Hole size

What is the minimum hole size? The material thickness and hardness limits the size of the hole we can punch. In mild steel, we can go down to the material thickness as the smallest hole -- the smallest CNC punch tooling we have is .031".
What is the maximum hole size? Tricky question. If multiple hits are allowed, we could CNC punch out a hole 50" x 72" (1270mm x 1828mm). The largest single CNC tool we can hold is 3.5" dia (88.9mm). The CNC equipment can press up to 30 tons so the material thickness and hardness create another limit.

Hole-to-fold

How close can a hole be to a bend? The hole can be right in the middle of a bend, but it will be distorted as the material stretches during forming. In general, the edge of the hole should be at least 2 material thicknesses away from the start of the inside bend radius.

Flange size

What is the smallest flange width? The quick answer is to keep the inside of the flange at least 3 times the material thickness + the inside bend radius. We don't like to do it, but we can "coin" flanges as small as 1.5 times the material thickness in thin ductile material (brass, mild steel, etc.).

Bend radius

What is the minimum inside radius? A rule of thumb is to keep the inside bend radius at least equal to the material thickness. Thick material requires a larger inside radius to minimize cracking and to reduce the pressure required. Another suggestion is to not be too fussy about inside radii (be generous in the allowable range).

Bend reliefs

Are bend reliefs required? Bend reliefs are cutouts or holes that are punched in anticipation of the material tearing or ripping near an edge when the part is formed. Bend reliefs reduce the propagation of stress cracking. By allowing the material to form without cracking, the accuracy and consistency during production is improved. They also reduce the debur effort and reflect good design practice. Use 1.5 times the material thickness for the size of the relief, and annotate them as "NOT INSPECTABLE" features so QC personnel don't overkill the inspection.

Drafting

Revision control: Always show a revision level on the print. Change the revision every time the drawing is changed! Even if all you did was fix a spelling error, change the revision of the print! The effort will reduce brain damage in the long term.

Title blocks & borders: At a minimum the title block should include the part description, part number, revision, and cad file name. Engineering & design authority, where used, and other routing and control info is recommended. The border around the print should be "gridded" in alphanumeric zones to facilitate telephone conversations. It is much easier to say "See the hole at zone A-5?" than it is describe it as "The hole on the side... no, the other side...".

Views and projections: When drafting side, top, left, right, bottom views, etc. use the "fish bowl" rule. Imagine your part resting in the bottom of a bowl and you are looking straight down at it. If you push the part to the right, it would move up the bowl and rotate to reveal what should be drawn as a side view. Push it left, up, or down to imagine the correct orientation of other views.

Tolerances: Don't over do it! Be generous! Be kind! (See sheet metal tolerances) Show tolerance notes for "hole size", "hole-to-edge", "hole-to-hole", and "hole-to-fold" or "fold-to-fold". The "X.XXX=+/-.005 or X.XX=+/-.020" is better than nothing, but leaves much to be desired.

Geometric dimensioning: This is a good thing, if it is properly drafted. It is an excellent method of showing important relationships between holes or bends and so forth. It helps inspection by showing how the holes will be used. This method of dimensioning requires much greater effort on the part of the draftsman and engineer, but it can contribute to cost effective design and production.