

# DFM Checklist for CNC, Stamping, Sheet Metal and Springs

Most of the cost of a part is locked in at the design stage. This checklist covers the points that most often drive price or cause rework, process by process. Run it before you release a drawing and you will remove most of the surprises.

## CNC machining

- Keep internal corner radii at least one third of the pocket depth - small radii need small tools and slow passes.
- Avoid pockets deeper than four times the tool diameter where you can.
- Make wall thickness uniform; thin walls chatter and need extra passes.
- Put tolerances and Ra callouts only on functional faces.
- Design a clear fixturing face so the part can be held for the tightest cut.

## Metal stamping

- Hole diameter should be at least the material thickness.
- Keep the distance from a hole to an edge at least one and a half times the thickness.
- Add a generous bend radius - at least the material thickness, more for high-strength alloys.
- Avoid sharp outside corners: they crack and wear the die.
- Tolerances across bends are looser than in-plane features; keep tight dimensions flat.

## Sheet metal

- Use a standard thickness from the mill's range to avoid special orders.
- Design bend relief notches so the bend does not tear.
- Keep holes at least two and a half times the thickness from a bend line.
- Allow for springback - the shop compensates, but very tight bend angles cost time.
- Specify the finish after the bend, since forming can mark a coated surface.

## Springs

- Spring index (mean diameter / wire diameter) of 5 to 12 gives the most stable coiling.
- Keep the number of active coils above three.
- Avoid index values below four - the spring is hard to coil and cracks easily.
- Specify load at a length, not only free length, so performance is measurable.
- Name the material and the expected life; stress relief depends on both.

## Heat sinks

- Keep fin thickness and spacing within the extruder's die limits for the profile size.
- Allow a machined face for the thermal interface if a flat base is critical.
- Tapped mounting holes need a minimum wall around them; design them into the base, not a fin.
- State the airflow direction so fin spacing matches the application.

## Frequently asked questions

### **What is DFM and why does it matter?**

Design for manufacturing means shaping a part so it is easy and cheap to make without losing function. Done early it removes operations, avoids rework and shortens lead time, often cutting cost by double digits.

### **Can you review my drawing for DFM before I commit?**

Yes. Send the drawing or a 3D model and our engineers return DFM feedback with the quote - recommended changes, cost impact and any tolerance that will be hard to hold.

### **Does DFM feedback cost anything?**

No. We review drawings for manufacturability free of charge and include the notes with the quotation, whether or not you place the order.

### **What is the most common DFM mistake?**

Over-tolerance. Tight tolerances and fine surface finishes applied to every dimension instead of only the functional ones are the single biggest driver of avoidable cost.