

ProstoCAD

User Manual

Inventor / Alibre-style CAD on the FreeCAD 1.1.3 engine

Version 0.12.0 — test build (formerly InnCad Plus)

Windows 10 / 11 (x64), test build

ProstoCAD 0.12.0. Screen names (tabs, buttons, dialog fields) are written as they appear on screen; what changed from version to version is in CHANGES.md. October 2026.

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1 About ProstoCAD

ProstoCAD is a CAD application with the look and the working habits of Autodesk Inventor and Alibre Design.

ProstoCAD (formerly InnCad Plus) is the ProstoCAD for FreeCAD workbench (formerly InCad) made into an application of its own: it brings its own FreeCAD 1.1.3 engine inside and starts straight into ProstoCAD's interface. You do not need FreeCAD installed, and an installed FreeCAD and its settings are never touched. What you get:

- a **ribbon** with tabs that change with what you are doing (part, sketch, assembly);
- a **Model Browser** with one history for the whole part, sketches under the features that use them and an **End of Part** marker you can drag;
- a **Parameters** panel where every dimension and feature value has a name and can be used in formulas;
- **feature dialogs** with a live preview, drag handles on the model, **OK**, **Apply +** and **Cancel**;
- **marking menus** on the right mouse button, and Inventor's mouse navigation;
- Inventor-like **sketching**, **assemblies** with constraints, joints and degrees of freedom.

FreeCAD underneath

Everything ProstoCAD makes is ordinary FreeCAD geometry: a part is a FreeCAD Part Design body, a sketch is a Sketcher sketch, an assembly uses FreeCAD's own Assembly joints. Your files stay normal **.FCStd** files that FreeCAD (with or without the ProstoCAD for FreeCAD workbench) opens too, and ProstoCAD opens files made in FreeCAD.

A file that was not made in ProstoCAD opens as well, with every tool: when it comes to the front it becomes a ProstoCAD part by itself (its largest Solid is the main one; a STEP opened as it is gets a Solid for each of its shapes, where they are). Save keeps that.

This version: 0.12.0, a test build

ProstoCAD (called InnCad Plus until 0.11) has extra tools beside the workbench's: STL files taken apart and rebuilt, and an AI that builds a part for you (chapter 12). New in 0.12.0: the new name — InnCad Plus is now **ProstoCAD** (home page prostocad.com); settings move over by themselves. New in 0.11.0: **Mirror Solid** — a mirrored copy of a whole Solid that follows the original until Break Mirror Link, or Join / Cut into itself, of the whole Solid or one side of the plane (6.14); sketch **Mirror** whose copy follows its original without copying dimensions, and InCad's own **Symmetric**; Shift-click selects a sketch's whole outline; the **Parameters** window; Split ► **Gap**; ⚙ ► **Report a Bug**; right-click menus list only what applies; bigger, coloured gear handles; a Boolean whose clearance can't be made cuts with 0 mm. Fixed in 0.10.4: every plane of a row (**Planes**) opens its own dialog with a double-click — its offset and Turn — and a click on another plane of the row moves the dialog there; Point fields take a 3D Line's points and work points; Angle around edge and Normal to edge take an axis; Midpoint of edge lies halfway along a curve; Edge meets plane and Two edges meet refuse what does not meet; a 3D Line and a Text hide and show as sketches do. Fixed in 0.10.3: Appearance ► **Hide Edges** works after the Shaded button. New in 0.10.2: Hole

▶ Concentric and Thread on several round edges or faces at once (6.4, 6.10); Apply + keeps the settings; Delete asks to delete what is built on the feature. Fixed in 0.10.1: Work Axis ▶ Axis of a cylinder on a cone or part of a cylinder; every Point field of the work features lights the part's marked points; the view turns while points are picked; Hide / Show no longer rebuilds the part. New in 0.10.0: **Hide**, **Show** and **Show All** said outright, in the ribbon and every right-click menu; in an assembly a part hides and shows in the assembly's tree (9.9); files made outside ProstoCAD — a script's, FreeCAD's, a STEP — open with every tool; Save of an assembly puts it and all its parts into a folder of its name (9.9); the assembly gizmo smaller, shown for a couple of seconds on a click; scroll arrows on the file tabs. New in 0.9.0: **Section in assemblies** — every part the plane meets, each in its colour, also STEPs, other programs' files and FreeCAD's own assemblies (8.4); **Interference**: overlapping parts in bright red; Hide / Show of axes, origin and sketches in assemblies; **Open** a component's part in its own window (9.9). New in 0.8.0: **3D Line** drawn on a virtual cube of points (a path for Sweep, 6.18); several work planes in one go (**Planes**) and turned with rings (**Turn**); Extrude **Extents 2** for the second side, and To an edge or a work axis; **Full** beside Fillet's radius; the Open window shows the file's picture; tips blue, errors red; Import of IGES and OBJ; an **Image** tab — a photo to sketch over, scaled by a known length; View ▶ **Screenshot**; the parts' axes in assemblies, to constrain to; fixed: a Sweep's wall stays even when the profile's sketch is flipped on its plane; a Sweep made away from the part no longer sends a dialog round and round. New in 0.7.0: Extrude ▶ **To** any face of any Solid, a curved one too, or any point; Emboss letters past the face run on round onto the faces that go on smoothly from it; End of Part above sketches, Texts and work planes; **Full Round Fillet** (two edges); Show / Hide All Sketches; a part goes see-through while you sketch on a plane inside it. New in 0.6.0: a **Compact** ribbon (View ▶ Ribbon Size: icons without names, the lowest ribbon), Work Plane ▶ Three points clicked on marked points as in Section; fixed: Offset and Symmetric on slots, Section on the XY / XZ / YZ plane. Fixed in 0.5.2: no more Qt warnings popping up at the pointer. Fixed in 0.5.1: ProstoCAD no longer hangs when FreeCAD shows a message pop-up (after a long Smooth). Fixed in 0.9.1: a feature made in a hidden Solid shows that Solid. New in 0.9.0: **Import** in an assembly (each solid a part file); Import and Export in Tools ▶ Convert. New in 0.8.0: **Import** of STEP files (File ▶ Import). New in 0.7.0: **Collinear** in the sketch, sizes typed while drawing (Tab), an **Assembly** tab in parts. New in 0.6.0: Spur Gears and Worm Gear make the pair in one dialog, Make Assembly puts each gear in a part file (6.17). New in 0.5.0: the assembly's gizmo on gears and an Axis for them (6.17). New in 0.4.0: a gear pair sized by **Ratio** or **Center distance**, and a Worm that meshes with a gear made before it (6.17). New in 0.3.0: **Gear** (spur and helical, a meshing pair set up for you) and **Worm** with its wheel (6.17). 0.1.0 was InCad 0.5.5 as an application of its own. New in 0.2.0: **Auto** Join / Cut for Extrude and Revolve (6.2, 6.3), **drag handles** on the model (6.1), loops and an edge list in Fillet and Chamfer (6.5), **Excluded** copies in patterns (5.9, 6.14), dimensions while drawing (5.2), the projected edges setting and projection of what a drawn point touches (5.7), an STL precision slider (3.10), negative values with the ▲▼ arrows and sketches that always face you (5.1). 0.2.1 fixes: dimensions to any edge or corner of the part (5.3), × in the value box, the version in the title strip — CHANGES.md has the full list.

Please: save your work often and work on **copies** of files that matter to you. If something looks wrong, tell us — chapter 11 says what to send.

2 Installation

ProstoCAD runs from its folder — no installation needed — or installs with its setup program.

Requirements

- Windows 10 or Windows 11, 64-bit (x64).
- About 600 MB of disk space. FreeCAD is not needed: ProstoCAD brings its own engine.

A first test build. ProstoCAD is developed on the Mac; expect problems on Windows, work on **copies** of your files and send the log with every report (chapter 11).

Portable: run it from its folder

1. Unzip **ProstoCAD-...-windows-x64.zip** (right-click ► **Extract All...**) into a short path, for example `C:\ProstoCAD`. Do not run anything from inside the zip.
2. Double-click **ProstoCAD.exe**. If Windows says "Windows protected your PC": **More info ► Run anyway**.

The folder can live on a USB stick; nothing is written next to it.

Install with the setup program

Run **ProstoCAD-...-windows-x64-setup.exe**. It installs for your user only (no administrator rights) into `%LOCALAPPDATA%\Programs\ProstoCAD`, adds ProstoCAD to the Start menu and the desktop, and offers ProstoCAD in **Open with** for `.FCStd` files (your default program for them does not change).

The first start

The first time, ProstoCAD asks what to make and in which units: **Part** (one part file) or **Assembly** (place and constrain part files), and **Millimeters** (mm, ISO threads listed first) or **Inches** (in, UNC/UNF threads listed first). With **Remember my choice** checked, **New** makes the same kind of file without asking next time. Click **Create**.

To bring the question back, turn on **Tools ► Options ► Ask on New** on the ribbon, or choose **⚙ ► Units for New Files ► Ask every time**.

Update to a new version

Portable: quit ProstoCAD and replace its folder with the new one. Installed: run the new setup program. Your settings stay.

Coming from InnCad Plus? ProstoCAD is its new name. On its first start your settings, recent files and AI Link move over by themselves. If you connected an AI by hand (an MCP settings file), point it at the new path (12.3), or click **Connect AI** once. You can then delete the old InnCad Plus app.

Uninstall

Portable: delete the folder. Installed: **Settings** ▶ **Apps** ▶ ProstoCAD ▶ **Uninstall**. The settings (%APPDATA%\ProstoCAD) and the log folder (%LOCALAPPDATA%\ProstoCAD) stay; delete them too if you like. Your .FCStd files keep working in FreeCAD.

3 The ProstoCAD window

A short tour: what each area of the window is for.

3.1 Title strip

The strip at the very top holds quick buttons — **New**, **Open**, **Save**, **Undo**, **Redo** and **Update** — the name of the file, and on the right **Light / Dark** and  (ProstoCAD settings).

Update recomputes the model and redraws the browser and the view. Use it if the view and the browser ever seem out of step with the model; in an assembly it also brings in parts that changed (see 9.10).

3.2 Menus

ProstoCAD keeps a short menu bar: **File** (New, New Assembly, Open, Save, Save As, Close, Quit), **Edit** (Undo, Redo) and **View** (Light / Dark, Zoom All, Home View, Shaded, Wireframe, Assembly Context, Parameters, Ribbon Size). Everything else is on the ribbon and in the marking menus.

3.3 The ribbon

The ribbon's tabs depend on what is open:

Mode	Tabs
No file, or a file not made in ProstoCAD	Get Started (New, New Assembly, Open, Import), View
A part	3D Model , Assembly , Inspect , Tools , View
Editing a sketch	Sketch , View
An assembly	Assembly , View

A button with a small arrow is a **split button**: it runs the command shown, and its arrow lists the variants (for example Rectangle ▸ Center Rectangle, 3-Point Rectangle ...). Hover over a button for its full name and a tip. **View ▸ Ribbon Size** (or  ▸ **Ribbon Size**) switches between **Compact** (icons without names, which are in the tooltips: the lowest ribbon), **Small** and **Large**. Appendix B lists every command of every tab.

3.4 The Model Browser

The **MODEL** panel on the left is the history of the part, as in Inventor:

- The top row is the part. Under it: **Origin** (XY, XZ and YZ Plane, X, Y and Z Axis, Center Point), then **Solid Bodies (n)** when the part has more than one solid (chapter 7), then the features in the order they were made — one history for every solid of the part.
- A sketch sits under the feature that uses it. A sketch row shows ✓ when it is fully constrained, or how many degrees of freedom are left (**2 DOF**).
- **End of Part** marks where the model ends. Drag it up to see (and work on) the part as it was at that point; features below it are greyed and not computed. It can also go above sketches,

Texts and work planes: they are greyed and hidden while they are below it. Right-click a feature ▶ **Move End of Part Here** does the same. Drag it back down to the end when you are done.

