



stlToSolid — User Manual

Mesh in • solid out
Version 0.5.7 • October 2026

Contents

#	Section	What it covers
1	Introduction	What stlToSolid is for and which tool to pick for your part.
2	Getting started	The screen layout, loading a file, units and scale, picking bodies and moving around the 3D view.
3	Mesh → Solid	Turns a machined or CAD-designed mesh into a clean STEP solid with real flat faces, cylinders and holes. Also gives you editable CadQuery and Fusion 360 scripts.
4	Sliced Loft	Slices a smooth, organic shape (a cap, handle or shell) along one axis and lofts the outlines into one smooth solid.
5	X-Ray	Turns slices of the mesh into real Fusion 360 sketches (lines, arcs and splines, not thousands of points) that you can select and extrude. Downloads a whole stack as one script, and can extrude them into solid slabs.
6	Blueprint	Reads a dimensioned drawing (a picture) into named parameters and sketch-and-extrude features. Builds a parametric Fusion 360 script and a STEP.
7	Fusion 360 add-in: Mesh to Outline	The X-Ray and single-slice tracing inside Fusion itself: pick a mesh body and planes, get sketches (and optionally a solid) straight in your design. Also covers the STL to Solid add-in preview.
8	Reading the report	What the result badge, the quality checks and the output table mean.
9	Using the results in Fusion 360	How to import the STEP and run the downloaded scripts.
10	Troubleshooting	Common problems and how to fix them.
11	Quick reference	File types, default settings and output files at a glance.

1. Introduction

A mesh file (STL, OBJ, 3MF and so on) describes a part as thousands of tiny triangles. That is fine for a 3D printer, but a CAD program cannot work with it: you cannot sketch on a triangle, measure a hole or fillet an edge. Fusion 360's free tier can turn a mesh into a solid, but every triangle becomes its own face, so the "solid" is just as hard to model against.

stlToSolid rebuilds the part the way a designer would have drawn it: real flat faces, real cylinders and cones, real holes. You get a STEP file you can import into Fusion 360 (or any CAD program) and design around.

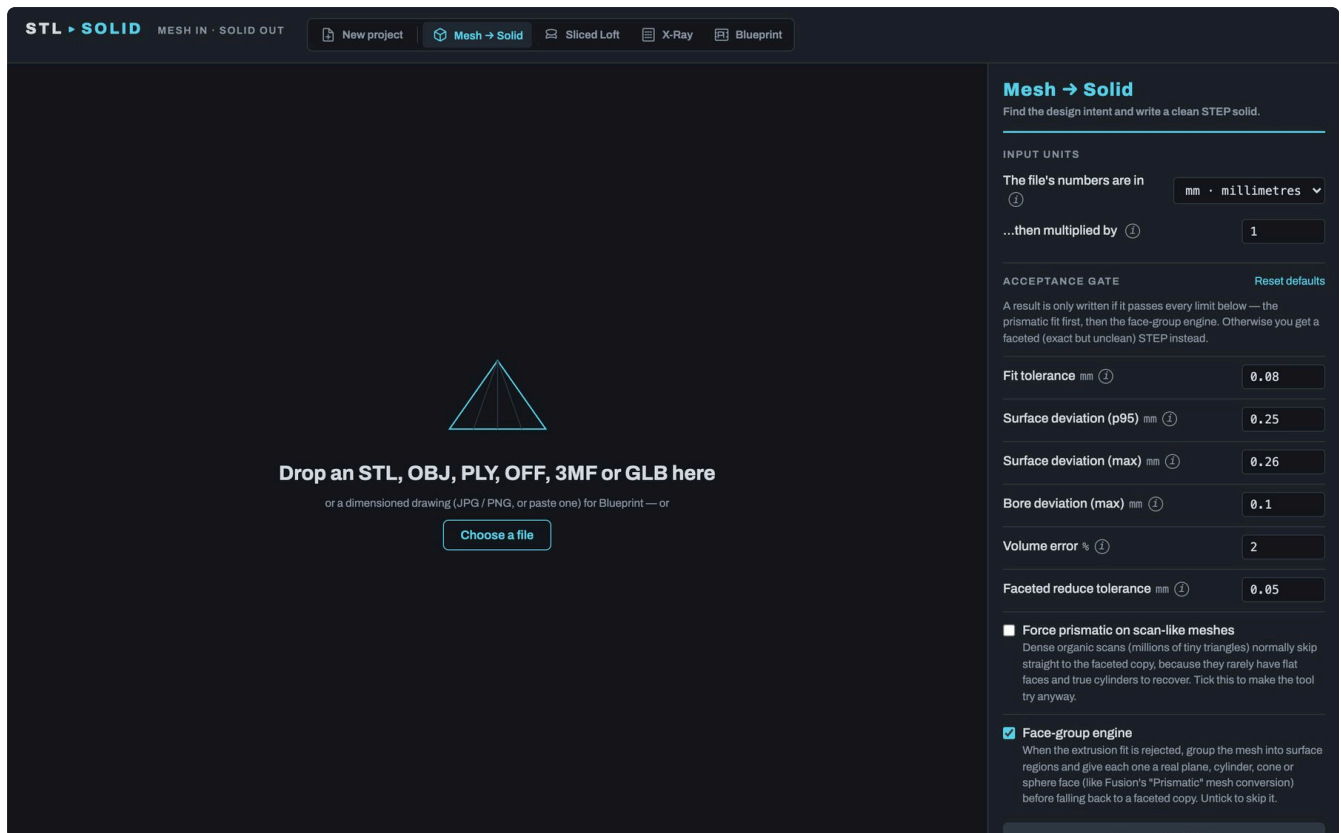
stlToSolid comes as a desktop app for Mac and Windows, as a Docker container for a home server or NAS (opened in a browser), and as add-ins that run inside Fusion 360. The desktop app and the browser version are the same program with the same four tools. This manual covers all of them.

Which tool should I use?

Your part looks like...	Use	You get
A machined or CAD-designed part: brackets, plates, housings, holes, slots and steps	Mesh → Solid	A clean STEP with true planes, cylinders, cones, spheres and tori
A smooth, flowing shape that changes gradually along one direction: caps, handles, shells, bottles	Sliced Loft	One smooth lofted solid, plus a Fusion script that repeats the loft
A part you want to rebuild yourself in Fusion from cross-sections	X-Ray	Real, selectable Fusion sketches, one per slice, ready to extrude (optionally extruded into slabs for you)
No mesh at all, just a dimensioned drawing or datasheet picture	Blueprint	A parametric Fusion script (User Parameters, sketches, extrudes) and a STEP
A mesh that is already inserted in a Fusion design	Mesh to Outline (Fusion add-in)	The same X-Ray sketches, drawn straight into your design, with no file to download

2. Getting started

2.1 The screen



The start screen, before a file is loaded.

The window has three areas:

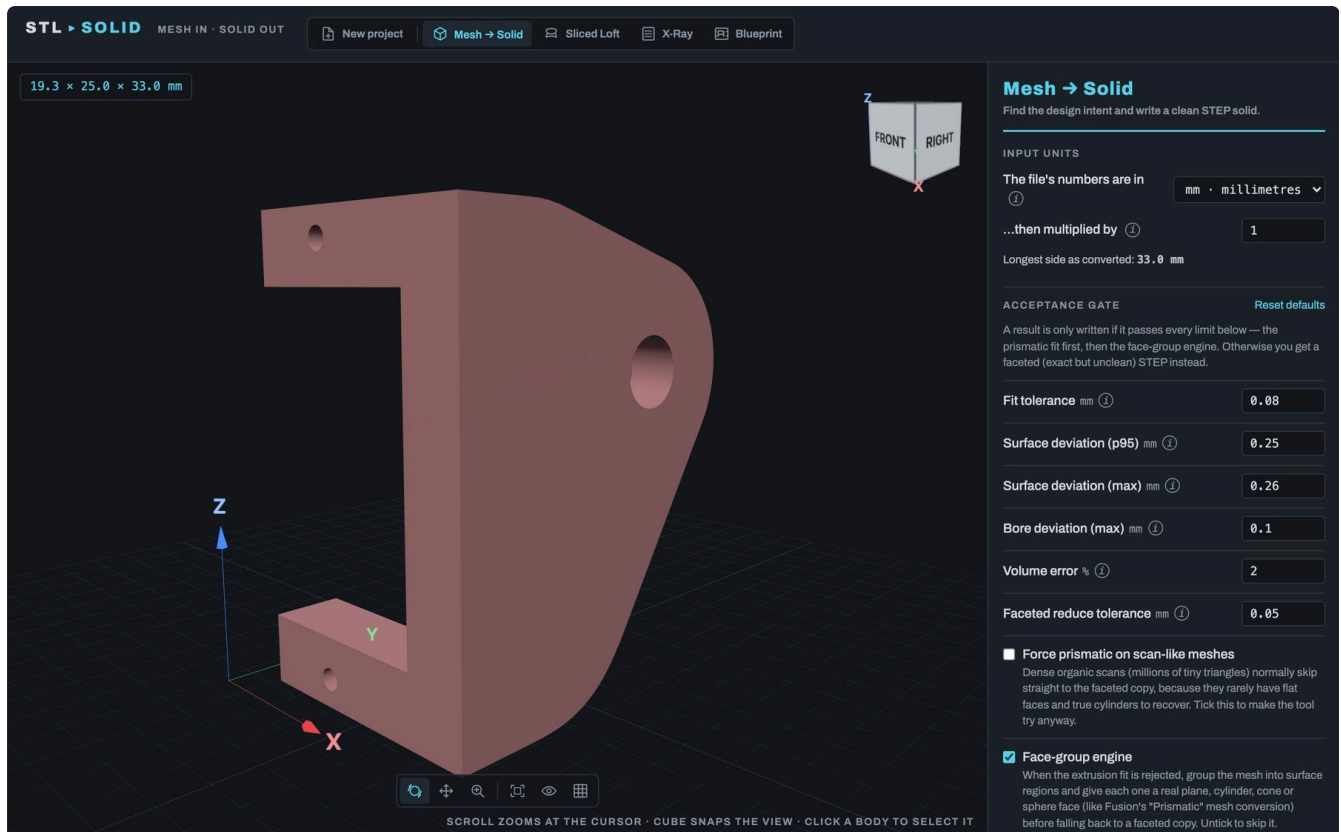
- **Top bar.** The **New project** button (start over with another file) and the four tool buttons: **Mesh** → **Solid**, **Sliced Loft**, **X-Ray** and **Blueprint**. The highlighted button is the active tool. Hover over a button for a one-line description.
- **Stage (left).** Drop a file here. Once loaded, it shows the part in 3D (or, for Blueprint, the drawing).
- **Rail (right).** The active tool's settings, its action button and, after a run, the report with the download buttons.

You can switch tools at any time. The loaded file stays, and each tool keeps its own settings.

2.2 Loading a file

Drag a file onto the stage, or click **Choose a file**. You can also paste a drawing image from the clipboard with **Cmd+V** / **Ctrl+V**.

- **Meshes:** `.stl`, `.obj`, `.ply`, `.off`, `.3mf`, `.glb`, `.gltf`. A mesh opens in the mesh tools. If Blueprint was active, the app switches to Mesh → Solid.
- **Drawings:** `.jpg`, `.png`, `.webp`. A drawing always opens in Blueprint.

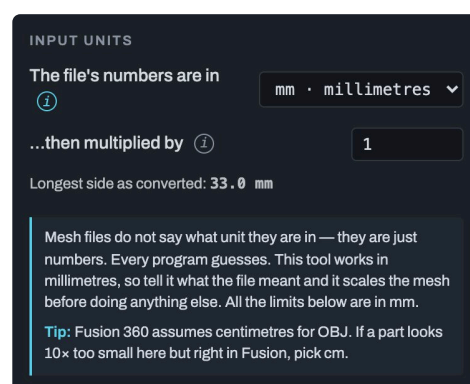


A servo bracket loaded. Its size is in the top-left corner of the stage.

The part's size appears in the top-left corner of the stage (for example $19.3 \times 25.0 \times 33.0$ mm). After a run, the **Input mesh** table in the report lists its triangle count, whether it is watertight (fully closed), how many bodies it holds, its bounding box, surface area and volume.

To start again with a different file, click **New project**.

2.3 Input units and scale



Input units. The small *i* opens a plain-language explanation, shown here for the unit.

Mesh files do not record their units. They are just numbers. stlToSolid works in millimetres, so tell it what the file meant. These two settings are shared by Mesh → Solid, Sliced Loft and X-Ray.

