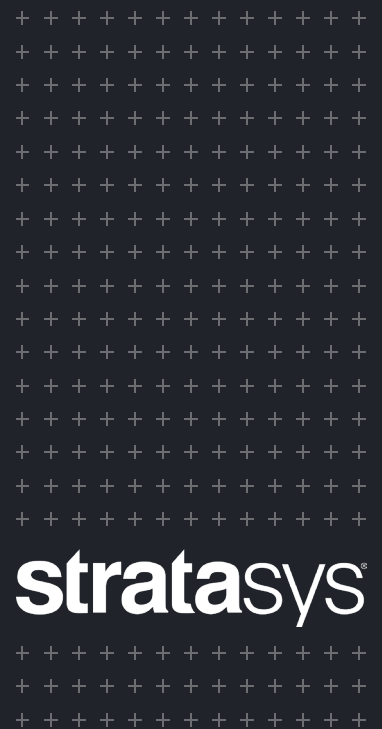


Additive Manufacturing for Custom Metal Form Tooling



stratasys



Additive Manufacturing for Custom Metal Form Tooling

This white paper presents the findings of a case study involving the use of FDM® additive manufacturing to create custom sheet metal form tooling. The design and processing techniques are specific to this use case but can be applied to most metal form tooling applications.

Section 1 - Application Overview

Traditional press brake tooling is machined from common tool steels such as A2, D2 or 4140. This tooling can be purchased “off the shelf” from several manufacturers (Wilson, Wila, Mate) in standard geometries. However, when a custom tool is needed, the cost and lead time increase drastically. Often, revisions occur during the design process after fit and function testing. These revisions add even more cost and lead time to an already expensive custom tool.

One way to minimize tooling costs during this development phase is to use additive manufacturing to produce custom-geometry tools. Additive manufacturing allows multiple iterations of custom, complex geometries to be printed without the costs and lead times associated with revising a metal tool. Stratasys FDM technology is capable of printing several high-strength thermoplastics that can withstand the pressure of forming thin-gauge metal. Additively manufactured tooling can be used as the production tool for volumes as high as 500 to 1000 pieces depending on the part material, geometry and tool material. For volumes that exceed 1000, it is recommended to use standard metal tooling.



Additive Manufacturing for Custom Metal Form Tooling

Section 2 – Custom Geometry vs. Standard Bends

Custom tools are commonly used to form complex geometry that can't be formed with traditional v-die press brake tooling as shown in Figure 2-1. Examples of these geometries include flares, offsets and ribs, where the primary function is to add rigidity and strength to a part without adding additional material.

2.1 Offset Bends

An offset bend shown in Figure 2-2, also referred to as a joggle bend, is a type of bending process commonly used in sheet metal fabrication in which two opposite bends are formed closer together than what regular v-die press brake tooling would allow.

Offset bends are primarily used to create additional stiffness in a part or create an offset to allow for a lap joint to mate two parts. Offset bends provide two main benefits. First, they're more accurate – the offset (distance between bends) and angle are set by the tool geometry rather than relying on the back gauges and ram of the press brake. Second, they lower the cost by reducing the cycle time. Rather than forming two individual bends, both bends are formed at the same time in one cycle of the ram.

It is recommended to design offset tools with a radius 2X the material thickness of the part being formed. Smaller radii are possible but the tool will deteriorate much faster due to a smaller surface area of the radius experiencing high loading conditions.

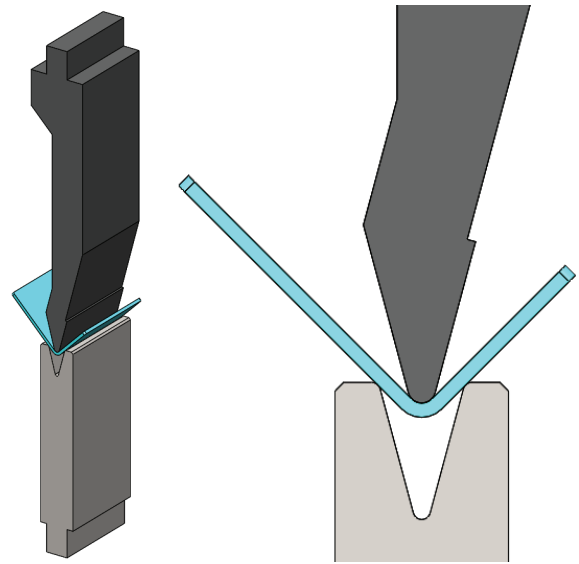


Figure 2-1: Standard v-die press brake tooling.

