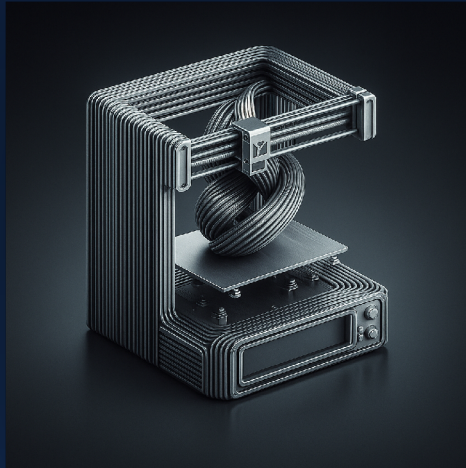




Additive Manufacturing Software Suite

Step-by-step user guide

One all-in-one suite: activate, repair, support, slice and cost a build — module by module, straight from the working application interface.

STL Repair (Free)

Support + SLM / EBM

DED — Wire & Powder

FFM — Filament & Pellet

Concrete Printing

ERP & Cost Reporting

CONTENTS

Introduction and document map.

This user guide walks through the ADDITIVEELITE suite module by module, from activation to a finished, costed build. It covers the free STL Repair tool, the Support Generator (CAD tools, settings and large-part handling), the AI agent that can drive supports and slicing in plain language, the slicer for each manufacturing technology (SLM/EBM, DED, FFM, Concrete), the ERP cost-reporting module, and how to verify a build before printing.

GETTING STARTED

Getting started	3
Workflow overview	4

SUPPORT GENERATION

Load, arrange and prepare parts	7
Mesh — refinement and decimation	8
3D Support Generator	9
Settings — spacing, tips, layout	10
Support type and spacing in detail	11
Display and Base tabs	12
Base Support in detail	13
CAD tools	14
Cut tools and the three views	15
Modify menu, Repair / Clean	16
Lattice — hollow and fill	17
Text Label — engrave text	18
Large parts — split + union	19
Quick reference	21

SLICING BY TECHNOLOGY

SLM / EBM Slicer	22
DED Slicer	23
FFM Slicer — filament and pellet	24
Concrete Slicer	25
Scan strategy — all slicers	26
ERP — manufacturing and cost report	27

CLOSING

Solution summary	29
------------------	----

STL REPAIR (FREE MODULE)

STL Repair Tool	5
Split + Multi-Core Repair and Union	6

AI AGENT

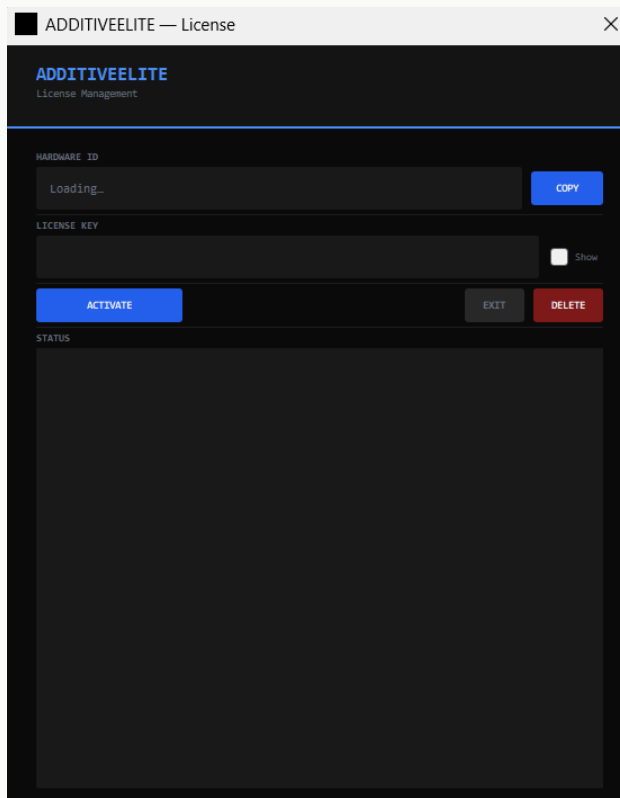
Control supports and slicing in plain language	20
--	----

VERIFICATION

Preview before you print	28
--------------------------	----

Activate the suite and get to know the six modules.

ADDITIVEELITE is an all-in-one modular additive-manufacturing suite: six modules share one geometry engine, so a part moves from repair to supports to slicing and costing without leaving the application. Each module opens as its own window; every module except the free STL Repair Tool starts with a license check.



License window on first launch, before activation — Hardware ID still loading, License key field empty.

- 1 **Hardware ID**
Shows “Loading...” while it is read, then fills in automatically (“Hardware ID ready” in the status log). Press COPY to copy it.
- 2 **License key**
The field starts empty. Paste your annual activation key; tick Show to reveal the characters.
- 3 **ACTIVATE**
Validates the key; the log shows the expiration date (later starts also list the days remaining), then “Ready to launch”.
- 4 **EXIT / DELETE**
EXIT closes the window; DELETE removes the current activation. Every module also has a License menu.

The six modules

Module	Process / feedstock	Axes	Use it to
STL Repair Tool · free	Any mesh	—	Check a model against 10 status tests and repair it; optional multi-core split repair for very large meshes.
3D Support Generator + SLM/EBM Slicer	Laser / electron-beam powder bed	3	Load STL or 3MF, nest, detect overhangs, generate and union supports, edit with CAD tools, slice. AI Agent included.
DED Slicer	Laser / e-beam wire or powder; WAAM (wire arc)	3/4	Slice for directed energy deposition; B axis for belt systems; arc-power G-code for WAAM.
FFM Slicer	Filament, pellet	3/4	Slice large or changing-table-size polymer parts; Filament, Pellet (virtual axis) and Pellet (direct volumetric) extrusion modes.
Concrete Slicer	Concrete	3	Large-format wall, infill, foundation and top passes with four infill strategies.
Manufacturing / Cost Report (ERP)	SLM, EBM, DED, FFM, concrete	—	Project documents, process data, costing, billing and reports; presets per technology.

WORKFLOW

From raw STL to a sliced, costed build.

Pick the route that matches your process. The same interface conventions repeat across every slicer, so once you know one, the others feel familiar.

METAL POWDER BED



WIRE / POWDER DED, FILAMENT / PELLET FFM, CONCRETE



ERP — manufacturing report and cost calculator across SLM, EBM, DED, FFM and concrete jobs

Common interface pattern

Area	What you find there
3D / layer view	Viewport with Iso · Top · Front · Side buttons and visibility toggles (Mesh, Paths, Plate, Plane) in the DED, FFM and concrete slicers. Left-drag rotates, right-drag zooms, middle-drag or Shift + left-drag pans, click selects a part (gold).
Layer slider	Scrub through sliced layers at the bottom of the view (or use < / >); the readout shows the layer number plus scan angle (DED), Z height (FFM) or both (Concrete).
Right-hand panel	Build volume, parts list (Add STL / 3MF, Reset All), part position and rotation (Center, Reset Rot), slicing orientation and process parameters.
Action buttons	SLICE ALL (or SLICE MODEL), then Export G-code — or Export CLI in the SLM/EBM slicer. Re-slice after any change: the preview always shows the last completed slice.
Status bar	Layer, contour and path counts after slicing, plus a LICENSED indicator. Quick Help in each module gives context help (F1 in the ERP).

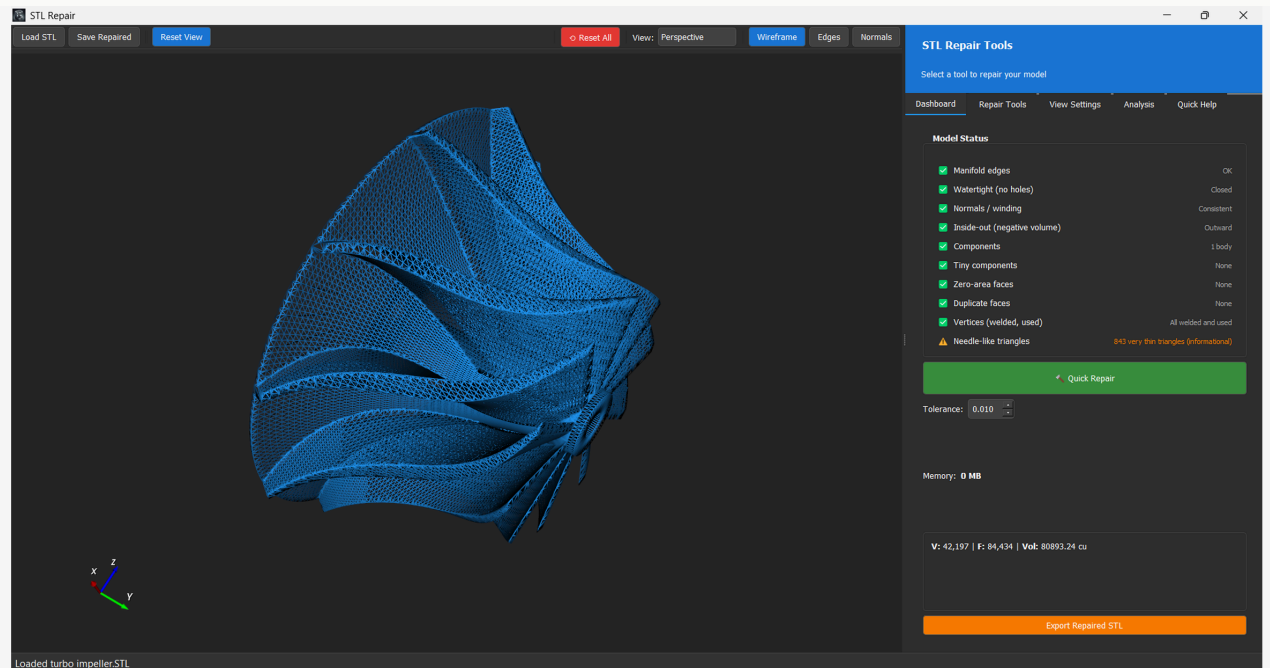
Performance decisions behind the engine

Mechanism	What it does
Multi-core parallel processing	Support generation, union, CAD tools, Simplify / Refine and the loading of large files run in the background on all CPU cores, so the window stays responsive (DED, FFM and Concrete slicing: see note below).
Shared memory optimization	Mesh data is held in shared memory rather than copied per worker process.
Disk-backed processing	Sliced layers are written to disk straight after slicing and cached on demand, so RAM stays low for very large, many-layer parts. Temporary layer files are removed automatically on exit.
Reliable large-scale union	A multi-stage fallback (several union methods tried in turn) keeps model-and-support union working on complex or imperfect geometry. Optional add-ons that are not installed are skipped automatically.
Adaptive resource scaling	Worker count and strategy adjust to available CPU and RAM; G-code export is RAM-aware. A recent CPU / RAM / GPU gives the best results on large models.

Note: DED, FFM and Concrete slicing switches to parallel mode automatically from 20 layers, but only if at least 2 workers and enough free RAM are available; otherwise it runs sequentially.

STL Repair Tool.

Check a mesh and fix it before it goes to supports or slicing. Works on the whole model in one pass or as a multi-part split repair for very large meshes.



STL Repair — sample turbo impeller loaded; Model Status checks and Quick Repair on the right.

1 Load the model

Press Load STL in the top bar. Large files are read in chunks with a vertex weld, so the window stays responsive; the progress dialog lets you cancel. Vertex, face and volume counts appear in the panel.

2 Read Model Status

The Dashboard tab lists ten checks: Manifold edges, Watertight (no holes), Normals / winding, Inside-out (negative volume), Components, Tiny components, Zero-area faces, Duplicate faces, Vertices (welded, used) and Needle-like triangles. Hover a status pill for a short explanation.