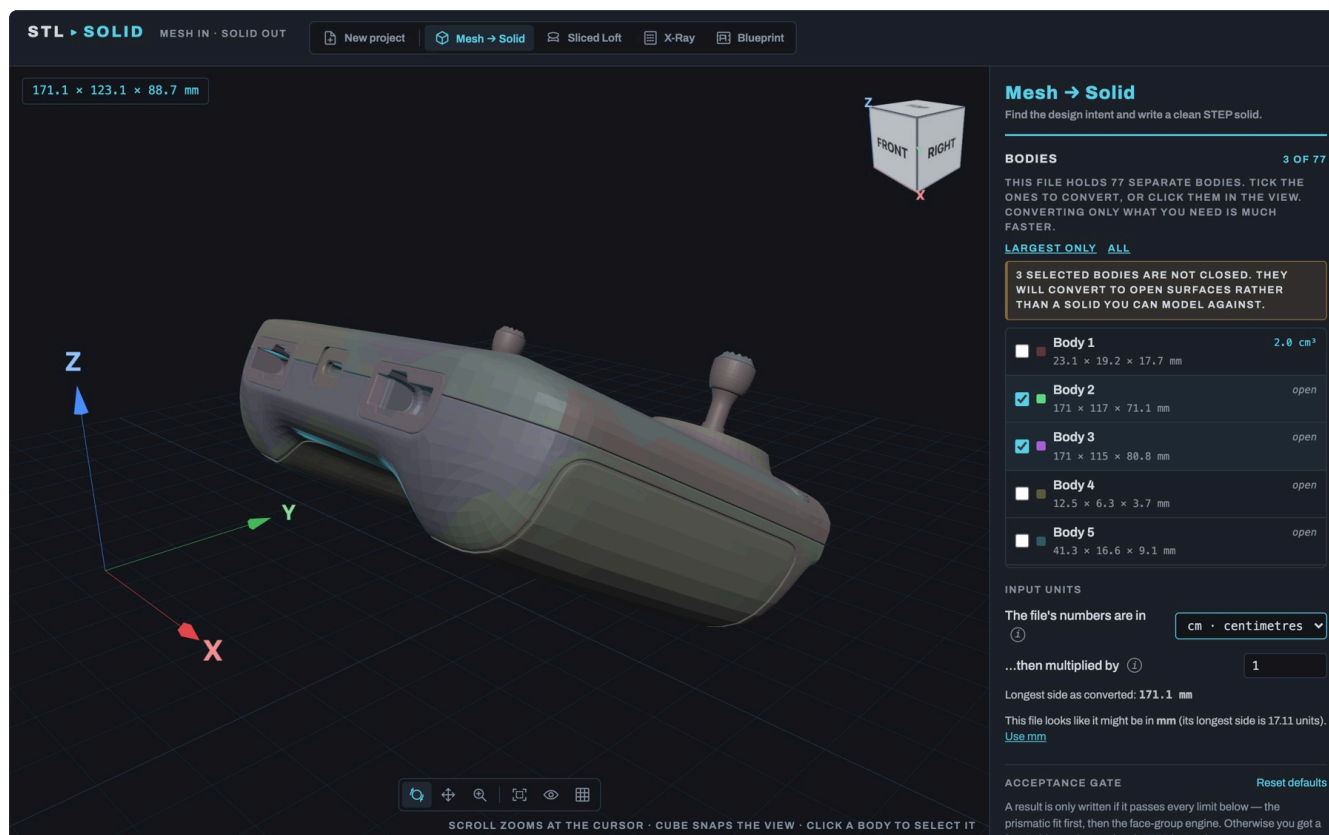
Setting	Default	What it does
The file's numbers are in	mm	The unit the file was saved in: mm, cm, inches, feet or metres. The mesh is scaled to millimetres before anything else happens.
...then multiplied by	1	An extra factor on top of the unit. Units can only make a part bigger (cm is $\times 10$, inches $\times 25.4$), so use this to shrink one. For example, 0.1 fixes a part designed in centimetres but saved as millimetres. Leave it at 1 unless the size looks wrong.

The line **Longest side as converted** shows the result, so you can spot a wrong scale before a long conversion, not after. If the part looks unusually large or small, a yellow **Check the size** note suggests a unit or factor and offers a one-click **Use ...** button.

Tip: Fusion 360 assumes centimetres for OBJ files. If a part looks 10× too small in stlToSolid but right in Fusion, pick **cm**.

2.4 Picking bodies

Some files hold several separate pieces ("bodies"). A scanned controller, for example, may have dozens. When that happens a **Bodies** panel appears at the top of the rail.



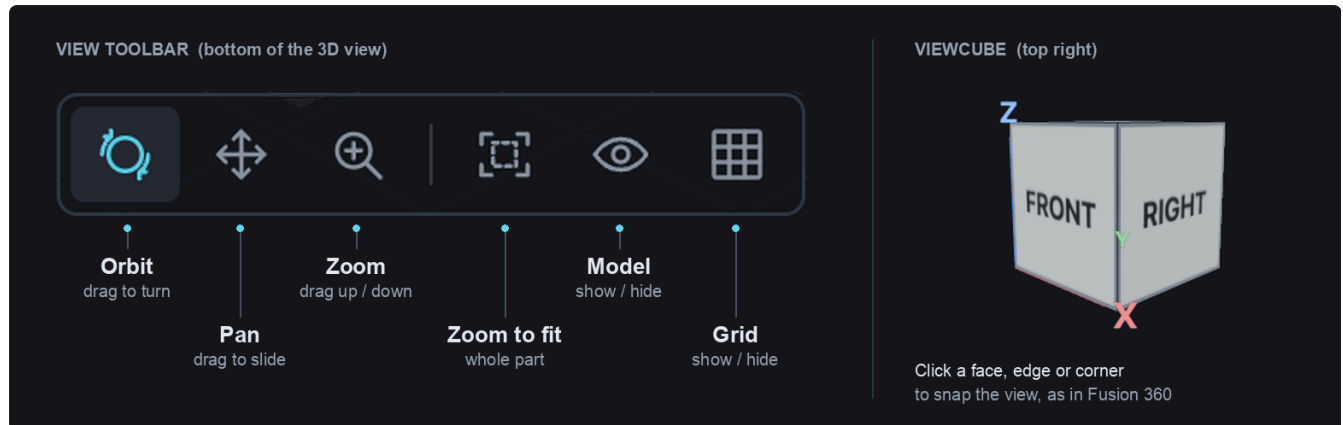
A controller with 77 bodies. Each body has its own colour in the view and in the list.

- Tick the bodies you want to convert, or **click them in the 3D view**. Hovering a row lights up that body.
- **Largest only** picks the main bodies. **All** ticks everything.

- Each row shows the body's size and volume. Bodies marked **open** are not closed. They convert to open surfaces rather than a solid, and a warning tells you when you have picked any.

Converting only what you need is much faster and uses much less memory.

2.5 Moving around the 3D view



The view toolbar sits at the bottom of the 3D view. The ViewCube sits in the top-right corner. The highlighted button is the active drag tool.

Control	What it does
Scroll wheel	Zooms in and out at the cursor.
ViewCube (top right)	Click a face, edge or corner to snap the view, as in Fusion 360.
Orbit (bottom toolbar)	Left-drag turns the view.
Pan	Left-drag slides the view.
Zoom	Left-drag up to zoom in, down to zoom out, at the point you press.
Zoom to fit	Frames the whole part.
Eye	Hides or shows the model. Slice traces stay visible.
Grid	Hides or shows the floor grid.

The coloured triad shows the axes: **X red, Y green, Z blue**, the same as Fusion 360.

3. Mesh → Solid

What it does: finds the design intent in a mesh (an extrusion direction, flat faces, cylinders, cones, spheres) and writes a clean STEP solid. It is the best tool for machined or CAD-designed parts.

3.1 How it works

The tool tries three routes, best first, and keeps the first one that passes every quality check:

1. **Prismatic.** If the part is an extrusion (a sketch pulled straight), it redraws the outlines with lines and arcs and extrudes them. This is the cleanest result. It also gives you editable scripts.
2. **Face-group engine.** Otherwise it groups the mesh into surface regions and gives each one a real plane, cylinder, cone, sphere or torus face. This is the same idea as Fusion's paid "Prismatic" mesh conversion.
3. **Faceted.** If neither passes, you still get a valid, closed solid. Flat areas are merged into single faces and curved areas are simplified, but it is made of facets.

You always get a solid. The report tells you which route produced it.

3.2 Settings: the acceptance gate

ACCEPTANCE GATE [Reset defaults](#)

A result is only written if it passes every limit below — the prismatic fit first, then the face-group engine. Otherwise you get a faceted (exact but unclean) STEP instead.

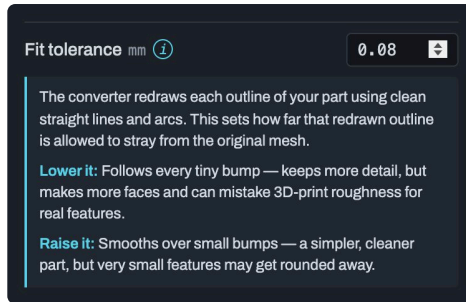
Fit tolerance mm ⓘ	0.08
Surface deviation (p95) mm ⓘ	0.25
Surface deviation (max) mm ⓘ	0.26
Bore deviation (max) mm ⓘ	0.1
Volume error % ⓘ	2
Faceted reduce tolerance mm ⓘ	0.05

☐ **Force prismatic on scan-like meshes**
Dense organic scans (millions of tiny triangles) normally skip straight to the faceted copy, because they rarely have flat faces and true cylinders to recover. Tick this to make the tool try anyway.

☒ **Face-group engine**
When the extrusion fit is rejected, group the mesh into surface regions and give each one a real plane, cylinder, cone or sphere face (like Fusion's "Prismatic" mesh conversion) before falling back to a faceted copy. Untick to skip it.

*The acceptance gate and the two options. Changed values are highlighted. **Reset defaults** puts them all back.*

A clean result is only written if it passes every limit. Click the small **i** next to any setting for a plain-language explanation of what raising or lowering it does.



The *i* button opens the explanation, with what happens when you lower or raise the value.

Setting	Default	What it means	Lower it	Raise it
Fit tolerance	0.08 mm	How far the redrawn lines and arcs may stray from the mesh.	Follows every tiny bump. Keeps more detail, but makes more faces and can mistake 3D-print roughness for real features.	Smooths over small bumps. A simpler part, but very small features may get rounded away.
Surface deviation (p95)	0.25 mm	95 % of the surface must be closer than this to the original. This is the main quality bar.	Demands a near-perfect match. More parts fall back to the faceted copy.	Accepts a rougher match. More parts convert cleanly, but may visibly differ.
Surface deviation (max)	0.26 mm	The limit for the single worst spot on the part. A taper, chamfer or rounded corner is usually what breaks it.	Even one slightly-off spot rejects the clean solid.	Lets one small area be off while the rest stays accurate.
Bore deviation (max)	0.10 mm	A stricter limit for round holes, so screws and pins still fit.	Holes must be almost exactly the right size.	Only sensible when the holes are cosmetic.
Volume error	2 %	The rebuilt part must hold about the same amount of material as the original. Catches a pocket that got filled in or a chunk that went missing.	Stricter overall shape check.	More forgiving. Rarely needs changing.
Faceted reduce tolerance	0.05 mm	Only used for a faceted result: how far simplified curved areas may move from the original (like Fusion's <i>Reduce by tolerance</i>). 0 keeps every triangle.	Keeps more triangles. Closer to the mesh.	Fewer triangles and a smaller STEP. Small details may be smoothed.

3.3 Settings: options

Option	Default	What it does
Force prismatic on scan-like meshes	off	Dense organic scans (millions of tiny triangles) normally go straight to a faceted copy, because they rarely have flat faces and true cylinders to recover. Tick this to make the tool try anyway.
Face-group engine	on	When the extrusion fit is rejected, try route 2 (fitted planes, cylinders, cones, spheres and tori) before falling back to a faceted copy. Untick to skip it.

3.4 Converting

Click **Convert to STEP**. The button shows *Converting...* (or *Waiting in queue...*) while it works, and a **Cancel** button appears. Your file and settings stay loaded if you cancel. Open **Pipeline log** under the report to watch the steps.

CHECK	MEASURED	LIMIT	
Surface deviation p95 (mesh → solid)	0.019 mm	≤ 0.25 mm	PASS
Surface deviation max (mesh → solid)	0.060 mm	≤ 0.26 mm	PASS
Surface deviation p95 (solid → mesh)	0.018 mm	≤ 0.25 mm	PASS
Surface deviation max (solid → mesh)	0.058 mm	≤ 0.26 mm	PASS
Bore deviation p95 (5 bores)	0.015 mm	≤ 0.1 mm	PASS
Volume error	0.00 %	≤ 2 %	PASS

OUTPUT STEP
1,644 triangles → 23 faces

BREP faces	23
Surface types	15 planes, 8 cylinders
Mean deviation	0.005 mm
Worst point at	(-60.91, -3.57, 1.97) mm
Solid volume	3,688 mm³
File size	85.4 KB

Download STEP
Download CadQuery script (.py)
Download Fusion 360 script (.py) — experimental

SCROLL ZOOMS AT THE CURSOR · CUBE SNAPS THE VIEW · CLICK A BODY TO SELECT IT

The servo bracket as a prismatic solid: 1,644 triangles became 23 faces (15 planes and 8 cylinders). Every check passed.

