSystems). For a lathe job, choose **Lathe** on the wizard's first page.

6. Uninstall

Windows Settings ▶ Apps ▶ Installed apps ▶ CAM Plug-in for TurboCAD 2026 x64 ▶ Uninstall. TurboCAD's own files are put back exactly as they were before the plug-in was installed. Your tool library and post-processor files stay.

If Windows still has a CAM file in use, the uninstaller finishes the job at the next restart. If it cannot, it stops without changing anything and asks you to restart and try again.

7. Good to know

Old jobs	Drawings with CAM jobs made in the 2019/2020 plug-in open in TurboCAD 2026 Platinum and produce the same G-code.
Licence check	The plug-in checks its licence when you use a CAM command. If the licence is missing, for example after a hardware change, it renews it online or offers to activate again.
Settings	CAM settings are kept per Windows user.

8. Help and support

- **Full reference:** the CAM Plug-in help file `TurboCADCAMHelp.chm` in the TurboCAD 2026 `CAM` folder (double-click to open).
- **This guide** is installed next to it as `CAM Plug-in Quick Start Guide.pdf`.
- **Support:** email support@imsidesign.com with your **order number** and your **TurboCAD build number**. For a G-code question, include the drawing and the post-processor you use.