3 Repair

Use Quick Repair on the Dashboard (Tolerance field, 0.010 in the sample) or the individual buttons in the Repair Tools tab. Every operation is validated and rejected if it empties the mesh or loses over 95% of the faces (details below).

4 Inspect and verify

Switch the view (Perspective, Front, Top, Right, Isometric) and toggle Wireframe, Edges and Normals. Re-check the status pills and the Analysis tab statistics after each repair.

5 Save

Press Save Repaired in the top bar or Export Repaired STL in the panel; the binary STL is written with live progress.

Good to know

- **Fix All / Quick Repair** runs only Fix Normals → Fix Non-Manifold → Fix Intersections → Fix Holes → Remove Small Parts. Normals, Non-Manifold and Holes are skipped when the check finds nothing to fix; Intersections and Remove Small Parts always run. Reduce Noise and Simplify are **not** part of Fix All — run them separately.
- **Validation.** A step is rejected if it returns an empty mesh, removes more than 95% of the faces or (topology steps) increases the open or non-manifold edges. A single button then leaves the mesh unchanged and shows an error dialog; inside Fix All only that step is rolled back, with a warning. A bounding-box change only prints a console warning. Simplify and Increase Detail are not validated this way.
- **Tolerance** is the Remove Small Components threshold: 0.01 removes components smaller than 1% of the largest one. **Apply Repair** (Repair Tools tab) runs Fix All with the Parameters box values and uses Small %.
- **Repair Tools tab:** Mesh Resolution, Topology Repair and Surface Quality groups, Apply Repair and the Parameters box. **Undo Last Repair** steps back; **Restore Original** returns to the loaded file and can itself be undone.
- **Large meshes:** above about 150,000 faces the heavy status checks run in the background and show “Calculating...”. Self-intersections are not tested in Model Status. The model loads as wireframe by default; View Settings and Analysis tabs hold display options and statistics.

Split + Multi-Core Repair and Union.

Optional. Tick Split + Multi-Core Repair + Union in the Parameters box and choose the part count; whichever repair you run next then goes through three stages.

Stage	What happens
1 • Split	The model is cut into the chosen number of parts (2 up to the CPU core count). Axis: Auto (longest), X (default), Y or Z. Each cut tries a primary and a backup method; a cut that fails both is skipped (one fewer part) and named in the warnings dialog.
2 • Repair	Every part is repaired in parallel in a shared, CPU- and RAM-aware pool with the operation you selected; a failed part is kept unrepaired and a warning is shown.
3 • Union	Repaired parts are unioned back with increasingly thorough methods, tried in turn. A part that cannot be unioned after every method is left out rather than patched in badly, and is named in the warnings dialog.

Seam overlap — Retry Overlap / Step / Tries

Parts that touch exactly at the cut line can confuse the union, so every seam needs a tiny overlap. Retry Overlap is the first distance tried (default 0.01 mm), Step is added after each failed attempt (default 0.01 mm) and Tries is the number of attempts (default 3: 0.01 → 0.02 → 0.03 mm). If models routinely fail to union, raise Tries or the starting overlap.

Continuing, retrying and undoing

- **Continue on the same parts.** After a Split + Union repair the repaired parts are kept. The next repair (checkbox still on, Parts and Axis unchanged) refines the same parts instead of re-splitting the unioned model. The cache is cleared when you load a new file, use Reset All, Undo Last Repair or Restore Original, change Parts or Axis, or run a repair with the checkbox off.
- **Retry Union Only.** If every part repaired but the final union failed at every overlap distance, a Union Failed dialog appears and this button (Repair Tools > Parameters) is enabled. Adjust Retry Overlap / Step / Tries and press it to repeat only the union — the repair pass is not repeated.
- **Undo Last Repair and Restore Original.** Undo keeps the last 3 mesh states; Restore Original returns to the file as loaded and can itself be undone. Reset All cancels running work and clears the model.

Recommended repair order

Fix Normals → Fix Non-Manifold → Fix Intersections → Fix Holes → Remove Small Parts. This is exactly what Fix All runs, in this order. Reduce Noise and Simplify are not part of Fix All; run them separately when needed.

Simplify and Increase Detail

Simplify reduces the face count to 1–100% of the original (default 50%) with DecimatePro or Quadric; the result is not repaired automatically, so run the repair steps afterwards. **Increase Detail** subdivides the mesh with 1–5 iterations (Linear or Butterfly). Neither step is validated like the repair operations. Restore Original reverts either.

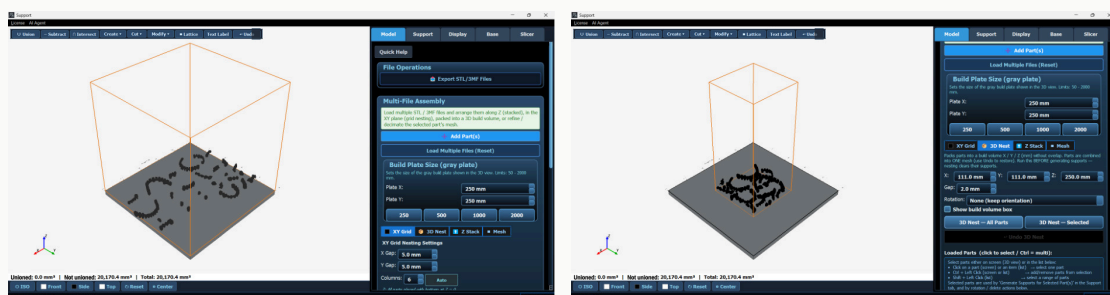
Load, arrange and prepare parts.

The Model tab loads STL or 3MF files, lays parts out on the plate, nests them and prepares the mesh before supports are generated.

Control	What it does
Volume info bar	Strip above the Iso / Front / Side / Top bar showing Unioned, Not unioned and Total volume (mm ³); it updates automatically. Not unioned includes parts not yet unioned plus supports not yet unioned; Unioned counts parts that went through a union step (Union, Split + Auto Support + Union). Z-stack and 3D Nest results stay Not unioned because they are simple concatenations. It shows "Volume: -" when the scene is empty.
Load Multiple Files (Reset)	Clears the whole scene (parts, supports, base plate), loads one or more STL / 3MF files and arranges them in an XY grid. Resets the camera.
Add Part(s)	Adds STL / 3MF files to the existing scene without clearing anything; supports, rotations and base support are kept. New parts are placed to the right with their bottom at Z = 0 and the camera is kept.
STL / 3MF files	Both formats load and behave exactly the same afterwards. 3MF files are converted to mm, a multi-object 3MF becomes one part, and colours / materials are ignored. Loading shows a progress bar and can be cancelled.
Export Model (STL / 3MF)	Enabled after a model is loaded or a union is completed. One item: normal Save As with an STL / 3MF filter. Several parts: choose STL (one file per part), 3MF (one file per part) or 3MF single file (all parts as separate objects at their current positions). Cancelling removes partial files.
Build Plate Size (gray plate)	Under the load buttons. Sets the size of the gray platform in the 3D view with Plate X and Plate Y (50 to 2000 mm): type a value and press Enter, or use the quick buttons 250 / 500 / 1000 / 2000. Ruler marks start from 0 at the plate's left / front edge and adapt to the size. Display only — parts are not moved or limited, and the 3D Nest volume has its own X / Y / Z.
Layout tabs	Inside Multi-File Assembly: XY Grid 3D Nest Z Stack Mesh .
XY Grid / Z Stack	XY Grid: X Gap, Y Gap, Columns (Auto Cols = $\sqrt{\text{part count}}$), Apply XY Grid. Z Stack: Apply Z Gap for all or selected parts, then the stacked parts are automatically combined into one list entry (placed in one mesh, not welded); Undo Z-Stack Merge restores the originals — only the most recent one. Drag a part in the view to move it in XY (Free XY / X only / Y only).
Selecting parts	Click on screen or in the Loaded Parts list; Ctrl + click adds or removes; Shift + click selects a range in the list. Screen and list stay in sync. Delete or Backspace (or the delete button) removes the selection and its supports.
Rotating parts	Rotate the selected part with the X / Y / Z buttons and a chosen angle; the part keeps its bottom on the plate. Reset Rotation restores the original orientation.

3D Nest

Packs parts into an X / Y / Z build volume (orange wireframe box, centred in XY and starting at Z = 0) without overlap, using each part's bounding box, largest first, lowest Z then Y then X. Set Gap and Rotation: **None** keeps orientation, **Rotate about Z only (90°)** swaps X and Y, **Any 90° orientation** tries all six axis orderings (densest, but overhangs change). Run it for All Parts or Selected. Parts that do not fit are placed outside the volume on the +X side and reported. After nesting the moved parts are combined into one entry (nest3d_merged_N_parts), which is not a union.



XY Grid with 27 files loaded (left) and 3D Nest packing the same parts into a 111 × 111 × 250 mm volume (right).

Nest before you generate supports

- Nesting concatenates the moved parts into one mesh and deletes their supports (you are asked first); Undo 3D Nest does not restore supports.
- Undo works as a stack — click Undo 3D Nest repeatedly until it greys out to return to the first state. Reset All and Load Multiple Files clear the stack.
- For per-part supports after a nest, undo the nest first.

Mesh — refinement and decimation

Works on the selected part (or the single loaded model) and clears that part's supports, so regenerate them afterwards.

Simplify: ÷2, ÷4, ÷8 reduce the triangle count by about 50 / 75 / 87.5 %. **Reduce:** custom reduction from 5 to 95 %, then Apply Decimation. Simplification is shape-preserving: triangles are removed where they matter least, so edges and the overall volume are kept. **Refine:** x1 (no change), x2 (about 4× triangles), x3 (about 16×). **Passes:** custom refinement with 1–4 passes, then Apply Refinement. **Restore Original** returns to the mesh as first loaded. Both run in the background — you can keep working while they finish. Decimate large meshes before generating — overhang detection gets much faster.

Background processing

Heavy work does not freeze the window. While a job runs, the progress line and the status bar keep you informed, and you can still rotate the view, select parts and read your settings.