3.5 Downloads

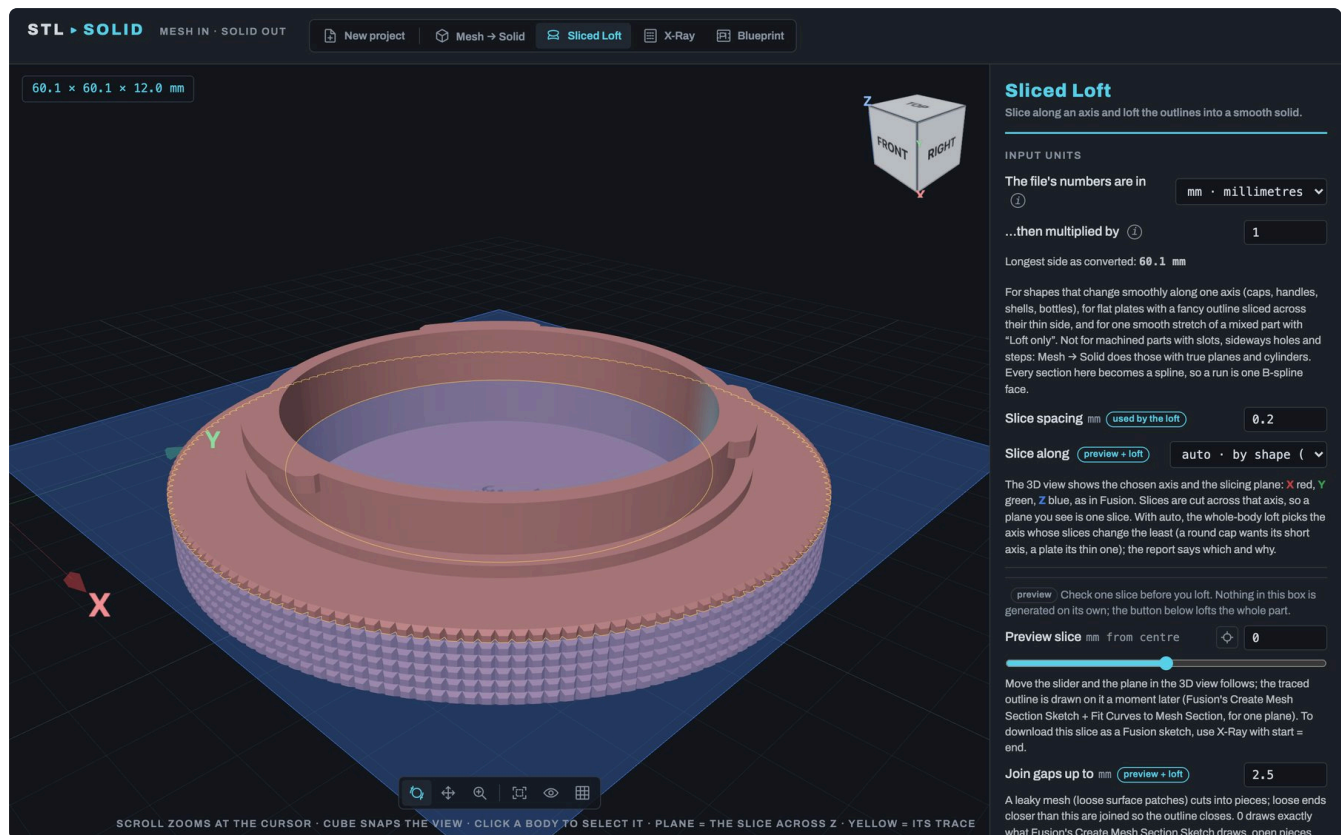
- **Download STEP:** the solid. Import it into Fusion 360 or any CAD program.
- **Download CadQuery script (.py):** for prismatic results. It rebuilds the recognised sketches and extrudes as an editable program. Change a radius or height and re-run it to get a new STEP.
- **Download Fusion 360 script (.py) — experimental:** for prismatic results. It rebuilds the part as a parametric Fusion timeline.
- **Download Fusion 360 script (Boundary Fill):** for face-group results (experimental). Fusion rebuilds every fitted surface and computes the edges itself, so the solid has clean edges. The label says whether a test run predicts it will build (*experimental*), probably fail (*likely to fail*), or could not be checked (*not checked*). The STEP is fine either way.

How to run a script in Fusion is in [section 9](#).

4. Sliced Loft

What it does: cuts the part into thin slices along one axis, redraws each outline as a smooth curve and lofts the stack into one smooth solid. It is what you would do by hand in Fusion with *Create Mesh Section Sketch* → *Fit Curves to Mesh Section* → *Loft*, done automatically.

Use it for: shapes that change smoothly along one direction (lens caps, handles, shells, bottles), flat plates with a fancy outline (sliced across their thin side), and one smooth stretch of a mixed part (with **Loft only**). **Not for:** machined parts with slots, sideways holes and steps. A hole drilled across the slicing direction becomes a trough. Use Mesh → Solid for those parts.



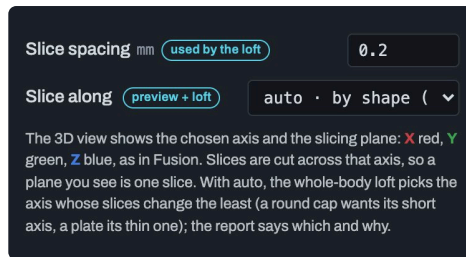
A camera body cap in Sliced Loft. The blue plane is the preview slice. The yellow line is the outline traced on it.

4.1 What the little tags mean

Some settings carry a small tag, so you can see which ones change the solid:

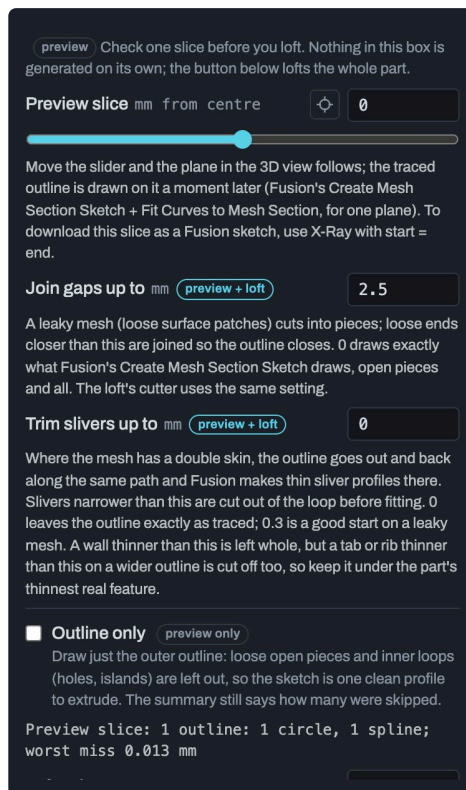
- **used by the loft:** changes the solid the Convert button makes.
- **preview + loft:** changes both the preview slice and the solid.
- **preview only:** changes only the preview slice. The solid ignores it.

4.2 Settings: spacing and axis



Setting	Default	What it does
Slice spacing	0.2 mm	Distance between slices. Smaller follows the shape more closely. The loft itself uses at most about 60 slices per run, but the checks still measure the result against every triangle of the mesh.
Slice along	auto	The axis to slice across. auto · by shape picks the axis whose slices change the least: a round cap wants its short axis, a plate its thin one. The list shows the part's length along each axis, and the report says which axis was picked and why.

4.3 Settings: the preview slice



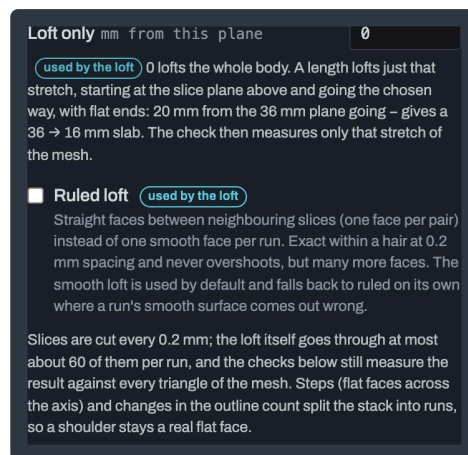
Everything in the preview box is for checking one slice before you loft. The summary line at the bottom says what was found.

Setting	Default	What it does
Preview slice	0 mm from centre	A slider and a number box that move one plane through the part. The traced outline appears on it a moment later. The crosshair button lets you click a face in the 3D view to put the plane there (Esc cancels). To download this one slice as a Fusion sketch, use X-Ray with start = end.

Setting	Default	What it does
Join gaps up to	2.5 mm	A leaky mesh (loose surface patches) cuts into open pieces. Loose ends closer than this are joined so the outline closes. 0 draws exactly what Fusion's own mesh section draws, open pieces and all. The loft uses the same value.
Trim slivers up to	0 mm	Where a mesh has a double skin, the outline goes out and back along the same path and makes thin slivers. Slivers narrower than this are cut out before fitting. Keep it at 0 for a clean, watertight mesh (a warning appears if you don't). 0.3 is a good start for a leaky one. Keep it below the part's thinnest real rib or tab.
Outline only	off	Draws just the outer outline and leaves out holes and loose pieces. Preview only.

The summary line reads like **Preview slice: 1 outline: 1 circle, 1 spline; worst miss 0.013 mm:** how many closed outlines were found, what they are made of, and how far the fitted curve is from the mesh at its worst point.

4.4 Settings: what to loft



Setting	Default	What it does
Loft only ... mm from this plane	0	0 lofts the whole body. A length lofts just that stretch, starting at the preview plane, with flat ends. The preview slider is then labelled Loft starts at . For example, 20 mm from the 36 mm plane going – gives a 36 → 16 mm slab. The check then measures only that stretch of the mesh.
Going	–	Shown when Loft only is above 0. + goes along the axis arrow, – against it.
Ruled loft	off	Straight faces between neighbouring slices (one face per pair) instead of one smooth face per run. Exact within a hair at 0.2 mm spacing and never overshoots, but many more faces. The smooth loft switches to ruled by itself for any run whose smooth surface comes out wrong.

The acceptance gate appears below these settings too (see section 3.2). With Sliced Loft the limits are **measured and shown, not enforced**: you chose the method, so its solid is written whenever it can be built. Only a loft that cannot be built at all falls back to a faceted copy.

4.5 Converting and downloads

Click **Convert to STEP**. The stack is split into *runs* wherever a flat step crosses the axis or the number of outlines changes, so a shoulder stays a real flat face. Each run becomes one smooth face. Holes along the axis are lofted and cut.

60.1 × 60.1 × 12.0 mm

STL → SOLID MESH IN · SOLID OUT

New project Mesh → Solid Sliced Loft X-Ray Blueprint

SLICED LOFT SOLID

87 sections along Z at 0.2 mm, 5 runs (split at 5 flat steps), 10 holes cut: one smooth face per run (4 runs ruled), like Fusion's Mesh Section Sketch + Loft. Axis chosen by shape: X 18 runs (10 of one slice) + 4 fast changes, Y 18 runs (14 of one slice) + 2 fast changes, Z 6 runs (3 of one slice) + 1 fast change. The gate below did not pass; the loft is written anyway because you chose the method — the deviation is usually a sideways hole or a corner the smooth surface cannot follow.

CHECK	MEASURED	LIMIT	
Surface deviation p95 (mesh → solid)	0.118 mm	≤ 0.25 mm	PASS
Surface deviation max (mesh → solid)	0.266 mm	≤ 0.26 mm	FAIL
Surface deviation p95 (solid → mesh)	0.043 mm	≤ 0.25 mm	PASS
Surface deviation max (solid → mesh)	0.146 mm	≤ 0.26 mm	PASS
Volume error	0.20 %	≤ 2 %	PASS

OUTPUT STEP

13,552 triangles → 274 faces

BREP faces 274

Surface types 231 planes, 41 freeform

Mean deviation 0.022 mm

Worst point at (9.39, 28.23, -2.42) mm

Solid volume 13,275 mm³

File size 19.73 MB

Download STEP

Download Fusion 360 script (.py) — experimental

SCROLL ZOOMS AT THE CURSOR · CUBE SNAPS THE VIEW · CLICK A BODY TO SELECT IT · PLANE = THE SLICE ACROSS Z · YELLOW = ITS TRACE

The body cap as a sliced loft solid. The summary explains the sections, runs and chosen axis. Here one strict check (max deviation, 0.266 mm against 0.26 mm) is just over its limit. It is reported, but the solid is still written.

Downloads: Download STEP and **Download Fusion 360 script (.py) — experimental**. The script repeats the loft as a Fusion timeline: one sketch per section on an offset plane, then one Loft per run, with holes as Loft cuts.

