Igniform

Operation Guide

Version 1.3.0

Design jigs the easy and quick way.

FabriAI, Inc.

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Introduction

This guide is intended for shop-floor users who are using Igniform for the first time. No CAD expertise is required: by following the on-screen prompts, you can produce jig data for 3D printers in a few simple steps.

Igniform offers two ways to create jigs. Chapter 1 explains how the two modes differ; Chapters 2 and 3 cover each mode in detail. After that, the guide moves on to export, settings, and creating jigs from photos.

> **A note on terminology**
> Throughout this guide, the part being made is consistently called a "jig," and the parts being machined or assembled are called "workpieces."

Chapter 1 The two modes of Igniform

When you launch Igniform, the first screen is "Let's start a project," where you choose between two ways of creating a jig.

1-1 How the two modes differ

	Freeform Editing	From a Template
Entry on the screen	"Start from your data"	"or start quickly"
How to start	Load a 3D model of the workpiece (STL / STEP) or a photo, then build the jig freely.	Pick one of eleven templates and fill in the dimensions step by step.
Best suited for	You already have a 3D model of the workpiece; you want to design freely.	You want to build a standard jig from scratch, quickly.
Main operations	Add Jig → Edit Jig → Add Support → Add Label	Pick template → fill each step → generate

In either mode, the finished jig can be exported the same way (see Chapter 4).

1-2 The buttons on the launch screen

Start from your data (Freeform Editing)

- "Open File": load an existing workpiece 3D model (STL / STEP) and start designing.
- "Create from Photo": import an outline from a workpiece photo and start designing (see Chapter 7).
- "Open Project": resume a previously saved project (.cad file).

or start quickly (Create from Template)

- Click one of the eleven templates to begin. The newly added "Mold" and "Pocket Jig" carry a NEW badge.
- The "Start with an empty scene" link at the bottom-right opens a blank workspace with nothing loaded.

If work from your last session ended without being saved, "Restore your previous work?" appears before the launch screen (3-6).

Chapter 2 Creating jigs in Freeform Editing

Freeform Editing lets you assemble a jig freely from a workpiece 3D model or photo. You enter this mode by choosing "Open file," "Create from photo," "Open project," or "Start with an empty scene" from the launch screen.

2-1 Understanding the screen

The left menu is divided into four groups corresponding to stages of the workflow.

Group	Menu items	Role
Scene	Load File / Initialize Grid / Reset View	Load workpieces and reset the view.
Design	Add Jig / Edit Jig / Add Support / Add Label	The core operations for creating and editing jigs.
Optimize	Simplify Shape / Printability	Tune the shape for printing and check for issues.
Quick	Templates / Create from Photo	Switch to template mode or launch photo-based creation.
(top)	Home	Returns to the launch screen.
(bottom)	Settings	Opens the Quick Settings panel (see 5-4).
(bottom)	Guide	Opens this guide (PDF).

The center area is the 3D view. "Scene Objects" in the upper-right lists the loaded workpieces and the jigs you have created. Rotate / Move / Scale / Apply buttons appear at the bottom of the screen (Rotate and Move are covered in 2-3).

In the "Initialize Grid" panel you can store the grid spacing and the base-plate hole pitch with "Save current value as default". Saved values are used for new projects on this PC; when you open an existing project, the values stored in that project take priority.

2-2 Basic workflow

1. Load the workpiece (Load File). Import the part you are building a jig for, as STL or STEP.
2. Set the position and orientation of the workpiece if needed, with Move and Rotate at the bottom of the screen (2-3).
3. Add a jig (Add Jig). Place the base shape for the jig. You can also generate a work nest automatically from "Quick" (2-4).
4. Form the jig (Edit Jig). Choose the molding mode and the "Gap between jig and work (XY)", then press "Apply Jig" to cut a pocket that matches the workpiece.
5. Add fixtures or text if needed: Add Support (2-5) and Add Label (2-6).
6. Lighten and check (Simplify Shape / Printability). Reduce material and print time, and confirm the jig prints without problems (2-7, 2-8).
7. Export. Write out an STL and pass it to your slicer (Chapter 4).

The "Gap between jig and work (XY)" can also be set with the preset buttons "Tight 0.05mm", "Standard 0.20mm" and "Loose 0.30mm" (default 0.20 mm, range 0–1 mm).

Choosing a molding mode (important)

When editing a jig, you choose one of three ways to shape the cavity around the workpiece. This setting affects both finish and usability.

Mode	Characteristics	Typical processing time
Precise Fit	Creates a jig that fits the workpiece exactly. Best for precision holding, but the workpiece may be hard to remove.	—
Slope Hugging (Recommended)	Fits the workpiece while allowing it to be lifted straight up. Handles slopes, steps, and complex shapes.	Depends on the shape. Intricate parts can take several minutes (10 minutes maximum).
Easy Removal	Creates a cavity matching the workpiece silhouette. Fastest and easiest to remove, but the fit is loose.	Under ~1 second

> **About the "Easy Removal" mode**
>
> This mode uses the largest silhouette seen from above. For stepped shapes or shapes that widen towards the bottom, steps are not reproduced and the cavity is merged into a single level. To preserve steps, use "Precise Fit" or "Slope Hugging".

While the jig is being formed

- The elapsed time is shown as "Processing... 1:23". Press "Cancel" to stop; this now works during a Slope Hugging calculation as well.
- For parts that are likely to take a while in Slope Hugging (intricate parts, or a part whose last run was slow), a notice appears at the start: "This part can take several minutes to process." Forming continues regardless.
- When it finishes, "Shaping finished (N s)." is shown.
- Slope Hugging stops after 10 minutes. Increase the layer thickness, or check the shape with "Easy Removal" first.

Jigs whose size cannot be changed, and jigs that cannot be formed

Jigs created with "Sketch" or "STL Import" cannot be resized in the Edit Jig panel; the panel shows "⚠ Why the size is locked". Create or import the jig again to change its shape.

If the jig shape cannot be determined, "Cannot shape the jig" is shown and nothing is formed. Select a shape in the Edit Jig panel, or select the jig again. A support base (2-4) cannot be edited in this panel — adjust it through the Point Support settings instead.

Chamfer on part-contact faces

After a jig has been formed, the rims of the cavity where the workpiece touches can be chamfered at 45° (a C chamfer). This keeps the workpiece from catching as it goes in and out and stops the rims from chipping. Select the jig, open "Edit Jig" and use the "Chamfer on part-contact faces" section.

- ① Pick the rims: "Select every rim on the top face" picks all the rims on the top face of the cavity at once. To pick them one at a time, press "+ Add / remove rims one by one" and click the rims (Esc to finish).
- ② Size: set the chamfer with the "0.3", "0.5" or "1.0" buttons or a number (default 0.5 mm, 0.2–2.0 mm).
- ③ Apply: press "Apply chamfer".

Chamfering comes after "Apply Jig". The outer edges of the jig, slots that run out to the side, and the inside corner where a cavity floor meets its wall are out of scope. It is not available on a point-support base or its pillars. When the jig is formed again, the rims you picked are chamfered again automatically. Your choice is saved in the project. If the chamfer is so large that a thin part would be cut loose, nothing is chamfered (use a smaller "Chamfer size").

2-3 Positioning and orienting the workpiece

Use Move and Rotate at the bottom of the screen to position an imported workpiece. The arrows and rotation rings are drawn at the centre of the workpiece (jigs and Scale behave as before).

Entering numbers

- Type values into "Translation (mm)" or "Rotation (°)" and press "Apply Changes".
- With a workpiece selected, the Move panel also shows "Height above reference plane (mm)". This is the height from the grid plane (the top face of the base plate) to the lowest face of the workpiece; entering a value moves the part along Z only. The value is not saved in the project file.
- The Move button menu offers "Grid snap (G)" and "Face snap (S)". With grid snap on, the centre of the workpiece lands on the grid steps.
- The "By angle" tab of "Rotate" has "Call this pose 0°" and "Back to the original basis". "Call this pose 0°" treats the current orientation as 0°, so you can turn the workpiece by angles measured from there. Neither button moves the workpiece. The basis you set is saved in the project. These buttons are not available in Template mode.