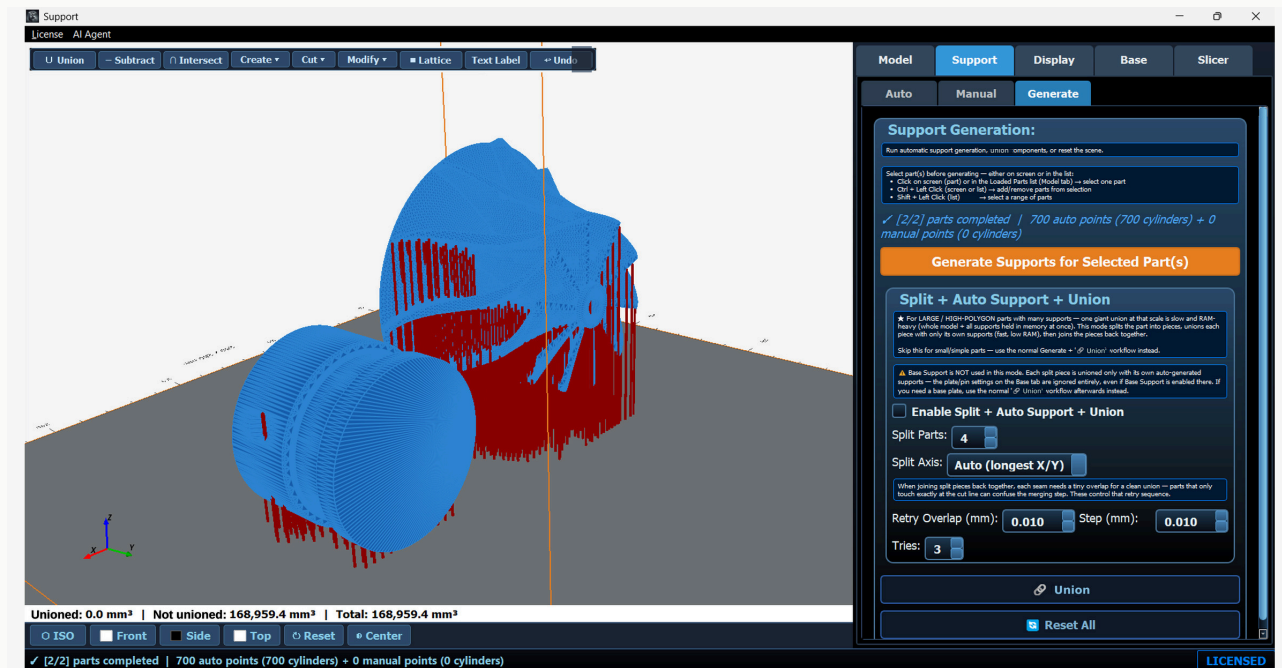
Area	What you notice
Opening large files	Very large meshes are prepared in the background — the progress bar shows “Computing normals...”. Loading can still be cancelled.
Simplify / Refine	Runs in the background; the part is swapped in when the result is ready.
Generate supports	Supports are built and tidied up on all CPU cores; the window stays responsive.
Union	The heavy joining step runs separately and shows its progress while you keep working.
CAD tools	Every operation, including Mirror and Scale, runs in the background with a progress line; if it fails, the part stays unchanged.
Manual supports	Supports snap to the nearest corner point of the mesh, shown as a green dot (see the box below).
Reset All / closing	All background helpers are stopped cleanly and made ready again for the next job.

Manual supports — vertex snap

- Select a part, open the Manual tab and press Add Mode.
- Move the mouse over the underside of the part: a green dot jumps to the nearest corner point that faces downward.
- Ctrl + click places the support exactly at the green dot. No dot? Move closer, or raise Vertex Snap Radius (Manual tab, default 2 mm).
- The first time you use Add Mode on a part it is prepared in the background, so the dot can take a moment to appear. On very coarse meshes, Refine x2 gives more points to choose from.

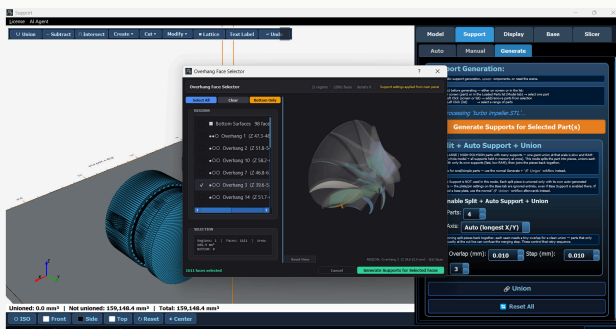
3D Support Generator — load, detect, generate.

Supports are placed automatically from detected overhang faces and run down to the base plate or to the lower face of the part.

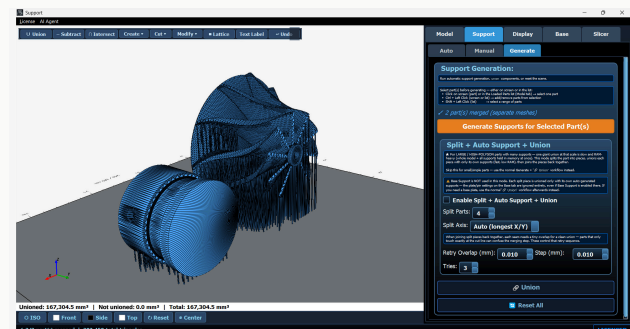


Support tab > Generate — red columns are the generated supports (**700 auto points** in this sample).

- Load and select parts**
 Add files in the Model tab. Select by clicking on screen or in the Loaded Parts list; Ctrl + Left Click adds or removes, Shift + Left Click selects a range in the list.
- Generate**
 Open Support > Generate and press Generate Supports for Selected Part(s). Only the selected parts get supports (parts without overhangs are skipped). A green line on the Generate tab and the status bar report the result; dialogs appear only for errors.
- Pick specific overhangs (optional)**
 Tick Enable Face Selection in the Auto tab. The Overhang Face Selector lists detected regions with checkboxes — Select All, Clear or Bottom Only — then Generate Supports for Selected Faces.
- Edit by hand (optional)**
 Manual tab: select a part first, then Add Mode. Hover over the part — a green dot snaps to the nearest corner point; Ctrl + click places the support there. Delete Mode: Ctrl + click near a support. Clear Manual / Clear Auto / Clear All remove support sets.
- Union**
 Press Union to join model and supports (and the base plate) into one mesh; the footer shows Unioned / Not unioned / Total volume. Reset All clears the scene after confirmation and also returns all settings to defaults (Base Support off).



Overhang Face Selector — choose regions by checkbox.



After Union — model and supports as one mesh.

Tune spacing, tips and layout.

The Auto, Manual and Base tabs control how supports look and how easily they come off. Defaults are listed where the Quick Help states them; the sample values are the ones visible in the screenshots.

Setting	What it controls
Overhang Angle	Faces steeper than this angle receive supports (0°–90°, default 45°; lower = more supports). If nothing is detected, try 30° or 20°. Highlight Overhangs shows detected faces in red.
Min Spacing / Auto Radius	Gap between support points (default 10 mm; 2.0 mm in the sample) and column radius (default 0.4 mm).
Support Type	Top of Support Configuration . Cylinder (default, constant radius), Cone (radius shrinks to Radius × Tip Ratio at the tip) or Neck (full-radius body with a short thin neck under the tip). Used for Auto, Manual and Split + Union supports. Details on the next page.
Tip Ratio / Neck Height	Tip radius as a multiple of the radius (0.3–1.0, default 0.6; the tip is never thinner than 0.15 mm). Neck Height (0.3–10 mm, default 1.5) is the length of the thin neck. Disabled when the chosen type does not use them.
Boundary Offset	Pulls support tips inward from the overhang edge (0–20 mm, default 0.0 mm).
Cylinder Count (1–4)	Single, Dual, Triple (triangle) or Quad (square) supports per point, used exactly as selected — the app never changes the count or the spacing for you. The label follows the Support Type (Cone Count, Neck Count). For very many supports you can pick Dual or more yourself: multi-cylinder groups get a real boolean union at generation time, which makes the final union more reliable at scale.
Cylinder Spacing	Used only when Count is 2, 3 or 4 (Cone Spacing / Neck Spacing for those types). A multiple of the base radius (default 1.5×; allowed 1.0–10.0 for Auto, 1.0–3.0 for Manual) that sets the shift s = radius × Spacing : how far each support sits from the support point (the centre) — not the distance between two supports. Details below.
Auto / Manual Indentation	Positive values push the tip slightly into the part (overlap; default +0.1 mm); 0.0 touches exactly; a negative value leaves a gap that makes removal easier.
Stop Mode / Lower Face Indent	Base: columns run to the build platform. Lower Face: columns stop at the next surface below — ideal for hollow or enclosed models (indent default –0.1 mm).
X / Y Offset	Shifts auto-supports radially from the model centre (positive = outward, negative = inward; kept 1.5 mm inside the bounds). Presets: Inward (–1, –1), Outward (+1, +1).
Vertical Thin Area Supports	On by default. Extra pass for near-vertical fins, blades and thin walls: Angle ≥ (70–89°, default 88°) and Max Thickness (0.2–3.0 mm, default 0.4 mm).
Manual radius / count / snap radius	Independent settings for manually placed supports (count and spacing labels follow the Support Type). Vertex Snap Radius (0.2–20 mm, default 2 mm) sets how far the green dot looks for a corner point; only downward-facing points are offered.

Cylinder Spacing in detail

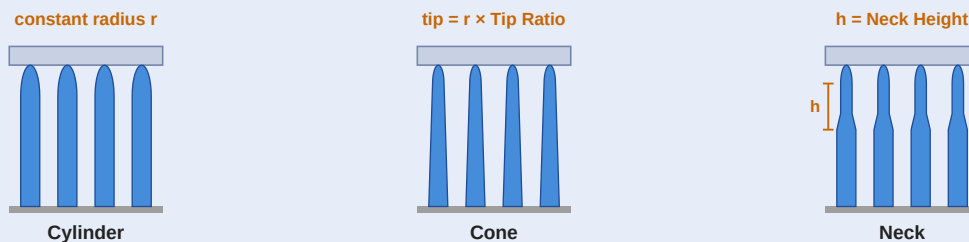
- **Dual** — one support moves left by s , the other right by s : centres end up $2 \times s$ apart.
- **Triple** — three supports s from the centre, 120° apart (triangle): centres about $1.73 \times s$ apart.
- **Quad** — four supports in the corners of a square, s from the centre in X and Y: neighbouring centres $2 \times s$ apart.
- **Example** (radius 0.4 mm, Spacing 1.5): $s = 0.6$ mm. In Dual the centres are 1.2 mm apart and the cylinders 0.8 mm thick, so the gap between them is 0.4 mm.
- **1.0** — Dual and Quad cylinders just touch; Triple cylinders slightly overlap and fuse. **Above 1.0** — they separate; Dual gap with a 0.4 mm radius: $1.5 = 0.4$ mm, $2.0 = 0.8$ mm, $3.0 = 1.6$ mm.
- **Small gap** — stronger, denser support, harder to remove. **Large gap** — separate thin pillars, easier to remove but weaker and taking more room on the surface.
- **Wanted centre-to-centre distance** (Dual): Spacing = distance $\div$ ($2 \times$ radius); for example 1.2 mm centres with a 0.4 mm radius $\rightarrow 1.5$.
- **Cone / Neck** — Spacing always uses the base radius, so the layout of a multi-support group is the same for all three types.

Support Type in detail

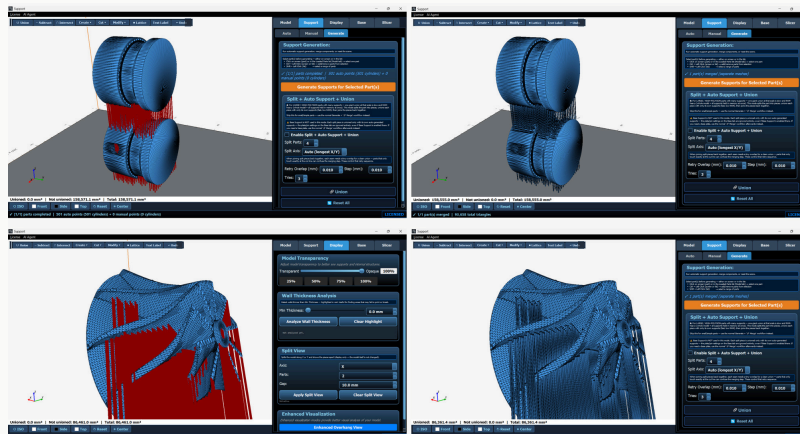
- **Where** — Support tab > Auto > **Support Configuration** (top of the group). The chosen type is used for Auto, Vertical Thin Area, Manual and Split + Auto Support + Union supports.
- **Cylinder** (default) — constant radius from the plate (or lower face) up to a rounded tip. Tip Ratio and Neck Height are disabled. The output is identical to earlier versions.
- **Cone** — the radius shrinks steadily from the full radius at the bottom to **Radius $\times$ Tip Ratio** at the top. Stable base and a small contact mark; good for tall supports.
- **Neck** — full-radius body, then a short thin neck of length **Neck Height** (radius = Radius $\times$ Tip Ratio) right under the tip. It breaks off cleanly and leaves the smallest mark while the body stays strong.
- **Tip Ratio** (0.3–1.0, default 0.6) — the tip is never thinner than 0.15 mm. Example: radius 0.4 mm $\times$ 0.6 $\rightarrow$ tip radius 0.24 mm. To see a clear difference use a larger radius (0.8–1.0 mm) with a ratio of about 0.4.
- **Neck Height** (0.3–10 mm, default 1.5) — if a support is shorter than twice this value, that support is built as a plain cylinder.
- **Names** — the Multi- header, Count and Spacing labels follow the type (Multi-Cone, Neck Count, Neck Spacing...). Reset All returns the type to Cylinder, Tip Ratio 0.6 and Neck Height 1.5.
- **Tip** — very thin tips are fragile: raise the ratio or the radius if they break during the build.