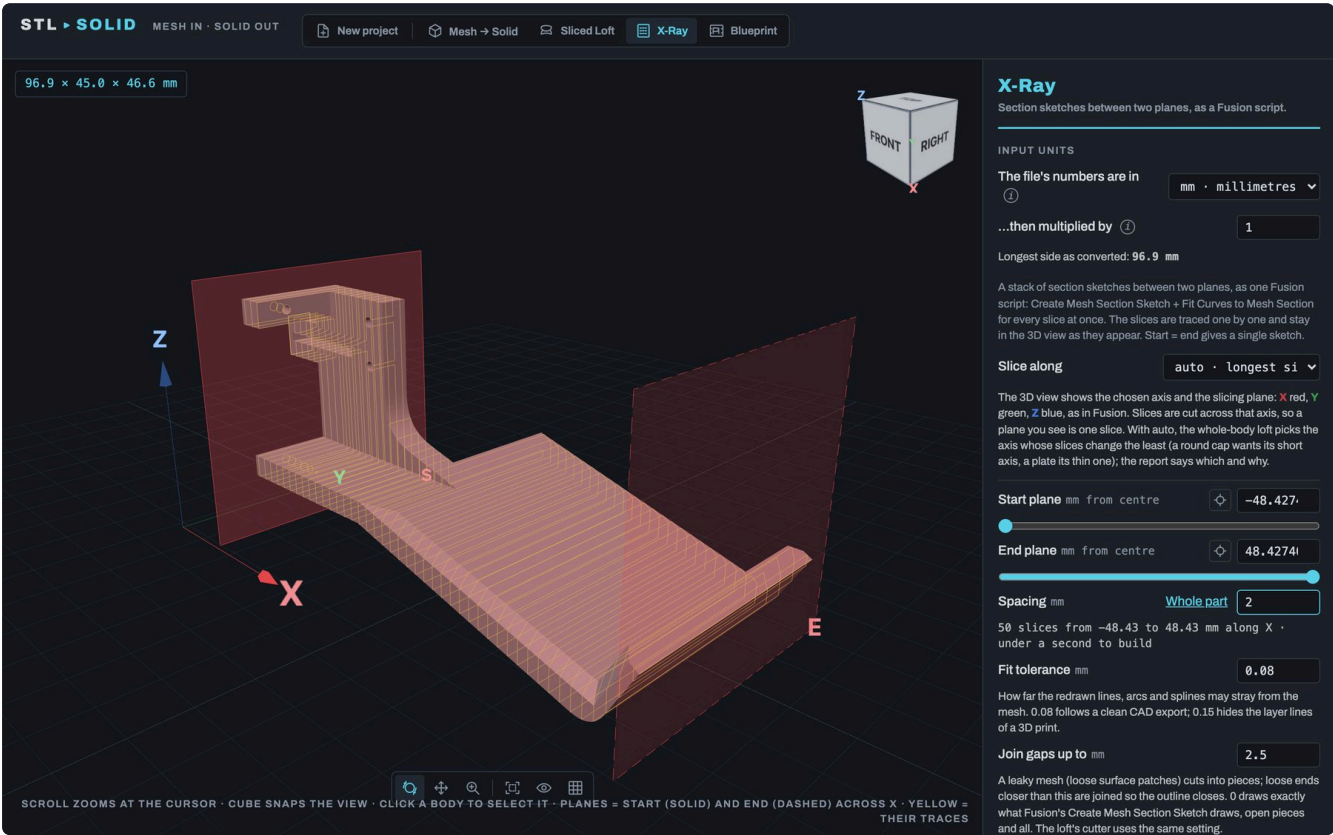
5. X-Ray

What it does: cuts the part at as many planes as you like and turns each cut into a **real Fusion 360 sketch**: clean lines, arcs and splines that join into closed profiles. You can click a profile and extrude it, offset it, or dimension against it, like a sketch you drew yourself.

Why this matters. Fusion's own *Create Mesh Section Sketch* gives you an outline made of hundreds or thousands of separate points. You cannot select it as a profile or extrude it, so in practice you have to draw a new sketch around the shape by hand. X-Ray does that redrawing for you, for every slice at once.

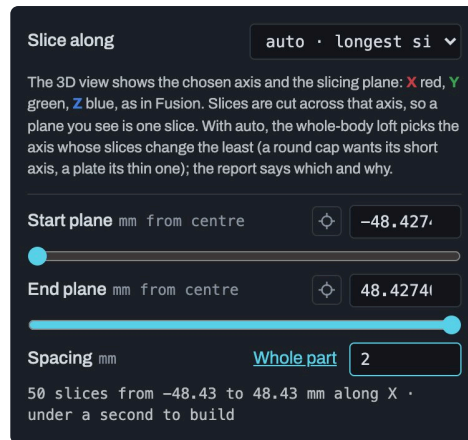
	Fusion mesh section sketch	X-Ray sketch
Made of	Hundreds or thousands of points	A few lines, arcs and splines
Closed profile you can select	No	Yes
Extrude, offset, dimension	Only after you trace it by hand	Straight away
Many slices	One plane at a time	A whole stack in one script

X-Ray does not make a STEP. It downloads one Fusion 360 script that draws the sketches, each on its own plane, so you can model in Fusion yourself. It can also extrude every slice into a solid slab for you. If the mesh is already in a Fusion design, the [Mesh to Outline add-in](#) does the same thing without a download.



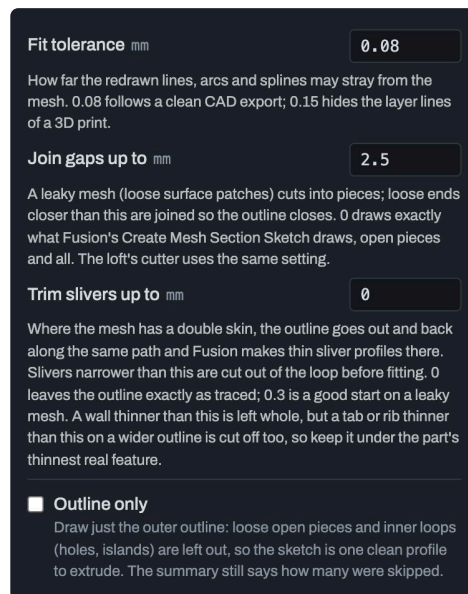
X-Ray on a frame, sliced along X every 2 mm over the whole part (50 slices). S is the start plane (solid), E the end plane (dashed). Each yellow outline is one traced slice, ready to become a Fusion sketch.

5.1 Settings: where to slice



Setting	Default	What it does
Slice along	auto	The axis to slice across. In X-Ray, auto uses the longest side.
Start plane / End plane	0 mm from centre	Where the stack begins and ends, in mm from the part's centre. Each has a slider, a number box and a crosshair button : press it, then click a face in the 3D view to put the plane there. Start = end gives a single sketch. Start past end is fine.
Spacing	0.2 mm	Distance between slices. Whole part moves the planes to both ends of the part. The line below shows the slice count and an estimated build time.

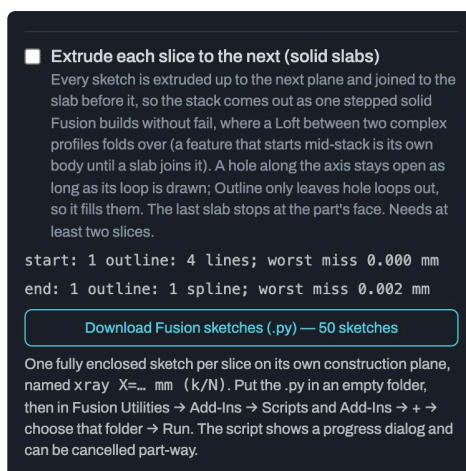
5.2 Settings: how each slice is traced



Setting	Default	What it does
Fit tolerance	0.08 mm	How far the fitted lines, arcs and splines may stray from the mesh. 0.08 follows a clean CAD export. 0.15 hides the layer lines of a 3D print.
Join gaps up to	2.5 mm	As in Sliced Loft (section 4.3). X-Ray keeps its own value.

Setting	Default	What it does
Trim slivers up to	0 mm	As in Sliced Loft. X-Ray keeps its own value, so a trim set for a leaky loft does not eat the thin walls of the next part. If slivers were trimmed anywhere in the stack, a note says in how many slices.
Outline only	off	Just the outer outline of each slice: holes, islands and loose pieces are left out. About five times faster on organic shapes.

5.3 Settings: slabs and the download



The start and end summaries say what the two end slices hold. The download button shows how many sketches the script will draw.

Setting	Default	What it does
Extrude each slice to the next (solid slabs)	off	The script also extrudes every sketch up to the next plane and joins the slabs, giving one stepped solid that Fusion always builds. (A Loft between two complex profiles often folds over.) Holes along the axis stay open as long as their loops are drawn. Outline only leaves hole loops out, so it fills them. Needs at least two slices.

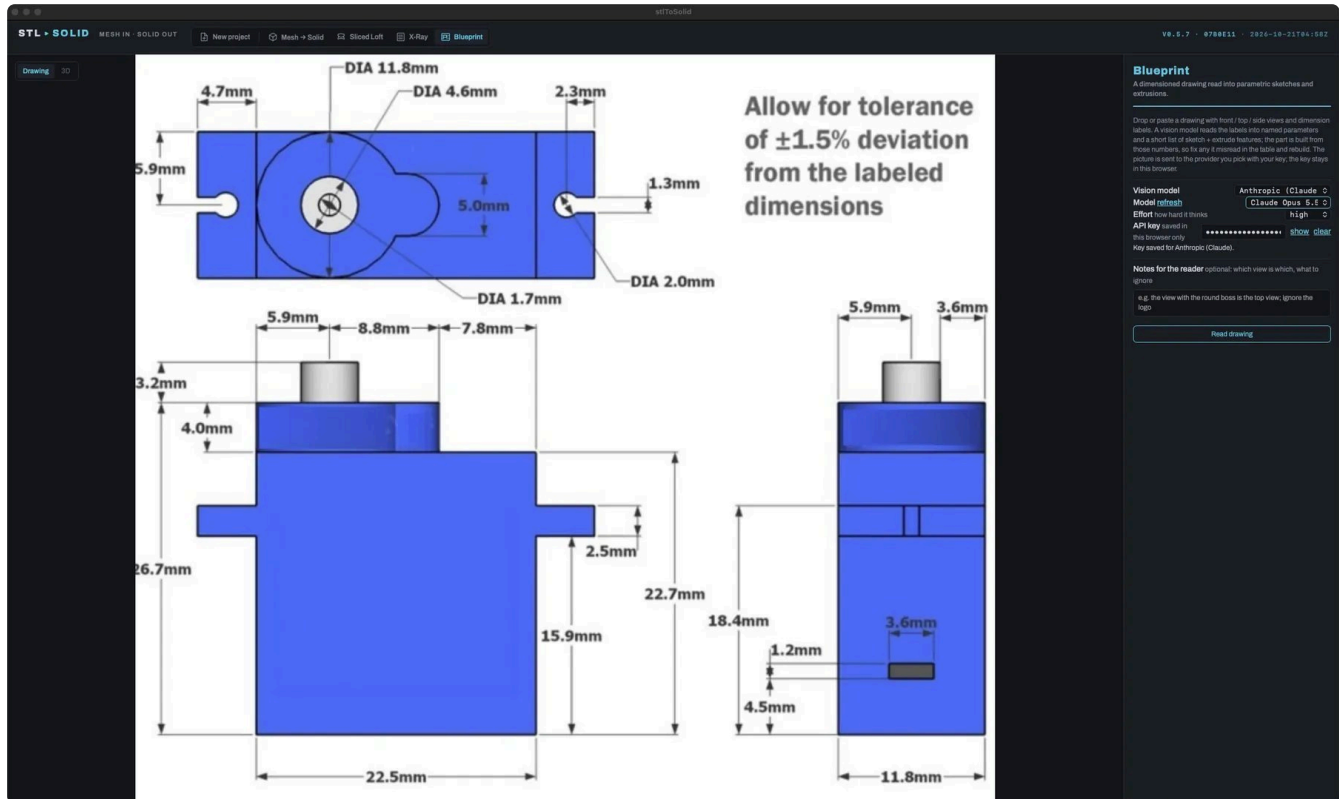
The slices are traced one by one and stay in the view as they appear. Click **stop** to halt the tracing and **trace the rest** to carry on.

Click **Download Fusion sketches (.py)**. The server fits every slice (*Working...* with a spinner) and then hands you the script. Each slice becomes a fully enclosed sketch on its own construction plane, named like xray X=-18.50 mm (3/24). In Fusion, the script shows a progress dialog and can be cancelled part-way.

Limits: at most 500 slices and about 4 minutes of fitting per download. If you go over, raise the spacing, narrow the range, or tick **Outline only**.

6. Blueprint

What it does: starts from a picture instead of a mesh. Drop or paste a dimensioned drawing (front, top and side views with their labels, like a datasheet). A vision AI model of your choice reads the labels into a small **recipe**: named parameters in millimetres and a short list of sketch + extrude features. The part is built from those numbers, so you can fix any it misread and rebuild. You get a **parametric Fusion 360 script**, a STEP and a 3D preview.



A micro-servo datasheet drawing loaded in Blueprint, ready to read.

6.1 Settings: reading the drawing

Drop or paste a drawing with front / top / side views and dimension labels. A vision model reads the labels into named parameters and a short list of sketch + extrude features; the part is built from those numbers, so fix any it misread in the table and rebuild. The picture is sent to the provider you pick with your key; the key stays in this browser.

Vision model Anthropic (Claude ▼)

Model [refresh](#) claude-opus-5

Effort how hard it thinks high ▼

API key saved in this browser only paste your key [show](#)

A key is needed here to read a drawing (or set STLTSOLID_ANTHROPIC_API_KEY on the server).