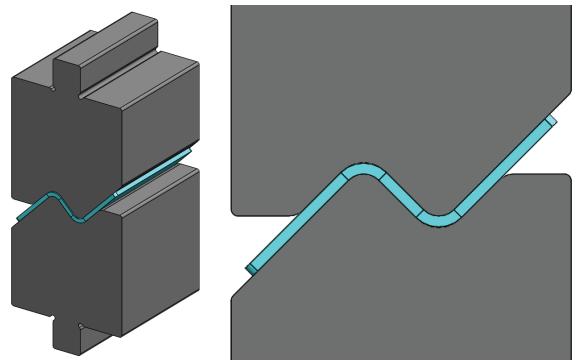


Figure 2-2: Offset bend tooling.

+ Additive Manufacturing for Custom Metal Form Tooling

2.2. Ribs

Ribs, like offset bends, are used to add stiffness to sheet metal parts. Common shapes include V, hat, and round (shown in Figure 2-3). These unique form shapes cannot be achieved with traditional v-die press brake tooling.

2.3 Dimples/Flares

In the aerospace and automotive industries, sheet metal parts often incorporate areas where material is removed to reduce weight. Forming tools that match the cutout shape are used at these locations to add a dimple or flare to restore rigidity. These tools can be purchased but fabricators are limited to the standard geometries and sizes and they can be very costly. As a comparison, the 2.5 inch circular flare tool shown in Figure 2-4 can be printed out of polycarbonate in 3.25 hours with a material cost of approximately \$58. In contrast, purchasing an off-the-shelf tool would cost \$130 and need to be shipped from a supplier. With the use of additive, it's possible to quickly and cost-effectively manufacture standard-geometry form tools as well as custom shapes such as triangular or rectangular flares as shown in Figures 2-4 and 2-5.

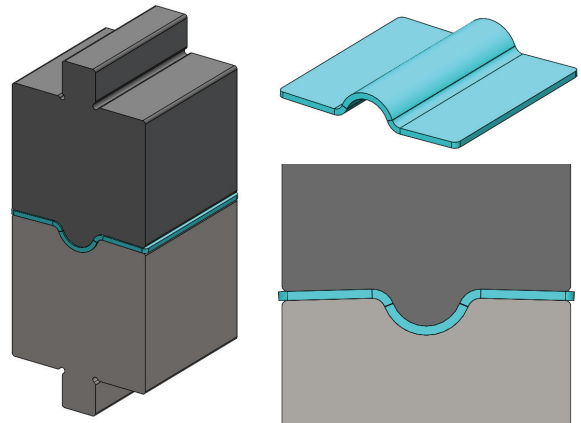


Figure 2-3: Round stiffener rib.



Figure 2-4: Custom flare tooling printed using polycarbonate.

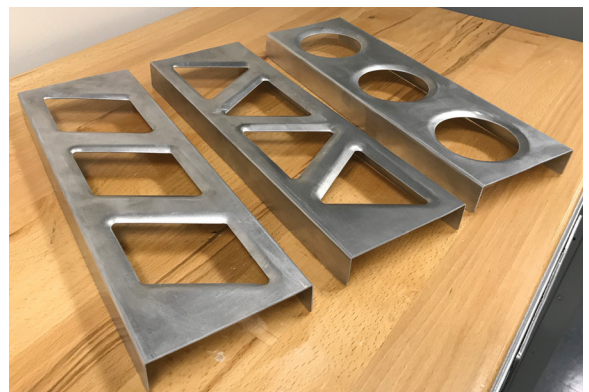


Figure 2-5: Examples of formed parts using the flare tooling shown in Figure 2-4.

+ Additive Manufacturing for Custom Metal Form Tooling

Section 3 – Material Recommendations and Processing Techniques

3.1 Materials

FDM technology has the capability to print a wide variety of thermoplastics with varying mechanical properties. For FDM forming tools, an important property to consider when selecting a material is the ultimate compression strength. Print orientation is a prime consideration because the compression strength of the material varies depending on the orientation the tools are printed in. Any FDM material can be used for metal forming applications but Stratasys recommends FDM Nylon 12CF, polycarbonate (PC), ULTEM™ 1010 resin or ULTEM™ 9085 resin, as they all offer high compression strength.

3.2 Processing

The offset dies were processed with solid infill and four contours for increased rigidity. The raster angle and contour are parallel at certain parts of the outer tool path, which creates an air gap because the raster tool path doesn't match up perfectly to the edge of the contour. To correct this, the raster angle was changed to 70° (see figure 3-1). With certain tool geometries, raster angle and width may need to be adjusted to eliminate air gaps in the tool.