Supports under a square part — 2D side view

Support Type **Neck** Tip Ratio **0.6 $\times$ r** Neck Height **1.5 mm**



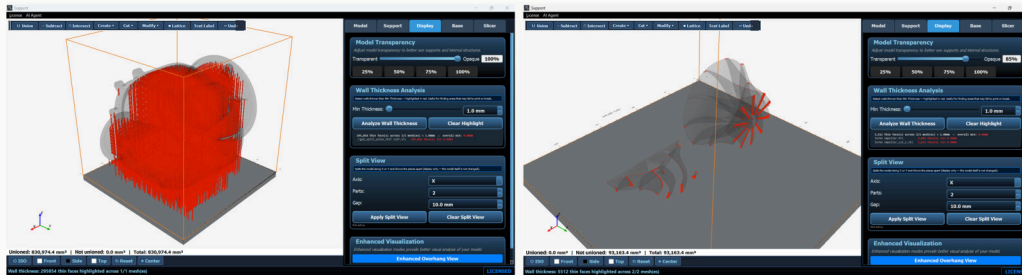
Schematic, not to scale. Orange labels show what the controls change.



Support generation and union, Display and union views — full application windows.

Inspect the part, then set the Base Support.

The Display tab changes how the scene looks without changing the model; the Base tab sets the plate or pins that every part sits on.



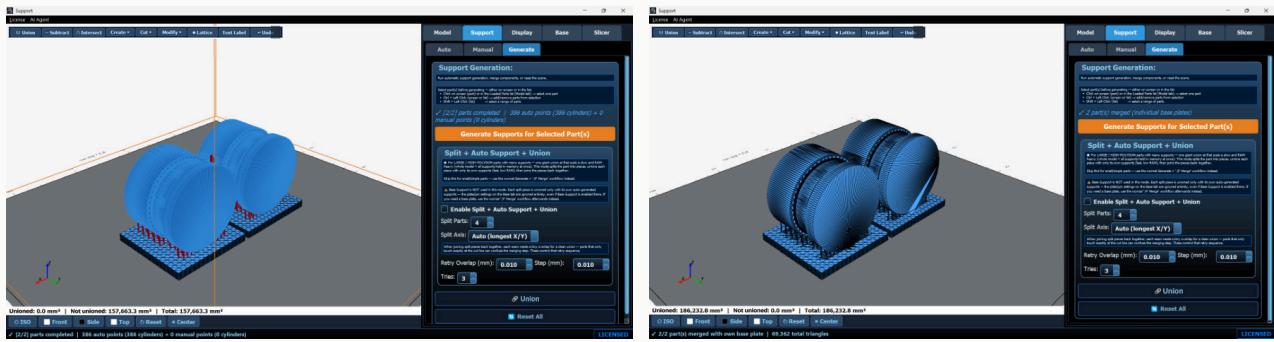
Display tab — transparency, wall-thickness analysis (5,512 thin faces across 2/2 meshes, Min Thickness 1.0 mm) and enhanced overhang view.

Display control	What it does
Transparency (10–100 %)	Opacity of model, all parts and supports together; presets 25 / 50 / 75 / 100 %.
Enhanced Overhang View / Normal View	Highlights every detected overhang face in red, dims the rest and reports the number of faces found; click again to re-scan with the current Overhang Angle. Normal View restores standard shading.
Split View	Cuts every mesh along X or Y into 2–16 equal slabs and shows them apart (Axis, Parts, Gap). Display only. Apply Split View cuts every part, colours each piece differently and hides the original parts and supports while active. The cut is open, so hollow parts and lattice stay visible. Clear Split View restores the scene (rotating, resetting or reloading also clears it); parts that cannot be cut are skipped and listed in the status line.
Analyze Wall Thickness	Marks walls thinner than Min Thickness in red, in any orientation, for every mesh in the scene, with a summary of thin faces and the smallest thickness found. Min Thickness runs from 0 to 2000 mm in 0.2 mm steps (slider or spin box) and starts at 0, where nothing is flagged — choose a value first (the screenshot uses 1.0 mm). Clear Highlight removes the overlay. The highlight also clears automatically when the model is rotated, nested, reset, reloaded or changed by a CAD operation.

Base tab

All Base settings are global: enabling Base Support or changing Type, Thickness, Z Height or Shape affects every loaded part, including parts added later. Each part still gets its own automatically sized, tight-fit plate (or pin array), rendered as one combined base — there is no manual plate size.

Base setting	What it does
Enable Base Support	On or off for all parts.
Type	Normal Base Support (default): solid plate with a Thickness control. Pin Base Support: an array of small cylinders.
Pin settings	Pin Radius, Pin Tip Height (above Z), Pin Stem (below Z), Pin Spacing centre-to-centre (auto-raised if pins would overlap or exceed 10,000).
Plate size	Automatic per part: 1 mm margin, 50–500 mm. (The gray Build Plate Size in the Model tab is a separate display setting.)
Z Height / Shape	Z of the base top surface, shared by all plates (starts at 0 mm; –1 / –0.1 / +0.1 / +1 buttons, or Reset Z to +0.1 mm). The plate is placed 0.3 mm above the Z value, and the gray platform follows both Z and the plate thickness / pin stem. Shape: square or circle.



Base Support on: **386 supports (386 cylinders)** stand on each part's own tight-fit Base Support (left); after Union the scene is one mesh per part, **69,362 triangles in total**, 186,232.8 mm³ (right).

Base Support (plate / pin base) is used in the normal Generate + Union workflow; it is not used in Split + Union.

Prepare the part with CAD tools.

The toolbar in the top-left corner of the Support viewport offers CAD-style editing built for manufacturing preparation, before supports and slicing.



Toolbar above the Support viewport: Union, Subtract, Intersect, Create, Cut, Modify, Lattice, Text Label and Undo.

Tool	What it does	Page
Union	Joins two or more bodies into a single solid.	—
Subtract	First selected part minus all other selected parts. The result replaces the first part (keeping its name and colour); the others are removed from the list.	—
Intersect	Keeps only the volume where the selected bodies overlap.	—
Create (menu)	Cube, Cylinder or Sphere (sizes in mm): added as a new part at the XY origin with its base at Z = 0; it can be dragged, rotated, combined and exported like any loaded part.	—
Cut (menu)	Cut by Plane (Preview or Closed), Cylindrical Hole and Box Cut.	15
Modify (menu)	Hollow, Offset, Convex Hull, Linear Array, Mirror, Scale, Repair / Clean, Split Bodies.	16
Lattice (button)	Hollows a part and fills the inside with a Cubic, BCC or TPMS lattice, with a 3D preview.	17
Text Label (button)	Engraves a text into a chosen face of the part.	18
Undo	Reverts the last operation (up to 20 steps); supports are not restored.	18

Recommended order

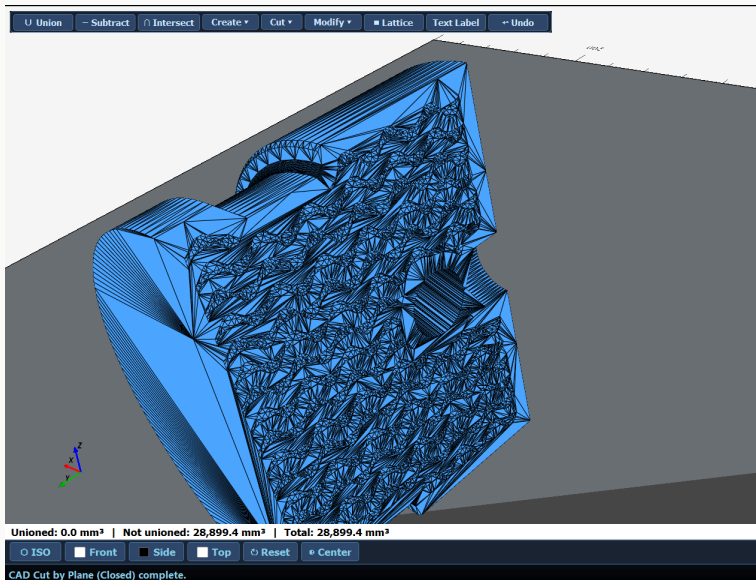
- 1 · **Repair / Clean** — make the mesh closed and clean.
- 2 · **Shape** — Hollow, Lattice, Offset, Scale, Mirror, Array.
- 3 · **Cut and mark** — Cut, Boolean operations, Text Label.
- 4 · **Supports** — Generate, then Union.

Good to know

- Select the part(s) in the Loaded Parts list first; most commands act on every selected part.
- Every CAD operation **deletes the supports**, and Undo does not restore them.
- Boolean, hollow, offset and lattice operations need closed meshes — run Repair / Clean first if a part is rejected.

Cut tools and the three views.

The Cut menu offers four commands. Cut by Plane (Closed) cuts a part with a plane and caps the cut face, so the result is a closed, printable solid.



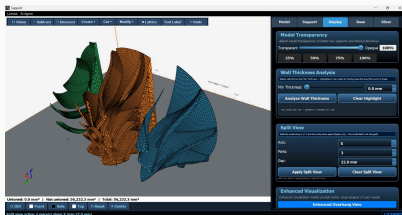
Result of Cut > Cut by Plane (Closed): the cut face is capped and the part is a closed solid. The status bar confirms "CAD Cut by Plane (Closed) complete."

Preview or Closed?

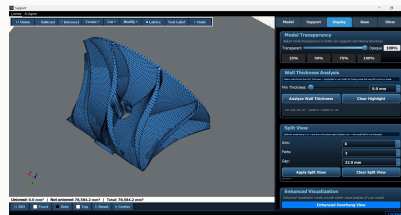
- **Preview** is an open inspection cut: it shows the inside, but the result is not a closed solid. Undo it before Boolean, Hollow or Offset.
- **Closed** caps the cut face. Use it when you want to keep working with the pieces or print them.
- Keep **Above**, **Below** or **Both**; the lower half becomes *name_below*.

Command	Settings and result
Cut by Plane (Preview)	Normal axis X / Y / Z and the plane offset from the part centre in mm (0 = middle). Open cut for inspection; Undo restores the part.
Cut by Plane (Closed)	Same settings as Preview, but the cut face is capped: a closed, printable solid.
Cylindrical Hole	Axis, diameter, depth (0 = through the whole part, measured from the + face) and two offsets from the part centre.
Box Cut	Removes a box of the given size at an offset from the part centre.

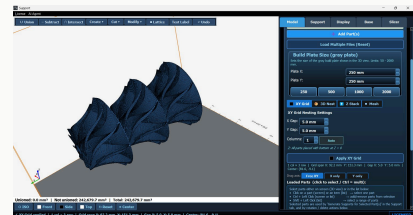
Split View vs Cut vs Split + Union



Display > Split View: the impeller as 3 pieces along X, gap 22.0 mm (display only).