Notes for the reader optional: which view is which, what to ignore

e.g. the view with the round boss is the top view; ignore the logo

[Read drawing](#)

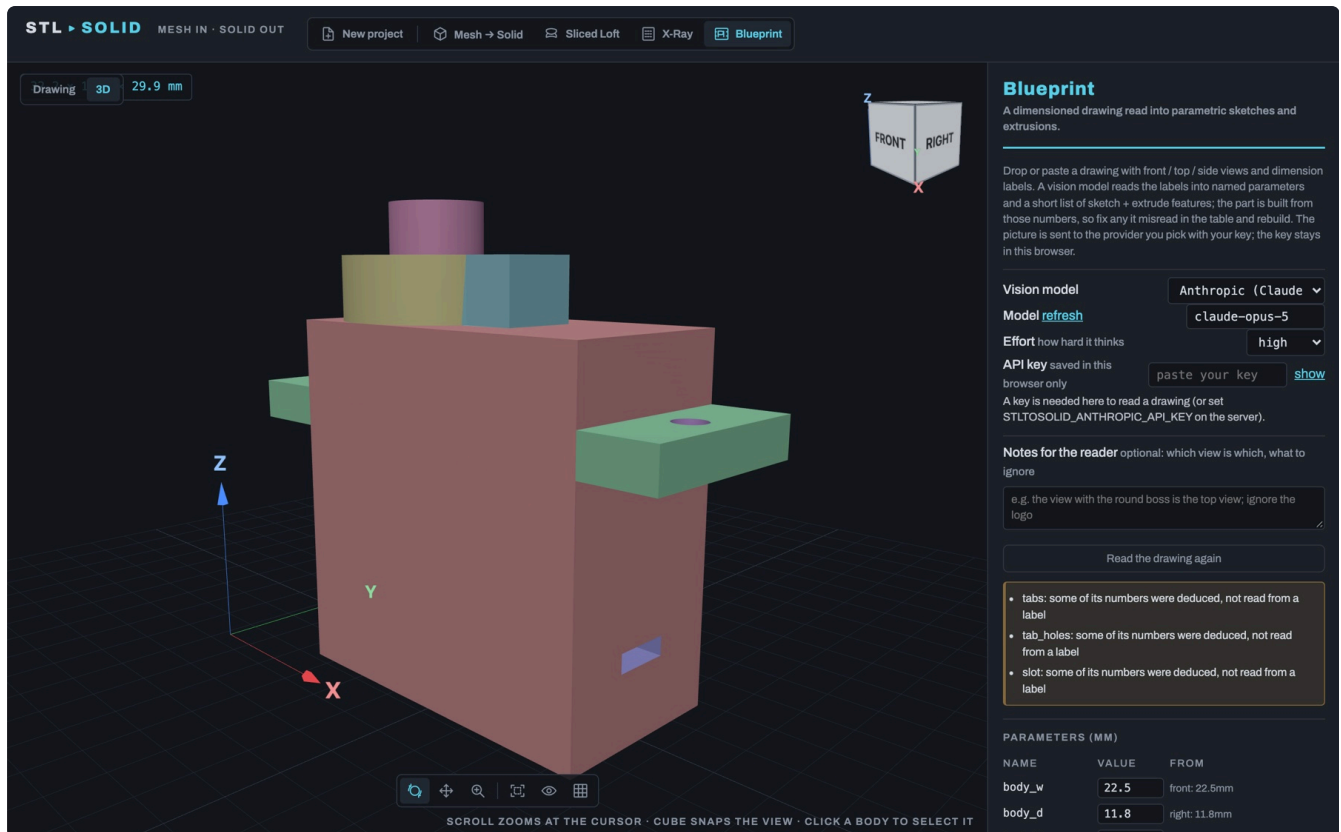
Setting	Default	What it does
Vision model	Anthropic (Claude)	The provider that reads the drawing: Anthropic (Claude), OpenAI, DeepSeek, or <i>Custom</i> (any OpenAI-compatible server, such as Ollama running a vision model on your own computer, or OpenRouter).
Model	the provider's default	Choose from the list, click refresh to ask the provider again, or pick <i>type a name...</i> to enter one.
Effort	high	How hard the model thinks. <i>high</i> is the most careful, slowest and most expensive. <i>medium</i> takes about half the time. <i>low</i> is a quick look.
Base URL	—	<i>Custom</i> only: the server's address, for example <code>http://localhost:11434/v1</code> for Ollama.
API key	empty	Your key for that provider. show reveals it, clear removes it. It is saved in this browser (or app) only and sent with each read. The server never stores it. Local servers usually need no key.
Notes for the reader	empty	Optional hints, for example " <i>the view with the round boss is the top view; ignore the logo</i> ".

Click **Read drawing**. A **Cancel** button appears while it reads.

Privacy: the drawing is sent to the provider you pick, with your key, and nowhere else. Choose a local model if the drawing must stay on your computer.

6.2 Checking and editing the recipe

When the read finishes, the part is built and the stage switches to the **3D** tab. Use the **Drawing | 3D** tabs to switch between the picture and the model.



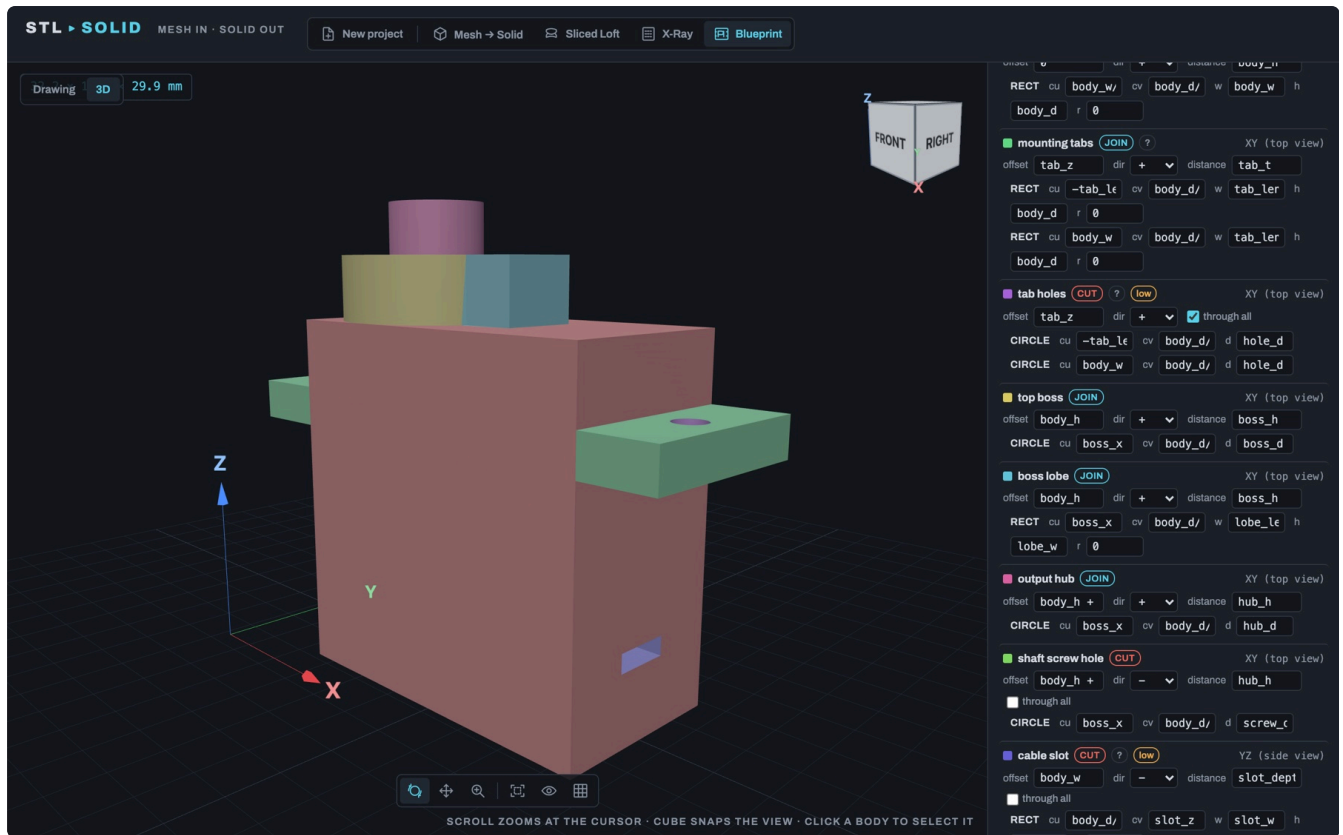
The built servo. Each feature has its own colour.

Before anything is built, the recipe goes through automatic checks: every value must resolve, shapes must have a size, cuts must remove something, and the overall size must match the drawing. Problems and guesses are listed in a yellow box.

Parameters (mm). Every number, with **From** saying where it came from (for example `front: 22.5mm`). A **?** marks a value the model had to deduce rather than read from a label. Check those first.

Features. Each feature has a colour swatch that matches the 3D view. Hover a feature to light it up. For each feature you can see and edit:

- the operation (**new body**, **join** or **cut**) and its sketch plane (XY = top view, XZ = front, YZ = right side),
- **offset**, direction (+, – or **sym**) and **distance**, or **through all** for cuts,
- its shapes (rectangle, circle, slot, polygon) with their positions and sizes (`cu / cv` = centre, `w / h` = width and height, `r` = corner radius, `d` = diameter). A **low** badge marks a feature the model was unsure of.

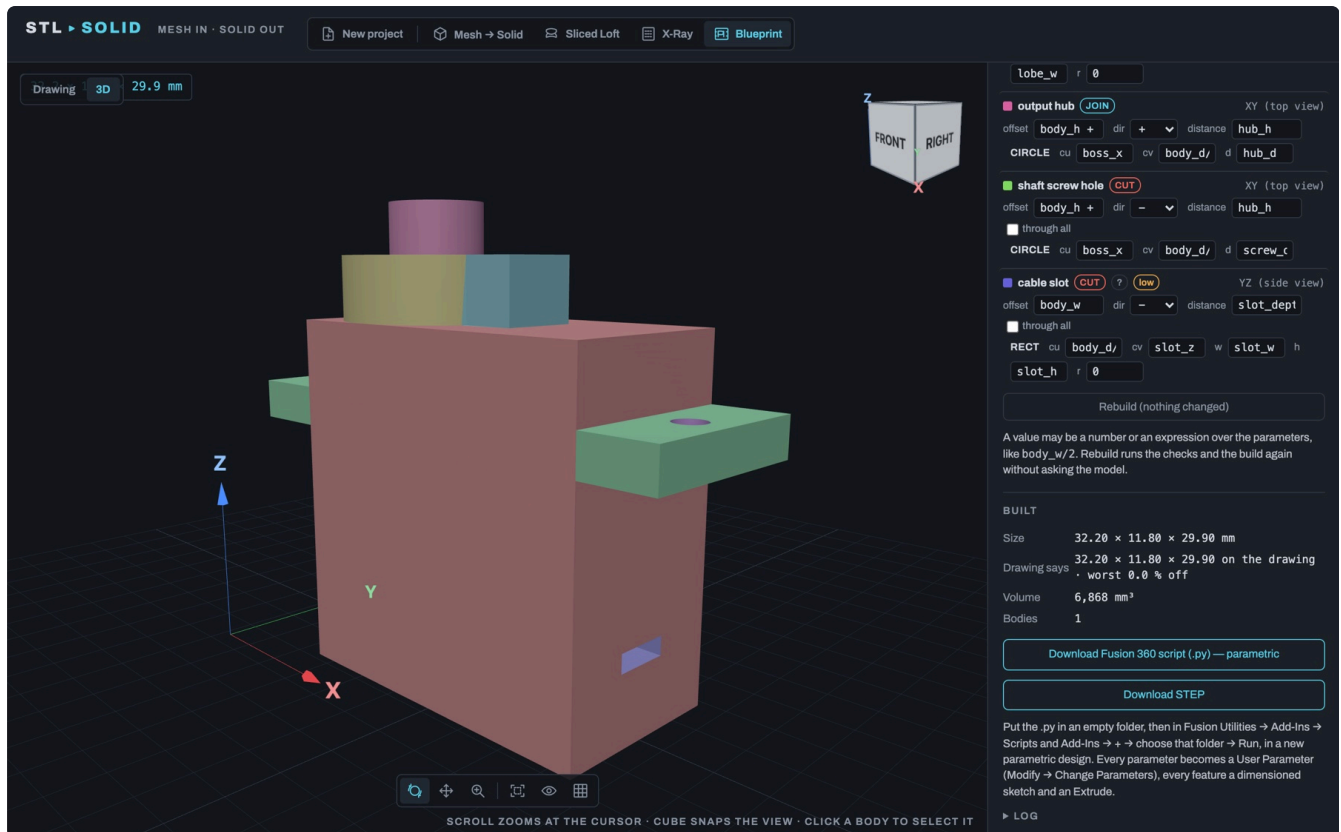


The feature list. Values can be numbers or expressions such as `body_w/2`.

Any value can be a number or an expression over the parameters, like `body_w/2`. Change a value and the 3D view redraws within about a second (*Redrawing object...*). Click **Revert** to undo your edits.

6.3 Rebuilding and downloading

Click **Rebuild with these values (STEP + Fusion script)**. This runs the checks and the build again **without** asking the AI model, so it costs nothing.



The Built table compares the model's size with the size stated on the drawing.