- Row colours: a feature that failed is **red** (hover over it for the reason); a suppressed feature is greyed and ~~struck through~~; a hidden item is dimmed.

Click a row to select it: the faces the feature made light up in the view. A hidden plane, axis, point or sketch is shown while its row is selected and hidden again when you select something else.

Double-click a row to edit it: a feature opens its dialog, a sketch opens for editing, an origin plane starts a new sketch on it, a work plane opens its dialog. With **Shift** or **Ctrl** you can select several rows.

Right-click a row for its marking menu (chapter 4):

Row	Commands
A feature or sketch	Edit, Move End of Part Here, Suppress / Unsuppress, Rename, Show in Window, Start 2D Sketch, Delete, Hide / Show
A solid (Solid Bodies)	Hide / Show, Rename, Show in Window, Delete
An origin plane or axis	Start 2D Sketch, Hide / Show

- **Show in Window** lights up the feature's faces and zooms the view to them. Its opposite, **Show in Browser** (right-click a face in the view), opens the browser at the feature that made that face.
- **Suppress / Unsuppress** turns a feature off without deleting it. A thread is suppressed together with the helper steps ProstoCAD made for it; picking a feature's profile sketch means the feature.
- **Delete** removes a feature together with everything ProstoCAD made for it (thread profiles, Boolean helpers, assembly cuts), as one undo step. **Backspace** or **Del** deletes the selection. When the delete affects more than the row — a solid used by a Boolean, the main solid, a Split with pieces — ProstoCAD asks first and says what will go.
- **Hide, Show** and **Show All** (View ▶ Visual Style ▶ Hide / Show, and in the right-click menus) hide or show the selected solid, assembly part, origin plane or axis, the Center Point or a sketch; a hidden one is picked by its browser row. **Show All** brings back every hidden solid of the part, every hidden part of the assembly, with nothing to pick.

3.5 The Parameters panel

Below the browser, **PARAMETERS** lists the part's values in three columns: **Name**, **Value** (what you typed — a number or a formula) and **Result**.

- **Model parameters** (shown in the accent colour) are made automatically: every sketch dimension and every feature value gets the next free name **d0, d1, d2 ...**; assembly constraint values are named after the constraint (**Mate1, Angle1 ...**). Type a new name over a model parameter to rename it — formulas that use it follow.
- **Your own parameters**: click **+**, type a name, press **Tab** and type the value (Width, 40 mm). Use the name in any value box or dialog field.
- Change a value in the Value column and the model updates.

- – removes the selected parameter. A value that is still used (by a feature or a formula) is not removed: the status line says what uses it. This is how you clean up values left behind by deleted features. A sketch dimension (d1, d2...) is deleted from its sketch the same way; one that a formula uses stays.
- While a value box or a dialog field is open, click a parameter's name to insert it into the formula.

View ▸ Windows ▸ Parameters (or the marking menu) opens the **Parameters window**: every parameter in one big table you can resize — **Name, Value, Result** and **Used by** (the features, sketches and formulas that use it; «— not used» in grey). It stays open while you work and follows every change. Type a value or a formula in **Value**, type a new name over a dimension or a feature value to rename it, **+ Add** and **– Remove** as in the panel, **Find a name** narrows the list, and **Delete Unused** removes every value nothing uses (values left behind by deleted features) in one undo step.

3.6 Values, units and formulas

Wherever ProstoCAD asks for a length or an angle you can type:

- a number, in the file's units (12), or with units (12 mm , 0.5 in , 30°);
- a parameter name (d3 , Width);
- a formula (d0/2 , Width – 2*d3 , d3*2).

A value that is a formula updates whenever the values it uses change.

3.7 The status line

At the bottom of the window: on the left, what to do next (for example "Pick a plane or a flat face for the new sketch") and messages — in orange or red when something did not work, so look here first when "nothing happens". On the right: the units of the file (**mm · deg**), while sketching the sketch's freedom (**DOF: 3** or **Fully constrained ✓** — click it to see the free geometry), and the size of the part (**2 solids · 14 features**).

3.8 Light, dark and the settings menu

Light / Dark on the title strip (also **View ▸ Appearance** and the the ☿ menu) switches the theme in one click. The ☿ menu holds ProstoCAD's settings:

⚙ menu	Choices
Theme	Match Windows, Light, Dark
Ribbon Size	Compact, Small, Large
Units for New Files	Ask every time, Always millimeters, Always inches
Dimension Display	How sketch dimensions are labelled: Value (40), Name (d3), Expression (d3 = 40)
Display Quality	Normal (faster) or Fine (smoother curves): how finely round faces are drawn. Fine keeps a highlight on a ball or a fillet free of facets; it changes only the drawing, never the part, and does not mark a file as changed. Normal by default on Windows (built-in graphics draw Fine more slowly).
Dimensions While Drawing	On: a rectangle, circle, slot, polygon, line or arc gets its dimensions as it is drawn (5.2). Off by default.
Type Sizes While Drawing (Tab)	On (as it comes): while a line, rectangle, circle ... is drawn, its size fields show at the pointer — type a value, Tab goes to the next (a rectangle's width, then its height), the click or Enter finishes.
Polygon Circle	The construction circle of a new polygon: Inscribed (it touches the sides — across flats; the default, as in Inventor) or Circumscribed (through the corners, as FreeCAD draws it)
Projected Edges	Normal geometry (the default, as in Inventor) or Reference (construction) ; and Project What a Drawn Point Touches (on by default) — 5.7
Show Log File	Shows ProstoCAD's log (ProstoCAD.log) selected in the File Explorer — the file to send with a problem report (chapter 11)
Report a Bug...	Sends a problem report straight to the public issue list of ProstoCAD on GitHub — no GitHub account needed: what happened, the steps, a contact if you want an answer, and the end of the log (your name is taken out of the file paths). The answer comes there (chapter 11)
Mouse Navigation	On: ProstoCAD's mouse (LMB+RMB orbit, right-click menu). Off: FreeCAD's own navigation style applies.

3.9 Undo and Redo

Ctrl **Z** undoes, **Shift** **Ctrl** **Z** redoes (also the title strip buttons and the NW / NE slots of the marking menu). Every operation is **one** undo step: a feature made in a dialog, a delete with its helpers, a hide, a drag of an assembly part. Inside a feature dialog, Undo steps back through the dialog's own changes. Undo also works in a sketch.

While a feature dialog is open, commands that would interfere with it (Save, Undo outside the dialog, editing a sketch) are refused with "Finish the ... dialog first". Click **OK** or **Cancel** first.

3.10 Files

New (**Ctrl** **N**) makes a part or an assembly (as chosen at the first start), **File ▶ New Assembly** always an assembly. **Open** (**Ctrl** **O**), **Save** (**Ctrl** **S**), **Save As** (**Shift** **Ctrl** **S**) work on the file in front. **Close** (**Ctrl** **W**) asks whether to save changes (**Save**, **Don't Save**, **Cancel**); **Quit** (**Ctrl** **Q**) asks about every changed file. Each open file has its own tab above the view. The **Open** window (and **Import**, **Place**) shows a picture of the file under the pointer — the view a FreeCAD file was saved with, an STL, STEP or IGES drawn shaded (a STEP over 40 MB is not read) — with its date and size.

File ▶ Import also reads **IGES** (.iges, .igs: loose faces are sewn into a solid) and **OBJ** meshes (as an STL). A **JPG**, **PNG** or **BMP** becomes a picture to sketch over (6.19). **Open** lists FreeCAD files, STEP and STL only; a STEP or STL opened makes a new part, as **Import** does.

View ▶ Capture ▶ Screenshot (also **File ▶ Screenshot**) saves the 3D view as PNG, JPEG or BMP at the screen's full resolution (twice the window's size on a Retina screen); a PNG can have a transparent background, for a web page. **Screenshot (Window)** (the arrow beside it) saves the whole window — ribbon, browser, 3D view and status bar.

File ▶ Export writes the part (every visible solid) or the assembly to **STEP** (exact solids) or **STL** (a mesh for slicers). For STL the **STL Precision** window asks how finely to mesh: a slider from **50 %** (coarser, a small file) to **100 %** (smoother curves), with the number of triangles and the file's size for the value chosen. ProstoCAD remembers the value.

File ▶ Import (also **Tools ▶ Files ▶ Import** and **Get Started**) reads a **STEP** file (.step, .stp) from another CAD: each solid in it becomes a Solid of the part (the first one fills an empty part's first Solid), drawn at its place in the file, in millimeters. Sketch on its faces, cut, fillet, split it or make an assembly of it like any Solid; the imported shape itself stays as it came. With no part open, **Import** makes a new part named after the file. In an assembly (**Assembly ▶ Component ▶ Import**) each solid becomes a part file next to the assembly, placed as a component where the file has it; the first is grounded. A new assembly is saved next to the imported file first.

4 Mouse, views and marking menus

ProstoCAD uses Inventor's mouse: the left button selects, both buttons orbit, the right button opens marking menus.

4.1 Mouse

Action	What it does
Left click	Select. Shift -click or Ctrl -click adds to the selection.
Left + right button drag	Orbit around the point under the cursor.
Middle button drag	Pan.
Wheel	Zoom at the cursor.
Right click	Marking menu (in the view and in the Model Browser).
Right button drag	Flick: runs the marking-menu command in that direction without waiting for the menu.
Double click	In the browser: edit the row (a feature, a sketch, an assembly part). In a sketch: edit a dimension, or the pattern of a copy.

No middle button? **Orbit** and **Pan** on the navigation strip (right edge of the view) or on the View tab turn the left button into orbit or pan; click the button again to stop.

⚙️ ▶ **Mouse Navigation** switches ProstoCAD's mouse handling off; FreeCAD's own navigation style then applies.

Sketch tools: while FreeCAD's line (polyline) or spline tool is drawing, a right click finishes the chain, as in FreeCAD, instead of opening the menu.

4.2 Marking menus

A right click opens a ring of eight commands around the cursor (north, north-east ... north-west) and a short list below it. What it offers depends on what is under the cursor — empty space, a face, an edge, a sketch, an assembly part, a browser row. Once you know where a command sits, **flick**: press the right button, drag towards the command and release — no need to wait for the menu. The list shows only what applies there: Hide on something shown, Show on something hidden, Show All only when something is hidden, Thread only on a round face, Make Assembly only with two Solids or more, the draw style you are not in; the ring keeps its places.

Under the cursor	Ring (N, S, E, W first)	List
Empty space (part)	N Start 2D Sketch, S Extrude, E Shaded, W Wireframe, NW Undo, NE Redo, SE Parameters, SW Measure	Zoom All, Home View, Hide / Show, Show All Origin, Hide All Origin, Make Assembly, Move Face
A face	N Start 2D Sketch, S Extrude, E Shaded, W Wireframe, NW Undo,	Show in Browser, Zoom Selection, Hole, Fillet, Move Face, Thread,

	NE Look At, SE Home View, SW Zoom All	Hide / Show
An edge	N Look At, S Zoom Selection, E Fillet, W Chamfer, NW Undo, NE Redo, SE Home View, SW Zoom All	Show in Browser, Shaded, Wireframe, Hide / Show
In a sketch	N Finish Sketch, S Dimension, E Circle, W Undo, NE Line, SE Rectangle, SW 3-Point Arc, NW Trim	only what applies: Delete for whatever is selected — lines, points and dimensions (a right-click on something not selected selects it first); on a line or point Construction, Show / Hide Constraints and its Constraints (n) ... (the Constraint List filtered to it: delete one or all there, section 5.6); Edit / Delete Pattern on a pattern's geometry, Break Mirror Link on a mirrored copy; on empty space Show / Hide Constraints and the Constraint List; Stop Tool first while a tool runs
Empty space (assembly)	N Place, S Mate, E Rigid, W Create, NW Undo, NE Zoom All, SE Home View, SW Ground	Edit, Hide / Show, Suppress, Remove from Assembly, Flush, Insert
A face of a part (assembly)	N Mate, S Flush, E Angle, W Tangent, NE Look At, SW Ground	Edit, Hide / Show, Suppress, Remove from Assembly, Rigid, Parallel, Perpendicular, Zoom Selection
An edge of a part (assembly)	N Insert, S Rotational, E Cylindrical, W Slider, SW Ground	Edit, Hide / Show, Suppress, Remove from Assembly, Rigid, Parallel, Perpendicular, Angle, Zoom Selection
No file open	N New, S Open, E Zoom All	—

The browser rows have their own menus (chapter 3; assembly rows in chapter 9).