Pointing a face down (By face)

You can pick a flat face of the workpiece and rotate the part so that face points down, towards the base plate. This is available only while a workpiece is selected.

1. Select the workpiece, press "Rotate" at the bottom of the screen and open the "By face" tab.
2. Press "Pick a face" and click a face of the workpiece. The face under the cursor is highlighted, and the area and triangle count of the picked face are shown.
3. If needed, set "Face extent" to "Narrower", "Standard" or "Wider".
4. Press "Point this face down" (double-clicking the face does the same).

- Right-click the workpiece and choose "Point this face down..." to start with that face already picked; nothing is rotated yet.
- After the rotation, the height of the lowest face is restored, and the amount restored is reported on screen.
- To stop picking, press Esc or "Cancel picking". "Clear picked face" releases the face you picked.

> **Rotating a workpiece that has jigs**
>
> If jigs overlap the workpiece or belong to it, "The fit with existing jigs will be lost" is shown. Choosing "Rotate and re-make the jigs later" rotates the workpiece, but the jigs are neither moved nor re-formed automatically. Re-make them after the rotation.

2-4 Adding a jig ("Add Jig")

"Add Jig" has two tabs: "Quick" for fast addition, and "Custom" for choosing a shape with detailed settings.

Quick

Generate commonly used jigs to fit the workpiece on the spot. Two options are available.

Feature	What it does	Main settings
Work nest	Automatically generates a holder shape that cradles the loaded workpiece. The workpiece is moved along Z during generation so that the bottom of the jig sits on the base plate.	Wrap depth (default 50%, 5–95%) / Wall X·Y·Z (10 mm each). Press "Generate work nest" after setting.
Point Support	Places a support pillar at each point you click on a face of the workpiece in the 3D view.	Pillar radius (10 mm) / Screw lock (off) / Support base (off; when on, 80 wide × 80 deep × 5 thick, corner R 3). Press "Start selecting", click the faces, then press "Generate".

The wrap depth is a guideline for how deeply the holder embraces the workpiece. A larger value cradles the workpiece more deeply.

When the workpiece is moved during work-nest generation, the distance is reported on screen. Pillars and support bases that belong to the workpiece move with it. If a jig overlaps the workpiece but has no record of which workpiece it belongs to, "Choose which jigs move with the work" appears: tick the jigs that should move and press "Move as selected". Jigs that already have the workpiece shape carved into them are ticked by default.

The plate under a point support is called the "Support base"; it is not the base plate of the scene. A support base and a screw lock both anchor the part to the table, so they cannot be used together: if you turn both on, a notice appears and you have to switch one off. Pillars can only be placed on faces of the workpiece — clicking the base plate or a jig shows a hint instead.

Custom

Pick the shape you want and add it as a jig. Choose the shape category first on the "Custom" tab.

- Shape category: Cylinder / Cone / Box / Other.
- Source of the shape: Preset (built-in shapes) / STL Import (your own data) / Sketch.

The following preset shapes are available:

Preset	Shape
L-bracket	L-shaped bracket.
T-bracket	T-shaped form.
Boss Plate	Plate with a boss.
Corner	Corner / edge reinforcement.
Ribbed Plate	Plate with a reinforcing rib.
U-channel	U-shaped form.

1. Pick the shape category and the source of the shape.
2. Enter the size. Presets use W / D / Height / Thickness; a cylinder uses Height and Diameter; a cone uses Height, Top D and Bottom D; a box uses Height, W and D. Height is always the vertical (Z) dimension.
3. For "STL Import", choose the "Basis Axis" (Height(Z) / W(X) / D(Y)) and the "Target Size (mm)". The imported shape is scaled so that the basis axis matches the target size.
4. Adjust placement in "Position (mm)" if needed.
5. Press the Add button to place the jig in the 3D view.

Preset jigs saved with an earlier version

In earlier versions the "Height" of a preset shape was not applied vertically, and the diameter of a cylinder ignored the value you entered. Preset jigs in projects saved before V1.2.0 keep their old orientation when reopened; create them again if that matters.

2-5 Adding a fixture ("Add Support")

Use this panel to attach screw-lock fixtures that fasten the jig to the holes in the base plate. The panel has two tabs: "Add Fixture" and "Fixture List".

1. Press "Add Support" in the left menu.
2. Choose the jig in "Select a jig".
3. In the "Screw Lock" card, set the screw hole, the gap, the flange, the arm shape and the screw length (see the table below).
4. Press "Set Screw Positions" and click grid holes to place the anchors. Click an anchor again to remove it.
5. Press "Add".

Item	What it covers (defaults for M3, in mm)
Screw Hole	Inner D (std) — M2 to M6, M3 by default / Outer D (6.5) / Depth (5.5)
Screw hole gap	Clearance between the hole and the screw; larger values make the screw easier to insert. Tight 0.05 / Standard 0.10 (default) / Loose 0.20
Flange	A flat plate added under the jig (off by default). Set the flange thickness (0.9) and radius (4.2)
Arm shape	Embed (1.35) / Root thickness (1.6x) / Root R (1.5) / Root width (1.6x)
Screw length	Table-side engagement (5). The panel shows the screw length you need, e.g. "Required screw length 10.5mm (M3)"

- When you change the standard, every dimension you have not edited by hand switches to the default for the new standard.
- If the arm dimensions do not suit the jig, Igniform adjusts them and tells you what it changed.
- If the bottom of the jig floats above the base plate and the fixture cannot reach it, "The fixture cannot reach the jig" appears. "Align, then add" moves the jig and its workpiece down so the bottom sits on the base plate and then adds the fixture. "Do not add" skips the fixture; the jig is then reported by the printability check as "Gap Above the Base Plate" (5-2).
- A support base (2-4) and a screw lock cannot be combined. While a support base, or a pillar that has one, is selected, "Set Screw Positions" and Add are disabled and the reason is shown.
- The "Fixture List" tab lists the fixtures per jig. Selecting a row highlights the fixture in the 3D view, and "Delete" removes it. Clicking a fixture in the 3D view selects its jig.
- Fixtures are merged into the jig body when you export (4-1).

Projects saved with an earlier version

Projects saved before V1.2.0 have no value for the root width, so the default (1.6x) is applied when they are opened and the fixture shape may change. If screw anchors no longer sit on grid holes, Igniform tells you on screen.

2-6 Engraving text (Add Label)

"Add Label" engraves text such as a part number or an orientation mark. The input fields and the basic steps are the same as in Template mode (3-4); this section covers what differs in Freeform Editing.

1. Press "Add Label" in the left menu and choose the jig under "Target". If no jig is selected, the support base or the first jig is picked for you.
2. Set the text, the dimensions and the position, then press "Engrave Label".

- A jig can carry several labels. To correct the text, use "Replace last label"; to remove it, use "Delete Label", which deletes every label on the jig. Both buttons appear only when the selected jig has at least one label.
- When the jig is re-formed by editing or by an auto-fix, the labels are engraved again automatically. If that fails, "The label could not be re-applied after re-molding, so the jig has no label." is shown — engrave it again with "Engrave Label".
- Adding a second label, or replacing one, re-forms the jig once and engraves every label again.
- Labels cannot be deleted while a molding job is running.

- Ctrl+Z restores the previous label state as well.

When a confirmation appears

On a jig with no workpiece (for example after the workpiece was deleted), a confirmation appears when you add a second label and when you delete a label. The jig is returned to its shape before molding, so the work cut-out, the warp-prevention grooves and any auto-fix results are lost.

