OCSWUG
1/22/09

SHEET METAL TIPS

1. Visit your manufacturer's facility – Get Bend Deduction tables and punch list
2. Understand the limitations (and advantages) of general purpose tooling & machinery.
3. Model sheet metal with open sketches.
4. Use gauge tables – see Tip #1.
5. Minimum outside flange length = 4x material thickness.
6. Maximum flange length < 29" – see Tip #1.
7. Lasers are happier cutting steel or stainless. Aluminum is harder to laser cut.
8. When punching, the smallest hole size = material thickness.
9. Model holes to the large size of the tolerance range – (e.g. +.000 -.003)
10. Keep holes at least 3 material thicknesses away from bend radius.
11. Use the hole wizard for most of your holes – especially CSK & tapped.
12. Countersinks are self-aligning! Use floating PEMS to avoid tolerance problems.
13. Countersink thru hole is a function of material thickness & angle. Leave a small cylindrical wall for burr management & strength.
14. PEMS – must follow minimum hole-to-edge and minimum material thickness!
15. PEMS – leave access for insertion tooling.
16. If you're making a sheet metal part, model it that way from the start.
17. Generally use the "Outside Bend" & "Virtual Outer Sharp" options.
18. Use Close Corner for matching weird angle flanges.
19. Use mitered flanges when 45° is useful (not so good with heavy gauges).
20. Allow for spring back – keep your gaps > .020"
21. Jogs can be tamed by nudging the radii and angle – see Tip #1.
22. Hems require generous tolerances – see Tip #1.
23. Sketched bends are great for internal tabs & weird stuff – use cautiously. Things that are hard to model are usually hard to build, too.
24. Use hidden lines to show bend direction in mechanical drawings.