4.3 View commands

On the **View** tab (and partly on the navigation strip and the View menu):

Command	What it does
Light / Dark	Switch the theme.
Zoom All	Fit the whole model in the view.
Home View	The standard isometric view.
Look At	Look straight at the selected face or plane; while sketching with nothing selected, straight at the sketch.

Zoom Selection	Fit the selection in the view.
Orbit, Pan	The left button orbits or pans until you click the button again.
Shaded, Wireframe	Visual style. Wireframe lines are drawn light in the dark theme and dark in the light theme.
Section	Cut the view open (section 8.4). Click again to close.
Section by 3 Points	Click three marked points for the section plane.
Hide / Show	Hide or show the selected solids, assembly parts, origin items or sketches.
Show All / Hide All	Show All Origin, Hide All Origin: every origin plane, axis and the centre point. Show All Sketches, Hide All Sketches: every sketch and Text of the part, the ones its features use too.
Assembly Context	In a part opened from an assembly: show or hide the other parts around it (9.9).
Degrees of Freedom	In an assembly: draw the DOF symbols on every free part (9.7).
Parameters	Open the Parameters window: every parameter, what uses it, Delete Unused (3.5).

5 Sketching

Sketches work as in Inventor: draw roughly, then dimension and constrain. Every dimension has a name you can use in formulas.

5.1 Starting a sketch

1. Select a flat face of the part or a plane (in the view or the browser) — or select nothing.
2. Press **S**, or click **3D Model ▶ Sketch ▶ Start 2D Sketch**, or right-click ▶ **Start 2D Sketch** (N in the marking menu).
3. With nothing selected, ProstoCAD shows the origin planes and a small panel **New sketch on** with **XY · Top**, **XZ · Front** and **YZ · Right**: click a button, or click a plane or a flat face in the view. **Esc** or **Cancel** gives up.

A sketch on a plane always faces you: ProstoCAD turns it so that you look at its front, whichever side of the plane you were looking from (seen edge-on, the front is the standard views' side). A sketch on a face looks out of the part.

Double-clicking an origin plane in the browser also starts a sketch on it (on a work plane: right-click ▶ New Sketch). To edit an existing sketch, double-click it in the browser (or right-click ▶ Edit).

To leave the sketch: **Sketch ▶ Exit ▶ Finish Sketch**, N in the marking menu, or **Esc** (the first **Esc** stops a running tool).

5.2 Drawing tools

All on **Sketch ▶ Create**. The tools are FreeCAD's Sketcher tools; split buttons hold the variants.

Button	Variants	Key
Line	Line (a chain of lines; M while drawing switches between line and arc), Single Line	L
Spline	Spline (through points), Control-Vertex Spline	
Circle	Circle (centre, radius), 3-Point Circle, Ellipse, 3-Point Ellipse	C
Arc	3-Point Arc, Center Arc, Elliptical Arc	A
Rectangle	Rectangle (corner, width, height), Center Rectangle, 3-Point Rectangle, 3-Point Center Rect., Rounded Rectangle. Each rectangle gets its two diagonals as construction lines and a construction point where they cross (a Rounded Rectangle's diagonals join its corner arcs' centres; a Center Rectangle has its own centre point): one constraint — that point on the origin — centres the rectangle. They follow the rectangle and can be deleted like any line.	R
Slot	Center-to-Center Slot, Overall Slot (the two outer ends, then the width), Center Point Slot (the centre, one end's centre, then the width), Arc Slot, Flat-End Arc Slot	
Polygon	Hexagon, Triangle, Square, Pentagon, Octagon, Polygon (N sides)	
Point	A sketch point (hole centres, for example)	

Sketch ▶ Modify: Fillet and **Chamfer** (sketch corners), **Trim** (**X**), **Extend**, **Offset**.

Sketch ▶ Pattern has the sketch patterns (5.9) and **Mirror**.

Offset (ProstoCAD's own): click a line — the whole shape it belongs to is taken (a rectangle, a polygon, a chain drawn line by line); **Ctrl**-click adds more shapes. The copy follows the pointer; a click places it. Corners stay sharp, a single line gives one parallel line, a circle or an ellipse gives a ring. One dimension drives the offset and the value box opens on it. Geometry selected before the command is taken at once. **Mirror** and the patterns also take the whole shape from a click on one side.

Whole shapes. **Shift**-click a line to select its whole chain or shape at once (it shows selected); **Shift**-click another to add it. A plain click still selects one line.

Mirror. Select the shapes (a click on one side takes its whole outline, which shows selected), click **Mirror**, then click the mirror line — any line, also one of the shape's own (a half profile mirrored about its closing line joins up), the X or Y axis — or a point. With the mirror line already selected last, the copy is made at once. The copy is held to the original by symmetry alone: it keeps the original's size and follows every change of it, and no dimension is copied. Right-click the copy ▶ **Break Mirror Link** frees it: it keeps its shape (its corners stay joined) and no longer follows the original.

Typing values while drawing. While a tool draws, type the length, radius or angle directly; **Tab** moves to the next field. **Esc** or **Stop Tool** (marking menu) ends the tool.

Dimensions while drawing. With **☞ ▶ Dimensions While Drawing** on, what you draw gets its dimensions at once, at the drawn values (nothing moves): a rectangle its width (bottom) and height (left), a circle its diameter, a slot its radius and the distance between its centres, a line — and each segment of a chain — its length, an arc its radius. A **polygon** gets the diameter of its circle and one side's length as a **reference** dimension; double-click that side's dimension to make it the one that drives the polygon (the circle's becomes the reference). What a snap or a typed value already fixes gets no second dimension, and one Undo takes the drawing with its dimensions.

Polygons. A polygon is drawn round a construction circle. With **☞ ▶ Polygon Circle ▶ Inscribed** (the default) the circle touches the sides, so its diameter is the size across flats (a nut's spanner size); with **Circumscribed** it runs through the corners. The polygon stays regular whatever you dimension.

Delete geometry or dimensions: select them and press **Backspace** or **Del**.

5.3 Dimensions

1. Press **D** (or **Sketch ▶ Dimension**, S in the marking menu).
2. Click a line, a circle, two points or two lines, and place the dimension. An edge or corner of the part counts too: click it and it is projected into the sketch (as construction) for the dimension — no **Project Geometry** first.
3. A value box opens: **d3 = 40 mm**. Type the value (or a formula) and press **Enter** or **✓**; **Esc** or **x** takes the new dimension away again. A value that cannot be used turns the box red and says why.

Double-click a dimension to change it later. Every dimension gets a name (**d0, d1 ...**) and appears in the Parameters panel, where you can rename it (**Width**) — see 3.5.

Formulas. In the value box, click another dimension in the sketch or a parameter in the Parameters panel to insert its name: `d0/2`, `Width - 2*d3`. A formula may use other dimensions and parameters, but not a reference (driven) dimension of the same sketch.

A slot's length, or the distance between two arcs or circles. It can be measured three ways, and the value box of such a dimension shows three buttons to choose: **Centers** (centre to centre), **Overall** (outer edge to outer edge — a slot's overall length) and **Gap** (inner edge to inner edge — what FreeCAD's Dimension gives when you click two arcs). A click switches the dimension at once; it keeps its name and the box opens on its new value.

Sketch ▸ Dimension ▸ Overall Dimension makes the overall one directly: click one edge of a slot (or two arcs or circles) and type the value. A new overall value makes the slot longer, not wider. If the slot's length is already fixed by other dimensions, the new dimension is a reference (driven) one that follows. A dimension used in a formula keeps its kind.

 ▸ **Dimension Display** chooses how dimensions are labelled: the value (40), the name (d3) or both (d3 = 40).

5.4 Constraints

Sketch ▸ Constrain: **Coincident**, **Concentric** (two circles, arcs or ellipses: their centres are tied), **Horizontal**, **Vertical**, **Parallel**, **Collinear** (two lines on one straight line), **Perpendicular**, **Tangent**, **Equal**, **Symmetric**, **Fix** and **List** (the Constraint List, 5.5). Select the geometry, then click the constraint (or click the constraint, then the geometry).

Symmetric takes whole pieces, not only points: click two points, two lines (their ends are paired up) or two circles / arcs (their centres, and equal radii), then the mirror line — any line, the X or Y axis — or a point. A click near an end or a centre takes that point, otherwise the line under the pointer; dimensions are never caught. What you picked shows selected. Once the pair is made the tool ends, so you can drag the pair at once (it moves as a mirror image); click **Symmetric** again for another pair.  cancels a pick.

The status line shows the sketch's degrees of freedom — **DOF: 3**, or **Fully constrained ✓**. Click it, or **Sketch ▸ Format ▸ Show Free Geometry**, to select the geometry that can still move.

Sketch ▸ Format ▸ Show / Hide Constraints hides or shows the constraint symbols (dimensions stay);  shows them,  hides them.

5.5 The Constraint List

Open it with **Sketch ▸ Constrain ▸ List**, or right-click in the sketch ▸ **Constraint List**. It lists every constraint of the sketch with what it ties and its value — including the ones that have no symbol to click, such as coincident points and a point on a line.

- Pick a line or a point in the sketch: the list shows only that element's constraints. Pick nothing to see them all.
- Select a row to see its constraint in the sketch.
- Check rows (or **Select all**) and press **Delete selected**. This is one undo step.

- Constraints that hold a sketch pattern together, and dimensions used in a formula, are kept; the window says why.

5.6 Deleting constraints with the right button

Right-click a line or a point in the sketch: the marking menu's list then has a **Delete ...** entry for each of its constraints (for example **Delete Horizontal**, or **Delete Distance (Line 2) d4 = 20 mm**), **Delete all constraints (n)**, and **Constraints...**, which opens the Constraint List for that element.

5.7 Project Geometry and construction

Sketch ▸ Insert ▸ Project Geometry brings edges and points of the part into the sketch. As in Inventor, projected edges are **normal lines**, ready to be part of a profile; with **☼ ▸ Projected Edges ▸ Reference (construction)** they come in as construction instead. They follow the part when it changes.

A point drawn onto the part. When a point you draw — a line's end, a corner, a centre — lands on an edge or a corner of the part that lies in the sketch's plane, ProstoCAD projects that edge (normal or reference, as the setting above says) or that corner and keeps the point on it. Turn this off with **☼ ▸ Projected Edges ▸ Project What a Drawn Point Touches**.

Sketch ▸ Format ▸ Construction works two ways:

- with geometry selected (also projected edges), it switches that geometry — only the selected edges — between normal and construction (for projected edges the status bar says which they are now: they look much the same either way);
- with nothing selected it is a mode: everything you draw is construction until you click it again.

Construction geometry does not take part in profiles; use it for axes and helper lines. It is drawn **orange** and dashed, thinner than the normal lines — a projected edge switched to construction too.

5.8 Seeing the sketch

- **Slice Graphics** (**F7**, **Sketch ▸ Format**) cuts away the part in front of the sketch plane, so you can see a sketch inside the part.
- **Look At** (marking menu) turns the view straight onto the sketch.
- In the dark theme the part is drawn darker while a sketch is open, so the sketch's lines stand out (the part's own colour does not change).
- After **Finish Sketch**, a sketch that no feature uses and that is an **open contour** (a path for a Sweep, say) stays visible in the model; a closed sketch, and any sketch a feature uses, is hidden. **Hide / Show** overrides this for any sketch.