On a jig with warp-prevention grooves, a confirmation warns that the grooves will be lost. They do not come back automatically — right-click the jig and choose "Add warp-prevention grooves" if you need them again.

2-7 Shape Simplification (lightweighting)

"Simplify Shape" lightens jigs (box-shaped) created with Work nest. By tidying the shape and removing internal material, it aims to reduce 3D-print time and material usage.

Shape simplification is available only while a box-shaped jig is selected. With any other jig selected, "Simplify Shape" in the left menu is disabled. A workpiece must be loaded as well.

Use the right-hand panel to set the level of lightweighting and detailed parameters.

Simplification Level

Choose the level of lightweighting from two modes. "Solid" is selected when the panel opens.

- Mild - keeps a top plate that follows the bottom face of the workpiece. Choose this when you want to support the workpiece over a surface.
- Solid - removes the top plate and supports the workpiece on pillars alone. Material and print time are reduced further.

Main settings

Section	Item	Meaning (example default)
Base	Base Plate Thickness	Thickness of the base plate (8 mm).
Base	Outer Margin	Margin left around the perimeter (5 mm).
Base	Fillet Radius	Radius of corner fillets (3 mm).
Shape	Contact Offset (Z axis)	Vertical gap around the part. Adjusts where the support pillars touch it (0.3 mm).
Shape	Clearance (XY axes)	Gap left around the part. Larger values make it easier to insert and remove (0.2 mm).
Shape	Cone Top Melt Depth	How far the top of the support cone embeds (0 mm).
Shape	Support Count	Number of support pillars (4). Not typed in directly: it changes with "+ Add Manual", right-click delete, and Reset.
Shape	Support Pillar Diameter	Support diameter (top 8 mm / bottom 12 mm).

Section	Item	Meaning (example default)
Shape (Mild only)	Top plate thickness	Thickness of the top plate (3 mm).
Shape (Mild only)	Pillar diameter	Diameter of the pillars under the top plate (10 mm).
Shape (Mild only)	Top Plate Melt Depth	How far the top plate embeds (0 mm).

"Contact Offset (Z axis)" and "Clearance (XY axes)" open the gap in different directions. Use Contact Offset for the vertical gap and Clearance for the gap around the sides. Clearance can be set from 0.0 to 1.0 mm.

In "Solid" mode you can also fine-tune the contact offset per pillar. Click a support pillar in the 3D view to open "Per-pillar fine tuning"; the value entered in "Adjust" is added to the global contact offset. Press "Clear adjustment" to revert.

You can also tune supports manually: "+ Add Manual" to add pillars and Reset to redo the layout. Existing pillars can be dragged to move or right-clicked to delete.

Jigs that carry fixtures

When a jig with screw-lock fixtures (2-5) is simplified, a flat counterbore is cut at each fixture so that the screw seat matches the "Depth" you set in the fixture panel. If the base plate is no thicker than that depth, the counterbore cannot be cut and Igniform shows the thickness it needs; increase the base plate thickness to at least that value.

Reviewing the effect and applying

1. After setting the parameters, press "Preview" to see the lightweighted shape on screen.
2. The lower panel shows estimated material reduction and time reduction (%).
3. If everything looks good, press "Apply" to confirm. Press "Undo" to revert.

> The estimated material/time reductions are guidelines only. Confirm exact figures in your slicer software.

2-8 Printability Check

"Printability", in the Optimize group of the Freeform menu, automatically verifies that the jig can be printed cleanly on a 3D printer. See Chapter 5 for the check items and settings.

Chapter 3 Creating jigs from a Template

In Template mode you pick one of eleven ready-made templates that matches your purpose, and just fill in dimensions step by step. Standard jigs can be produced quickly without any CAD knowledge.

3-1 Choosing a template

On the launch screen, the eleven templates are tiled in four columns under "Create from Template"; the number of templates is shown next to that heading. The empty space in the last row shows "More coming soon". New templates carry a NEW badge. Click the one closest to the jig you want to make.

Template	When to use
Shadow Board	Lay out tools or parts on a tray with shape-matched cutouts. Can also be created from a photo.
Storage Box	Organize and transport parts or tools in a box with dividers, handle, and lid.
Position-Fixing Plate	Fix the position of a workpiece. Suitable for welding, assembly, machining, etc.
Clamp Fixture	Clamp the workpiece from both sides for cutting or welding.
Positioning Plate	Position parts accurately and repeatably with pin holes and screw holes.
V-Block Jig	Hold round bars or shafts steadily in a V-shaped groove.
Assembly Guide	Drop parts into a frame for accurate assembly. Suited for low-volume, high-mix production.
Gauge Tool	Perform Go/No-Go judgement or dimension checks. For QC processes.
Sign Plate	You want a safety or zone sign. Finished by engraving text.
Mold	You want a casting mold that copies the shape of a product exactly. Urethane, silicone, plaster and similar materials are poured in at room temperature.
Pocket Jig	You want to drop a panel into a pocket and line up a backing sheet with locating pins — for example, for applying double-sided tape.

Step structures and detailed settings for each template are summarized in Appendix A.

3-2 Filling in the steps

After choosing a template, the editor opens. The left panel shows one step at a time. Within a step, the settings are grouped under headings such as "A · Base plate" and "B · Work Setup".

1. The top of the left panel shows the current step and total with a progress bar, e.g. "STEP 1/5 — Base Setup".
2. Fill in each step's fields from top to bottom. Some items provide S/M/L preset buttons. When you type a size by hand, the preset button that matches it lights up.
3. For finer control, open "Advanced".
4. When done, press "Next — (next step name)" at the bottom to advance. Use "Back" to return to the previous step.
5. On the last step, press "Generate Shape" to produce the jig ("Generate & Review" for the storage box, "Generate Plate" for the sign plate, "Build the mold" for the mold).

To pick a different template

Click "Back to template selection" in the upper-left. "Return to template selection?" appears; press "Return" to go back to the template list. The current work is not saved, so save it first if you need it (3-5).

You can also add labels to the finished jig in Template mode.

3-3 Progress and completion messages

While the shape is generated, the notice area at the bottom shows "Generating shape...". When it finishes, "Shape generation completed" appears there and the left panel shows "Jig complete" together with an "Add Label" button. Complex shapes can take longer; please wait.

When it is finished, the heading at the top of the left panel changes to "Done — all N steps" and the progress bar runs all the way. When you close the completion display, the heading goes back to the step you were on.

For the mold, the left panel shows "The mold is ready". While it is being built, the current stage is shown, for example "Building the mold...".

3-4 Engraving text on a jig (labels)

Once the jig is finished, use "Add Label" to engrave text such as a part number or an orientation mark.

- ① Press "Add Label" on the completion screen.
- ② Type the text, then choose "Engrave" or "Emboss", the line alignment (Center / Left), the depth and size, the position and the rotation. You can also drag the text in the 3D view to move it.
- ③ Press "Engrave Label".

Engraving several labels

Each press of "Engrave Label" adds one more label; it does not replace the previous one. Pressing it again without changing anything stacks a second label in the same place.

- To correct the text, press "Replace last label". Only the label you engraved last is replaced with what the panel now holds. The button appears only when there is at least one label.
- "Delete Label" removes every label on that plate. Labels cannot be deleted one by one.
- Labels other than the last one cannot be picked and edited. To change one, delete them all and engrave them again.

Saving and regenerating

The engraved text is kept as part of the jig's settings.

- Change the dimensions and regenerate the shape: every label is engraved again.
- Reopening the label panel shows the content and position of the label you engraved last.
- Save the project and open it again: the text, position, size, rotation and engrave/emboss setting of every label are preserved.

Text on a storage box lid is the one exception. It does go into the exported file, but it disappears if you then change the settings and regenerate the shape, and it is not saved in the project file. On the lid, "Replace last label" and "Delete Label" are not shown, and each press of "Engrave Label" adds another line of text. Engrave it again after regenerating. With no lid, the text is engraved on the box body and behaves the same way.