Split View cleared: the impeller as one mesh (78,584.2 mm³).



Model tab > XY Grid: 3 parts in 1 column x 3 rows, X / Y gap 5.0 mm.

How the three views relate

- **Display > Split View** only shows the pieces apart; the model itself is unchanged.
- **XY Grid (Model tab)** only arranges the parts on the plate; to make copies of a part use CAD > Modify > Linear Array.
- **CAD > Cut** changes the part (Undo restores it): Preview is open, Closed is a capped solid.
- **Support > Split + Auto Support + Union** really cuts, adds supports and rejoins the part.

Modify menu — how it works.

The Modify menu changes the shape of the selected part(s) before supports and slicing. Each command applies to every part selected in the Loaded Parts list.

Command	What it does	Settings
Hollow (Shell)	Removes the inside and leaves a wall of the given thickness; the wall must be less than half the part's thinnest section.	Wall thickness (mm)
Offset / Grow-Shrink	Moves every surface outward (positive) or inward (negative) by a fixed distance — useful for fit tolerances.	Distance (mm); negative = shrink
Convex Hull	Wraps the selected part(s) in their convex hull; several parts become one.	—
Linear Array	Repeats the part along one axis, as one joined part or as separate parts.	Axis X / Y / Z; copies (2–100); spacing
Mirror	Mirrors each part about its own centre (a Z mirror keeps the part on the plate).	Mirror direction X / Y / Z
Scale	Scales X, Y and Z independently. Footprint centre stays fixed and the bottom stays on the plate.	Scale X / Y / Z (%)
Repair / Clean	Repairs the part with the same steps as the STL Repair tool, only where problems are actually found (details below).	—
Split Bodies	Separates disconnected shells into individual parts; the largest keeps the original name.	—

Inside the Modify tools

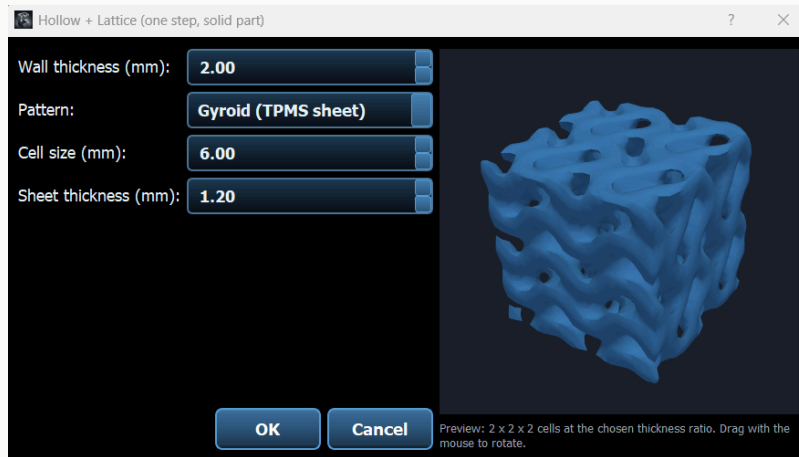
- Select the part(s) in Loaded Parts (Ctrl + click for several), then choose the command. For Union, Subtract and Intersect the first selected part is the base.
- Boolean, hollow and offset operations need closed meshes. If a part is rejected, run Repair / Clean first (and, for very dense meshes, reduce the mesh in Model tab → Mesh → Simplify) and try again.
- After boolean, hollow and similar operations tiny leftover fragments are removed automatically; the part list and volume bar refresh and the camera is kept.
- To check a hollow result use Display > Split View (open cut, part unchanged) or Cut by Plane (Preview), then Undo.
- The AI Agent can run the same operations from plain-language commands.

Repair / Clean in detail

- Steps run in this order: Fix Normals → Fix Non-Manifold → Fix Self-Intersections → Fill Holes → Remove Small Parts.
- Only detected issues are repaired: a step runs only when its problem is found (wrong normals; non-manifold, duplicate or zero-area faces; open edges; small bodies below 1 % of the largest). A part with no issues is left untouched and gets no undo step.
- Self-Intersection repair runs only when non-manifold edges remain after the earlier steps, because there is no cheap way to detect self-intersections.
- Every step is checked afterwards. If a step empties the part, removes almost all of its faces or increases open / non-manifold edges, that step is rolled back and the part continues with the previous result.
- With several parts selected, a part that fails is left unchanged and the others are still repaired; a summary lists unchanged parts and rolled-back steps.
- Self-intersection repair can be slow and memory-heavy on large parts; reduce the mesh first in Model tab → Mesh → Simplify if needed.

Lattice — hollow and fill in one step.

On a solid part the **Lattice** button makes the wall and fills the inside with a lattice in one step. The dialog shows a 3D preview of the chosen pattern that you can rotate with the mouse.



Hollow + Lattice dialog: Gyroid (TPMS sheet), wall 2.00 mm, cell size 6.00 mm, sheet thickness 1.20 mm; the preview shows $2 \times 2 \times 2$ cells.

Dialog settings

- **Wall thickness** — outer wall that stays around the lattice.
- **Pattern** — Cubic grid, BCC, Gyroid, Schwarz-P or Diamond.
- **Cell size** — size of one lattice cell.
- **Strut / Sheet thickness** — strut thickness for cubic and BCC; sheet thickness for TPMS (at most half the cell size).

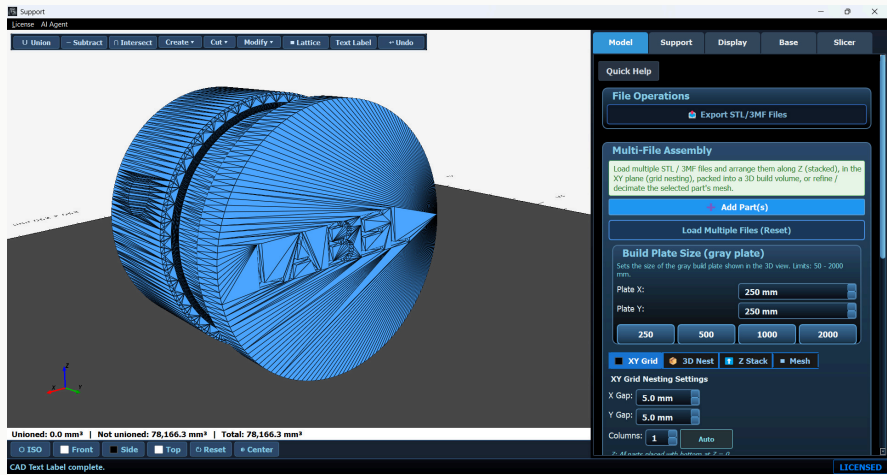
Pattern	What it is	Good for
Cubic grid	Square grid of struts along X, Y and Z.	Simple, predictable infill
BCC	Diagonal struts from each cell centre to its 8 corners.	Stiffer than the cubic grid
Gyroid (TPMS)	Smooth, continuous sheet with no corners or nodes.	SLS / metal; self-supporting
Schwarz-P (TPMS)	TPMS sheet aligned to the cube axes, open straight channels.	Light parts, easy powder removal
Diamond (TPMS)	Denser TPMS sheet that follows the diamond crystal pattern.	Higher fill, higher strength

Lattice settings and limits

- TPMS patterns (Gyroid, Schwarz-P, Diamond) need the scikit-image package.
- Use it on the original, un-hollowed solid part (Undo any earlier Hollow first). Strut thickness must be smaller than the cell size; very thin struts are made thicker automatically.
- If too many cells would be needed, the cell size is increased automatically and you are told the new value. The lattice fills the box around the inner space and is then cut to the part, so cells at the surface can be cut off. The result is one solid inside the original outline.
- It needs a lot of memory: on big parts use a larger cell size and apply it before supports and Union. If the main engine fails, OpenSCAD is used as a backup (slow for TPMS).

Text Label — engrave text into a face.

The **Text Label** button at the end of the CAD bar engraves (subtracts) a text into the chosen face of every selected part — for part numbers, versions or marks.



Text Label result: the text is engraved into the face of the part. The CAD bar ends with the Lattice and Text Label buttons.

Setting	What it does
Text	The text to engrave (one line; basic Latin letters, digits and symbols).
Face	+Z (top), -Z (bottom), +X, -X, +Y or -Y. The text reads correctly when you look at that face from outside.
Text height (mm)	Height of the capital letters.
Engrave depth (mm)	How deep the text goes into the surface; it must be smaller than the part thickness.
Offsets (mm)	Move the text from the face centre: along the text direction and perpendicular to it.
Rotation (deg)	Turns the text in the plane of the face.

Text Label tips

- The text must fit on the face; otherwise a message asks for a smaller height, a shorter text or another offset.
- The text is cut into the bounding-box face of the part; on curved or uneven faces it may only partly touch the surface. Undo restores the part.
- For printing keep the text at least 6–8 mm high and about 0.5–1 mm deep.

Undo, Union and side effects

- **Undo** reverts the last CAD operation; the last 20 are kept.
- Every CAD operation **deletes the supports**, and Undo does not restore them. Do CAD work first.
- **Union has no separate undo step.** Undoing a CAD operation after union returns the part to its state before that operation, so finish CAD work before union.
- If an operation fails on one of several selected parts, that part stays unchanged and the others are still processed.
- If the program closes the calculation on a very large part, restart and try a smaller or simplified part.

Split + Union and the union step.

One giant union of a big model plus thousands of supports is slow and RAM-heavy. Split + Auto Support + Union cuts the part into pieces, generates supports on each piece, unions each piece with only its own supports, then joins the pieces back together.

1 Enable

Tick Enable Split + Auto Support + Union in Support > Generate (below the Generate button). Because this mode generates its own supports, all existing supports (auto and manual) are removed after you confirm.

2 Set the split

Split Parts (2–16; 4 in the sample) and Split Axis: Auto (longest of X / Y), X or Y. Z is not offered because support columns would cross the cut.

3 Set the retry sequence

Each seam needs a tiny overlap for a clean union: Retry Overlap (0.010 mm), Step (0.010 mm) and Tries (3; 1–8), the same logic as in STL Repair.

4 Generate

Press Generate Supports for Selected Part(s). All Auto-tab settings are used exactly as selected and Vertical Thin Area supports are included when enabled; the status line shows "Split+Union complete". Result parts are renamed `unioned_split_union_...`; if a piece cannot be unioned, the original part is left unchanged.



Large part with a high mesh count: Split + Union generated **1,672 support points (3,344 cylinders)**, and the part with all its supports was then sliced. Right: SLM/EBM Slicer view of the unioned part with supports — **846 layers**, layer 615 shown (**2,531 contours, 23,880 paths**).

Large parts with a high mesh count get a very large number of supports — and the part with all its supports can be sliced.

Base Support note

Base Support is not used in this mode — plate and pin settings on the Base tab are ignored. If you need a base plate, use the normal Generate + Union workflow. Skip Split + Union for small or simple parts.

Union (normal workflow)

Union combines model and supports — and, when Base Support is on, a tight-fit base plate per part — into one solid mesh, using a two-stage union with fallbacks so complex or imperfect geometry still unions. With Base Support enabled, each selected part first unions with its own supports and then gets its own plate in a second step; afterwards the global Enable Base Support box is switched off, because the plates are already part of each mesh. Parts made with Hollow + Lattice union like any other part. The volume bar then moves the supports' volume from Not unioned to Unioned. Export as STL or 3MF, or use Send All to Slicer in the Slicer tab (union and finish CAD work first).