Table 3-1: Compression strength of FDM materials in the XZ and ZX orientations.

Material	Compression Strength (PSI)	
	XZ	ZX
PC	10,000	9,200
ULTEM™ 1010 resin	19,500	15,100
ULTEM™ 9085 resin	26,200	13,100
FDM Nylon 12CF	9,670	13,310

XZ = X or "on edge"

XY = Y or "flat"

ZX = or "upright"

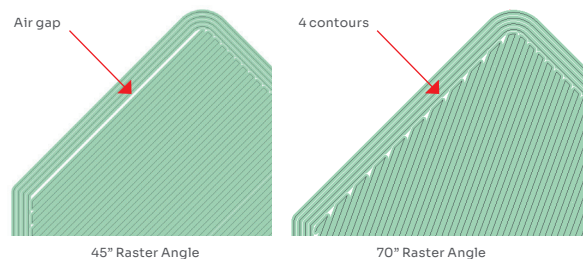
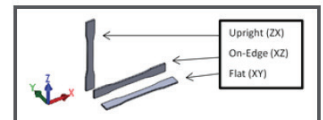


Figure 3-1: Air gap between raster and contour.

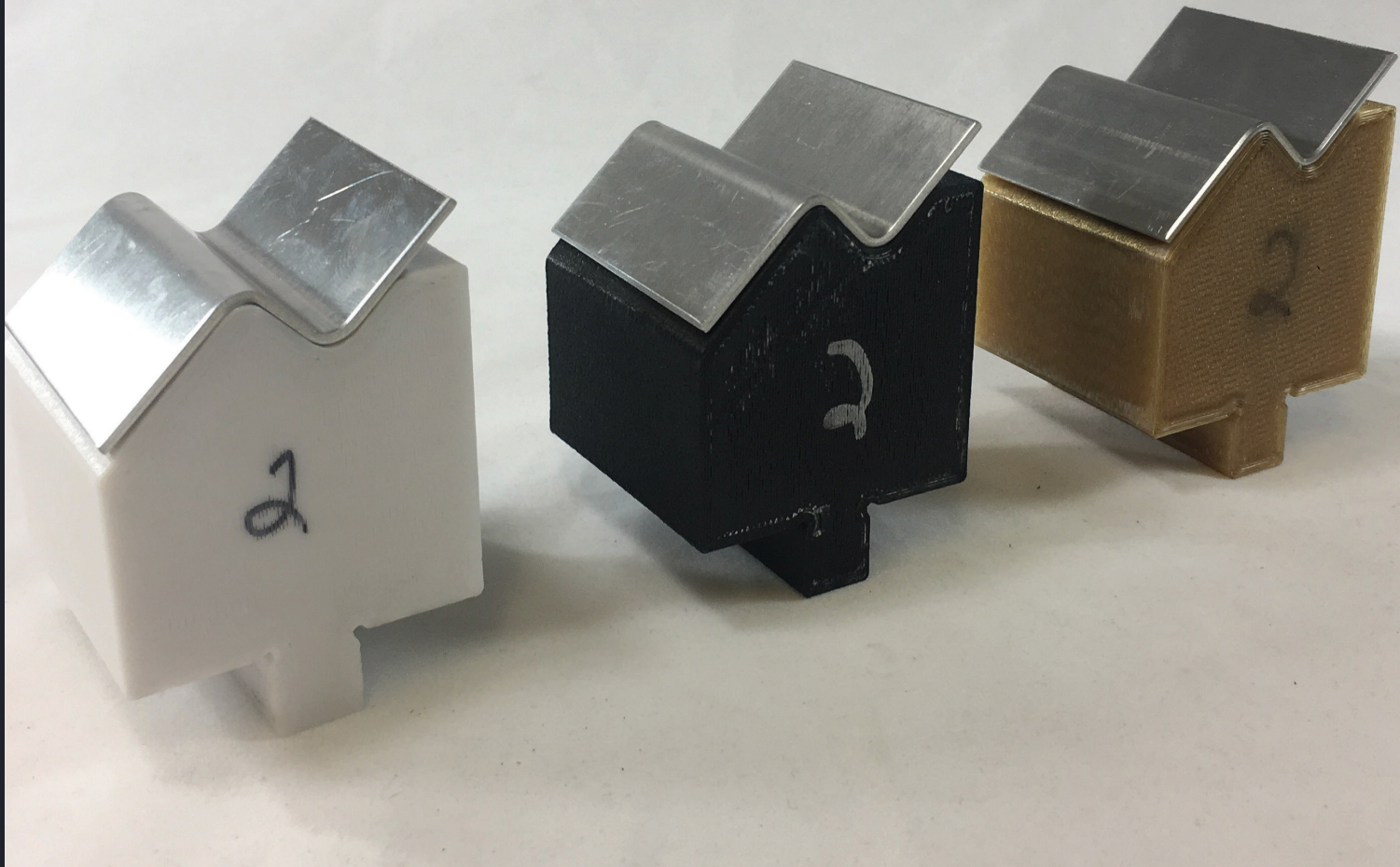


Figure 4-1: PC, Nylon 12CF, and ULTEM™ 1010 resin offset tools.