The **Built** table shows the size, what the drawing says (and how far off the worst dimension is), the volume, the number of bodies, and which model read the drawing. Then:

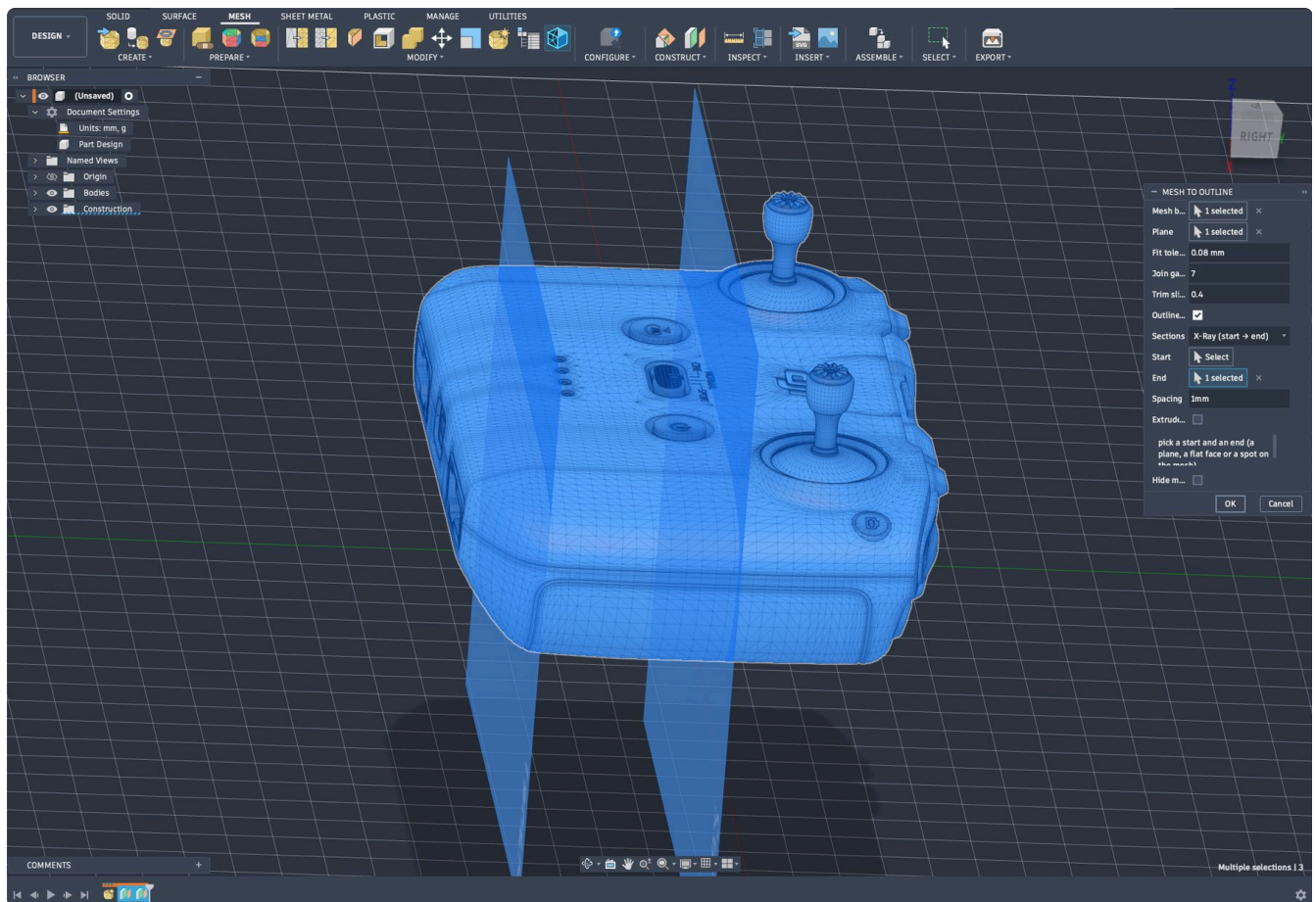
- **Download Fusion 360 script (.py) — parametric.** Every parameter becomes a Fusion **User Parameter** (*Modify* → *Change Parameters*), and every feature becomes a dimensioned sketch and an Extrude. Change `body_h` in Fusion and the parts on top move with it.
- **Download STEP.** The same part as a plain solid.

Click **Read the drawing again** to start over with a fresh read.

7. Fusion 360 add-in: Mesh to Outline

What it does: the X-Ray and single-slice tracing, running inside Fusion 360 itself. Pick a mesh body and a plane, and it draws the section as a normal sketch of lines, arcs, circles and fitted splines. Or pick a start and an end and it draws a whole stack, and can extrude it into one solid. It is Fusion's *Create Mesh Section Sketch* followed by *Fit Curves to Mesh Section*, in one button, for as many planes as you like.

It runs on Fusion's own Python. Nothing is uploaded, and no server or app is needed.



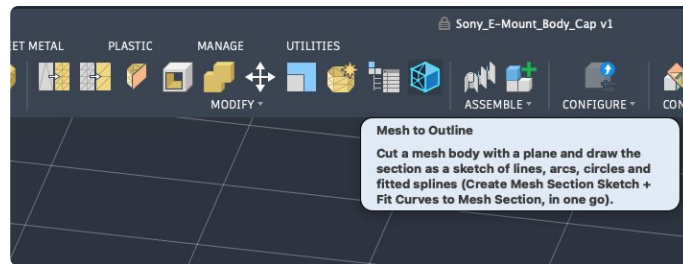
Mesh to Outline on a scanned controller in Fusion 360.

7.1 Installing

The add-ins are not in a public release yet. The website's add-ins button says *coming soon* until they are.

1. Download the add-ins zip from the latest release and unzip it.
2. Copy the `MeshSectionSketch` folder into Fusion's add-ins folder: - **Mac:** `~/Library/Application Support/Autodesk/Autodesk Fusion 360/API/AddIns/` - **Windows:** `%APPDATA%\Autodesk\Autodesk Fusion 360\API\AddIns\`
3. In Fusion: **Utilities** → **Add-Ins** → **Scripts and Add-Ins** → **Add-Ins** tab, select **MeshSectionSketch** and click **Run**. It starts with Fusion from then on.

It needs a current Fusion (with its built-in Python 3.14). numpy comes inside the add-in.



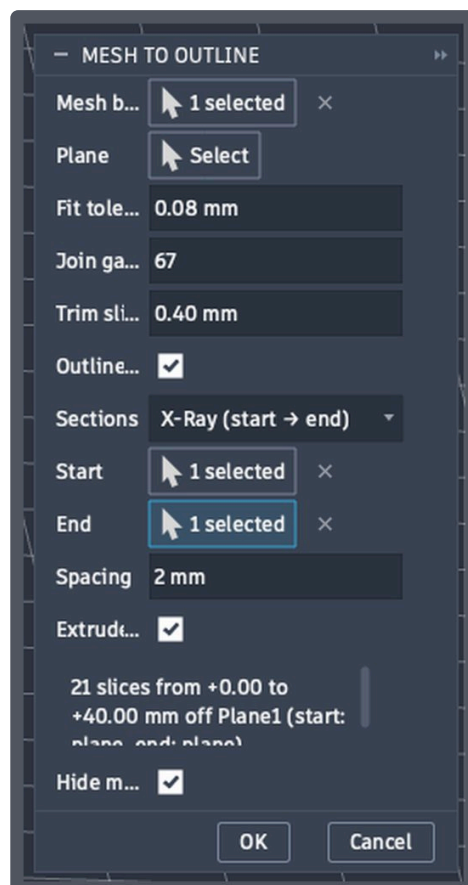
The button sits on the **Mesh** tab, in the **Modify** panel, next to **Convert Mesh**.

If the Mesh panel is not found, the button goes under **Utilities** → **Add-Ins** instead. The Text Commands palette says where.

7.2 Before you start

1. **Insert** → **Insert Mesh**, and set the units to mm.
2. For a single section or a stack: make a construction plane where you want the cut (**Construct** → **Offset Plane**), or use any flat face.
3. **Mesh** tab → **Modify** → **Mesh to Outline**.

7.3 The settings



The dialog in X-Ray mode. The values shown are from one run on a leaky scanned controller, not the defaults. Fusion cuts the labels short. The full names are in the table below.

Setting	Default	What it does
Mesh body	—	The mesh body to cut.

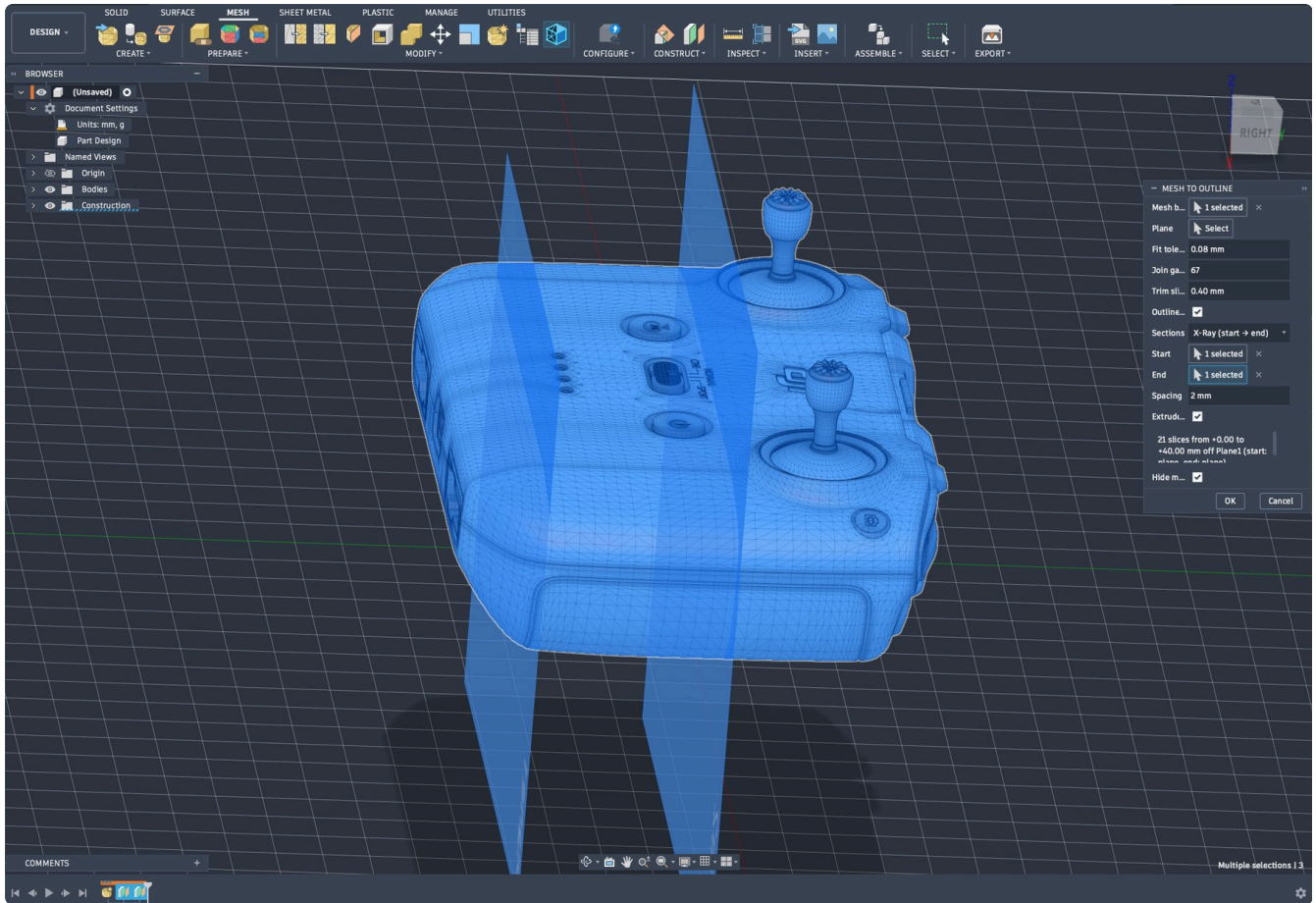
Setting	Default	What it does
Plane	—	A construction plane or a flat face to cut on. In X-Ray mode it is optional: it only sets the slice direction (see below).
Fit tolerance	0.08 mm	How far a fitted curve may sit off the raw section.
Join gaps up to	2.5 mm	Loose ends closer than this are joined, so an outline broken only by hairline cracks between surface patches comes back as one closed profile. 0 draws exactly what Fusion's own mesh section draws. Raise it if the outline still comes back in pieces. Lower it if two unrelated pieces got joined.
Trim slivers up to	0 mm	Where the mesh has a double skin, the outline goes out and back along the same path and Fusion makes thin sliver profiles. Slivers narrower than this are cut out before fitting. 0.3 is a good start on a leaky mesh. 0 leaves the outline as traced.
Outline only	off	Just the outer outline. Loose open pieces and inner loops (holes, islands) are skipped, so each sketch is one clean profile to extrude. The summary still says how many were skipped.
Sections	One	One : a single sketch on the plane. Stack (count + direction) : several sketches on offset planes. X-Ray (start → end) : a stack between two picks, like the app's X-Ray.
Spacing	0.2 mm	Stack and X-Ray: the distance between sketches.
Sections (count)	10	Stack only: how many sketches, 2 to 400.
Direction	Along the plane normal (+)	Stack only: along the plane's normal (+), against it (–), or both sides of it.
Start / End	—	X-Ray only: where the stack starts and ends. Pick a construction plane or flat face parallel to the slicing plane, or click on the mesh. A click on a flat mesh face across the slice direction snaps to that face (just inside the material, so the cut catches it). A click anywhere else takes the height of the spot clicked. The end is always included, and start past end is fine.
Extrude each slice	off	Stack and X-Ray: each sketch's profile is extruded to the next slice and joined into one body, a stepped solid. Use this instead of Loft, which usually fails between two complex profiles. With Outline only ticked, only the outer shape is extruded. Without it, inner loops are filled as well. At 0.5 mm spacing the steps are hard to see.
<i>(info line)</i>	—	X-Ray only: how many slices your picks and spacing make, and between which heights.
Hide mesh after	off	Hides the mesh body when it is done, so you can see the result.