5.9 Sketch patterns

Sketch ▸ Pattern repeats sketch geometry. The copies are real sketch geometry, tied to the original: change the original and the copies follow.

Edit Pattern: double-click a copy, or select a copy and right-click ► **Edit Pattern**. **Delete Pattern** (right-click on a copy) removes the copies; the original stays. If you delete single copies, the pattern is rebuilt the next time you edit it.

Excluded (every sketch pattern): click the field, then click the copies the pattern does not need — they are left out. The list under the field names them (**Copy 3**); × puts one back, **Clear** puts them all back. Copies are numbered from 1, the original (which always stays); with two directions the first direction counts first.

Rectangular

Field	Meaning
Geometry	The lines, arcs, circles and points to repeat.
Distribution	By spacing (a direction, a count and a spacing) or Equal spacing (a count spread evenly between two limits).
Direction 1, Flip direction	A line of the sketch, or the X or Y axis.
Count, Spacing	How many (2), how far apart (10 mm).
From, To	Equal spacing: the limits — a line or a point.
Two directions	By spacing: a second direction with Direction 2, Count 2 and Spacing 2.

Circular

Field	Meaning
Geometry	What to repeat.
Axis	The centre point (any point of the sketch, a projected point or the origin).
Count, Angle	How many (4) over which angle (360°).
Distribution	Fitted (the copies fill the angle), Between copies (the angle is the step from one copy to the next) or Equal spacing (the copies spread round the axis between From and To — a point or a line of the sketch — as far from From as from To).
Flip	The other way round.

Along Path

Field	Meaning
Geometry	What to repeat.
Path	The path's lines, arcs and splines, in order.
Count	How many (3).
Distribution	By spacing (with Spacing, measured along the path) or Fit to path .
Orientation	Identical (every copy as the original) or Follow path (turned with the path).
Flip	Go along the path the other way.

5.10 Text

Sketch ▶ Create ▶ Text writes words in the sketch; Extrude raises them (Join) or engraves them (Cut), Emboss wraps them round a cylinder or a cone, or projects them onto a sphere (6.16).

1. Press **Text** — the cursor shows an orange **Aa** — and click two corners: a construction box (orange, dashed, never extruded) is drawn and the dialog opens on it.
2. Type the words (several lines are fine), pick the font and the size, and press **OK**.

Field	Choices and meaning
Text	The words; Return starts a new line. Cyrillic works.
Font, Bold, Italic	The fonts installed in Windows; Bold / Italic use the font's own Bold and Italic files.
Mirror	Flips the text left-right, to read it from the plane's other side (a sketch on the underside of a part, or on a plane turned the other way).
Size	The height of the capital letters (5 mm), a named value like any other (3.6).
Placement	In a box or Along a curve .
Box	The box the text sits in: the one drawn by the two clicks, or a line of any rectangle of the sketch. Dimension or constrain the box (centred on a face, 3 mm from an edge) and the text follows; a tilted box tilts the text.
Horizontal, Vertical	Left / Centre / Right and Top / Middle / Bottom in the box, by the letters themselves. A text longer than the box runs past it.
Fit to the box	The letters scaled so the widest line fills the box's width (never taller than the box); Size then hides.
Curve, Position, Shift, Distance, Flip side	Along a curve: any line, arc, circle, ellipse or spline of the sketch (construction too). Select it before pressing Text and the text runs along it — each letter turned with it, Centre at a closed curve's top, Shift moves it along, Distance lifts it off, Flip side puts it on the other side (inside a ring it then reads at the bottom, as on a coin).
Letter spacing	Extra space between letters (0 mm).

The text is one object (a **Text** row in the browser): double-click it to change the words or the font — its sketch opens with the dialog. While the sketch is open the text follows its box or curve at once, and its outline is drawn on top. A Text that an Extrude or Emboss uses is hidden and shows under that feature.

6 Part features

Every feature opens a dialog with a live preview. This chapter first explains how the dialogs work, then each feature with what to pick and what each option means.

6.1 How feature dialogs work

- **Select first or pick later.** Select the faces, edges or sketch first and then click the command — or click the command and pick in the view or the browser while the dialog is open. The highlighted field takes the picks. Click another pick field (**Select...**) to make it take the next picks. While you pick, the shape as it was before the feature is shown, so you can reach every face.
- **The profile.** Extrude and Revolve take the only finished sketch no feature uses yet automatically; if there are several, select one. If a sketch has crossing lines, ProstoCAD shows its areas: click the one to use, or **Shift**-click several and then click the last one (areas side by side become one outline).
- **Preview.** Every change shows on the model at once. While you pick edges or faces, Fillet, Chamfer and Move Face draw the result's new edges over the part in orange. The full check of the result's geometry runs on **OK**; if it finds a problem, the dialog stays open and says what.
- **Values.** Type numbers with or without units, names or formulas (3.6). The ▲ ▼ arrows next to a value — and the **↑** **↓** keys — change it by one unit (1 mm, 1°, 1/16 in); with **Shift** by ten. Values that may be negative (a work plane's offset, Move Face's distance) step through zero. A formula cannot be stepped. The parameter's name is shown after the field's label, for example **Distance (d19)**; click a parameter in the Parameters panel to insert its name.
- **Drag handles.** Extrude shows an arrow at the end of the extrusion, Revolve an arc with a knob where it ends, Move Face an arrow on the face. Drag one to change the distance or the angle; the preview follows as you move. A drag steps in whole millimetres or 5°; hold **Ctrl** for any value. Over a handle the pointer and the status line say how to drag it.
- **Choices.** Options are buttons; long lists wrap four to a row. A choice that cannot be used with the current picks is greyed out. Choosing an option that needs a pick starts that pick.
- **Buttons.** **OK** makes the feature and closes the dialog. **Apply +** makes it and keeps the dialog open for one more of the same kind, with the same settings (only the picks start empty). **Cancel** or **Esc** leaves everything as it was.
- **Errors** appear in red at the bottom of the dialog, in plain words ("Pick the cylindrical face to thread"). Sometimes the dialog offers a button with a way around the problem.
- **Editing.** Double-click a feature in the browser (or right-click ► **Edit**) to open its dialog again with its values. As in Inventor, while the dialog of an earlier feature is open the part is shown as of that feature, and only that feature is recomputed as you change values; the features after it are recomputed once, on **OK**. If one of them fails then, the status line says which.

6.2 Extrude **E**

3D Model ► Create ► Extrude. Pulls a closed sketch profile into a solid, or cuts with it.

Field	Choices and meaning
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Operation	Auto (the default, as in Fusion: where the extrusion goes mostly into the material it is a Cut, otherwise a Join — flip the direction and it switches), Join (add material), Cut (remove), Intersect (keep only what the extrusion and the part share), New Solid (a separate solid, chapter 7).
Extents	Distance; To Next (up to the next face of the part — if nothing is in the chosen direction, it goes toward the part); Through All; To (up to what you pick: a face of any Solid, flat or curved, a work or origin plane, or a point — a corner, a sketch or work point: it stops on the plane through it parallel to the sketch — an edge or a work axis, through its middle — and it follows when that moves).
Distance	The length (10 mm).
Direction	→ the sketch's front side, ← the other side (the same for Join and Cut), ⇔ symmetric (half each way), ⇔ 2 two sides, the second with its own Extents 2 : Distance (Distance 2 , 5 mm), To Next, Through All or To (To 2).
Taper	Draft angle of the sides (0°).

Drag the **arrow** at the end of the extrusion to change the Distance (6.1); with ⇔ it moves both ends. A Join that does not touch the part becomes a solid of its own; the status line says so. Join and Cut can be switched later in the dialog, unless other features are built on this Extrude.

6.3 Revolve R

3D Model ▶ Create ▶ Revolve. Turns a sketch profile around an axis.

Field	Choices and meaning
Operation	Auto (the default: mostly into the material is a Cut, otherwise a Join), Join or Cut .
Axis	Sketch vertical or Sketch horizontal — the sketch's own axes. Construction / projected line — a construction line of the sketch or a projected straight edge, chosen in the Line list. If the sketch has exactly one such line, it is chosen for you. To revolve around an edge of the part, project that edge into the sketch (an edge perpendicular to the sketch projects to a point and is not offered). Line, axis or edge — Pick a sketch line, an origin or work axis, or a straight edge in the view. Two points — click two Points on the part: corners, edge middles, circle centres. They are projected onto the profile's plane, so the axis always lies in that plane.
Extents	Full (360°), Angle (with Flip for the other way), Two angles (Angle and Angle 2 , one each way). Angles start at 90°.

With Angle or Two angles, an **arc** runs round the axis from the profile to where the revolve ends: drag its knob to change the angle (5° steps, Ctrl for any value). For a projected line or two points, ProstoCAD keeps a hidden **Revolve axis** under the feature; it is deleted with the feature.

6.4 Hole H

3D Model ▶ Create ▶ Hole. Drilled, counterbored, countersunk and threaded holes.

Field	Choices and meaning
Placement	From Sketch : a hole at every point and circle centre of the picked Sketch .

Concentric: click round edges or round faces (**Round**; several at once — a hole at the centre of each); the holes start on the flat face beside them, or on the **Face** you pick.

Hole	Simple (a Diameter , 5 mm), Clearance (for a screw: Standard, Size and Fit — Close, Normal, Loose), Tapped (a modeled thread).
Standard	Clearance and Tapped: ISO metric, ISO metric fine, UNC, UNF, UNEF, NPT, G / BSP, BSW, BSF.
Size	The thread size, or Auto (from Diameter) .
Seat	None (drilled) , Counterbore (C'bore Ø 10 mm, C'bore depth 3 mm), Countersink (C'sink Ø 10 mm, C'sink angle 90°), Spotface (Spotface Ø 10 mm, Spotface depth 0.5 mm).
Termination	Distance (a Depth , 10 mm), Through All (the default), To (up to To face , a face or work plane).
Drill point	For Distance and To: Flat or Angle (Point angle 118°) .
Thread	Tapped: Whole hole or Depth (Thread depth, 8 mm) .
Print clearance (on diameter)	Tapped: how much larger the thread is made so that a printed part fits (0.2 mm, measured on the diameter).
Direction	Tapped: Right hand or Left hand .
Flip direction	Drill the other way.
Chamfer	None , Start , Both ends , with a Chamfer size (0.5 mm): a 45° lead-in at the mouth and, for a through hole, at the exit. A tapped hole is chamfered from the thread's diameter, so a screw starts easily. With a seat only the exit can be chamfered (Start is greyed out).

Other threads in a hole: for Trapezoidal, ACME, print-friendly or custom threads, drill a Simple hole first, then use **Thread** (6.10) on its face.

6.5 Fillet and Chamfer

3D Model ▶ Modify ▶ Fillet / Chamfer (a split button). Select the edges, then the command — or pick the edges in the dialog. Also on the marking menu of an edge (E Fillet, W Chamfer).

Feature	Fields
Fillet	Edges , Radius (1 mm). Full (beside the radius arrows): a full round between the two picked parallel edges instead — the face between them becomes a half cylinder (the radius is greyed); off again, an ordinary fillet.
Full Round Fillet	Pick the two parallel Edges of a flat face (the end of a bar, a plate's side): the face becomes a half cylinder that runs on into both sides — two fillets meeting on it.
Chamfer	Edges ; Type: Distance, Two distances (Distance, Distance 2) or Distance + angle (Distance, Angle 45°); Flip swaps the sides for the last two types. Distances start at 1 mm.

The edges must be on the current shape of the part. A click on an edge takes its whole smooth chain (a slot's lines and arcs, a ring in pieces); **Shift**-click takes the whole **loop** round the face you are looking at (a top face's outline, a hole's rim); a click on a **face** takes all its edges, and a second click on it takes them off. The picked edges are listed under the field (**Edge 12**): × takes one off, **Clear** all; a long list scrolls. While you pick, the part shows as it is and the result is drawn over it as orange outlines.

6.6 Shell

3D Model ▸ Modify ▸ Shell hollows the part.

