Smart Sheet Metal PRO

SketchUp Sheet Metal Extension

User Guide — Version 1.0

Developed by Atelier Createur Lab
support@lab-atelier.com | www.lab-atelier.com

1. Overview

Smart Sheet Metal PRO is a SketchUp extension that transforms sharp-edged 3D sheet metal models into realistic folded parts with accurate bend radii, automatically generates the flat pattern, and exports a DXF file ready for laser cutting or CNC press brake operation.

No need to switch to SolidWorks or Fusion 360 for simple to moderately complex sheet metal parts — everything stays inside SketchUp, in just a few clicks.

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2. System Requirements

Software	SketchUp 2021 or above
OS	Windows (Mac: not tested)
Units	Metric (mm) and Imperial (inches) — auto-detected
Language	EN / FR / DE / ES / IT — auto-detected

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3. Preparing Your Model

3.1 Mandatory Rules

Before using Smart Sheet Metal PRO, your model must follow these rules:

- The part must be a SketchUp Group or Component.
- The part must be a closed solid — no missing faces, no open geometry.
- Wall thickness must be constant throughout the part — no thicker or thinner areas.
- Fold edges must be sharp — no pre-existing fillets or radii. The plugin generates the bend radii automatically.
- Part edges must be perpendicular to the face — no chamfers or bevels.
- At each end of a fold, the adjacent edges must be perpendicular to the fold axis over a minimum length equal to the sheet thickness, measured from the interior fold edge. Without this clearance, the plugin cannot correctly generate the fillet or calculate the flat pattern.

> *! Any part that does not meet these rules will be rejected by the plugin with a color-coded error message displayed directly in SketchUp (see Section 5).*

3.2 Modeling Guidelines

Model your part as you normally would in SketchUp, but with sharp right-angle edges. Examples:

- An angle bracket: two perpendicular rectangles joined by a sharp edge.
- A U-channel: a base plate with two vertical flanges at right angles, with perpendicular clearance at each end of each fold equal to at least the sheet thickness.

> *i Holes, slots, and complex cutouts in flat faces are fully supported — they will be preserved in the flat pattern and exported to the HOLES layer in the DXF.*

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4. Step-by-Step Workflow

① Corner Rounding (optional)

Before the sheet metal processing, you can round sharp corners (cut relief notches) using the Corner Fillet tool.

1. Activate the Corner Fillet tool from the Smart Sheet Metal panel.
2. Hover over the part — it automatically switches to X-Ray mode.
3. Click the short edges you want to round.
4. Adjust the radius in the panel (default: 2 mm).

> *! This tool only works on a raw part, before sheet metal processing.*

② Sheet Metal Processing — 1 click

5. Click the base face of your part (the face you want to keep fixed during unfolding).
6. The plugin automatically detects the sheet thickness and the fold edges.
7. Select your material in the panel:

Material	K-Factor	Typical use
Mild Steel S235	0.44	Fabrication, metalwork
Stainless 304/316	0.42	Food industry, cladding
Aluminium	0.46	Lightweight structures, bodywork
Custom	Free input	Any specific material

8. Click Process — the plugin generates the bend fillets and produces the flat pattern in a single operation.

> *i The inside bend radius is automatically set equal to the sheet thickness. This parameter is not user-adjustable — it is the standard empirical rule for the majority of sheet metal parts.*
>
> *i The flat pattern is calculated using: L = (R_inside + K x t) x angle (in radians), which guarantees accurate developed lengths for press brake operation.*

③ View Toggle

Once processing is complete, two groups are created on two separate layers:

- Layer FOLDED — the 3D part with bend radii.
- Layer FLAT — the 2D flat pattern.

Use the toggle button in the panel to switch between the two views with a single click.

④ Laser Etching (optional)

Before DXF export, you can add etching or marking elements directly on the flat pattern:

- Text with font, size and bold options.
- Free-form shapes using native SketchUp tools (line, arc, rectangle...).
- Imported logos.

These elements are automatically assigned to the ETCH layer in the exported DXF.

⑤ DXF Export

Click Export DXF in the panel. The generated file contains 6 layers:

Layer	Content	Machine use
OUTLINE	Outer perimeter of the flat pattern	Laser / waterjet / plasma cutting
BENDS	Bend zones	Reference for CNC press brake operator
BEND_AXIS	Fold center lines	Bending reference on machine
BEND_NOTES	Bend annotations (angle, radius, material)	Operator information
HOLES	Interior holes and cutouts	Laser cutting of perforations
ETCH	Text, logos, markings	Laser engraving / marking

> *i Layer names are in English and are not translated — they follow standard CAM software conventions. The exported DXF is in standard 2D R12/2000 format, compatible with most CAM software (Lantek, Radan, JetCAM, LaserCut, etc.).*