The plane in X-Ray mode. If you leave **Plane** empty, the slices run parallel to the Start plane (or the End plane). You need a Plane only when both Start and End are clicks on the mesh.

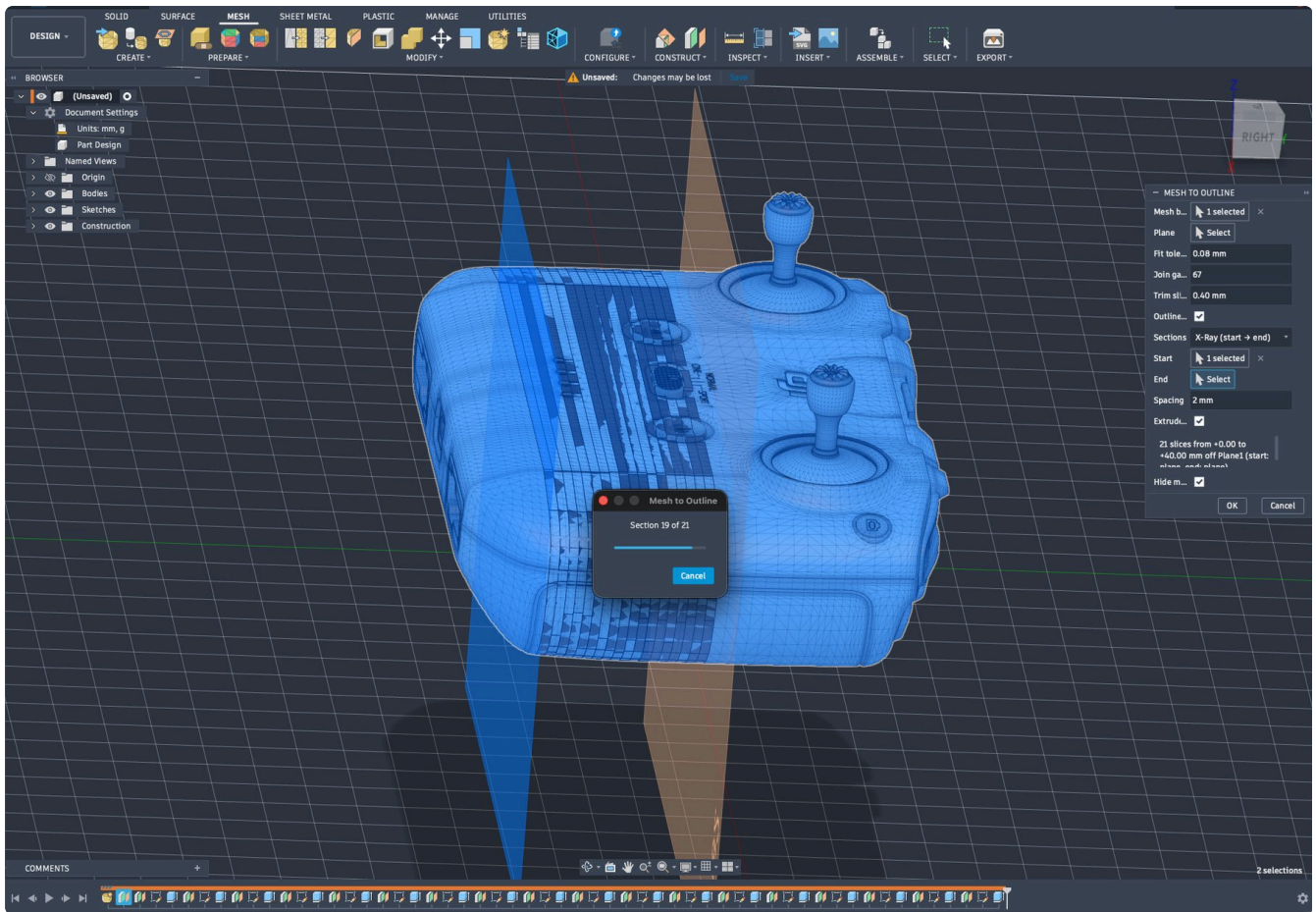
7.4 Walk-through: from mesh to solid with X-Ray

This is a real run on a scanned controller: 21 slices, 2 mm apart, then extruded into one body.

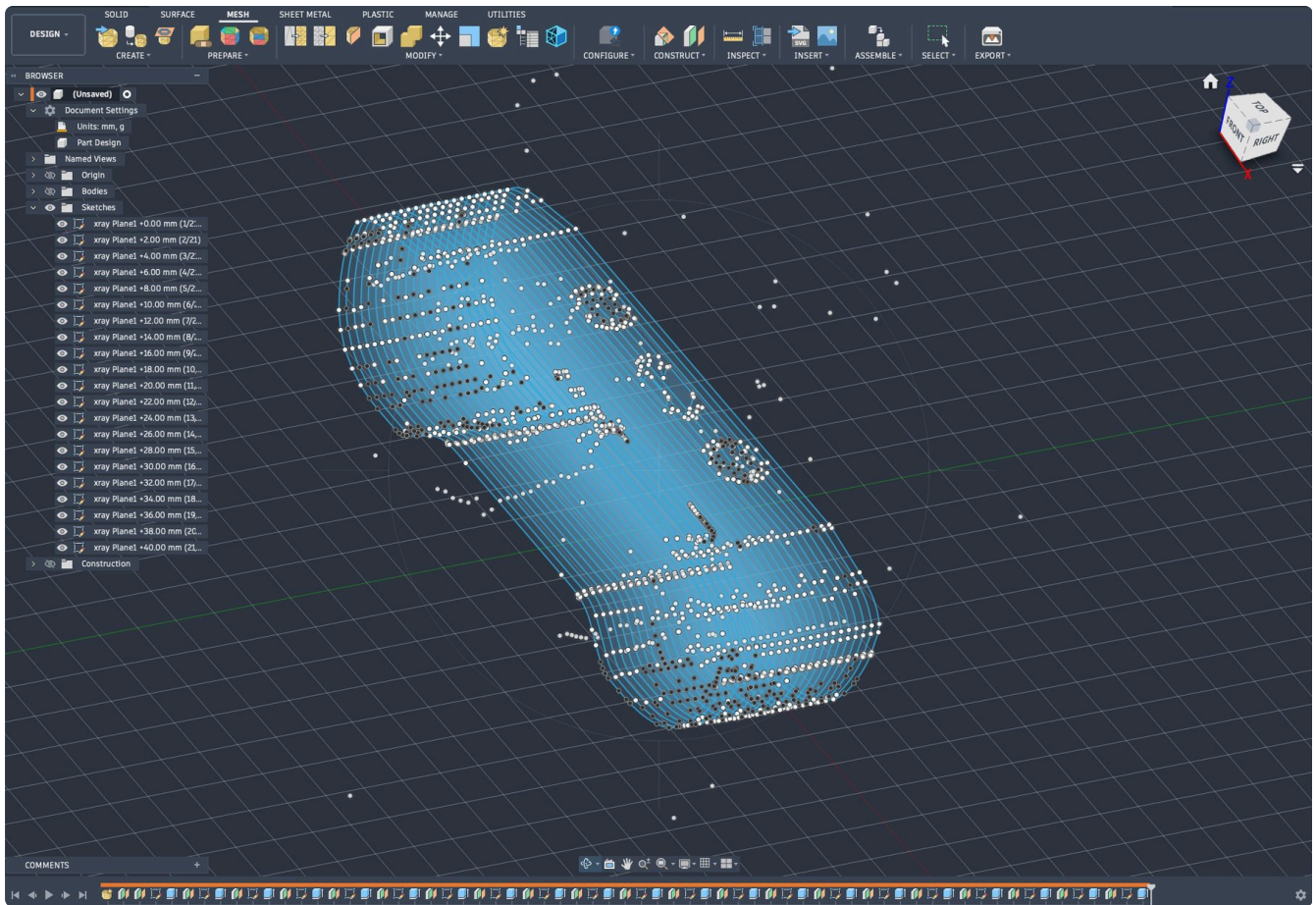
1. Pick two planes. Choose the mesh body, set **Sections** to **X-Ray (start → end)**, and pick a Start and an End. Set the **Spacing**. The info line says how many slices that makes (here 21, from +0.00 to +40.00 mm). Tick **Extrude each slice** if you want a solid, then click **OK**.



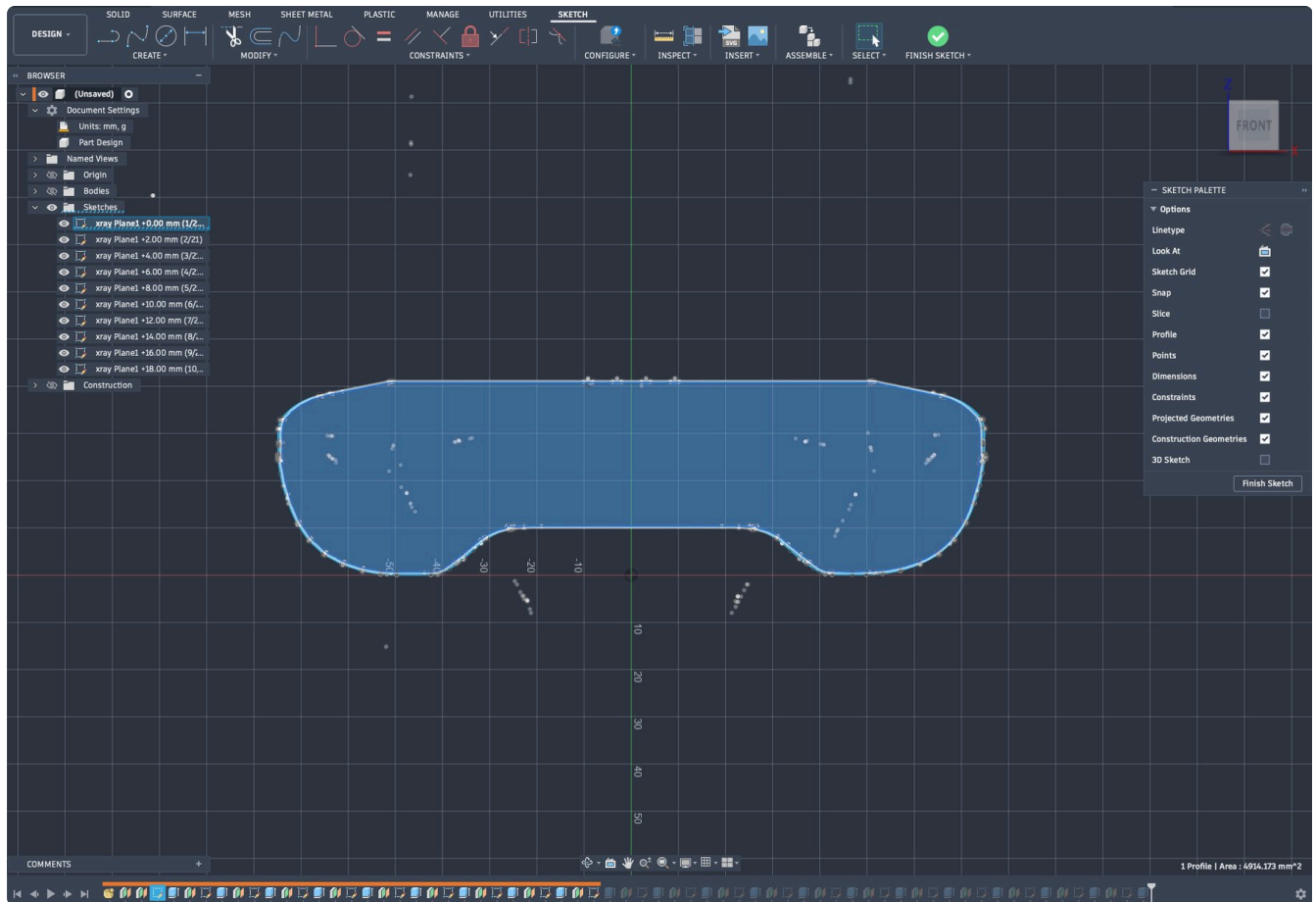
2. Watch it trace. Each slice is cut and fitted in turn. A progress dialog counts the sections. **Cancel** stops it after the current one.