Section 4 – Testing Results

4.1 Offset Bend Tooling – Aluminum

The offset tools shown in Figure 2-2 were printed in ULTEM™ 1010 resin, Nylon12 CF and PC for testing purposes. Using a 60 ton Accurpress press brake, each tool was cycled 100 times at a forming pressure of 6 tons, bending 16 gauge (1.62 mm [0.064 in.]) 5052 aluminum. Parts 1, 50 and 100 were measured for offset distance and angle using a Keyence VHX 6000 microscope. Both angle and offset stayed consistent throughout the samples with very little variation on all three materials. Figure 4-2 shows the 100th formed part using an FDM Nylon 12CF tool, and Table 4-1 shows how close to nominal the desired measurements were. Common tolerances in sheet metal are $\pm 1^\circ$ for angular dimensions and ± 1 mm (0.039 in.) for linear dimensions. The parts formed using FDM tooling fall well within those tolerances, proving it is a feasible application.

Table 4-1: Part measurements.

	Nominal	Actual	Difference
Angle 1	90°	89.88°	-0.12°
Angle 2	90°	89.57°	-0.43°
Offset Distance	15.88 mm (0.625 in.)	15.88 mm (0.625 in.)	-0.3 mm (-0.012 in.)

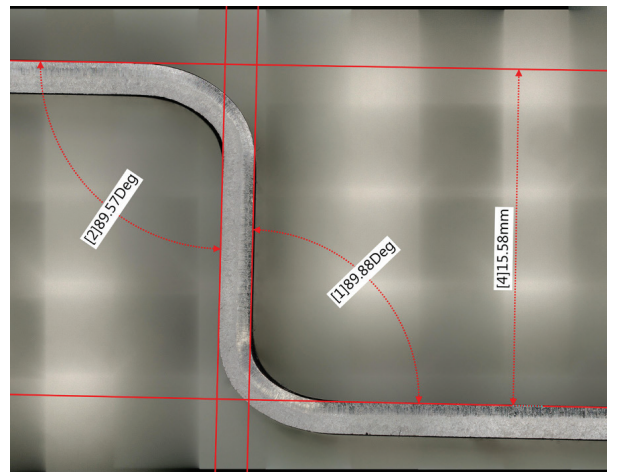


Figure 4-2: Part measurements on the 100th formed part.

+ Additive Manufacturing for Custom Metal Form Tooling

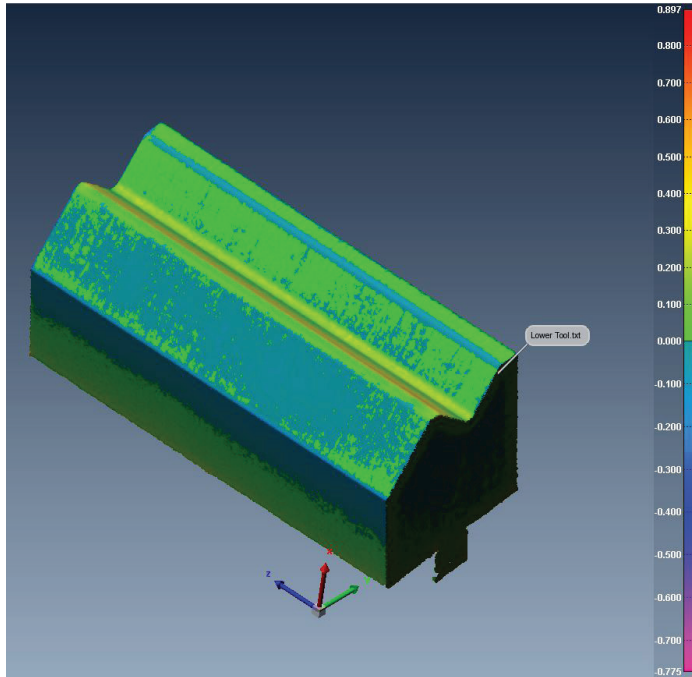


Figure 4-3: 3D scan of printed tool.

