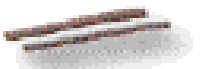
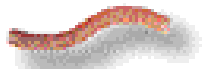
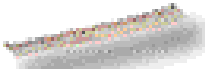




# TECH NOTES

## LAIRD TECHNOLOGIES



#130

---

## SPECIFYING TOLERANCES

---

By: Manny Cambria

If there is one specification or drawing requirement that can be identified as the major cause of late deliveries, lost orders, or customer dissatisfaction, it would most certainly be the specification of tolerances and how they have been assigned on a particular part. Cost and delivery are directly affected by the use of unnecessarily close or improperly applied tolerances. After much time has been spent in the research, development, and testing of a part, only a minimal amount of time is given to tolerance values. Too often, an arbitrary "Block Tolerance" is assigned. This, in turn, affects every dimension on the drawing, regardless of its importance to the function of the part, or to the effect it may have on the cost and lead times of both the tooling and the manufacture of the part. The effects are usually inversely proportional: as the tolerances get tighter, the costs go up and the lead times get longer.

When a tool is built to "Block" tolerances, every dimension must be considered critical, thereby increasing the time and cost to complete the tool, and increasing the possibility of error. While block tolerances are convenient, they can be very costly if not applied in conjunction with key or control dimensions. The control dimensions should have tolerances which are practical and consistent with the function of the part and are usually the dimensions to which statistical process control (SPC) is applied. In these cases, the block tolerances should be liberal enough to satisfy the form and fit requirement of a part.

Ideally, the design engineer and the manufacturer of the part will meet and determine which dimensions are critical and which should have liberal tolerances. This approach allows the tool to be built with the focus on those few dimensions which must be closely controlled, thereby reducing both build time and cost. Also, it allows for the use of functional gages which reduce the

time required to inspect or check parts. It is strongly suggested that the above procedures be implemented whenever possible as a means of simplifying the manufacturing and inspection process.

The term "Practical Tolerances," is often misinterpreted to mean that tolerances should be very loose or wide open in order to make the part easier to manufacture. It should be interpreted to mean, if the dimension is not critical, make the tolerance loose, but if the function of the part requires tight tolerances, then they should be judiciously applied to those dimensions which control its function and only after consulting with the manufacturer.

In the past few years, geometric tolerancing has become the accepted method of assigning values to dimension and features.

This procedure has greatly simplified the tolerancing process especially in the area of forms, profiles of surfaces, parallelism and run out which formerly were addressed by vague or ambiguous notes. However, the same problem as with numerical tolerance will occur if the reasoning behind geometric dimensioning and tolerancing is not clearly understood when applied to a part. Tolerancing then should be applied to parts only after carefully considering the ramifications behind them, such as:

- Can the part be made to the assigned tolerances?
- Can the part be inspected to the assigned tolerances?
- What is the cost and delivery impact on the part and tooling?

When these questions can be answered to the satisfaction and agreement of both the buyer and the manufacturer, only then will there be a properly toleranced part, of minimum cost and lead time.