Recommended workflow

1 Model tab: set Build Plate Size if needed, then Load / Add parts → 2 XY Grid, 3D Nest or Z Stack → 3 Mesh tab: simplify or refine the selected part → 4 Rotate into print orientation → 5 CAD work (Repair / Clean, union, hollow, lattice...) **before** supports and Union → 6 Wall Thickness check and Split View → 7 Auto tab settings (45° is a good start) → 8 Select parts and Generate (Split + Auto Support + Union for large parts) → 9 Enhanced Overhang View, manual add / delete → 10 Base tab: enable Base Support (Pin recommended) and set Z height → 11 Union → 12 Export or Send to Slicer.

Control supports and slicing in plain language.

The Support module and the SLM/EBM slicer each have an AI Agent menu. Open it (or press Ctrl + Shift + A in the Support module) to dock a chat panel on the right, type what you want, and the agent applies the settings and runs the step.



AI Agent panel docked in the Support module — type a command in the box at the bottom. | AI Agent in the SLM/EBM slicer — same panel, e.g. “load these STL files, slice, and export”.

1 Open the panel and add your key

Choose AI Agent in the menu bar. On first use, if no key is found, paste your API key into the field at the top of the panel and press Save (or Enter). The key is stored securely on this computer and never shown or saved as plain text; once active, a masked version is shown with change (□) and delete buttons. The panel docks on the right and starts hidden — the menu button toggles it.

2 Type a command

Examples: “set angle 25, spacing 8mm, generate supports”, “Nest all parts in a 250 × 250 × 100 volume with 3 mm gap”, “Make part 0 hollow with a 2 mm wall and a 6 mm cubic lattice inside”. Press Enter or Send.

3 Let it work

The agent checks the scene, changes settings, selects parts and runs the step, then reports what it did. Tick Details to see each tool call; Clear Chat resets the conversation.

What the Support-module agent can do

- **Settings** — any numeric setting, checkboxes and option lists (stop mode, base type and shape, split axis, nest rotation, manual mode).
- **Actions** — generate, clear supports, union, reset, export, send to slicer, wall-thickness analysis, grid / stack / nest and their undo, split view, delete or select parts, rotate, mesh operations.
- **CAD** — union, subtract, intersect, hull, create, cut by plane (open or closed), text label, hole, box cut, hollow, offset, hollow + lattice (cubic, BCC, gyroid, Schwarz-P, diamond), array, mirror, scale, repair (optional small-component threshold), split bodies, undo. To reduce triangles, ask for mesh decimation (÷2, ÷4, ÷8 or a custom percent).
- **See the view** — takes a live screenshot and describes it. **Status** — reports parts, supports, settings and undo steps.

Things to know

- Reset, Nest, CAD, decimation and refinement delete supports or open confirmation dialogs you still answer.
- Export, Send to Slicer and the load dialogs open the normal dialogs.
- Manual supports cannot be placed by the agent (you Ctrl + click); it can only arm Add / Delete mode.
- In multi-part scenes, make sure the right part(s) are selected first.

In the slicer the agent can load files, tune power, speed, hatch and layer thickness (globally or per part), nest parts, switch between SLM and EBM, slice, estimate build time and export — for example “load these STL files, set infill 200W/900mm/s, slice, and export”.

Mouse, keyboard and tips.

Controls of the Support module's 3D view and the habits that save the most time.

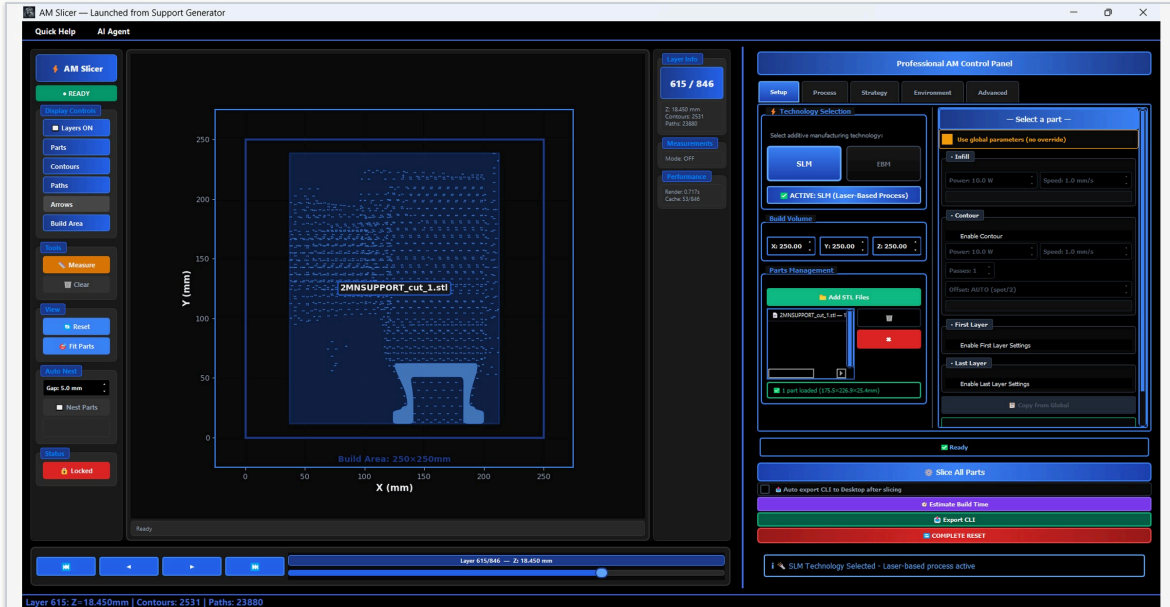
Input	Action
Left drag (empty space)	Rotate the camera
Left drag (on a part)	Move that part in XY (the Drag axis buttons limit it to X or Y)
Scroll wheel	Zoom
Shift + left drag	Pan
Double click (empty space)	Reset to the default isometric view (empty scene: centres on the build plate). Not active on a part or in Add / Delete Mode
Ctrl + left click (Add Mode)	Place a manual support at the green dot (nearest corner point)
Ctrl + left click (Delete Mode)	Remove the nearest support
Click (screen or list)	Select one part
Ctrl + click (screen or list)	Add / remove parts. For CAD the first clicked part is the base
Shift + click (list)	Select a range of parts
Delete / Backspace (list)	Delete selected part(s) and their supports (not while typing in a field)
Enter (3D Nest X / Y / Z)	Apply and redraw the build-volume box
Enter (Plate X / Y)	Apply the new gray plate size
Ctrl + C	Fit all parts in view (parts are not moved); ☐ Center moves parts to the origin
Ctrl + Shift + A	Open / close the AI Agent

Tips

- **Order matters:** CAD first, then Nest, then supports, then Union. Nest and CAD delete supports.
- **Keep your work:** use Add Part(s) instead of Load Multiple Files. **Camera:** most actions keep your camera angle; only Reset All and Load Multiple Files reset it.
- **Large parts / many supports:** use Split + Auto Support + Union, or increase Min Spacing / use Dual cylinders.
- **Lattice on big parts:** use a larger cell size and apply it to the original part.
- **No overhangs detected?** Lower the Overhang Angle (try 30° or 20°). **Too many supports?** Increase Min Spacing.
- **Hollow models:** Stop Mode → Lower Face. **Thin walls / fins:** Vertical Thin Area Supports are on by default.
- **Weak spots:** run Wall Thickness Analysis with a value that matches your printer limits (Min Thickness starts at 0).
- **Large meshes:** Simplify (÷2 / ÷4 / ÷8) before generating speeds up detection a lot.
- **Damaged parts:** run CAD → Modify → Repair / Clean before Boolean, Hollow or Offset.
- **Send to Slicer** is in the Slicer tab; **AI Agent:** Ctrl + Shift + A opens a chat that can control the app.

SLM / EBM Slicer.

Opens from the Support module's Slicer tab or on its own. Nest several parts on one plate, give each part its own parameters and preview every layer before export.



AM Slicer — nesting, per-part parameters and layer preview (layer 615 of 846).

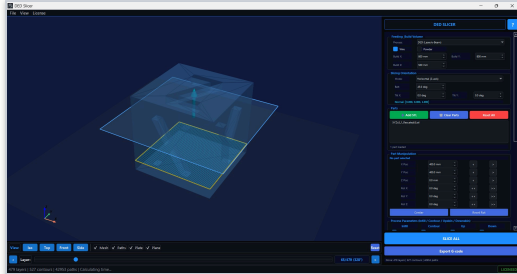
- 1 **Choose the technology**
In Setup > Technology Selection pick SLM (laser) or EBM (electron beam).
- 2 **Set the build volume**
Enter X / Y / Z (250 × 250 × 250 mm in the sample).
- 3 **Add parts**
Add STL / 3MF Files (Ctrl + O) in Parts Management. Each entry shows its size; use the red buttons to remove or clear.
- 4 **Nest**
Set the Gap (5.0 mm in the sample) and press Nest Parts.
- 5 **Set parameters**
Leave Use global parameters ticked, or untick it and set Infill, Contour (passes), First Layer and Last Layer values per part; Copy from Global restores defaults. Process, Strategy, Environment and Advanced tabs hold further settings.
- 6 **Choose the scan strategy**
Strategy tab. **Line Scan**: Angle (first layer), Inc (turn per layer, default 67°) and the Bidirectional checkbox (default on). **Stripe Scan**: 45° hatch inside stripes — Stripe size (default 15 mm), Stripe Gap (0 = AUTO = hatch ÷ 4), Shift (mm per layer) and Rot (° per layer); Shift and Rot at 0 mean no shift and no rotation.
- 7 **Slice and review**
Press Slice All Parts, then scrub with the layer slider. Toggle Layers, Parts, Contours, Paths, Arrows, Build Area; the Measure tool checks distances. Parts stay locked after slicing until Complete Reset (Ctrl + Shift + R).
- 8 **Estimate and export**
Estimate Build Time, then Export CLI (Ctrl + E). Auto export CLI to Desktop after slicing is ticked by default and saves SliceJob_<N>layers.cli on the Desktop; untick it to export manually.

Good to know

- Slicing and export run in separate processes, so the window stays responsive; closing the app or Complete Reset (Ctrl + Shift + R) stops them.
- Parts are locked during and after slicing. Complete Reset unlocks them and clears the sliced layers.
- Process tab presets: Save / Load / Delete; "Default SLM" and "Default EBM" are included. Presets also store the stripe size, gap, shift, rotation and Bidirectional setting.
- Shortcuts: Ctrl + O add files, Ctrl + S slice, Ctrl + E export CLI, Ctrl + Shift + R complete reset, F1 Quick Help.

DED Slicer.

Laser / e-beam DED (wire or powder) and arc-based WAAM, in 3 or 4 axes, exporting 4-axis G-code (X / Y / Z / B) for your controller.



DED Slicer — wire and powder feed, WAAM.

1 Process and feeding

Process: DED (Laser / e-Beam) or WAAM (Wire Arc). Feeding: Wire or Powder (powder is disabled in WAAM). Build X / Y / Z (800 × 800 × 500 mm in the sample).

2 Slicing orientation

Horizontal, Tilted X / Y / XY, or Belt DED (default 45°). Z is always absolute; the B axis only moves between layers, in Belt mode.

3 Parts and position

Add STL / 3MF, then X / Y / Z position and rotation; Center and Reset Rot.

4 Parameters and slice

Infill, Contour, Upskin, Downskin zones, Layer Configuration, then SLICE ALL and Export G-code.