On a mold, the text is engraved on the top face of an open mold, or only on the top face of the upper half of a split mold (never on the lower half). Each press of "Engrave Label" adds another label, and "Replace last label" and "Delete Label" are not shown. To remove text, use "Remove the engraved text" in the result section to remove all of it, then engrave again. Engraved text stays when you rebuild the mold and when you reopen the project. If the text overlaps the sprue or a vent, a notice appears; move the label.

Compatibility with earlier versions

Projects saved with a version earlier than V1.1.0 do not contain the label information. If you open one and regenerate the shape, the engraved text is lost (it stays as long as you do not regenerate). Igniform tells you on screen when you are about to regenerate, so engrave the text again afterwards if you need it.

If you open a project saved with V1.2.0 in V1.1.0, only the first label is shown.
V1.2.0 did not save the second and later labels engraved in a template. When you open a template project saved with V1.2.0, only the first label is shown. Engrave the rest again if needed, then save.

3-5 Saving and picking up where you left off

Press the "Save" button at the top of the screen (or Ctrl+S) at any point and your work is saved as a project file (.cad). "Saved: (file name)" appears when it succeeds; if it fails, the reason is shown on screen.

Saving removes the draft kept so far (3-6). When you close the app with unsaved changes, "Confirm exit" appears and you can choose "Save and close", "Close without saving" or "Cancel". It does not appear if you close right after saving.

Open a saved file from "Open Project" on the launch screen. If the project was made from a template, Igniform returns you to that template's editing screen with the step, the values you entered and the workpiece layout exactly as you saved them, so you can carry straight on.

When you open a mold project, the mold is rebuilt from the saved settings ("Rebuilding the mold from the saved settings..." is shown). If it cannot be rebuilt, the reason is shown and the project opens in the state before the mold was built.

- A file that uses a template this version does not have, or a damaged file, is not opened; the reason is shown instead.
- Projects saved with the Mold or Pocket Jig template cannot be opened in V1.2.0 or earlier.
- An older file with no template information opens in Freeform Editing. Such a file cannot be re-edited as a template.
- When you open a project saved with V1.2.0 or earlier and rebuild the shape, the warp-prevention grooves now avoid holes, so their length and the way they are divided may change slightly (their depth and width stay the same).

Notices shown when a project is opened

If part of a project cannot be restored, Igniform tells you as soon as the file is opened. The most common notices are below; Appendix B lists what to do.

- "Some shapes could not be restored" — this can appear for files saved with an earlier version. What you see on screen is correct; save the project once and carry on. Saving repairs the file.
- "Some jigs lost their original shape" — the jigs listed are opened with the shape you see now. Deleting a label will leave the engraving in place, so avoid re-molding them; create them again if you need to change their shape.
- "Some objects could not be loaded" — they are kept inside the file as they were, and saving will not remove them. You can carry on working.
- "The source work could not be found" — the jigs opened unchanged, but they cannot be rebuilt to fit the workpiece.

- "This project contains jigs that cannot be exported" — the jigs listed cannot be written out by this version. Create them again if you need them.

The same notices may also appear when you restore from a draft (3-6).

3-6 Automatic drafts and restoring your work

Work in progress is kept automatically as a draft, even if you have not saved it to a file. A draft is made about 20 seconds after your last action and is then updated every 1–5 minutes, depending on how heavy the work is. While a calculation such as forming is running, it waits until the calculation ends.

The draft is removed when you save, and when you close the app normally (including when you choose "Close without saving").

If the app ended without saving — a power cut, a forced shutdown, the display stopping unexpectedly and so on — "Restore your previous work?" appears before the launch screen the next time you start it.

- One item: "There is work from a session that ended without saving (date / name). Restore it?" appears. Choose "Restore" or "Discard".
- Two or more items (for example, when more than one Igniform window was open): choose "Restore" or "Discard" for each item in the list. You can restore one item per launch. The rest, and any you set aside with "Later", are offered again at the next launch.
- "Discard all" shows "All N items will be discarded. This cannot be undone." They are removed only when you press "Discard" again. "Cancel" takes you back to the list.

Restored work opens as an "Untitled" project that has not been saved yet. The original file is not overwritten, so press "Save" and choose where to save it if you want to keep going. Work done in a template comes back on the same step of the template editor.

Drafts are kept only on this PC and are removed automatically after 14 days. If you start the app again within 3 minutes of a forced shutdown, the prompt may not appear; it will appear at the next launch.

3-7 When the 3D view stops

If the graphics processing of the PC stops for a moment, the 3D view may stay white. In most cases it recovers by itself within a few seconds and "The 3D view has been restored" is shown.

If it has not recovered after 5 seconds, the "The 3D view has stopped" screen appears. Your work has not been lost at this point.

- ① Press "Save now" to save. Once the save succeeds, the "Restart" button is highlighted.
- ② Press "Restart" to restart the app. If you have not saved, a draft is kept before the restart and you are asked whether to restore it after the app starts (3-6).

If you close the screen with "Close", a bar saying "The 3D view is stopped — please restart." stays at the bottom of the screen. From the bar you can open the screen again, or restart straight away with "Restart (your work will be restored)". If the view stops several times in a short time, "The 3D view keeps stopping" is shown; if it switches to a slower drawing method, "The 3D view is running in a slow fallback mode" is shown. Restarting puts both back to normal.

If the whole display stops, "The display stopped unexpectedly" is shown. Press "Restart" to restore from the latest draft. In this case the last few minutes of work after the latest draft may not come back.

Chapter 4 Exporting data

Export the finished jig with the "Export" button in the upper-right. The flow is the same in both Freeform and Template modes.

> Note: many of the export screens are still shown in Japanese.

4-1 Export procedure

1. Press the "Export" button in the upper-right.
2. Choose the file format — "STL" or "STEP" — and press "Next". STL is for 3D printing and exports quickly; STEP is for re-editing in CAD software.
3. The first time you export, Igniform asks which way up to write the data (see 4-4).
4. Choose which parts to export and where they go (see 4-2). If there is only one part, this screen is skipped and you go straight to choosing where to save.
5. Press "Export" to start. "Export complete" appears when done.

> To save your work itself so you can continue later, use the "Save" button at the top of the screen (or Ctrl+S) rather than Export (see 3-5).

The mold and the pocket jig cannot be exported until the mold (jig) has been built. In the pocket jig, loading a different panel after building detaches the jig that was built.

Merging fixtures and support bases

A jig with screw-lock fixtures is exported as a single body: the fixtures are merged into the jig. A support base is merged with the pillars standing on it in the same way, so the pillars are not listed separately among the parts to export. "Merging fixtures into the jig body..." is shown while this runs.

If the merge fails, "Could not merge the fixtures" is shown and the parts are exported separately, as before. Most slicers merge overlapping bodies automatically, so 3D printing is usually unaffected — but note this if you need a single solid in CAD.

4-2 Choosing where the files go

When there is more than one part (a storage box body and its lid, the upper and lower halves of a split mold, or several jigs exported together), you can choose between three destinations.

Destination	What it does
Combine into one file (keeping the layout) [default]	Writes everything as a single file with the jigs in the same positions as on screen. When you load it into your slicer, the layout is exactly what you saw. The shapes themselves are not changed.
Separate files (choose a folder)	Pick a folder and save one file per part. Use this when printing them one at a time.
Single ZIP	Pack all parts into one ZIP file.

"Combine into one file" is offered for STL only. STEP is meant for re-editing in CAD, and combining would stop the parts being handled individually. With STEP, choose "Separate files" or "Single ZIP".

The storage box does not offer this choice, because the body and the lid are printed separately.

A split mold exports two parts, the upper and the lower half. For STL the default is "Combine into one file": the upper half is turned with its parting face down and placed next to the lower half (not as it is laid out on screen). With "Separate files" you get two files whose names end in _lower (lower half) and _upper (upper half). Either way, the parts are written in an orientation that is ready to print.