Field	Meaning
Remove faces	The faces to open (pick them).
Thickness	The wall (1 mm).
Direction	Inside (the outer size stays) or Outside (the wall grows outward).

FreeCAD cannot shell a part that already has a modeled thread. The dialog then offers to make the Shell **before** the thread; you can also drag End of Part above the thread, make the Shell there and move End of Part back.

6.7 Draft

3D Model ▸ Modify ▸ Draft (with Shell) tilts faces for moulding: pick the **Faces**, the **Neutral plane** (the faces keep their size there), the **Angle** (1°); **Reverse** tilts the other way.

6.8 Move Face

3D Model ▸ Modify ▸ Move Face. Direct editing: pushes or pulls flat faces of the part, the way Inventor does, without going back to the sketch. Select the faces first, then the command (also on the marking menu of a face and of the part) — or pick them in the dialog. Everything is a step at the end of the part's history, a row **Move Face** in the browser: double-click it to change the values, Suppress or Delete it like any feature. One **OK** is one undo step; **Finish move** on top of the right-click menu of the view does the same as OK. A round wall the move makes longer stays one face, so a chamfer or fillet on its rim can be as large as the wall allows.

Field	Choices and meaning
Faces	One or more flat faces. Several faces must face the same way (a boss top and the floor of a pocket, for example); they all move together. Faces looking in different directions are refused: <i>The faces must be flat and face the same way.</i> A round wall (a cylinder, an extruded ellipse or spline) moves outward or inward by a Distance (To is greyed out for it); one click takes the whole wall.
Extent	Distance (a number) or To (up to a face or plane).
Distance	The move along the face's outward direction: positive grows the part, negative cuts into it, 0 changes nothing. Units, parameters and formulas work as everywhere (3.6). It starts at 0 mm: type a value or drag the arrow (whole millimetres; Ctrl for any value).
To	Pick a flat face or a work / origin plane parallel to the faces, an edge that runs parallel to them (all of it at one height: a level line, an arc or a circle), or a point (a corner or a work point); the faces move to its height. The target may be on another Solid. The distance is the gap, found again on every change, so the faces follow the target if it moves. A tilted target is refused: <i>Pick a face, plane, edge or point parallel to the faces.</i>
Neighbours	What the faces next to the moved face do: Inventor (as it opens), Step or Together — see below.

The arrow. While the dialog is open an arrow stands on the first face, along its direction (drawn like the assembly gizmo's arrow). Drag it with the mouse: the face moves, the preview follows, the **Distance** field changes in steps of 0.1 mm and a label at the arrow tip shows the value ("12.5 mm", "-3 mm"). The arrow lights up and the cursor becomes a hand when you are on it. Type a distance and the arrow and the label follow. Dragging replaces a formula or parameter in **Distance** with the dragged value in mm. With Extent **To** the arrow only shows where the face goes; it cannot be dragged.

The three ways the neighbours behave. Take a block with a chamfer on one top edge and pull the top face up:

Neighbours	What happens	When it refuses
Inventor	The neighbours keep their angle and grow or shrink: the chamfer gets taller and the top face narrower. A hole through the face stays a hole, a boss wall slides along.	A neighbour that is round across the move (a fillet, a cone): <i>A neighbouring face is round: Inventor mode can't follow it yet — try Step or Together.</i>
Step	Only the face moves, as a straight slab. A slanted or round neighbour stays as it was and a straight wall appears at its edge.	Never because of a round or slanted neighbour; only "too large" (below).
Together	The face and the chamfers or fillets next to it move as one piece, keeping their size; the walls around them grow.	The walls around the chamfer or fillet are not along the move (slanted or round themselves): <i>The walls around the chamfer / fillet must run along the move — try Inventor or Step.</i>

Other messages: *The move is too large: a neighbouring face would vanish* (for example a face pushed in further than the chamfer next to it is wide; nothing changes), and *A face to move is no longer there* (an earlier feature removed the face; the Move Face row turns red); a face picked on another Solid is refused: *The faces must be on this Solid*. ProstoCAD finds the faces again by their geometry, not by their names: in their plane or — if an earlier change moved them — in the nearest parallel plane where they overlap, so Move Face keeps working when an earlier feature is edited, for example when the block is made taller. The To target is kept by its name, like other features' references.

6.9 Sweep, Loft and Coil

On the **3D Model ▶ Create** split button after Hole. Each has an **Operation**: Join or Cut.

Feature	Fields
Sweep	Profile (a closed sketch) along Path (another sketch or a 3D Line, 6.18: lines, arcs, splines). Wall (Join only) makes a tube: Thickness (1 mm) and Ends — Both open, Start closed or End closed.
Loft	Sections : two or more sketches, in order. The list shows them numbered; ▲ ▼ move a section earlier or later, × removes it. Ruled makes straight faces between sections; Closed joins the last section back to the first.
Coil	A spring: Profile (a sketch), Axis (an axis, a sketch line or a straight edge), Pitch (5 mm), Height (20 mm), Direction Right hand or Left hand.

The path of a Sweep is usually an open sketch; open sketches stay visible after Finish Sketch, so you can pick them.

6.10 Thread

3D Model ▶ Modify ▶ Thread. A real, helical thread — geometry you can print. Select the outside of a shaft or the inside of a hole (a full round face), then click Thread. Also on the marking menu of a face.

Field	Choices and meaning
Face	The round face to thread — or several (holes or shafts): a thread on each, with the same settings; afterwards each is a thread of its own.
Start face	Optional: a flat face at the end where the thread starts. Without it the thread starts at the free end (the tip of a shaft, the mouth of a hole); on a through hole or a loose rod, at the chamfered or rounded end.
Offset	Distance from the start (0 mm).
Length	Full length or Distance (10 mm).
Flip direction	Start from the other end.
Standard	ISO metric, ISO metric fine, UNC, UNF, G / BSP, NPT (tapered), Trapezoidal, ACME, Print-friendly (flat crests, sizes from M3 up) or Custom (Pitch 1.5 mm, Flank angle 60°, Depth 0.9 mm).
Size	A size of the standard, or Auto (from the face) : the size that matches the face (a Ø3

hole gets M3, a Ø4 hole M4; a tap-drill hole gets its thread: Ø5.2 → M6, Ø8.5 → M10).

Hand	Right hand or Left hand .
Chamfer	None , Start or Both ends , with Chamfer size (1 mm).
Print clearance (on diameter)	How much room the thread leaves so printed parts fit (0.2 mm on the diameter; 0.3 mm for Print-friendly, set when you choose it). 0 gives the nominal thread.

Sizes that do not match the hole. Any sensible size works in a hole: in a Ø3 hole, M3 fills the hole back to the M3 tap size first, M4 bores it out to the M4 tap size first. These helper steps sit right before the thread in the history; the fill goes only where the hole has a wall all round. A size far off the hole is refused with a message.

On a rod the rod is first turned down to the thread's crests (a **ThreadTrim** step under the Thread), so no thin fin is left on the crests.

Suppress, Delete and Edit on a thread act on the thread together with its helper steps.

6.11 Threads Off / On

3D Model ► Modify ► Threads. Modeled threads make redraws slow. Click **Threads** to suppress every thread while you work, and again to bring them back. A thread you suppressed yourself stays off. It also turns off the modeled thread of tapped holes (the holes stay). A suppressed tapped hole keeps its modeled thread off until it is unsuppressed, so it costs no time while it is off.

6.12 Boolean

3D Model ► Modify ► Boolean (Inventor's Combine) joins, cuts or intersects solids of the part. Select the base solid, then the tool solid(s), then click Boolean — or pick them in the dialog: click anywhere on a solid to pick it. As in Inventor, the first click is the base (the solid that stays), the next ones are the tools.

Field	Choices and meaning
Base	The solid that is changed.
Tools	One or more other solids. A click on the base while picking tools is ignored.
Operation	Join , Cut (the default) or Intersect .
Clearance (per side)	Cut only: a gap left on each side (0.2 mm), for printed parts that must fit. A tool that can't be grown or shrunk (a thread, an Emboss) is cut with 0 mm: the field shows 0 mm and the status line says why.
Clearance on	Cut only: Cavity (the hole is made larger), Tool (the hole is exact; a shrunk copy "<Tool> fit" is made for printing), Split (half on each).
Keep tool	Off (as it opens): the tool is used up and shows under the Boolean in the browser. On: the tool stays a separate, visible solid — print the pin and the box.

The tools stay editable: change a tool and the result follows. Threads and other complex shapes may refuse a clearance; the dialog then says so.

6.13 Split

3D Model ▶ Modify ▶ Split. The lines of a sketch (lines, arcs, splines) become thin cutting surfaces through the whole part.

Field	Choices and meaning
Sketch	The sketch whose lines cut the part (select it before the command, or pick it).
Mode	Keep all: every piece stays; this solid keeps the largest piece and every other piece becomes a solid of its own — hide it, work on it, make it a part (Make Assembly). The pieces keep following the sketch. Keep picked: only the pieces you click (Keep) stay. Gap leaves the pieces that much apart (its arrows step 0.05 mm; a parameter in the Parameters list); Gap from says which piece gives the material: Both (half each), This Solid (the piece that stays in this Solid) or New Solids (the others). With Keep picked the gap comes half from each side.

Deleting a Split joins its pieces again; ProstoCAD asks first and names the solids that go.

6.14 Patterns and Mirror

3D Model ▶ Pattern: Rectangular, Circular and Mirror (one split button), and **Mirror Solid**. They repeat features: pick them in the browser or click one of their faces. Picking a fillet, a chamfer or a thread stands for the feature it modifies.

Feature	Fields
Rectangular	Features; Include dependent features (also repeat what is built on them — the hole in a boss); Distribution: By spacing (Direction 1 — a straight edge or an axis —, Count 2 , Spacing 10 mm) or Equal spacing (Count copies spread between From and To : a flat face, an edge or a work plane); Flip direction; Two directions (Direction 2, Count 2, Spacing 2).
Circular	Features, Include dependent features, Axis: Axis or edge (an origin or work axis, a sketch line, a straight or circular edge, a round face — the origin's X, Y and Z show while you pick) or Two points (click two points on the part: a hidden work axis through them, drawn as an orange dashed line while the dialog is open), Count (4) ; Distribution: By angle (Angle , 360°) or Equal spacing (the copies spread round the axis between From and To — a corner, an edge, a face or a work point on the circle: the first copy as far from From as the last is from To).
Excluded	Rectangular and Circular: click the field, then click the copies you do not need on the model; they are left out at once. × in the list puts one back, Clear all. The original always stays. The copies left out are remembered with the pattern.
Mirror	Features, Include dependent features, Mirror plane (a flat face or a plane). Right-click its row ▶ Break Mirror Link : the mirrored copies keep their shape and no longer follow the features they copy (the features before it still change the part).
Mirror Solid	A mirrored copy of a whole Solid as a new Solid, as Mirror works in a sketch: select the Solid (a part with one Solid takes it; with several, click the one), click Mirror Solid , then click the Mirror plane (an origin or work plane, or a flat face). The copy follows the original — every change, every feature added to it later — and takes features of its own. Right-click its row or its Solid ▶ Break Mirror Link : the copy keeps its shape and stops following; both Solids stay. Deleting the original keeps the copy as it is; Make Assembly gives it its own part file. Result: New Solid (the copy above), Join (the mirror image

joined into the same Solid: a half made whole) or **Cut** (the image cut out of it) — a step at the end of that Solid that mirrors everything before it; the Solid must stay one piece.

Mirror: Whole, or only the part on **One side** / the **Other side** of the plane (a part crossing the plane: with Join it becomes symmetric, the other side's own material replaced by the mirror image). A Solid almost symmetric about the plane gives a copy right on top of itself: pick another plane. A Boolean may use the copy as a tool on its original (Join, Cut, Intersect): the copy then follows the original up to the step before that Boolean.