Parameter	What it does
Process type	DED: Power (W) and Spot Diameter; G-code uses M03 S{power} for source on/off and M68 for feed. WAAM: Voltage, Current and WFS; M11 Q{arc_power} switches the arc. Choosing WAAM forces Wire feeding and shows the WAAM Arc Parameters panel.
Strategy	Line Scan: Start Angle (first layer) and Rotation (turn per layer, default 67°); Bidirectional reverses alternate lines. Stripe Scan: 45° hatch inside stripes — Stripe sz (default 30 mm), Stripe gap (AUTO = Hatch ÷ 3), Shift (mm per layer) and Stripe rot (° per layer); 0 = off. Controls the chosen strategy does not use are inactive.
Contour Offset	Compensates for melt-pool width. AUTO (0) = Spot Diameter ÷ 2; a value above 0 is a fixed offset in mm. Outer contours shrink inward, hole contours grow outward. Spot Diameter is DED only; in WAAM the Spot field is disabled but AUTO still uses Spot ÷ 2, so enter a manual offset (about bead width ÷ 2).
Hatch and Contour Gap	Hatch is the line spacing (default 1.2 mm) and is used exactly as entered — it is not widened automatically. Contour Gap is the distance in mm between the offset contour and the start of the infill: AUTO (0) = Hatch ÷ 3, a value above 0 is used as entered (no ratio). Total infill inset = Contour Offset + Contour Gap.
Deposition zones	Downskin = first layer (1400 W / 10 mm/s), Infill = middle layers (1500 W / 15), Upskin = last layer (1300 W / 13), Contour runs before infill (1200 W / 12). In WAAM the Power fields are informational only.

WAAM Arc Parameters

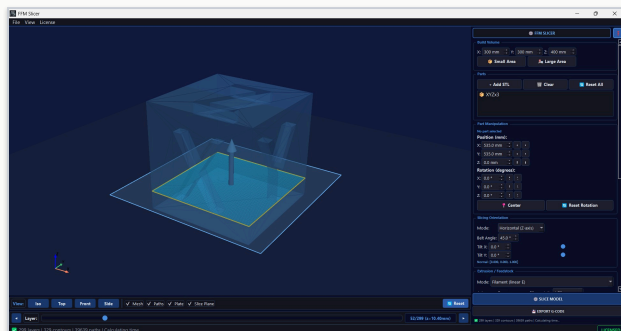
Setting	What it does
Arc Mode	CMT (low heat, thin walls), Pulse (controlled droplet transfer), Short Arc (low heat, thin sections), Spray Arc (high deposition, bulky sections). Written to the G-code header.
Voltage / Current	Arc Power (W) = Voltage × Current, shown live and written into the arc-on commands in place of laser power.
WFS	Wire feed speed in m/min, written as the M68 feed value. Weaving adds a sinusoidal oscillation (Weave Width mm, Weave Frequency Hz).
Interpass Temp Max	Header information only; not enforced by the G-code.

Good to know

- Layer thickness: WAAM typically 1.5–4 mm, laser DED 0.2–1 mm. Time estimates ignore acceleration — expect builds 10–20% longer.
- Downskin and Upskin only change the infill; each zone's On box turns its power and speed on or off in the export, while the contour always uses the Contour values.
- In WAAM the Power and Feed fields are unused (WFS drives the feed). Reset All restores every setting to its default and removes the parts and results; Clear Selected removes only the selected parts and keeps the settings.

FFM Slicer — filament and pellet.

Large or changing-table-size polymer parts in 3 or 4 axes, with three extrusion modes and nozzle-compensated contours.



FFM Slicer — filament or pellet, 3/4 axes.

- 1 **Build volume**
X / Y / Z (300 × 300 × 400 mm in the sample) or the Small Area (300 × 300 × 400) / Large Area (1070 × 1070 × 1118) presets for large-area printing.
- 2 **Parts and manipulation**
+ Add STL / 3MF, Clear Selected or Reset All; position and rotation, Center, Reset Rotation.
- 3 **Slicing mode**
Horizontal; Tilted X / Y / XY (−89° to +89°; above about 60° bed adhesion may suffer); Belt FFM, where the B axis advances between layers and Z stays absolute.
- 4 **Extrusion and slice**
Choose the extrusion mode, then SLICE MODEL and EXPORT G-CODE.

Extrusion modes

Mode	How it works
Filament	Classic linear E axis in millimetres of filament, for desktop and belt filament printers.
Pellet — Virtual axis (recommended)	Computes the axis value like filament mode, using a virtual diameter you set. Matches most pellet retrofits and controllers calibrated around a nominal diameter.
Pellet — Direct volumetric	Writes the raw cubic-millimetre volume with no diameter conversion. Correct only if your controller is configured to read the axis that way.

You must calibrate pellet output

The software cannot know your machine's steps/mm or axis letter. Before a real print run a short calibration move, weigh the output and adjust the virtual diameter (or axis letter) until the numbers match. The max throughput is only a warning threshold and in pellet modes the warning is just a comment in the G-code; Screw $\varnothing$ is informational.

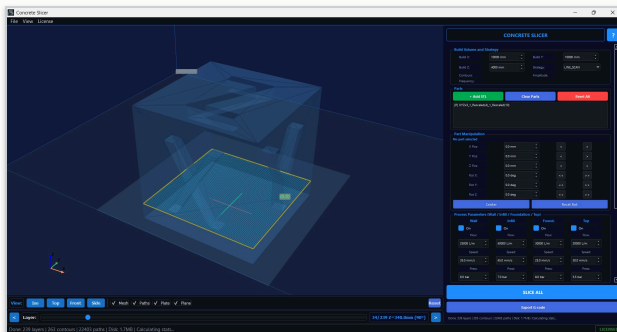
Infill, offsets and tips

Setting	What it does
Strategy / density	LINE pattern with LINE_SCAN (whole polygon; Start Angle and Angle Inc turn the lines each layer) or STRIPE_SCAN (stripes first, for thermal stress on very large parts): 45° hatch, Stripe size, Gap (AUTO = line spacing ÷ 3), Shift (mm per layer) and Rot (° per layer); 0 = off. The Bidirectional checkbox (default on) prints alternate lines in opposite directions. Density guide: 5–10 % decorative, 15–20 % standard, 30–40 % functional, 50–100 % maximum strength.
Contour Offset	Compensates the extruded line width. 0 = AUTO = nozzle diameter ÷ 2; above 0 is a manual value in mm. Outer contours shrink inward and holes grow outward.
Contour Gap (infill offset)	How far inside the compensated contour infill starts. Default is a fixed 0.133 mm; enter 0 for AUTO = Line Width ÷ 3 (needed at pellet scale, where 0.133 mm is far too small). Total infill inset = Contour Offset + Contour Gap.
Quality and troubleshooting	Pellet scale: layer height 0.5–5 mm, line width 2–9 mm, close to nozzle diameter. Parts not slicing: check for an invalid or empty mesh, a model larger than 5000 mm or an empty section; slicing does not check the build volume. Missing infill: raise density or check contours. Wrong extrusion on hardware: re-check mode, axis letter, virtual diameter. Peak flow warning: lower speed, width or layer height.

Speeds and G-code: the Speed field is the infill speed; contours print at 80% of it. Fixed values: travel 150 mm/s, retraction 1.5 mm at 45 mm/s, fan 100%. The G-code uses generic RepRap commands (G28, M83, M104/M109, M140/M190, M106) — verify them against your controller.

Concrete Slicer.

Large-format concrete slicing with its own wall, infill, foundation and top passes.



Concrete Slicer — 3 axes, large format.

1 Build volume and strategy

Build X / Y / Z (10000 × 10000 × 4000 mm in the sample), shown as a wireframe box with the origin and corners labelled, and a Strategy.

2 Parts

+ Add STL / 3MF (several at once); select by name or click; X / Y / Z Pos and Rot with < / > nudges, Center, Reset Rot, Reset All.

3 Layer and process settings

Layer H, Line W, Nozzle, Rotation, Start Angle, Wait, Infill Gap, Wall Offset, Bidirectional, Stripe W / Gap / Shift / Rot; Flow, Speed and Pressure for four passes.

4 Slice and export

SLICE ALL, review with the layer slider, then Export G-code. The summary shows layers, contours, paths and file size.

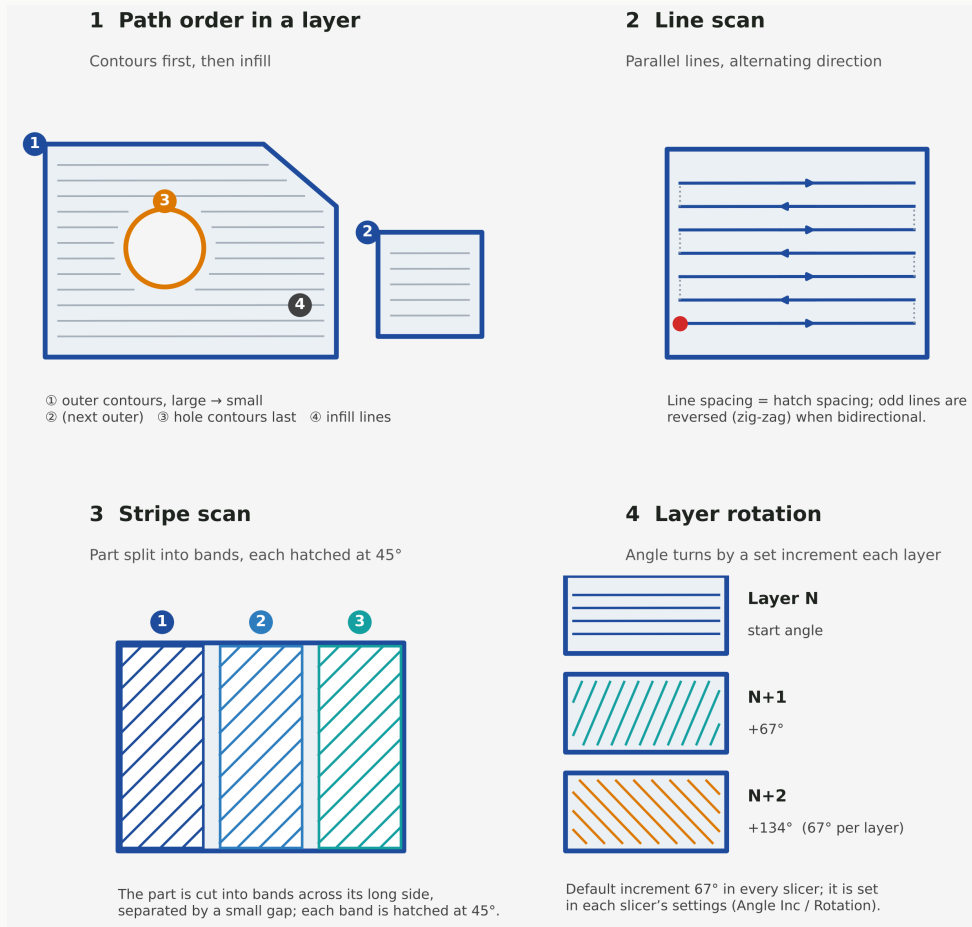
Strategy	What it does
LINE_SCAN	Evenly spaced straight infill lines; Start Angle sets the first layer and Rotation turns the line direction each layer for better bonding.
STRIPE_SCAN	Layer divided into parallel stripes (Stripe W), each filled independently — for very long or large flat shapes. 45° hatch; Stripe Gap AUTO = $0.75 \times \text{Line W}$; Shift (mm per layer) moves the stripe boundaries and Stripe Rot (° per layer) turns the stripe grid; 0 = off.
CONTOUR_ONLY	Only the outer wall outline on every layer; Infill controls are disabled. For thin-walled shapes and test prints.
CONTOUR_PARALLEL	Rings following the wall: Contours = ring count, Amplitude = waviness (0 = smooth), Frequency = waves per ring. Rings print with the Infill parameters, the outer wall with the Wall parameters.
Setting	What it does
Wall Offset	Compensates bead width: wall path moves inward for outer boundaries and outward for holes. AUTO (0) = $\text{Nozzle} \div 2$; a manual value fixes it. Infill is generated relative to the offset wall, cumulative with Infill Gap. Infill Gap: AUTO (0) = $0.75 \times \text{Line W}$; a value above 0 is used as entered.
Layer controls	Start Angle and Rotation are active only in LINE_SCAN; Stripe W, Stripe Gap (AUTO), Shift and Stripe Rot only in STRIPE_SCAN. Bidirectional (default on) reverses every second infill line.
Wait	Pause between layers so concrete can firm up; too short may sag, too long slows the job.
Process passes	Wall and Infill = every layer except first and last; Foundation = first layer only (slower, more material); Top = last layer only (slower, cleaner finish). Each of the four checkboxes (Wall, Infill, Foundation, Top) switches its own pass on or off (Foundation or Top off: that layer uses the Wall / Infill values).
Mix and system	Slump, W/C and target Strength are stored with the job for records only. Standoff (default 5 mm) is added to every Z value in the G-code.