About the STEP format

STEP takes longer than STL, because the data is converted into a precise form that CAD software can edit. Progress is shown while it converts, so please wait. If it cannot finish in time, "STEP conversion timed out" is shown — export as STL instead, or simplify the shape. A file in another format is never saved silently in its place.

A jig whose shape is broken (not a closed solid) cannot be written as STEP. "Cannot export STEP" is shown and no file is created. Fix it with Optimize → Printability and try again.

If faces are found to be missing, or seams not closed, after the conversion, "The STEP shape is incomplete" is shown. The file has already been saved, so open it in CAD and check it.

4-3 The warning before export

If the last printability check found at least one 🔴 Critical issue, a "Printability warning" appears before the export. It does not appear for 🟡 Warning or 🔵 Info items alone. You have three choices.

- Open printability check [default] — stops the export and shows the results of the printability check. Press "Highlight location" on an issue to show where it is in the 3D view. Items that can be fixed automatically get a fix button right there. Fix them, then export again.
- Ignore and export — exports as it is, in full knowledge of the warning.
- Cancel — stops the export.

See Chapter 5 for the Printability Check itself.

4-4 Export orientation

Which way is "up" in the exported data depends on the software you use. The first time you export, Igniform asks which one you want.

- Z axis up (standard) — for 3D printers and slicers. This is the orientation Igniform has always used.
- Y axis up — opens with the correct orientation in SolidWorks.

Choose an orientation and press "Export with this orientation" to continue. "Remember this and do not ask again" is ticked from the start, so the orientation you choose is remembered and you are not asked next time. Untick it if you would rather choose every time. You can change it at any time under "Export orientation" in the Quick Settings panel.

Either choice keeps the shape and the size exactly the same; only the way the part is laid down changes.

Chapter 5 Printability Check and its settings

Printability Check (in Freeform Editing) automatically verifies that the jig can be printed cleanly on a 3D printer. This chapter explains the results panel and the settings.

5-1 Reading the results

- Findings are shown at three levels: "🔴 Critical · 🟡 Warning · 🔵 Info", with counts in the header (for example "🔴 Critical 2 · 🟡 Warning 1") and a verdict line underneath, such as "✓ Printable" or "⍰ Critical errors remain".
- When everything is clear, "✅ No issues detected" appears.
- 🔴 Critical cards are listed expanded. 🟡 Warning items go into a collapsed "Quality notes" group and 🔵 Info items into a "Details" group.
- Click an item to view details and "💡 Recommended action". On a critical item with no auto-fix button, "How to fix" is shown instead, highlighted.
- Press "Highlight location" to show where the problem is in the 3D view. The panel also offers "3D highlight" and "Re-check".
- Open "📊 Print estimate" in the panel for an estimate of print time and material (it uses the settings in 5-3 ④).
- Checks that could not run are reported as "Items that could not be checked" under 🔵 Info, so that they are not mistaken for a clean result — for example when the self-intersection check is skipped because the shape is too large, or when the data-integrity and overlapping-data checks are skipped because the forming calculation has already confirmed that the shape is properly closed.

Auto-fix

Some items have an auto-fix button so you can resolve them in one click. A confirmation appears; press "Apply". Auto-fixes can be undone with Ctrl+Z. "Fix all" at the top of the panel applies them together.

Button	Check	What it does
Fix data integrity	Manifold check	Re-forms the jig to repair the data. The check is run again automatically.
Fix overlapping data	Self-intersection	Re-forms the jig to remove intersecting faces. The check is run again automatically.
Remove separated parts	Floating island	Removes only small detached pieces (under 5% of the model). Large separate pieces are kept — select the one you do not need and delete it, or re-form the jig.
Add warp-prevention grooves to the base	Warp risk	Adds warp-prevention grooves to the bottom face of the jig.
Scale jig to N%	Build volume	Scales the workpiece down so that the jig fits the printer. Not available while no workpiece is loaded. Re-form the jig after applying it.

Auto-fix results and warp-prevention grooves are now reflected in the exported STL/STEP and in the project file as well. If the critical count does not drop after an auto-fix, a separate notice is shown; follow "How to fix" and revise the dimensions or the shape.

5-2 What is checked

Checks come in two kinds: "Common checks" (same on every printer) and "Per-profile checks" (dependent on the selected printer). The severity column shows how each finding is reported.

Common checks (always on)

Item	Meaning	Severity
Manifold check (data integrity)	Detects data defects that may cause STL export to fail. The check is skipped for a shape already confirmed by the forming calculation.	Critical
Floating island detection (disconnected parts)	Detects small chunks separated from the main body, which may detach during printing.	Critical
Self-intersection detection (overlapping data)	Detects intersecting faces, which can fail at export. The check is skipped for very large shapes (over 200,000 triangles) and for shapes already confirmed by the forming calculation.	Critical
Base plate contact (gap above the base plate)	Detects a jig left floating above the base plate with no fixture attached — a jig for which you chose "Do not add" when adding a fixture (2-5).	Warning
Warp risk	Evaluates the chance of corners warping during printing.	Info

Per-profile checks (printer-dependent)

These can be toggled on/off, and their thresholds adjusted.

Item	Meaning	Default	Severity
Overhang (needs support)	Detects overhanging ceilings that may require supports.	Max 45°	Warning above 10% of down-facing area, otherwise Info
Thin wall (thin walls)	Detects walls that may be too thin for adequate strength.	Min 0.8 mm	Critical under half the threshold, otherwise Warning
Sharp edge (sharp corners)	Detects sharp corners prone to stress concentration. The vertical edges of a box get an automatic fillet, while the top and bottom edges and cylinders or cones are deliberately left sharp. The rims of a cavity, where the workpiece touches, can be chamfered with "Chamfer on part-contact faces" in Edit Jig (after forming; 2-2).	Min 30°	Warning
Bed contact area (bed adhesion)	Checks whether the area in contact with the bed is sufficient.	Min bed contact area 100 mm², and more	Warning
Build volume (printer size limit)	Detects a jig that exceeds the printer build volume.	The build volume of the profile	Critical

5-3 Opening the Printability Check Settings

Press the "Settings..." button on the "Printability check" row of the "Quick Settings" panel to open the "Printability Check Settings" dialog. Here you select the printer and adjust the thresholds.

① Printer Profile

Select your 3D printer. Choosing a profile sets the build volume (X/Y/Z), nozzle diameter, layer height, and process (FDM/SLA/SLS). Profiles are grouped as "Built-in", "Preset" and "User-created". All seven profiles below use a 0.4 mm nozzle, a 0.2 mm layer height and FDM.

Printer	Build volume (X × Y × Z mm)
Generic FDM (default)	220 × 220 × 250
Raise3D Pro2	305 × 305 × 300
Raise3D Pro2 Plus	305 × 305 × 605
Raise3D Pro3	300 × 300 × 300
Raise3D Pro3 Plus	300 × 300 × 605
Raise3D E2	330 × 240 × 240
Raise3D E2-CF	330 × 240 × 240

> The built-in and preset profiles (the seven above) cannot be edited. To change values, press "Duplicate" to create your own copy (User-created) and edit that. Copies can be removed with "Delete".

② Common Checks

Manifold check, floating island detection, self-intersection detection, base plate contact and warp risk — five items that are always active and cannot be disabled.

③ Per-profile Checks

Overhang, thin wall, sharp edge, bed contact area and build volume can each be toggled with a checkbox. Each item shows its default value. Tick "Use custom value" to override the default, and press "Reset to default" to go back. Build volume has no adjustable threshold: the build volume of the profile is used.

If a value is out of range, "Input value is out of range" is shown and the settings cannot be saved.