6.15 Work features

3D Model ▶ Work Features: Plane, Axis, Point (one split button). Choose the **Method**, then pick what it asks for. Work features follow the geometry they were made from.

Work feature	Methods
Work Plane	Offset from plane (Offset 10 mm, negative for the other side), Parallel through point, Midplane (between two planes), Angle around edge (Angle 90°), Three points (click the marked points, as in Section: corners, edge middles, circle centres), Normal to edge at point, Tangent to surface at point. With Offset, Planes makes that many planes at once, 2×, 3×... the offset from the same base (for a loft: a new offset spaces them all, each can still be moved on its own); Turn turns the plane about its own point — X, Y, Z about the previous plane's axes (Prev) or the global ones (Global), typed (the wheel steps them) or dragged on the red, green and blue rings by their knobs; turning a plane leaves the next one as it was. Each plane of the row opens its own dialog (double-click it); while it is open, a click on another plane of the row moves the dialog and its rings there. Point fields also offer a 3D Line's ends and the points it passes through, and work points; Angle around edge and Normal to edge at point take an axis too.
Work Axis	On edge, Two points, Intersection of two planes, Normal to plane through point, Axis of a cylinder, Parallel to edge through point.
Work Point	At a point (a marked point: corner, edge middle, circle centre), Center of circle, Midpoint of edge, Edge meets plane, Two edges meet.

A "plane" pick takes a flat face or a work or origin plane. Double-click a work plane to open its dialog; right-click ▶ New Sketch starts a sketch on it.

6.16 Emboss

3D Model ▶ Create ▶ Emboss (Inventor's Emboss / Engrave) raises a Text or a sketch from a face or cuts it in — any closed profile: letters, a star, a logo. On a **cylinder or a cone** it is wrapped round the face — the letters keep their size and run round it, their height along the radius; on a **flat face** it goes straight; on a **sphere** it is projected along the sketch's normal onto the side the sketch faces, its height along the radius.

1. Put the Text (or sketch) on a plane parallel to the round face's axis: the origin plane through the axis, or a plane touching the face. It lands on the side of the face that looks at that plane.
2. For a sphere: put the sketch on any plane, best one through the sphere's centre or in front of it; the profile lands on the half of the sphere that faces the sketch.
3. Press **Emboss**: the Text (or the selected sketch) is taken as the profile and the **Face** field waits

— click the round, spherical or flat face. A sketch left in another solid moves to the solid of the face you click, with the emboss.

Field	Choices and meaning
Operation	Emboss (raise) or Engrave (cut in). Engrave may cut right through a thin wall, like a stencil: the loose middles of O, A, D fall out.
Depth	The height or depth (1 mm), along the radius on a round face.
Face	A cylinder, a cone, a sphere or a flat face of the part (a flat face must be parallel to the profile).
Around	As drawn (the spacing of the Text) or Evenly round (the letters keep their widths and spread with equal gaps once round the whole face, the middle one facing the sketch). Greyed out on a sphere.

Emboss is a step in the history like Move Face: it finds its face again when an earlier step changes (a bigger cylinder). Letters past the face's edge run on round onto the faces that go on smoothly from it (the flat side and the rounded ends of a slot-shaped part), laid on them without stretching; where no face goes on smoothly they are refused: *Some of the profile is off the face*; on a sphere, so is a profile that reaches past the sphere's outline seen from the sketch.

6.17 Spur Gears and Worm Gear

3D Model ▶ Create ▶ Spur Gears (split button with Worm Gear) makes a pair of gears in mesh in one dialog, as Inventor's Design Accelerator — no sketch. Each gear is a Solid of its own; **Make Assembly** (the Assembly tab) then puts each into a part file of its own and builds the assembly, so any gear can be opened and saved anywhere. Involute teeth (the standard tooth: addendum 1 module, dedendum 1.25 modules).

1. Press **Spur Gears** and press **OK**: a pair 20 and 30 teeth, module 2 mm, ready. The line under the fields says what the numbers make: teeth, pitch diameters, ratio, centre distance.
2. **Both gears**: Module, Pressure angle (20° standard), Helix angle (0° spur; 5–45° helical — gear 2 gets the other hand by itself), Hand of gear 1. **Make**: Pair or One gear.
3. **Gear 1**: Teeth, Width, Bore Ø (8 mm; 0 = none). **Gear 2**: Size by **Teeth, Ratio** (1.5: half as many teeth again) or **Center dist.** (the nearest whole number of teeth), Width, Bore Ø.
4. **Place (optional)**: **Axis** (a work axis, a sketch line — a construction line too — or a straight edge), **Center** (a circle: a shaft's end, a hole, a sketch circle — its centre and plane; a corner or a work point) or **Plane**. Nothing picked: a Solid of its own at the part's origin — beside the gears already there.
5. **Mesh with** (Both gears) puts gear 1 in mesh with a gear already there — a third gear for a pair: click that gear. Module, pressure and helix angle then come from it; **Around it** turns gear 1 round it (degrees from its X), **Around gear 1** (Gear 2) does the same for gear 2. A worm can mesh with a gear already there the same way.

Moving it. Gear 1 carries the assembly's gizmo — while the dialog is open, and whenever you click a gear of the set in the part: drag an arrow to slide the pair along gear 1's X, Y or Z (whole mm; hold **Ctrl** for any value), an arc to turn it round its axis or tilt it round X or Y (5°). The value

shows at the pointer and in **Offset X / Y / Z, Turn, Rotate X / Y**, where you can also type it. Gear 2 always stays in mesh; a drag without the dialog is one undo step.

3D Model ▶ Create ▶ Worm Gear: the worm and its wheel in one dialog. **Worm**: Module (axial), Pressure angle, Hand, Starts (1–6), Pitch Ø, Length, Bore Ø; **Make**: Worm and wheel or Worm only. **Wheel**: Size by Teeth, Ratio (30: thirty turns of a one-start worm per turn of the wheel) or Center dist., Width, Bore Ø (10 mm). The worm's Bore Ø (8 mm) gives its axis to pick: for a joint, an Axis or a Center. The wheel lies across the worm at its middle, its teeth leaning as the worm's thread (the lead angle). The thread runs out one module before each end: the worm's ends are round.

Double-click either gear of a set to change the set. Deleting the first gear (or the worm) takes the second with it. Every number is a parameter (d0, d1 ...): rename it in the Parameters panel and use it in formulas.

6.18 3D Line

3D Model ▶ Sketch ▶ 3D Line (also on the Sketch tab). A line in space — a path for Sweep — drawn on a virtual cube of points round the origin, so every click knows its depth.

Field	Choices and meaning
Line	Lines (straight pieces), Spline (the points steer it, as control points) or Through points (a smooth line through every point). Curvature : the violet comb along the line — the longer the spikes, the tighter the bend; untick it to hide the comb.
Cube, Step	The cube's size (100 mm) and the distance between its points (10 mm), written grey along its edges. Nearer lines are thicker and brighter, far ones thin and dotted, so the cube reads in space as you turn the view.
Layer, At	XY, XZ or YZ : one layer of the cube is bright — its lines, points and a blue frame named «XY · 20 mm» — and only it takes clicks; the rest is a faint cage. At is its height (or depth); its arrows — or Tab / Shift Tab over the view — move it one step of the cube. Switching to another layer puts it through the last point. Every click lands on the bright layer, so the line stays flat until the layer moves between clicks (Tab , At, another layer, or Cube). Cube : the whole cube, as before.
Points	Click a point of the cube, or anywhere on one of its lines: there the point slides freely, and grey numbers give the distances to the cube's points either side. A dashed line runs from the last point to the pointer. Drag a point of the line (an open hand over it) to move it the same way. Undo removes the last point, Clear all of them.

OK keeps the line (orange) in the Solid, as a sketch; the cube shows only while the dialog is open. Sweep takes the line as its **Path** (6.9); Edit on it opens the dialog again.

6.19 Images to sketch over

The **Image** tab: a photo or a drawing behind the model, sized by a length you know.

Button	What it does
Insert Image	A JPG, PNG or BMP on the XY plane, centred on the origin, 100 mm across (its proportions kept); it gets a row of its own in the browser (with no part open, a

new part).

Scale / Transparency	Click two points of the picture whose real distance you know (a dashed line and its length in grey follow the pointer), type the Real length : the whole picture scales so it is, the first point staying where it is. A third click starts over. The Transparency slider (0–90 %) sets how see-through the picture is — alone it can be kept with OK too. A double-click on the picture's row opens it. Line ▶ Horizontal or Vertical turns the image about the first point until the line between the two points lies so (a photo taken at a slant set straight). A magnifying glass beside the pointer shows the spot while you put a point.
On XY / On XZ / On YZ	The picture onto that origin plane (seen from the top, the front, the right), about its middle.
Turn 90°, Mirror	A quarter turn in its plane; mirrored left to right.
Show / Hide	Every picture of the part at once.

A picture is only something to look at: sketches drawn over it do not depend on it. Its row's menu has Hide / Show, Rename and Delete.

7 Several solids in one part

A part can hold several separate solids — for a tool and the part it cuts, or a part you will print in pieces.

How separate solids are made

- Extrude with **Operation ▸ New Solid**.
- An Extrude ▸ Join that does not touch the part (a box drawn away from it): it becomes its own solid automatically.
- Split ▸ **Keep all**: every piece except the largest becomes a solid.
- Boolean with **Keep tool**, or a "<Tool> fit" copy (Clearance on Tool).

The browser then shows **Solid Bodies (n)** with a row for each solid. The history stays one list: features of other solids name their solid.

Sketching on another solid

A new solid sketched on a face of another solid holds only its own piece; the solid below stays as it is. The sketch follows that face only when it **projects edges** of the other solid; otherwise the face only gave it its plane.

Hide, show, rename, delete

Right-click a solid's row: **Hide / Show, Rename, Show in Window, Delete**. A hidden solid is dimmed in the browser. Delete asks when the solid is used by a Boolean (both go together) or when it is the main solid (the next one becomes the main solid). Selecting a piece in the view and pressing Backspace deletes that solid.

Make Assembly

When the pieces are ready, **Assembly ▸ Make Assembly** turns every separate piece into a part file of its own and links them in a new assembly where they were. See [9.12](#).

8 Inspect and tools

Measuring, mass, material, section views and options.

8.1 Measure

Inspect ► Measure (a split button). Measuring stays live: every change of the selection measures again, and the results show in a small panel. **Esc** finishes.

Command	Select	Shows
Measure	A face, an edge, a solid, or two items	Area; Length, or Radius and Diameter of a round edge; Volume, Surface and Size of a solid; for two items Distance (and Angle when both have a direction)
Distance	Two faces, edges or points	Distance; between two points also ΔX , ΔY , ΔZ
Angle	Two flat faces or straight edges	Angle
Length	Edges (a loop)	Total length and the number of edges
Area	Faces	Total area and the number of faces

8.2 Mass Properties

Inspect ► Measure ► Mass Properties shows the part's Volume, Surface area, Mass and Size. The mass uses the density of the part's material (8.3).

8.3 Material

Tools ► Material ► Material sets the part's material: its density for Mass Properties and its colour. The default is PLA.

Material	Density, g/cm ³	Material	Density, g/cm ³
PLA	1.24	Nylon	1.14
PETG	1.27	Resin	1.18
ABS	1.04	Aluminium	2.70
ASA	1.07	Steel	7.85
TPU	1.21		

Appearance

Tools ▶ Material ▶ Appearance (also in the right-click menu of a Solid, a face or the part) colours the picked faces or Solids — every Solid with nothing picked. Pick a colour from the palette or any colour (More...), a finish — Matte, Satin, Glossy, Metal, Chrome, Glass (see-through), Rubber —, a texture over the colour — Brushed, Wood, Carbon, Knurl, Sand, at its real size on any part — and **Hide Edges** to draw the Solid without its lines. The part shows each choice at once; OK keeps it (one Undo step), Cancel puts it back, Reset returns to the material's colour (picked faces: to the Solid's). A coloured face keeps its colour when a feature moves it, and an assembly shows its parts as coloured.

