

REFERENCE

How to Choose a Manufacturing Process

How material, volume, tolerance, and geometry point to CNC, sheet metal, 3D printing, vacuum casting, injection molding, or die casting, with a quick decision path.

The single biggest cost-and-quality decision on a part is made before any design rule: which process makes it. The same bracket can be machined, molded, cast, printed, or folded from sheet, and each gives a different cost curve, tolerance, finish, and lead time. This guide is a short map from what you know about a part to the process that fits.

The trade-offs below draw on standard design-for-manufacture practice (Boothroyd DFMA) and on the capability of the processes we run. For the design rules within each process, see its dedicated guide.

The four questions

Four attributes decide most cases: material, volume, tolerance, and geometry.

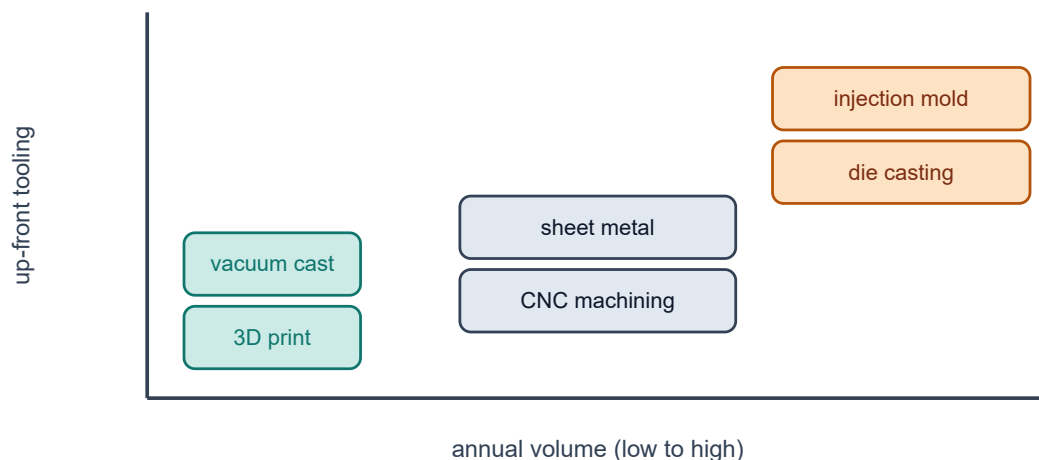
1. Material

Metal or plastic is the first fork. Metals go to CNC machining, sheet metal, or die casting; plastics go to injection molding, 3D printing, or vacuum casting. A few processes cross over (CNC and 3D printing run both), but material narrows the field immediately.

2. Volume

Volume decides whether to pay for a tool.

Process by volume and tooling



Low volume favors no-tooling processes; high volume amortizes a mold, so molding and die casting win.

- Low volume (ones to hundreds): use a process with no dedicated tooling. 3D printing and vacuum casting for plastics, CNC and sheet metal for metals. You pay per part but nothing up front.
- High volume (thousands and up): pay once for a mold or die and the per-part cost drops far below any no-tooling process. Injection molding for plastics, die casting for metals.

Vacuum casting bridges the two: a soft silicone tool made from a printed master gives up to about 25 cast-plastic parts without hard tooling.

3. Tolerance and finish

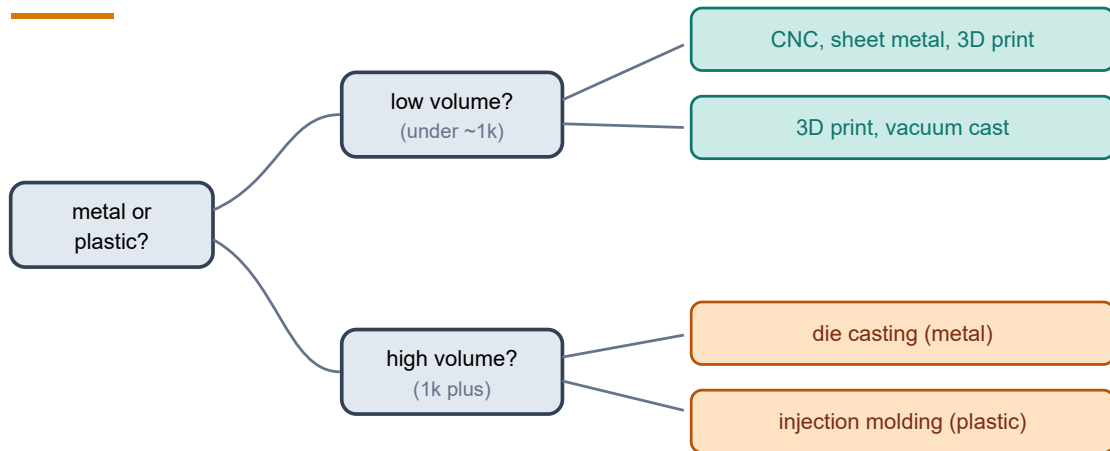
If the part needs tight tolerances or specific finishes, machining holds the tightest bands and any finish; molding and casting hold looser as-formed tolerances and are machined only where a feature is critical; 3D printing is the loosest and is best for form, fit, and function checks rather than precision fits.

4. Geometry

Some shapes favor a process. Prismatic metal parts with pockets and holes machine well; thin-walled enclosures mold or cast well; flat parts with bends are sheet metal; organic or internal-channel shapes that no tool can reach are where 3D printing wins outright.

A quick decision path

A quick decision path



Material and volume narrow the field fast; tolerance and finish then pick the final process.

Start with material and volume to get to a shortlist, then let tolerance, finish, and geometry pick the final process. When more than one fits, the cheaper-at-your-volume option usually wins.

A rough comparison

Process	Best volume	Tolerance	Tooling
CNC machining	1 to thousands	Tightest	None
Sheet metal	1 to thousands	Good	Low
3D printing	1 to hundreds	Loosest	None
Vacuum casting	about 10 to 25	Moderate	Soft (silicone)
Injection molding	Thousands and up	Good (resin-dependent)	Hard (steel)
Die casting	Thousands and up	Good	Hard (steel)

How FabWorks helps

Upload a model and our DFM engine reads the geometry and, together with the material and volume you give, checks it against each offered process, so a part that will not mold cleanly or cannot be machined as drawn is flagged before you commit. If a different process suits the part better, that shows up early, when

changing it is free. The result is the right process chosen for the part, not defaulted to.