SHEETMETAL PARTS AND ASSEMBLY DESIGN FUNDAMENTALS

by Jeff Dayman, Dayman Design Inc.

https://www.daymandesigninc.com

This whitepaper will discuss some fundamentals about sheetmetal part and assembly design work.

There have been many misunderstandings about gauge / thickness standards for different materials in sheetmetal work. For example 12 gauge aluminum sheet is nominally 2.053 mm thick, and 12 gauge mild steel sheet is 2.656 mm thick, and 12 gauge stainless steel is 2.778 mm thick. Very confusing. Luckily, vendors of sheetmetal such as www.metalsupermarket.com publish free charts of gauges/sizes for various metals. It is wise to look at these charts to model your parts at correct and available thicknesses. A call to the vendor to verify that a particular gauge or a certain metal is available at all is also wise. There are many gauges of different metals no longer obtainable in Canada and the USA. A note about units - sheetmetal sold in Canada is still made primarily in the USA to the existing traditional inch-based gauge systems. As more sheetmetal is imported from Far East and Europe, this may eventually change to nominal metric gauges like 0.8 , 1.0 , 1.2 mm etc. Again it is wise to check with suppliers about availability of metals if metric stock is required.

Tolerances for commercial sheetmetal fabrication are also not universally understood. The tolerance capability for cut features, usually the first operations when making sheetmetal parts, are much closer than for bent features. From any cut edge to any cut feature, and across the cut openings, typically tolerances can be held to +/-0.12 mm on sheet up to 3 mm thick. For bends on metal up to 1.6 mm thick, cut edge to inside face of first bend can usually be held +/-0.4 mm. To next bend, another +/-0.4 mm tolerance should be applied. So for a part with two bends, from one side to the other at the bends the design should allow for +/-0.8 mm variation due to bending. Thicker sheet needs wider bend tolerance. The variation can be planned for by allowing gaps and using cut feature tabs in slots on the other part to align parts to each other. Another technique if aligning parts with screws is to have the part the screws pass through with one hole and one slot, to allow the hole to hole variation of the other part. Never use a hole if a slot will do. Bend angle tolerance is usually +/- 2 degrees. Stiffness varies across any sheet of metal, and even if the bend brake tool movement is extremely precise, the springback for different bends will vary within the tolerance noted.

It is good practice to design all sheetmetal parts with a 3 mm radius or larger on all external corners if possible. This makes it much less likely to have cut and scratch injuries to fab shop staff and assembly staff by sharp corners. When planning for internal cuts in parts, other than round holes, it is wise to plan 0.4 mm to 1mm radii in all internal corners because laser and waterjet cutting (often used for first ops now) do not cut dead sharp corners. If sharp corners are required on these features they can be CNC punched, but if the part is designed with cut relief holes at all internal corners centred on the vertex of the required sharp corner, the part can be laser or waterjet cut into these holes leaving a "virtual sharp" corner the mating part can fit in. The corner can still be CNC punched if need be, the programmer just ignores the corner hole reliefs.

For installation of pressed-in fasteners like Pemnuts or Pemstuds it is important to allow room for tooling access to install the items. If you visualize a 20 mm diameter cylindrical tool 100 mm above and 100 mm below each installed fastener, on the fastener centreline, and make sure nothing on the part is in that zone, there should be no access problems for the install of the fasteners. For the fasteners themselves, they need to be the manufacturer's minimum recommended distance from fastener /hole centre away from the radius-start line of any bend or from any part edge. This distance is designed to prevent distortion of the sheetmetal due to the fastener install force and also to ensure it holds in the part with the rated strength. Typically if this distance is 10-11 mm or more there will be no issues, but it is wise to check the manufacturer's recommendations before designing the parts. Note that all of these fasteners are designed for certain material gauge. Be sure to select the correct fastener for the gauge to be used.

If you need any help or assistance with design for sheetmetal parts and assembly, or cost-down activities for this type of parts and assemblies, please contact me.

https://www.daymandesigninc.com info@daymandesigninc.com 519-888-6462