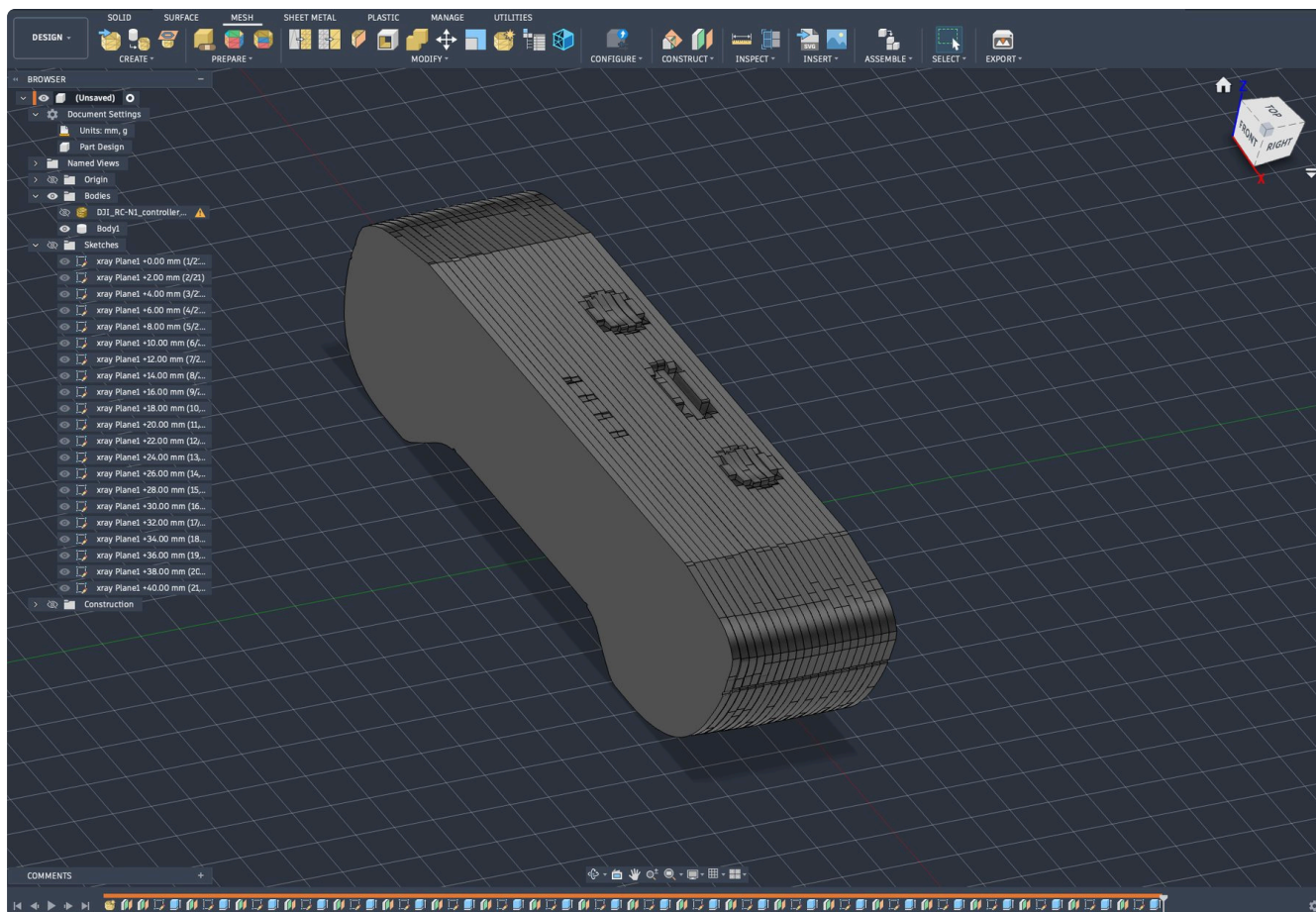
3. One sketch per slice. Every slice lands as its own sketch on its own plane, listed in the browser from first to last and named like `xray Plane1 +4.00 mm (3/21)`. When it finishes, a message box lists each sketch with what it holds (outlines, open curves, lines, arcs, circles, splines, gaps joined, worst miss).



4. A closed profile, every time. Open any sketch: it is one closed outline that Fusion sees as a profile, ready to extrude, offset or dimension.



5. A solid you can design on. With **Extrude each slice** ticked, the stack becomes one stepped solid body in the timeline, next to the mesh.



A single section takes a second or two on a 300,000-triangle mesh. The sketches go in the root component. A mesh inside a component is read in root coordinates, so the curves land on the mesh either way.

7.5 Messages you may see

Message	What to do
<i>Empty cut skipped</i>	The plane misses the mesh there. Fusion's own section sketch would be empty too.
<i>WARNING: the outline did not close in N slices</i>	The mesh has holes wider than Join gaps at those slices (a vent grille's slots, say). Those outlines stay open, Outline only drops them, and those slices get no slab, which leaves a groove in the solid. Raise Join gaps to the value the warning gives and run again.
<i>WARNING: N slabs could not be extruded</i>	The sketch was drawn, but Fusion found no profile to extrude there. The per-slice list says why.
<i>numpy could not be imported / Fusion's Python is now X.Y</i>	Fusion updated its Python. Install the matching add-in version.

7.6 STL to Solid add-in (preview)

A second add-in, **STL to Solid**, sits next to Mesh to Outline. It does the face-group conversion (section 3.1, route 2) inside Fusion: it fits planes, cylinders, cones, spheres and tori to a mesh body, builds each one slightly oversized as a surface, and lets Fusion's Boundary Fill keep the material. It is an early preview. It works on clean, CAD-like meshes, and refuses open shells and scans.

Setting	Default	What it does
Mesh bodies	—	The mesh bodies to convert.
Fit tolerance	0.08 mm	How far a triangle may sit off a fitted surface.
Expand surfaces	1.0 mm	How far each surface is grown past its region, so neighbours overlap for Boundary Fill.
Max regions	200	Bodies with more fitted surfaces than this are refused.
Keep tool surfaces	off	Keeps the surface bodies (hidden) after the fill, for checking.
Run Boundary Fill	on	Untick to only create the surfaces and run Boundary Fill yourself.
Hide mesh after	on	Hides the mesh body when a solid was made.
Diagnose rejected tools (slow)	off	If Fusion refuses the fill, finds which surface it rejects.

If the log says *Fusion could not compute the cells*, try Expand 0.5 or 2 mm, or use Mesh → Solid in the app instead.

8. Reading the report

After a Mesh → Solid or Sliced Loft run, the rail scrolls to the report.

PRISMATIC SOLID

Clean BREP with true planes and cylinders — every gate below passed.

CHECK	MEASURED	LIMIT	
Surface deviation p95 (mesh → solid)	0.019 mm	≤ 0.25 mm	PASS
Surface deviation max (mesh → solid)	0.060 mm	≤ 0.26 mm	PASS
Surface deviation p95 (solid → mesh)	0.018 mm	≤ 0.25 mm	PASS
Surface deviation max (solid → mesh)	0.058 mm	≤ 0.26 mm	PASS
Bore deviation p95 (5 bores)	0.015 mm	≤ 0.1 mm	PASS
Volume error	0.08 %	≤ 2 %	PASS

OUTPUT STEP

1,644 triangles → 23 faces

BREP faces	23
Surface types	15 planes, 8 cylinders
Mean deviation	0.005 mm
Worst point at	(-60.91, -3.57, 1.97) mm
Solid volume	3,688 mm ³
File size	85.4 KB

Download STEP

Download CadQuery script (.py)

Download Fusion 360 script (.py) — experimental

The script rebuilds the recognised sketches and extrudes as an editable program — change a radius or height and re-run it to get a new STEP.

The result badge, the checks table and the output summary for the servo bracket.

Input mesh. The file as read: triangles, vertices, watertight or not, bodies, bounding box, edge lengths, surface area, volume and file size.

INPUT MESH · SERVO_BRACKET_1.STL	
Triangles	1,644
Vertices	818
Watertight	yes
Bodies	1
Bounding box	19.26 × 24.96 × 33 mm
Edge length min / mean / max	0.042 / 2.188 / 6.340 mm
Surface area	2,443 mm ²
Volume	3,685 mm ³
File size	80.4 KB

Result badge. Which route made the solid:

Badge	Meaning
Prismatic solid	A clean extrusion with true planes and cylinders. Every check passed. Scripts are offered.
Face-group solid	Surface regions fitted with real planes, cylinders, cones, spheres and tori.
Sliced loft solid	The sliced loft. Checks are shown for information.
Faceted solid	No clean fit passed. The solid is exact and closed, but made of facets.

Bodies table (multi-body files). One row per body: its triangle count, which route it took and details.

Checks table. Each check with the measured value, the limit and **PASS** or **FAIL**. Deviation is measured both ways: from the mesh to the solid and from the solid to the mesh.

Output STEP. The headline shows the reduction, for example 1,644 triangles → 23 faces. Below it: face count, surface types, mean deviation, where the worst point is, the solid's volume and the file size.

Surfaces, not solids. If the bodies you picked were not closed, the STEP holds surfaces instead of a solid. You can measure them and snap to them, but you cannot cut or join them.

9. Using the results in Fusion 360

Importing a STEP

In Fusion 360 choose **File** → **Open** → **Open from my computer**, or drag the `.step` file into the Data Panel. The part arrives as a solid body with real faces you can sketch on, measure and constrain.

Running a downloaded script

All the Fusion scripts (Mesh → Solid, Sliced Loft, X-Ray and Blueprint) run the same way:

1. Put the `.py` file in an **empty folder of its own**.
2. In Fusion 360: **Utilities** → **Add-Ins** → **Scripts and Add-Ins**.
3. Click **+**, choose that folder, then click **Run**.

For Blueprint, run it in a **new parametric design**. Longer scripts show a progress dialog with a Cancel button. For more detail, open **View** → **Show Text Commands**.

10. Troubleshooting

Problem	What to do
The part is 10× (or 25.4×) too big or small	Change The file's numbers are in , or use ...then multiplied by (for example 0.1). Watch Longest side as converted .
I got a Faceted solid instead of a clean one	Open the checks table to see which limit failed. Try a slightly larger Fit tolerance or Surface deviation (max) . For smooth organic shapes, try Sliced Loft .
Conversion runs out of memory (<i>oom</i>)	Tick fewer bodies. Each body converted at the same time can use several gigabytes.
Conversion <i>crashed</i>	The geometry kernel gave up on one face. Pick fewer bodies, raise the tolerance, or try Sliced Loft.
The result is surfaces, not a solid	The picked bodies are not closed (marked open). Pick closed bodies, or repair the mesh first.
A slice outline is broken into pieces	The mesh is leaky. Raise Join gaps up to .
Thin slivers or hairpins in the outline	Raise Trim slivers up to a little, but keep it below the part's thinnest real feature.
A thin rib or tab vanished from a slice	Trim slivers up to is too high. Lower it, or set it to 0 on a watertight mesh.
X-Ray says too many slices or over the time limit	Raise the spacing, narrow the start and end planes, or tick Outline only .
Sliced Loft turns a sideways hole into a trough	That is a limit of slicing. Use Mesh → Solid for machined parts.
Blueprint says a key is needed	Paste an API key for the chosen provider, or pick a local <i>Custom</i> server.
Blueprint misread a number	Fix it in the Parameters table and click Rebuild . No new read is needed.
Mesh to Outline left a groove in the extruded solid	Some slices did not close. Raise Join gaps up to to the value the warning gives.
A red banner appears at the top	Read the message. Open details for more, then dismiss it with x . Your file and settings stay loaded.

11. Quick reference

File types

Kind	Extensions	Opens in
Mesh	.stl .obj .ply .off .3mf .glb .gltf	Mesh → Solid, Sliced Loft, X-Ray
Drawing	.jpg .png .webp (or paste from the clipboard)	Blueprint

Default settings

Setting	Default
Units / multiplier	mm / 1
Fit tolerance	0.08 mm
Surface deviation p95 / max	0.25 mm / 0.26 mm
Bore deviation (max)	0.10 mm
Volume error	2 %
Faceted reduce tolerance	0.05 mm
Slice spacing (Sliced Loft, X-Ray, Mesh to Outline)	0.2 mm
Join gaps up to	2.5 mm
Trim slivers up to	0 mm
X-Ray limits	500 slices, about 4 minutes of fitting per download
Mesh to Outline stack	10 sections (2 to 400)
Blueprint effort	high

What each tool gives you

Tool	STEP	Fusion 360	Other
Mesh → Solid	Yes	Script: parametric timeline (prismatic) or Boundary Fill (face-group)	CadQuery script (prismatic)
Sliced Loft	Yes	Script: section sketches + Loft per run	—
X-Ray	—	Script: one sketch per slice, optionally extruded into slabs	—
Blueprint	Yes	Script: User Parameters, sketches, extrudes	3D preview
Mesh to Outline (add-in)	—	Sketches drawn straight into the design, optionally extruded into one body	—

stlToSolid 0.5.7 · Licensed under PolyForm Noncommercial 1.0.0 · Fusion 360 is a trademark of Autodesk, Inc. stlToSolid is not affiliated with Autodesk.