

Laser Micromachining CAD Design Rules

To aid us in processing your laser micromachining job efficiently there are a few rules which should be followed when creating CAD files.

Rule 1: Draw all geometries as zero-width closed polylines.

All outlines or areas to be machined should be presented as zero-width polylines. Areas to be machined should be drawn as polygons rather than adjustments to the width of the drawing pen. In most cases, our translation process will ignore the width of a line.

Rule 2: Draw curves and arcs as segmented zero-width polylines.

It is possible to create curved entities such as circles, ellipses, curves and arcs by a number of different methods. The interpretation of such entities varies between different CAD packages and in some cases the entity can be interpreted as a straight line. To prevent this, all circles, ellipses, curves and arc should be made up of multiple straight polyline segments. There is no limit on the number of segments that can be used. However, using a segment length less than the desired resolution of your machined part can increase the machining time. Most CAD packages have a command to automatically convert a curve to a segmented polyline with the optimum number of segments.

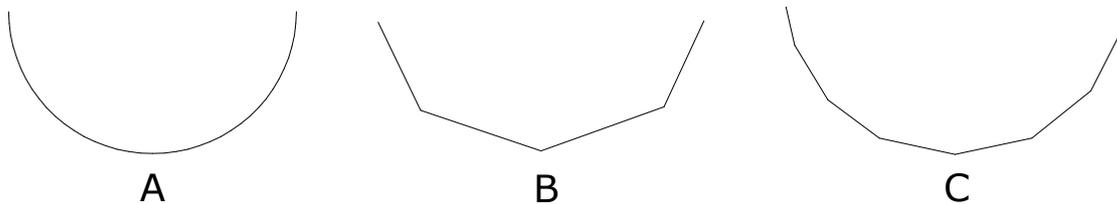


Figure 1. Drawing A shows a curve as originally drawn. Drawing B shows the curve represented as a segmented polyline. The curve is approximated by a number of straight polyline segments. As Drawing C shows, increasing the number of segments gives a more accurate representation of the curve.

Rule 3: Join all individual geometries into a single zero-width polyline

A geometry comprising of several entities, such as that shown in figure 1, should be presented as a single zero-width polyline. All curves or arc in the geometry should be converted to segmented polylines and consecutive geometry entities should have a start point at the end point previous entity. The use of your drawing package's 'Snap to end point' feature is strongly encouraged. Additionally, the 'Join' or equivalent command should be used to join consecutive entities together to form a single polyline. It should be noted that this is not the same as 'Grouping' all the entities together.



Figure 2. Drawing A shows a geometry made up of four different entities, three lines and a curve. Such a geometry should be presented as a single, zero-width polyline as shown in drawing B. The curve has been converted to a segmented polyline.

If the geometry comprises of curves or arcs these should be converted to polylines and consecutive segments of the geometry should be linked together rather than being left as individual CAD entities. Presenting your geometries as a complete objects helps our

translation software know exactly what laser path to take and, on complex geometries, can even reduce the machining time and hence cost.

Rule 4: Draw all areas as re-entrant closed polylines.

Complex area geometries which have 'island' regions should be drawn as a single closed polylines. Such polylines are allowed to touch each other and are referred to as re-entrant polylines. This rule is illustrated in figure 2 below.



Figure 3. Drawing A shows an area geometry that contains a single island in the centre the area to be machined is between the outer and inner boxes. To ensure there is no confusion translating such geometries, it is better to draw them as in drawing B where the machined area is defined by a single outline which goes into the interior of the geometry and return to the edge of the geometry by the same route.

Rule 5: Don't use hatching

Areas that are to be machined should not be hatched on your drawing. Instead the regions should be clearly labelled using text.

Rule 6: Keep geometries to be machined on a single, separate layer

Your drawing can contain multiple layers. However, to ensure that all parts of the geometry to be machined are correctly identified, all such geometries should be placed on a single layer containing only geometries to be machined. Additional information such as explanatory text, guidelines, dimensions and any line indicating drawing scale and units should be placed on a different layer.

Rule 7: Do not overlay or duplicate drawing entities

The entities in your drawing are used to calculate the motion of our laser tools, it is important that your drawing does not contain entities drawn on top of each other or duplicated as this can lead to confusion in interpreting your drawing and inaccuracies in the machining process. When generating CAD files for machining, the key point to remember is not that the drawing 'looks' correct but that the correct entities are used and positioned in the correct manner.

Rule 8: Clearly state the units of your drawing.

You may use any preferred units for your drawing. However, to ensure that the drawing dimensions are interpreted correctly by our translation process, the drawing units should be clearly marked in text on the drawing. We strongly encourage you to include an additionally straight line on your drawing, the length of which is clearly marked. 'Dimension' drawing entities can be misinterpreted by some CAD packages so length of the line should be given in simple text. We recommend that the drawing units are mm

Rule 9: Save file as AutoCAD 2000 DXF format.

DXF is a widely used file format for CAD designs and is supported by the majority of CAD packages. However, the DXF format has evolved over the years. To ensure maximum compatibility use the oldest version of the DXF format you have access to. We suggest using the version of DXF released with AutoCAD 2000.