Preview and tips

- Gold = wall paths (at the compensated Wall Offset position), cyan = infill, pale blue square = current layer plane. Mesh, Paths, Plate and Plane can be toggled separately.
- The Volume & Production Estimate shows layers, path length (m), concrete volume (L) and print time; the time estimate uses the Wall / Infill speeds for all layers, including Foundation and Top.
- G-code uses centred coordinates (X from $-Bx/2$ to $+Bx/2$) and the M200–M204 concrete commands; flow is written in ml/min (M202 S...) although the panel label reads “L/m”.
- Reset All restores every setting to its default and removes the parts; Clear Selected removes only the selected parts.
- Sagging while building: increase Wait. Rough shape: reduce Layer H. SLICE ALL greyed out: add a visible part.

Scan strategy at a glance.

SLM/EBM, DED, FFM and Concrete order and fill every layer the same way; only the stripe gap and the Concrete extras differ.



Scan strategy — contour order, line scan, stripe scan and layer rotation (schematic, not a real slice).

Slicer	Angle (default)	Stripe gap (AUTO)	Stripe shift / rot	Strategies
SLM / EBM	Angle 0°, Inc 67°	hatch spacing ÷ 4	mm / ° per layer	LINE_SCAN, STRIPE_SCAN
DED	Start Angle 0°, Rotation 67°	hatch spacing ÷ 3	mm / ° per layer	LINE_SCAN, STRIPE_SCAN
FFM	Start Angle 0°, Angle Inc 67°	line spacing ÷ 3	mm / ° per layer	LINE_SCAN, STRIPE_SCAN
Concrete	Start Angle 0°, Rotation 67°	0.75 × line width	mm / ° per layer	LINE_SCAN, STRIPE_SCAN, CONTOUR_ONLY, CONTOUR_PARALLEL

Same in all four slicers

- Contours first: outer contours from large to small, then hole contours, then every infill line. Concrete calls the contours “walls”.
- Line Scan: parallel lines at the layer angle, starting from the Start Angle (Angle in the SLM / EBM slicer); with Bidirectional on (a checkbox in all four slicers, default on), every second line is reversed (zig-zag).
- Stripe Scan: the layer is cut into bands, each hatched at 45°, with a gap you can set (AUTO = the value in the table). With Shift and Rot at 0 the bands are cut across the long side of the layer.
- Stripe Shift and Rot: when either is above 0 the bands lie on a fixed grid anchored at the origin. Shift slides the grid by that many mm every layer (wrapping around stripe size + gap); Rot turns the grid and the hatch by that many degrees every layer, so band edges do not stack from layer to layer. Both are 0 (off) by default.
- Layer rotation: in Line Scan the angle turns by the Scan Angle Increment every layer. The increment and the strategy are set in each slicer's settings; controls that do not apply to the chosen strategy are inactive.
- CONTOUR_ONLY (walls only) and CONTOUR_PARALLEL (rings) exist in the Concrete Slicer only.
- While exporting, the progress bar advances in steps through the layers instead of waiting until the end.

ERP — manufacturing and cost report.

An ERP-style manufacturing and cost report for every technology. Pick the technology first; each one keeps its own presets.

ERP — technology selection dialog over the Project tab.

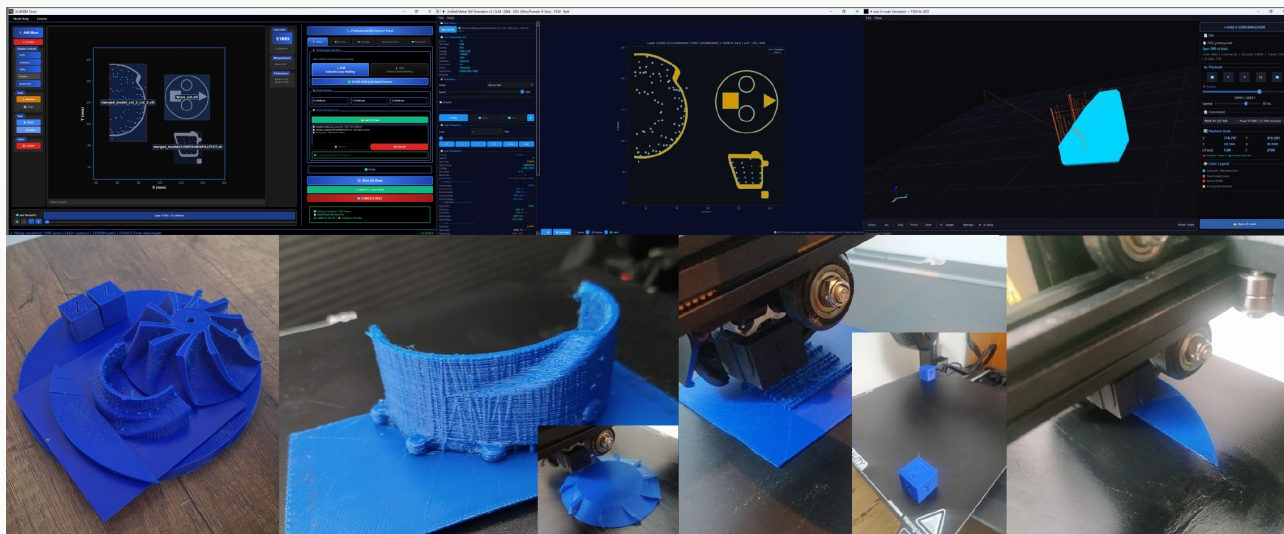
- Select the technology**
 Pick SLM, EBM, DED, FFM or CONCRETE in the Select Manufacturing Technology dialog; the title and status bar show the choice.
- Fill in the project**
 Project tab: Document Information (number, publication date, revision) and Project Details (name, code with Generate Code, customer, priority Low / Medium / High / Critical, status Planning / In Progress / Completed).
- Enter process data**
 Process tab: material, power, speed, hatch spacing, layer thickness and build volume, entered manually for your process.
- Calculate cost**
 Costing tab: machine cost, lifetime, working days and hours, Production Time in hours (depreciation = hourly depreciation × Production Time); material cost per kg and amount; gas, electricity, labor; VAT. The total updates automatically, always in “\$”.
- Reuse settings**
 Tools > Save Current Settings as Preset saves the material, process values and costing inputs; Tools > Manage Presets loads them. Projects and presets are stored in a local database.

Shortcut	Action
Ctrl + N / Ctrl + O / Ctrl + S	New, load and save project
F1	Quick Help

Save, Billing and Reports — details

- Presets** store 20 fields: the material, five process values (power, speed, hatch spacing, layer, volume) and 14 costing inputs — not project information, billing data or post-processing text.
- Save / Load Project** stores project, process, costing and post-processing data plus client name, e-mail and phone; it does not store the billing table, company information or invoice number. Save requires a project name; Load lists only projects of the current technology.
- Billing:** From / Bill To, invoice number and dates (due date = +30 days), currency, line items and Add Manufacturing Cost (including VAT); Generate Invoice PDF. The currency only changes the symbol on the PDF — amounts are not converted.
- Reports:** post-processing text with quick-insert buttons; Export PDF Report needs a project name; Export Excel is available only if xlsxwriter is installed.

Preview before you print.



Test and prototype results — multi-part nesting, unified metal AM simulator, 4-axis G-code simulator, and physical FFM test prints.

Toolpaths can be previewed layer by layer in each slicer. The simulators and the FFM test prints shown above are test and prototype work from development; they are not released product features. The FFM test parts were a curved shell with supports, a calibration cube and a curved-surface test.

Quick reference

Situation	What to do
Very large or high-polygon mesh in STL Repair	Tick Split + Multi-Core Repair + Union; raise Tries or Retry Overlap if a seam will not union; use Retry Union Only to repeat just the union.
Very large part with many supports	Use Split + Auto Support + Union; adjust Retry Overlap, Step and Tries if a seam will not union.
Need Base Support with a split part	Run the normal Generate, then Union (Base Support is ignored in Split + Union).
Only some overhangs need support	Enable Face Selection and tick the regions in the Overhang Face Selector.
No overhangs detected / too many supports	Lower the Overhang Angle (30° or 20°) / widen Min Spacing.
Thin walls and fins	Run Analyze Wall Thickness; enable Vertical Thin Area Supports.
Hollow or enclosed part	Stop Mode: Lower Face.
Packing several parts	3D Nest before generating supports; Undo 3D Nest to step back.
Edited the part with CAD tools	Generate supports afterwards — CAD operations delete supports.
Contours look off-size (DED / FFM / concrete)	Keep Contour Offset / Wall Offset on AUTO; set a manual value only if the real bead width differs from spot or nozzle size.
Pellet extrusion amounts wrong	Calibrate virtual diameter and axis letter on a short test move.
Slow processing	Heavy jobs run in the background so the window stays usable; the engine scales workers to your hardware (a recent CPU / RAM / GPU gives the best results).
Not sure what a control does	Use the Quick Help button in each module.

More information: additiveelite.com

SOLUTION SUMMARY

The problems ADDITIVEELITE is built to solve

PROBLEM	ADDITIVEELITE SOLUTION
Fragmented software tools	→ All-in-one: 6 modules on one platform
High license costs	→ €3K/year (competitors €15K–20K)
Large-part processing failures	→ Disk-backed, resource-aware processing
No cost reporting	→ Built-in ERP module
Single-technology lock-in	→ Metal + polymer + concrete together

WHERE IT IS USED

Example parts by industry

35%



Aerospace & space

- Turbine impellers and blades
- Fuel injector nozzles
- Lightweighted mounting brackets

MATERIAL
Ti-6Al-4V, Inconel 718

PROCESS
SLM, EBM, DED

25%



Automotive

- Conformal-cooled mold inserts
- Lightweight engine / suspension brackets
- Prototype and fixture parts

MATERIAL
AlSi10Mg, SS316L

PROCESS
SLM, FFM

20%



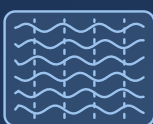
Medical & dental

- Acetabular cup (hip implant)
- Dental bridge / crown framework
- Patient-specific surgical guides

MATERIAL
Ti-6Al-4V, CoCrMo

PROCESS
SLM, EBM

20%



Energy & defense

- Heat-exchanger channel structures
- Gas-turbine components
- DED repair of worn parts

MATERIAL
Inconel 718, SS316L

PROCESS
SLM, DED

Parts are representative application examples; drawings are illustrative only. Industry shares match the estimates on the market page.