8.4 Section views

View ▶ Visual Style ▶ Section cuts the view open — the model itself is not changed.

- Select a flat face, a plane (XY, XZ, YZ from the browser too) or three points first; with nothing selected the section is on the XZ plane. Both faces of the split button (Section, Section by 3 Points) cut at the selection, and a face or plane selected while sectioned moves the cut there.
- A small **Section** panel moves the cut: **Offset** along the plane's normal, **Flip** to look from the other side, **Done** to keep the cut view, **Off** to close it. Clicking Section again also closes it.
- **Section by 3 Points** (the same split button): click three of the marked points — corners, edge middles, circle centres — for the plane.

In an assembly the section cuts every shown part the plane goes through: each cut face in its part's colour, with its own hatching angle, so neighbouring parts stand apart. It cuts anything shown, too: a STEP, shapes made in other programs, and assemblies made in FreeCAD itself (their parts as Bodies or shapes in the assembly), each part in its colour.

8.5 Options

- **Tools ▶ Options ▶ Ask on New**: on, **New** asks Part or Assembly and the units; off, it makes what you chose last time.
- The other settings are in the ⚙ menu (3.8).

9 Assemblies

An assembly links part files and holds them together with constraints and joints, as in Inventor. The parts stay separate .FCStd files.

9.1 Starting an assembly

- **File** ▶ **New Assembly**, or **Get Started** ▶ **New Assembly** with no file open.
- **New** (**Ctrl** **N**) makes an assembly too if you chose Assembly with **Remember my choice**.
Tools ▶ **Options** ▶ **Ask on New** brings the question back.

An assembly links to part files, so it must be saved before the first part is placed. ProstoCAD does this for you: when you place the first part, a new assembly is saved next to that part under its own name; if a file of that name is already there, ProstoCAD asks for a name and a folder. **Create** asks you to save the assembly first.

9.2 Placing and creating parts

1. **Assembly** ▶ **Component** ▶ **Place** (**P**) — or **N** in the marking menu.
2. Choose a saved part (or assembly) file.
3. The first part placed is **grounded** (fixed in place); the next ones are put beside it.

Create asks for a **New part file name** and makes a new, empty part file next to the assembly, placed in it. **Ground** grounds or ungrounds the selected part.

9.3 The assembly browser

In an assembly the browser shows the assembly, one row per part, and **Relationships (n)** with the constraints and joints.

- Each part row shows its **freedom: grounded, DOF 0 ... DOF 6**, or **DOF ?** when it could not be counted (hover for the reason). A suppressed part shows nothing.
- Under each part row is the part's own history (its sketches and features). A click on one of those rows selects the part; a double-click opens the part file.
- A hidden part is dimmed, a suppressed one struck through.
- Right-click a part (row or in the view) for **Edit**, **Hide / Show**, **Suppress**, **Remove from Assembly**, **Ground** and the constraints (4.2).

9.4 Adding a constraint or a joint

1. Click the constraint on **Assembly** ▶ **Constrain** or **Assembly** ▶ **Joint** — **C** is Mate, **J** is Rigid — or use the marking menu on a face or edge of a part.
2. Pick the **First** item on one part and the **Second** item on another, in the 3D view. The preview moves the part at once.
3. Set the **Offset** or **Angle** if the constraint has one, and **Flip** if the part faces the wrong way.
4. **OK** (or **Apply** + for the next one).

The part that moves is the free one; a grounded part never moves. A constraint that cannot be met together with the others is refused: "Mate conflicts with the other constraints (it can't be met)". Offsets and angles are parameters named after the constraint — **Mate1, Flush1, Angle1** ... — and appear in the Parameters panel.

9.5 Constraints and joints, one by one

Constraint	Pick (First, Second)	What it does	Value, Flip
Mate 	Two faces	Faces against each other, facing each other.	Offset (0 mm). Flip: the faces face the same way instead.
Flush	Two faces	Faces side by side, facing the same direction.	Offset. Flip: which way the faces face.
Angle	Flat faces or straight edges	The two at an angle.	Angle (90°).
Parallel	Flat faces or straight edges	The two stay parallel; the distance stays free.	—
Perpendicular	Flat faces or straight edges	The two stay at 90°.	—
Tangent	A round face and a flat or round face	The faces touch. A round face sits outside the flat face's part by default.	Offset. Flip: outside or inside.
Insert	Two round edges	Axes aligned, the faces of the edges together (a screw in a hole); the part can still turn.	Flip: direction.
Rigid 	A face, edge or point on each part	The two parts are fixed together.	—
Rotational	A face, edge or point on each part	A hinge: turns about one axis only.	Flip: direction.
Slider	A face, edge or point on each part	Slides along one direction only.	Flip: direction.
Cylindrical	A face, edge or point on each part	Turns about and slides along one axis.	Flip: direction.
Ball	A face, edge or point on each part	Turns freely about a point.	—

Names: Make Assembly names each part file after its Solid, as the browser shows it, and the assembly after the part with **_assy** (Train_assy); a new assembly saved on its first Place is named after that part + **_assy**.

Axes: every turned body and hole of the parts (a cylinder or cone going half way round or more) shows its axis as a light dash-dot centre line on a dark rim. Click an axis while Mate, Flush, Insert, Rotational, Slider, Cylindrical, Parallel, Perpendicular or Angle waits for a pick: the axis is taken (for Insert, the round edge of it nearer the click). Mate or Flush on two axes (or two round faces) puts them in line.

For joints, a round edge gives its centre and its axis — pick round edges for hinges and shafts.

9.6 The Relationships rows

Every constraint and joint has a row under **Relationships**. **Double-click** a row to edit it. Right-click it for **Edit**, **Delete**, **Suppress / Unsuppress** and **Rename**. A suppressed constraint is struck through; the solver ignores it and its parts stay where they are. Bringing back a constraint that can no longer be met is refused.

9.7 Degrees of freedom

A part's degrees of freedom are the ways it can still move: up to three slides and three turns. ProstoCAD counts them against what holds the part (the other free parts are held still while it counts):

- the part row says **grounded**, **DOF n** or **DOF ?** (9.3);
- **View ▶ Visual Style ▶ Degrees of Freedom** (View tab of the assembly) draws Inventor's DOF symbols on every free part — a double arrow for each direction it can slide, an arc for each axis it can turn about;
- with that button off, the part under the pointer shows its symbol as a tip.

A fully constrained part (DOF 0) and a grounded part show no symbol.

9.8 Moving parts

Drag a free part with the left button: it moves only the ways its constraints still let it — a Mated block slides and turns on its face, an inserted pin turns in its hole. The other parts stay where they are, even free ones. A fully constrained part does not move (the status line says "... is fully constrained"); a drag on a grounded part says it is grounded.

The gizmo. Click a free part: its DOF symbol becomes a gizmo with an **arrow** for each direction it can slide and an **arc** for each axis it can turn about.

- Grab an arrow to slide the part, an arc to turn it. Handles are easy to catch: the one under the pointer lights up.
- The first arrow is up and down — straight up, or up the slope of a tilted face — where the part can move that way.
- A click on empty space hides the gizmo.

Every drag that moved the part is **one undo step** ("Move part");   puts the part back. While a constraint dialog is open, dragging is off.

9.9 Editing a part in the assembly

Right-click a part ▶ **Edit** (or double-click its row): the part opens in its own tab, with the other parts of the assembly drawn around it — grey, semi-transparent, at their real places. They are for reference only: they cannot be picked and are not saved in the part. Change the part as usual and save it. **3D Model ▶ Return** takes you back to the assembly, which shows the change; the part's tab stays open until you close it.

View ▶ Assembly Context (View menu, or the View tab of the part) turns the other parts off and on.

Right-click a part ▶ **Open** (also Assembly ▶ Component ▶ Edit | Open) opens its file in its own window as a part of its own — no assembly around it and no Return, as Inventor's Open.

Saving an assembly: the first Save (or Save As) asks for a name and a place and makes a folder of that name there: the assembly and every part file it uses (a sub-assembly's too) are saved into it, and the assembly links to those copies. The files the parts came from are left as they were. Later Saves keep it all in that folder.

Hide / Show in the assembly

View ▶ Visual Style ▶ Hide / Show has **Show / Hide All Axes** (the light centre lines of the parts' turned bodies and holes; remembered — a constraint dialog still shows them while it waits for a pick), **Show / Hide All Origin** (the assembly's and its parts') and **Show / Hide All Sketches** (the parts' sketches: set in the parts' files).

Interference

Assembly ▶ Analyze ▶ Interference shows where parts go into each other: every overlap in bright red over the parts, and a panel with each pair and its volume. Faces that only touch (a shaft in its hole) are no interference. It is redrawn as parts move; click Interference again to turn it off.

9.10 Updating the assembly after a part changed

Back in the assembly's tab after a part changed, ProstoCAD asks: **Parts of ... changed. Update the assembly now?**

- **Yes** brings the changes in.
- **No** leaves the assembly as it was; the top row of the browser then says ⚡ **Needs update** until you click **Update** on the title strip.

9.11 Hide, Suppress, Remove

Command	What it does
Hide / Show	Hides or shows the selected part; the row is dimmed while hidden.
Suppress	The part is hidden and its constraints are off; again to bring it back.
Remove from Assembly	Takes the part out of the assembly; its part file stays.
Ground	Fixes the part in place, or frees it again.

9.12 Make Assembly from a part

Model a product as one part with several solids (chapter 7), then turn it into an assembly in one step.

1. Save the part — the new files go into its folder.
2. **Assembly** ▶ **Make Assembly**, or right-click in empty space ▶ **Make Assembly**.
3. The dialog lists the new files: the **Assembly** name and one name per piece (**Piece 1 (grounded)**, Piece 2 ...) with its volume. Hover over a row to see the piece marked in the view. Change the names if you like.
4. Click **OK**. The new assembly opens.
 - Every separate piece becomes a part file **with its history**; the assembly links them where they were, the largest piece grounded.
 - **Gears** (6.17) are not grounded: each stands on its own axis — a Rotational joint to *Gear Axes*, a hidden grounded copy of their bore circles. Select a gear and drag the gizmo's arc: it turns, and the gears in mesh with it turn by their ratio (a worm and its wheel too).
 - The original part is not changed.
 - Running Make Assembly again on the same part offers **Replace existing files** (with the names of the files already there), so the files of the earlier run are overwritten instead of asking for new names. The part itself is never replaced. If one of those files is open with unsaved changes, ProstoCAD asks first.
 - A part of one piece makes an assembly of one component (a single gear, say). To get several components, split it first (Split ▶ Keep all) or make pieces as New Solid. Gears of a set (6.17) each go into a part file of their own, still in mesh.

9.13 Boolean between parts

Assembly ▶ **Component** ▶ **Boolean** cuts one part by others, for example a pocket for a bearing:

Field	Meaning
Cut this	The part to cut.
With	The part(s) that cut it.
Clearance	A gap for printed parts (0.2 mm).
Clearance on	Cavity , Tool or Split , as for the part Boolean (6.12).

The pocket is written into the cut part's own file and follows the tools when they move or change. Save that part file too. The cut gets a row under Relationships (Edit, Delete, Rename).

9.14 Threads Off / On in an assembly

Assembly ► Component ► Threads turns every thread of every part off (faster redraws) and back on. It changes the part files: save them with the assembly.

9.15 Saving

Save saves the file in the active tab. A part edited from the assembly, cut by a Boolean or changed by Threads Off / On is a file of its own: save it as well (switch to its tab, or let **Quit** ask about every changed file).

10 Tips for 3D printing

ProstoCAD was made with printed parts in mind. A few habits save reprints.