④ Print Estimation Parameters

These values are used for the print time and material cost estimate ("Print estimate" in 5-1) and can be matched to your own environment. They cannot be edited on built-in or preset profiles, so adjust them on a duplicated profile.

Item	Default (range)
Print Speed	50 mm/s (10–300)
Infill Density	20% (0–100)
Filament Density	1.24 g/cm³ (0.5–3.0)
Filament Price	3000 per kg (0–50000)
Setup Time	5 min (0–60)

Press "Save" when done, or "Cancel" to discard (closing the dialog with × or Esc also discards the changes).

5-4 Other display settings (Quick settings)

The "Quick Settings" panel lists the following, from the top:

- Parallel Projection: switch the 3D view between orthographic and perspective projection.
- Interface Language: change the UI language.
- Theme: switches the interface between light and dark colours.
- Export orientation: switches which way up exported data is written (see 4-4).
- Camera Sensitivity: adjusts pan and zoom sensitivity separately, in five steps (Slow / Slower / Normal / Faster / Fast).
- Printability check: "Settings..." opens the Printability Check Settings (see 5-3).
- License: shows the expiration of your license (see 6-2).
- Version: shows the version of Igniform you are running.

Chapter 6 License Management

Igniform requires a registered license. Licenses are issued tied to your PC.

6-1 First launch and registration

Step 1: Send your Machine ID to support

When Igniform is launched for the first time, the license registration screen appears. The "Machine ID" identifying this PC is shown on screen — please send it to support.

Step 2: Receive the license file

Based on the Machine ID you provided, support will issue a license file (or license key) for this specific PC. Follow the support team's instructions to receive it.

Step 3: Register the license

Drag and drop the license file onto the designated area on the registration screen, or paste the license key into the input field, then register. Once registered, Igniform will launch normally.

> Licenses are bound to the Machine ID of the PC they were issued for. To use Igniform on a different PC, send that PC's Machine ID to support and request a new license.

6-2 Checking and renewing expiration

The current expiration is shown in the "License" row of the "Quick Settings" panel, just above "Version". "Expires: " is followed by the date and the time (the format follows the regional settings of your PC). If the license has lapsed, "Expired: " is shown instead.

> **Renew well before expiration**
>
> A reminder appears in the application from 30 days before the expiration. It becomes more prominent as the date approaches, and is marked with ⚠️ in the last 7 days. When you see it, send your Machine ID to support and request a renewal in good time. If the license expires, Igniform cannot be used until you register a new one.

Chapter 7 Creating jigs from photos

Capture a workpiece outline from a photo and use it to build a jig. The outline can be placed onto a jig plate or exported directly as an STL file.

Start from "Create from Photo" on the launch screen or the "Create from Photo" button in other screens (also available from the Shadow Board template). If you minimise the photo window, press "Create from Photo" in the left menu again to bring it back.

7-1 Four steps

Step	What you do
① Shooting guide	Review tips for taking good photos. You can also download a capture surface (capture tray / capture mat) here.
② Import image	Load the photo. The app uses the four corner markers to auto-rectify the perspective.
③ Trace outline	The tool's outline is extracted automatically. Click on the image to fix areas that aren't captured well.
④ Finalize shape	Choose the cross-section type (flat or cylinder) and thickness, set the margin (offset), and you're done. You can also save the result as an STL file.

7-2 Preparing the capture surface

For shooting, use a "capture surface" with markers at the four corners. You can download it from the Shooting Guide screen; two types are available, depending on how you want to use it.

Surface	Characteristics	How to prepare
Capture tray (3D-printed, recommended)	Gives the cleanest outline and a stable platform (recommended).	Download the STL and the marker-sticker PDF. 3D-print the tray and attach the marker stickers at the four corners.
Capture mat (paper print, easy)	Place the workpiece on the printed sheet and shoot. Handy when no 3D printer is available.	Print at 100% (actual size) on A4 or A3 paper, place the workpiece on top, and shoot.

> We recommend 3D-printing the capture tray first. It is stable, gives clean outlines, and is reusable. If you can't 3D-print right away, start with the printed capture mat.

7-3 Tips for good photos

1. Place the workpiece on the capture surface (tray or mat) and shoot from directly above.
2. Keep all four corner markers in frame (three are enough for correction).
3. Don't let the workpiece or your hand hide the markers.
4. Use bright, even lighting (avoid harsh shadows and glare).
5. Highly polished metal or blades can be hard to capture; adjust lighting or fix the trace manually on the next screen.

> If outline extraction fails, switch to a darker background, keep the tool within the capture area, and retake the photo. You can also edit the outline manually in Step ③.

7-4 Editing the outline (Step ③)

You can refine the auto-traced outline with the mouse. There are two modes: "Edit contour" and "Add contour".

- Wheel: zoom in/out around the cursor.
- Shift + drag, or middle-click drag: pan the image.
- "Edit contour" mode: drag a point to move it; click near an edge to insert a point.
- "Add contour" mode: left click to add a point.
- Right click: delete a point (works in either mode).

When ready, press "Send to Igniform" — the workpiece is placed onto the jig plate exactly as positioned in the photo. To keep just the outline as a file, use "Save STL" to export an STL.

Appendix A Settings per template

Step structure and key inputs for each template. Numbers in parentheses are typical defaults (mm, etc.). "A · ..." and "B · ..." are the headings of the setting groups inside a step.

A-1 Shadow Board (5 steps)

Tray-style storage jig with tool-shaped cutouts.

Step	Main inputs
Plate setup	Small / Large presets (S = 150×210 mm, L = 227×320 mm) / Width (150) / Depth (210) / Thickness (18). Advanced: corner radius (5), anti-warp groove (on)
Add workpiece	"Add from STL", "Create from photo", Move / Rotate (Z) / Flip vertically (numeric entry available), Alignment spacing (5), Auto-align
Pocket (Work Fit)	Pocket Depth (10) / Clearance (0.5; tight 0.2, std 0.5, loose 1.0). Advanced: Pocket Corner R (2), Floor Shape (Flat / Tapered)
Layout & tuning	D · Finger Notch (off by default): shape (Semicircle (U) / Rectangle), Notch Width (20), Notch Length (10), Placement (Automatic — you can limit the target edges — or Manual) Wall thickness: Wall Between (8), Wall Edge (10) Mounting Screw Holes (off by default): hole Ø9, layout (corners / edge midpoints), hole shape (round / X slot / Y slot)
Generate	"Generate Shape"

The position and orientation of a workpiece can also be entered as numbers. Select "Move" or "Rotate" and input fields appear just below the buttons: Move takes left-right (X) and front-back (Y) in millimetres, Rotate takes the angle on the plate in degrees. Positions are measured from the centre of the base plate. The numbers update as you drag, too.

If the workpieces do not fit in a single row, they are automatically arranged in several rows. If they still do not fit on the plate, Igniform tells you on screen how large the plate needs to be.

A-2 Storage Box (6–8 steps)

A box for organizing parts and tools. Built from workpiece dimensions or 3D data, with dividers.

Step	Main inputs
Workpiece	"Enter dimensions" (default) or "From data". For dimensions: width (60) / depth (40) / height (25). With "From data", load the part and its outer dimensions are shown
Orientation	Shown only with "From data". Adjust the orientation in the 3D view (5° steps). Advanced: rotation X/Y/Z, Origin Reference (centre of the outline / file origin)
Work Layout	"By count" (default): columns (1) / rows (1), each 1–20. "By size": width (200) / depth (150)

Step	Main inputs
Cells and dividers	Clearance (0.5) / cell size (fit to workpiece or manual: 61 wide × 41 long) / Divider Thickness (2, when there is more than one cell) / Divider Height (30) / Pocket Depth Mode (automatic or manual = 20, with "From data"). The cell count and box dimensions are fixed here
Pocket shape	Shown only with "From data". Drop-in cell (Recommended) / Slope Hugging / Easy Removal
Box build	Height above workpiece (10; with "By size", Height 60) / wall thickness (3) / floor thickness (3) / corner radius (2) / anti-warp groove (recommended, on)
Handle	Faces (front and back on by default, left and right off) / protrusion (20) / width (60) / thickness (5) / height (30)
Lid	Add lid (on) / lid type (inset by default / skirt) / thickness (3) / clearance (0.3) / skirt depth (4). With a skirt, Skirt Wall Thickness (3) as well. Finish with "Generate & Review"

Choosing "Enter dimensions" hides "Orientation" and "Pocket shape", giving 6 steps; "From data" gives 8 steps. The outer dimensions of the box are calculated from the workpiece size, cell count and clearance. To add dividers, set the columns or rows to 2 or more.