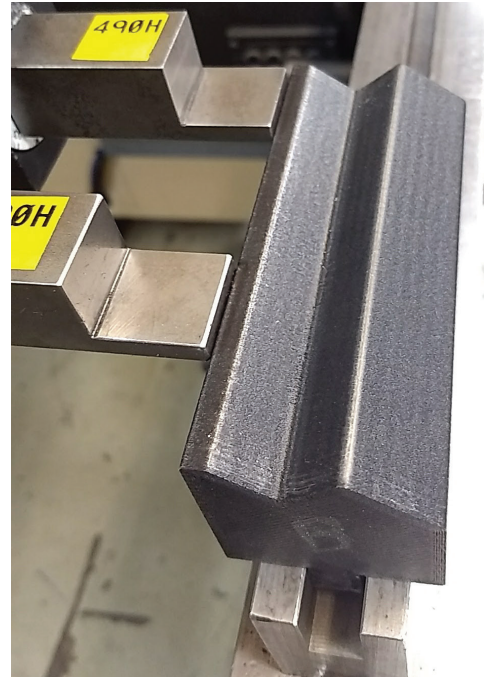


Figure 4-4: Printed tool shown in press brake.

4.2 Offset Bend Tooling – Steel

A second set of testing was done using a similar offset geometry with an FDM Nylon 12CF tool but with 16 gauge (1.52 mm [0.60 in.]) cold rolled steel. The tool was cycled 250 times at a forming pressure of 90 kN (9 tons) and measured in 50 cycle increments using a CMM (coordinate measuring machine). Again, the tool did not see significant wear. The only major feature that saw dimensional change was the tool radius. It began the testing measuring 3.2 mm (0.126 in.) and flattened out slightly to 3.6 mm (0.142 in.) – a 12% drop. But it still produced dimensionally accurate formed parts. A 3D scan of the tool after 250 cycles was compared against the nominal CAD data and it confirmed the 0.4 mm (0.016 in.) dimensional change that was seen with the CMM measurements.

Table 4-2: Part measurements.

	Nominal	Actual (250 cycles)	Difference
Angle 1	104.84°	105°	0.16°
Angle 2	104.84°	105.5°	0.66°
Offset Distanc	19.05 mm (0.75 in.)	18.59 mm (0.732 in.)	0.46 mm (0.018 in.)

Section 5 – Considerations

5.1 Tonnage Ratings

The following tools saw maximum forming pressures ranging between 50 and 100 kN (5 – 10 tons) and experienced no issues under load. Note that the max rating for a press brake tool is a function of its length and forming surface area. Taking the two examples previously mentioned above into account, the per-length rating of those tools is around 0.8 – 1.2 kN/mm (2 – 3 tons/inch) for max loading conditions. This general rule of thumb can be used for similar style tools printed with PC, FDM Nylon 12CF or ULTEM™ 1010 resin.

5.2 Max Material Thickness and Cycles

Aluminum and steel were the only materials tested in this case study. There are many other types of metals that have the capability of being formed using additively manufactured tooling. The following table is a general guideline for the max thicknesses that could successfully be formed in substantial quantities. Pushing the limits into thicker gauges is possible but it's dependent on the type of material being formed and the geometry of the tool.

The maximum number of cycles that tools are capable of lasting depends on the material being formed and the tool geometry. The above examples ranged from 100–250 cycles which is a good baseline. Maximum cycles can be expected to reach 500. In unique circumstances, tool life can get into the 750-cycle range. After 750, it is recommended to use metal tooling.

5.3 Spring Back

Spring back should be treated the same as when forming sheet metal with metal tools. Tools should be designed accordingly so parts can be over-bent with the printed tool and then spring back to the desired angular dimension. This is dependent on material type and angle for the bend but some general guidelines are listed in Table 5-1.

Table 5-1: Max material thickness.

Material	Thickness Range
Steel	Up to ~2 mm (0.090 in.)
Stainless Steel	Up to ~2 mm (0.090 in.)
Aluminum	Up to ~3 mm (0.125 in.)

Table 5-2: Springback values.

Material	Spring back (degrees)
304 Stainless Steel	2 – 3
Mild Aluminum	1.5 – 2
Cold-Rolled Steel	0.75 – 1
Hot-Rolled Steel	0.5 – 1
Copper and Brass	0 – 0.5

Conclusion

Additively manufactured metal form tooling is a viable option for forming unique and custom geometries for volumes ranging from 100–500 cycles. FDM tooling is well suited for offset and flare tooling in thinner gauges of sheet metal. It offers significant time and cost savings over traditional purchased metal form tooling.