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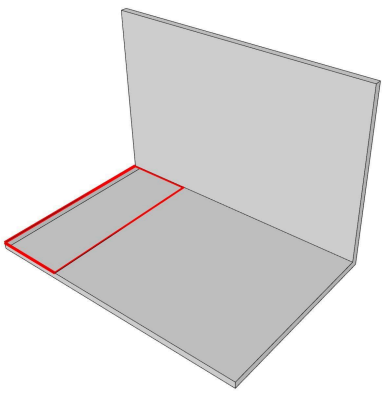
5. Quality Validation System

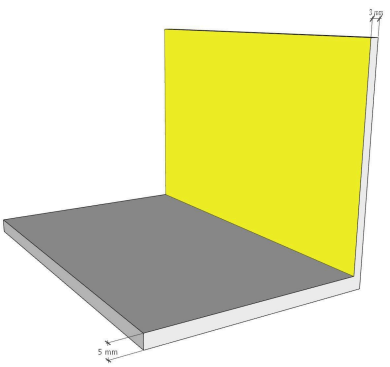
Before any processing, the plugin checks the geometry and displays errors directly in SketchUp by color-coding the problematic elements.

Error	Color	Cause and fix
Non-manifold	Red	Open solid — missing face or hole in geometry. Fix using SketchUp's Solid Inspector tool.
Inconsistent thickness	Yellow	Some faces have no parallel opposing face at the same distance. Check that the part has uniform wall thickness.
Beveled edge	Orange	An edge face is not perpendicular to the main face. This geometry cannot be laser cut. Correct the angle.
Partial fold / non-90°	Magenta	Angled cut or insufficient clearance at a fold end. Adjacent edges must be perpendicular to the fold over a length >= sheet thickness.

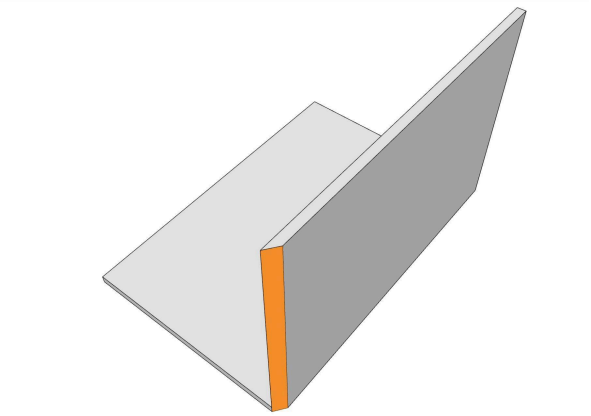
Visual examples of detected errors:



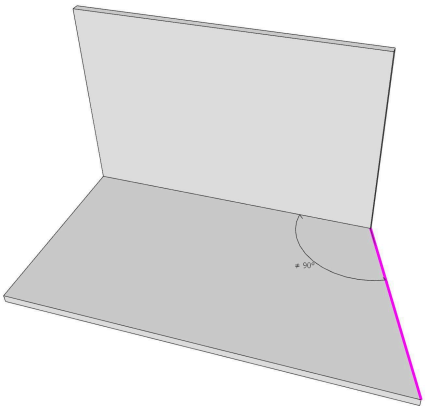
Non-manifold — edges in red



Inconsistent thickness — faces in yellow



Beveled edge — face in orange



Partial fold / non-90° — edge in magenta

6. Limitations — Version 1.0

What the plugin supports

- Constant-thickness parts with variable fold angles (not only 90°).
- Multiple folds on the same part, in different directions.
- Holes, slots, and complex cutouts in flat faces.
- Steel, stainless, aluminium, or any material with a custom K-factor.

What the plugin does not support (v1.0)

- Multi-part assemblies — each part must be processed separately.
- Variable-thickness parts (ribs, reinforced areas).
- Pre-existing fillets in the model — the part must have sharp edges before processing.
- Curved folds (cylindrical or conical developments).
- Automatic corner relief at adjacent fold intersections — must be modeled manually.
- 3D DXF export — only the 2D flat pattern is exported.

> *! The plugin has been tested on SketchUp 2026 on Windows. Mac compatibility is not guaranteed.*

7. Frequently Asked Questions

The plugin does not detect my sheet thickness — what should I do?

Make sure your part is a Group or Component. Open the group (double-click) and check that there are no floating faces unconnected to the main solid.

The flat pattern has incorrect dimensions — why?

Check the selected material and its K-factor. If your material is specific (heat-treated steel, composite...), use the Custom option and enter the K-factor provided by your press brake operator.

My DXF is read correctly by my CAM software but the layers are not displayed — why?

Some CAM software requires layer configuration at import. Check that DXF import is set to read layers. The exported format is DXF R12/2000 — if your software requires a specific version, please contact us.

Can I modify the part after processing?

Yes — use the Undo Bending button in the panel. This restores the part to the exact state it was in before you selected the base face, with no data loss.

Is the default K-factor accurate for my workshop?

The default values (K = 0.44 for mild steel S235) are widely-used industry standards. For maximum accuracy, ask your press brake operator for their specific bend tables and use the Custom option.

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8. Support & Contact

For any question, bug report, or suggestion:

- Email: support@lab-atelier.com
- Website: www.lab-atelier.com
- Extension Warehouse: search for 'Smart Sheet Metal PRO'