A-3 Position-Fixing Plate (5 steps)

Generic position-fixing jig for welding, assembly, and machining.

Step	Main inputs
Base Setup	A · Base plate: S (200×150) / M (250×200) / L (350×280) presets, W (250) / D (200) / Thick (25, thick recommended). Advanced: corner (3), anti-warp groove (on)
Work Setup	B · Work Setup: S (40×60) / M (80×120) / L (150×200) presets, W (80) / H (50) / D (120) / heat margin (10)
Stopper & Spatter Catcher	C · Stopper: shape (Block / Cylinder / L-Shape), layout (3 points by default / left-right / front-back / four sides), stopper height (20) and width (20) D · Spatter Catcher (on): Catcher Width (8) / Catcher Depth (3)
Fix Setup	E · Clamp holes (on): bolt size (M6 / M8 by default / M10 / M12), layout (corners / edge midpoints) F · Mounting Screw Holes (on): Ø11, layout, hole shape
Generate	"Generate Shape"

The "D · Spatter Catcher", "E · Clamp holes" and "F · Mounting Screw Holes" groups are shown collapsed in every step.

A-4 Clamp Fixture (4–6 steps)

Clamps the workpiece from both sides. Step count varies with settings.

Step	Main inputs
Base setup	Base plate presence and size
Workpiece selection	Specify block or import STL/STEP
Clamp setup	W (30) / H (50) / Clearance (0.1) / Bolt M8 / 0–4 bolts
Workpiece tuning (when imported)	Position/orientation, pocket on/off, clearance
Mount setup (when base present)	Mounting Screw Holes / T-slots (width, orientation)
Generate	"Generate"

A-5 Positioning Plate (5 steps)

Position parts precisely via pin and screw holes.

Step	Main inputs
Plate setup	S (150×100) / M (200×150) / L (300×200) presets, W (200) / D (150) / Thick (12). Advanced: corner (3), anti-warp groove (on)
Pin holes	B · Datum pin holes: pin diameter (3 / 4 / 5 / 6 / 8 / 10, default 6), Pattern (X-axis pair by default / Y-axis pair / triangle / square — the number of pins follows the pattern), pin spacing (100). Advanced: pin hole depth (10), pin hole type (through / blind)
Clamp holes	C · Clamp holes: Clamp Type (Screw Hole by default / Clamp Seat / None) Screw Hole: Screw Diameter (M3–M6, default M5); advanced: screw hole type (through / tapped / countersunk) Clamp Seat: seat diameter (20) / seat depth (3) / bolt hole diameter (8) Both: layout (corners / six points); advanced: edge margin (25)
Mount setup	D · Mounting Screw Holes (on): Ø9 E · Relief Slots (off): slot width (10) / slot depth (5) / direction (X / Y / cross) F · Datum Face (off): Stop Height (3) / edge; advanced: stop width (10)
Generate	"Generate Shape"

A-6 V-Block Jig (4 steps)

V-groove for steady holding of cylinders and shafts.

Step	Main inputs
Block setup	W (100) / D (80) / Block Height (60). Advanced: corner (3), anti-warp groove (on)
V-Groove	V angle (60° for small, precise shafts / 90° general purpose, default / 120° for large shafts) / shaft diameter (30); groove dimensions are computed automatically. Advanced: V Bottom (pointed by default / flat)
Mount setup	C · Clamp Slot (off): slot width (T12 / T14 by default / T18) / slot depth (8) / Slot Position (side / bottom) D · Mounting Screw Holes (on): Ø9
Generate	"Generate Shape"

A-7 Assembly Guide (4 steps)

Drop parts into a frame for accurate assembly.

Step	Main inputs
Frame size	A · Frame: S (100×80) / M (150×120) / L (200×160) presets, W (150) / D (120) / Block Height (30) / Wall Thickness (12). Advanced: corner (5), anti-warp groove (on)
Guide holes	Part shape (rectangular by default / round). Rectangular: part width (60) / part depth (40); round: part diameter (40). Clearance (tight 0.1 / standard 0.3 by default / loose 0.5; inner dimensions are computed) / Guide Depth (20). Advanced: inner corner radius (2)
Options	C · Guide pins (off): pin diameter (2–6, default 4) / layout (2 on X / 2 on Y / corners) / Pin Height (8) D · Mounting Screw Holes (on): Ø5, layout (edge midpoints by default)
Generate	"Generate Shape"

A-8 Gauge Tool (4 steps)

For Go/No-Go judgement and dimension checks.

Step	Main inputs
Plate setup	S (100×80) / M (150×120) / L (200×160) presets, W (150) / D (120) / Thick (15)
Gauge setup	Gauge Type (Go/No-Go by default / Flush / Height) Go/No-Go: Go Ø (20.0) / NoGo Ø (20.1) / Hole Pattern (along X by default / along Y / single hole) / Hole Spacing (60). A single hole has no NoGo diameter or spacing Flush: window shape (rectangular / round), part size, clearance (tight 0.02 / standard 0.05 / loose 0.1) Height: nominal height (10.0) / tolerance (±0.1) / gauge width (30)
Stop & Relief	C · Stop Face (on): stop face height (5), position (bottom edge by default) D · Relief Hole (off; shown only for Go/No-Go): Ø22 / depth (2) E · Mounting Screw Holes (on): Ø5, edge midpoints
Generate	"Generate Shape"

A-9 Sign Plate (3 steps)

A plate for safety and zone signage, finished by engraving text.

Step	Main inputs
Plate & Frame	Size presets (Small (100×60) / Medium (150×100) / Large (200×150)) / Width (150) / Height (100) / Thickness (3) / Add frame (on), frame height (0.8), frame width (2). Advanced: corner radius (3)
Mounting	Bolt holes (M4 by default / M5; countersink off by default) / Hanging holes (Ø5) / Magnet pockets (Ø10, depth 2.5) / None (double-sided tape)
Generate	"Generate Plate"

Once the plate is made, the text panel opens straight away (see 3-4), with "Emboss" preselected. Choosing magnet pockets or a countersink automatically raises the plate to the thickness those need.

A-10 Mold (3 steps)

A casting mold that copies the shape of the product (master) exactly, for materials poured at room temperature.

Step	Main inputs
Import the product	A · Product (master): import the product with "Import STL"; its file name, outer size and volume are shown. To load it again, use "Import a different STL". If the product is made of several separate lumps, "Use the largest lump only" (off by default)
Orientation and mold type	B · Orientation (draw direction): "Flip upside down", Rotate about X / Y / Z (0° each, −180 to 180°; you can also turn it with the gizmo in the 3D view). Faces that cannot be drawn upwards are shown in red, with their share C · Mold type and parting plane: Mold type (Open mold (poured from above) = default / Split mold (upper and lower halves)). For a split mold, the parting height (the recommended height by default) and "Back to the recommended height"
Mold settings	D · Outer shape and walls: Side wall (5 for an open mold / 10 for a split mold, 2–30), Bottom wall (5, 2–30), Top wall (5, split mold only), Vertical edge radius (3, 0–10), Top extension (0, 0–20, open mold only) E · Cavity: Shrink compensation (0%, −1.0 to +3.0%) F · Sprue and vents (split mold only): Sprue diameter (6, 3–15) with "Pick the sprue position by clicking" / "Back to the recommended position", Vent diameter (2, 1–4) with "Add a vent" / "Back to the suggested vents" When the settings are done, press "Build the mold". When the mold is ready, the result section shows its outer size, volume and the casting volume

The cavity copies the product exactly (clearance 0). The mold is drawn upwards. If faces that cannot be drawn upwards remain, or if the top of the product has no flat face, a split mold solves it (with an open mold, a "Switch to a split mold" button appears).