- **Threads that fit.** Thread and Hole ▶ Tapped have a **Print clearance (on diameter)**, 0.2 mm by default (0.3 mm for a Print-friendly thread). Try it on a small test piece with your printer; raise it if a printed bolt is tight, lower it if it is loose. The **Print-friendly** thread standard has flat crests that print cleanly.
- **Lead-ins.** Use Hole ▶ **Chamfer** (Start or Both ends) and Thread ▶ **Chamfer** so screws and nuts start easily.
- **Parts that fit into each other.** Boolean ▶ Cut with **Clearance (per side)**: on the **Cavity** to print the part with the hole, on the **Tool** to get a slightly smaller "fit" copy of the tool to print, **Split** for half on each.
- **Print the tool as well.** Turn on **Keep tool**: the tool stays a visible solid of its own — the pin and the box with its hole.
- **Too big for the bed?** Draw the cut lines in a sketch, **Split ▶ Keep all**, then **Make Assembly**: each piece becomes a part file you can export and print, and the assembly shows them together.
- **Weight and filament.** Tools ▶ Material, then Inspect ▶ Mass Properties for the mass.
- **Fast work.** Turn threads off (Threads) while modeling; turn them on before exporting.

11 Troubleshooting and reporting a problem

When something does not work

1. **Read the status line** and the red text in the dialog: ProstoCAD says why a command was refused and what to pick.
2. A dialog is open? Commands like Save and Undo wait for **OK** or **Cancel**.
3. A feature in **red** in the browser: hover over it for the reason, then edit it (double-click).
4. The view or the browser looks out of step with the model: click **Update** on the title strip.
5. Undo (**Ctrl** **Z**) the last step.
6. Still stuck: save under a new name, quit and restart ProstoCAD.

Reporting a problem

Please send:

- what you did, step by step, and what you expected instead;
- the version (the title strip, *ProstoCAD 0.12.0*);
- your Windows version;
- the log `%LOCALAPPDATA%\ProstoCAD\ProstoCAD.log` —  **Show Log File** opens it in the File Explorer with the log selected (Windows search does not look into this hidden folder);
- if you can, the .FCStd file you were working on;
- a screenshot if something looks wrong.

If ProstoCAD crashed or froze, please say what you did just before; the log ends there.

Known limits of this build

- Windows 10 / 11 (x64): a first test build.
- A test build. Save often and work on copies.
- The interface is in English.
- FreeCAD cannot shell a part that already has a modeled thread (ProstoCAD offers to shell before the thread).
- Some offsets fail on threads and other complex shapes (an Emboss). A Boolean's clearance then falls back to 0 mm (the field and the status line say so); a fit copy is not made.
- Mirror Solid ▶ Join / Cut and Boolean need a result in one piece: a mirror image crossing the part in many places can leave separate pieces (the dialog says so — try another plane, One side / Other side, or New Solid). A part almost symmetric about the plane gives a copy right on top of itself.
- A mirrored Solid used as a tool by a Boolean on its own original follows the original only up to the step before that Boolean. Make Assembly keeps mirrored Solids as they are (no link between part files).
- Move Face moves flat faces along their own direction and straight round walls outward / inward. In Inventor mode it cannot follow a neighbour that is round across the move yet (use Step or Together); Together inward next to fillets or chamfers on a round wall is refused for now.

12 ProstoCAD: STL and AI

An STL is a mesh of triangles: a slicer prints it, but you cannot fillet it or change a size. ProstoCAD turns it back into a part — with tools you drive, or with an AI that drives ProstoCAD for you.

12.1 An STL in three steps

The **Tools ▶ STL** buttons go left to right; each step shows its result, and Undo takes it back.

Step	What it does
Import	Tools ▶ Files ▶ Import (or File ▶ Import) brings the STL in as it is, its triangles drawn. The status line says how many there are.
STL → STEP	Makes the mesh a solid of its triangles. Export can already write it as STEP for another CAD program, but its faces are still triangles.
Mesh Section	Pick a plane or a flat face: the STL is cut there and a new sketch holds the cut as lines, arcs and circles with round sizes, opened to edit. Revolve or Extrude it, or draw over it. The sketches go to a Solid of their own beside the STL.
Smooth	Rebuilds a turned part as features, the way it was drawn: a sketch and a Revolve, partial Revolves for what only part of the turn holds, square cuts as Extrude and Groove, a Thread with the measured pitch and depth. The result is a Solid of its own beside the mesh; the triangles are hidden.

Smooth works best on turned parts (about the Z axis). For other shapes, use Mesh Section and your own features, or the AI (12.2).

12.2 AI Link and Connect AI

An AI on your computer can look at the STL (sections, sizes, the surface at a point, a picture) and build the part with ProstoCAD's own features — sketches, Extrude (also as a cut), Revolve, Fillet, Chamfer, Thread — in your window, checking each step against the mesh. The features stay in the Model Browser like your own: edit them, delete them, or Undo each step.

1. Install an AI's command-line tool and sign in with your own account: **Claude Code** (claude.com/claude-code), **Codex** (OpenAI) or **Gemini CLI** (Google). With none installed, **Tools ▸ AI ▸ Connect AI** shows each one's page and the command that installs it, ready to copy into a command prompt (Codex and Gemini CLI need Node.js, nodejs.org); start it there once to sign in. Nothing is downloaded for ProstoCAD itself: its MCP server is built in.
2. In ProstoCAD, import the STL (or open the part).
3. **Tools ▸ AI ▸ Connect AI**. A command prompt opens in the part's folder (for a new STL, the folder it came from), ProstoCAD is connected to the AI and the AI starts there. It starts Claude Code when it is installed (or the Codex / Gemini CLI used last), at once — no questions; with none installed it shows how to get one.
4. Tell the AI what you want, in your words: *"Rebuild the STL in ProstoCAD as a part, sharp edges first."*

AI Link (on by default) lets the AI in. It listens only on this computer and only to clients that know the key ProstoCAD keeps in its settings folder. Turn it off and no AI can reach ProstoCAD.

Ask for the plain shape first — cylinders, boxes, holes, sharp edges — and add fillets and chamfers after: it is easier to check and to fix. How well the part comes out depends on the model: ProstoCAD was tried with Claude; small local models lose track of sizes and planes.

12.3 Other AI programs

AI Link speaks **MCP**, the open protocol many AI programs use, so any MCP client on this computer can work in ProstoCAD: Claude Desktop, Cursor, VS Code (Copilot's agent mode), Windsurf, Cline, or LM Studio with a local model. Connect AI sets up the three above; for the others, add a local (stdio) MCP server named `innCAD` in the program's MCP settings:

Field	Value
Command	<ProstoCAD folder>\runtime\bin\python.exe
Argument	<ProstoCAD folder>\runtime\Mod\InCad\incad\ai_bridge.py

ProstoCAD must be open with AI Link on; the program finds ProstoCAD's tools by itself.

Connect AI, with no command-line AI installed, shows these two of this very installation, each with Copy, and the entry for a program that keeps its MCP servers in a settings file (Claude Desktop: Settings ▸ Developer ▸ Edit Config; Cursor, Windsurf, LM Studio's mcp.json; VS Code names the list `servers`):

```
{"mcpServers": {"innCAD": {"command": "<Command>", "args": ["<Argument>"]}}}
```

Restart the program after changing its settings.

Experimental. ProstoCAD is a test edition: work on copies, look at what the AI built before you trust a size, and send the log (chapter 11) when something goes wrong.

A Keyboard shortcuts

Single keys work in the mode named.

Key	Command
Part	
S	Start 2D Sketch
E	Extrude
R	Revolve
H	Hole
F	Fillet
Sketch	
L	Line (M : line / arc while drawing)
C	Circle
A	3-Point Arc
R	Rectangle
X	Trim
D	Dimension
Tab	Next value field while drawing
F7	Slice Graphics
F8 / F9	Show / Hide Constraints
Assembly	
P	Place
C	Mate
J	Rigid

Key	Command
Everywhere	
Ctrl N	New
Ctrl O	Open
Ctrl S	Save
Shift Ctrl S	Save As
Ctrl W	Close (asks to save)
Ctrl Q	Quit (asks to save)
Ctrl Z	Undo
Shift Ctrl Z	Redo
Backspace / Del	Delete the selection
Esc	Cancel the dialog, stop the tool, finish the sketch
In dialogs and value boxes	
Enter	Apply the value (value box)
↑ / ↓	Dialog value one step up / down
Shift + ↑ / ↓	Ten steps

B Ribbon map

Every tab, group and command. "A | B | C" is one split button: A is shown, B and C are on its arrow.

Part

Tab	Group	Commands
3D Model	Sketch	Start 2D Sketch · 3D Line
	Create	Extrude · Revolve · Hole · Sweep Loft Coil · Emboss
	Modify	Fillet Full Round Fillet Chamfer · Shell Draft · Thread · Threads (Threads Off / On) · Boolean · Split · Move Face
	Pattern	Rectangular Circular Mirror · Mirror Solid
	Work Features	Plane Axis Point
Assembly	Assembly	New Assembly · Make Assembly · Open
Inspect	Measure	Measure Distance Angle Length Area · Mass Properties
Tools	Files	Import · Export
	STL	STL → STEP · Mesh Section · Smooth
	AI	AI Link · Connect AI
	Material	Material · Appearance
	Options	Ask on New
Image	Insert	Insert Image
	Scale	Scale / Transparency
	Place	On XY On XZ On YZ · Turn 90° · Mirror
	Look	Show / Hide
View	Appearance	Light / Dark
	Navigate	Zoom All · Home View · Look At · Zoom Selection · Orbit · Pan
	Visual Style	Shaded · Wireframe · Section Section by 3 Points · Hide / Show Show All Origin Hide All Origin Show All Sketches Hide All Sketches · Assembly Context
	Capture	Screenshot
	Windows	Parameters

Sketch

Tab	Group	Commands
Sketch	Create	Line Single Line · Spline Control-Vertex Spline · Circle 3-Point Circle Ellipse

3-Point Ellipse · 3-Point Arc | Center Arc | Elliptical Arc · Rectangle | Center Rectangle | 3-Point Rectangle | 3-Point Center Rect. | Rounded Rectangle · Center-to-Center Slot | Overall Slot | Center Point Slot | Arc Slot | Flat-End Arc Slot · Hexagon | Triangle | Square | Pentagon | Octagon | Polygon (N sides) · Point · Text

	Modify	Fillet Chamfer · Trim · Extend · Offset
	Pattern	Rectangular Circular Along Path · Mirror
	Dimension	Dimension Overall Dimension
	Constrain	Coincident · Concentric · Horizontal · Vertical · Parallel · Collinear · Perpendicular · Tangent · Equal · Symmetric · Fix · List (Constraint List)
	Insert	Project (Project Geometry)
	Format	Construction · Constraints (Show / Hide Constraints) · Free DOF (Show Free Geometry) · Slice (Slice Graphics)
	Exit	Finish Sketch
View	Appearance	Light / Dark
	Navigate	Zoom All · Home View · Orbit · Pan
	Visual Style	Shaded · Wireframe · Section Section by 3 Points
	Capture	Screenshot
	Windows	Parameters

Assembly

Tab	Group	Commands
Assembly	Component	Place · Create · Import · Ground · Edit Open · Boolean · Threads (Threads Off / On)
	Constrain	Mate · Flush · Angle · Parallel · Perpendicular · Tangent · Insert
	Joint	Rigid · Rotational · Slider · Cylindrical · Ball
	Analyze	Interference
View	Appearance	Light / Dark
	Navigate	Zoom All · Home View · Look At · Zoom Selection · Orbit · Pan
	Visual Style	Shaded · Wireframe · Section Section by 3 Points · Hide / Show Show / Hide All Axes, Origin, Sketches · Degrees of Freedom
	Capture	Screenshot
	Windows	Parameters

No file open

Tab	Group	Commands
Get Started	New	New · New Assembly · Open
View	Appearance	Light / Dark

Commands only in the marking menus: Show in Browser, Show in Window, Move End of Part Here, Suppress / Unsuppress, Rename, Delete, Edit, Stop Tool, Edit Pattern, Delete Pattern, Suppress (part), Remove from Assembly.