A split mold is shown "open", with the upper half lifted. "Show closed" shows the halves together. To change the sprue or vents after the mold is built, press "Edit the sprue and vents".

For shrink compensation, enter the shrinkage from the datasheet of the material you use. For materials that expand as they set, such as plaster, enter a negative number. After casting, the sprue and vents leave small stubs; cut them off. Molds for injection molding cannot be made.

A-11 Pocket Jig (4 steps)

A plate with a pocket the panel drops into. Locating pins line up the backing sheet.

Step	Main inputs
Panel	A · Panel: load one panel with "Add from STL". It is turned automatically so the longest side runs along X and the thinnest direction is the height (Z). "Rotate 90°", "Flip upside down", and the Size (W × D × T) display. To load it again, use "Load another STL"
Pocket & plate	B · Pocket: Pocket depth (default: panel thickness − 0.5; 0.5 up to plate thickness − 2), clearance (Tight 0.2 = default / Standard 0.5 / Loose 1.0; or Value (per side) 0–2.0). The panel protrusion is shown C · Plate: Margin (each side) (15, 5–100), Plate thickness (10, 4–30). Corner radius (5, 0–20) under the advanced settings
Locating pins	D · Locating pins: Pin type (Blind hole (for a metal pin) = default / Integral pin (printed with the plate)), Layout (2-2 (4 pins) = default / 2-3 (5 pins)), Nominal diameter (all pins) (3 / 4 / 5 / 6 / 8 / 10, default 5), Distance from pocket (4, 2–20). For holes, Hole clearance (0.2, 0–0.5) and Hole depth (6, 2–28); for pins, Pin height (5, 1–20)
Generate	"Generate shape"

Move a pin by changing its numbers in the list or by dragging its marker in the 3D view. Moving one switches the layout to "Manual". "Add pin" adds a pin, and "×" in the list removes one. "Reset to auto layout" goes back to the layout you had chosen. A pin that cannot be placed is shown in red in the list, and you cannot go on until it is fixed. If the plate margin is too small, "Set margin to N mm" fixes it.

An integral pin (printed with the plate) gets a C0.5 chamfer at its tip. Thin pins (nominal diameter under 4 mm) break easily, so holes are recommended for them. Even if the panel has holes, the pocket is made from its outline only. Whether the jig fits the build size of the printer is also shown.

Appendix B Troubleshooting

Common in-app messages and how to respond. Messages marked (JP) are still shown in Japanese.

Situation / Message	Suggested action
"Shape generation failed: " appears	The shape may be too complex. Try simpler dimensions or workpiece shape.
"Workpiece is not loaded" (JP)	Workpiece-linked modes need an STL/STEP file. Go back and load one.
"Clamp block is outside the base plate" (JP)	Enlarge the base plate or adjust clamp position/size.
"Please run 'Generate & Review' in Step 5 first" (JP)	In the storage box, run "Generate & Review" on the last step before exporting.
"The workpieces do not fit on the ◯×◯ mm board" (JP)	Enlarge the plate to the size shown, or use fewer workpieces. They have been laid out as far as they fit.

Situation / Message	Suggested action
"Generation was stopped because it exceeded the time limit (10 minutes)."	Slope Hugging ran for more than 10 minutes. Increasing the layer thickness shortens it; if you cannot wait, check the shape with "Easy Removal" first.
"Critical" issues remain in the printability check	Run auto-fix and re-check. Some critical items have no auto-fix button — thin walls under half the threshold, or a build-volume overflow with no workpiece loaded. Follow "How to fix" and revise the dimensions or the shape.
"Could not merge the fixtures"	The jig and its fixtures were exported as separate bodies. This is usually fine for 3D printing, but note it if you need a single solid in CAD.
"STEP conversion timed out" (JP)	The shape may be too complex. Export as STL, or simplify the shape and try again. A file in another format is never saved silently in its place.
"Cannot export STEP"	The shape is broken, so it cannot be converted. Fix it with Optimize → Printability and try again.
"The STEP shape is incomplete"	The file has been saved. Open it in CAD and check for missing faces or open seams.
"Some shapes could not be restored"	This can appear for files saved with an earlier version. Save the project once and carry on; saving repairs the file.
"Some objects could not be loaded"	They are kept inside the file as they were and saving will not remove them. You can carry on working.
"This project was saved by an older version and has no label information."	The file predates V1.1.0. Regenerating the shape loses any engraved text. Engrave it again afterwards if you need it.
"The plate was generated, but the label could not be engraved."	Review the text, size and depth, then press "Engrave Label" again. The current shape has no label in it.
"The label could not be re-applied after re-molding, so the jig has no label."	Press "Engrave Label" to engrave it again.
"Restore your previous work?"	Some work ended without being saved. "Restore" opens it where you left off. Restored work is "Untitled", so save it again (3-6).
"Another window is already restoring this."	Another open Igniform window is restoring that work. Use that window.
"Could not restore"	The draft is damaged and cannot be restored. Use a project file you saved yourself.
"The 3D view has stopped" / "The 3D view keeps stopping" / "The 3D view is running in a slow fallback mode"	Your work is not lost. Press "Save now", then "Restart". Even if you cannot save, you can restore at the next launch after restarting (3-7).
"The display stopped unexpectedly"	Press "Restart". You can restore from the latest draft (3-6).
"The warp-prevention grooves were left out because holes are in the way."	The grooves are left out to keep material around the holes. The jig can be used as it is. If you need the grooves, review the hole positions or the plate size.

Situation / Message	Suggested action
"The chamfer is too large and would cut thin parts loose." / "The chamfer could not be re-applied after re-molding."	Make the chamfer size under "Chamfer on part-contact faces" smaller and press "Apply chamfer" again (2-2).
"The product shape is not closed" / "The panel shape is not closed"	Export the data again as a closed shape from the software that created it (Mold / Pocket Jig).
"Even split top and bottom, N% of the surface cannot be released." / "There is no flat face at the top of the product, so the mold has no pour opening."	In the mold, change the orientation, or switch to a split mold and press "Back to the recommended height".
"There is no room for the dowels." / "There is nowhere to put the sprue"	In the mold, increase the side wall (the "Set the wall to N mm" button shown also does this), or review the parting height and the sprue position.
"Press [Build the mold] first."	Build the mold before exporting it.
"The mold could not be rebuilt from the saved settings, so the project opened without it."	Fix the settings using the reason shown, then press "Build the mold".
"There is not enough margin for the locating pins." / "N locating pin(s) cannot be placed."	In the pocket jig, press "Set margin to N mm", or move the pins in the red rows of the list at least 2 mm away from the pocket and the edge of the plate.
"The pocket is too deep" / "The holes are too deep" / "The panel does not stick out above the jig surface."	In the pocket jig, make the depth no more than the value shown, or make the plate thicker. If you want the panel to stick out a little, make the pocket shallower.
"The mold (jig) came out in N pieces and cannot be exported."	For the mold, review the walls, the orientation and the parting height; for the pocket jig, review the margin and the depth. Then build it again.
License registration screen or expiration warning appears	Send your Machine ID to support and request a license file. If expiration is approaching, request a renewal in good time.
"The project file is damaged and could not be read" / "This project uses a template that does not exist in this version"	Use another file or a backup. For the latter, open it with a version that has that template.