

Dimensioning and Tolerancing

Prepared by: Michael Hypes
Cornell University

Preparation:

One of the most common problems for new designers is choosing dimension that do not reflect the purpose of the part. Let us consider a nominal 1/4" hole. Most new designers will simply put 0.25" or 0.250", but in many cases this isn't what was intended. Tapped holes, ones where you thread the hole, require a smaller hole be drilled, while clearance holes require a slightly larger hole be drilled. Press fits are generally the only 1/4" hole that actually requires a 1/4" drill. Once you know the intended uses of all your dimensions it is much easier to select the proper dimensions. A list of drill sizes for tapped and clearance holes is available online or in the machine shop.

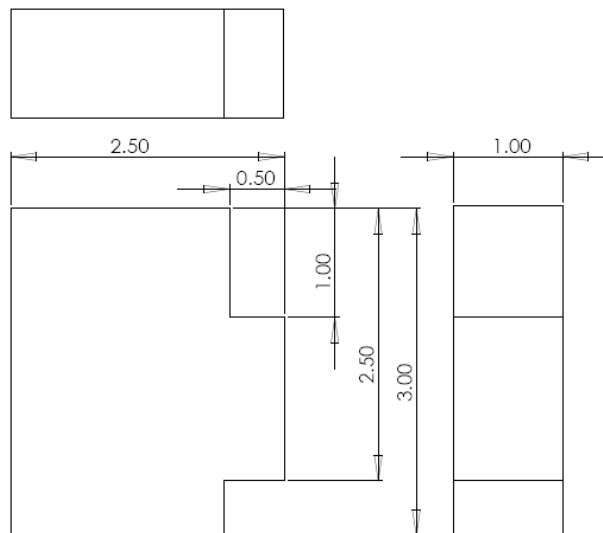
Hole Type	Drill Size
1/4"-20 Tapped	.2010 (#7)
1/4" Dowel Pin	.2500 (1/4")
1/4" Close Fit	.2570 (F)
1/4" Free Fit	.2660 (H)

Table 1: Drill sizes for 1/4" holes in aluminum

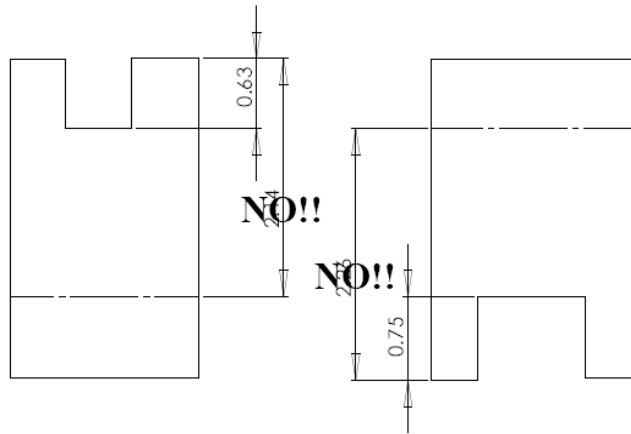
Dimensioning:

There are a few simple best practices which can help us dimension a working drawing:

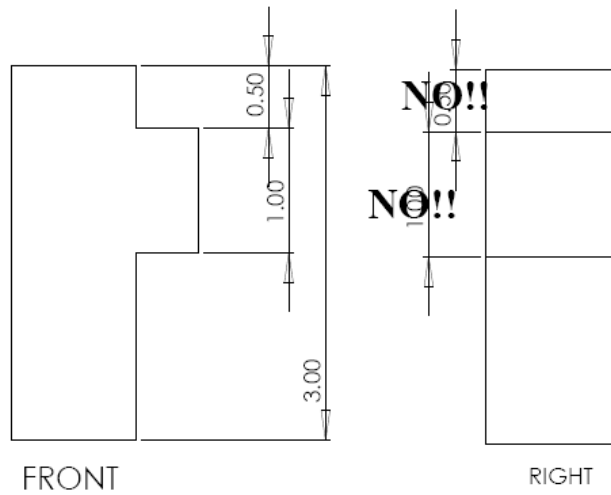
1. Place dimensions between views sharing dimensions, when possible



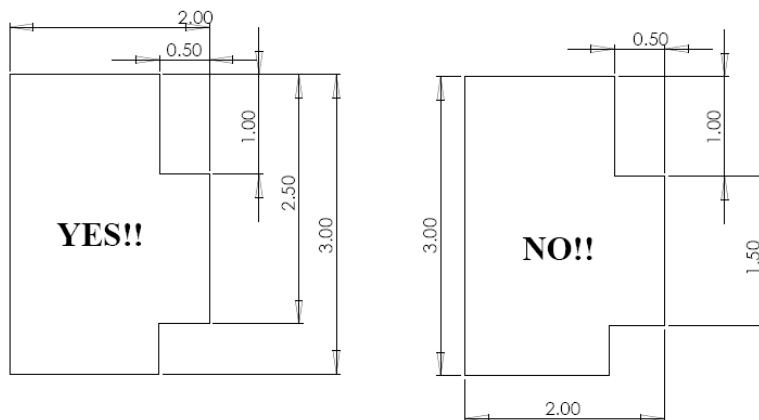
2. Do not dimension to hidden lines



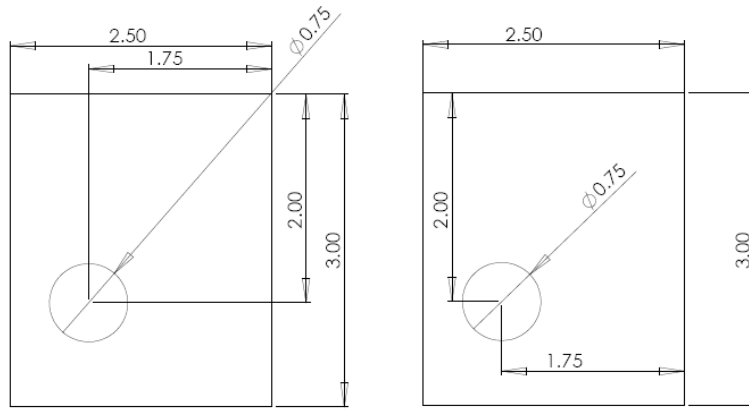
3. Dimension in a view that shows the geometric characteristics of the part



4. Group and organize Dimensions



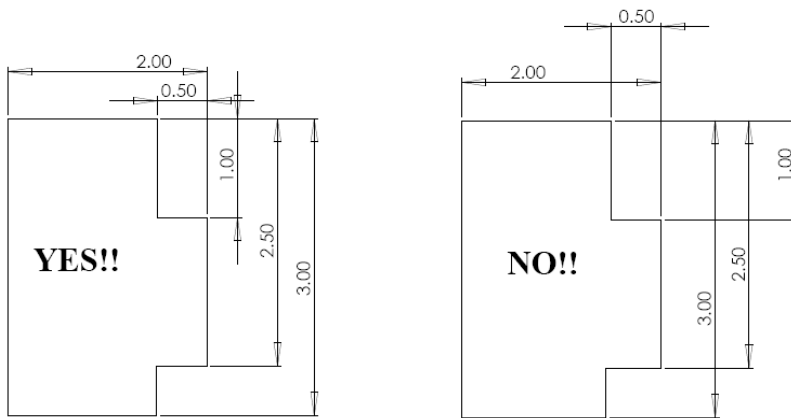
5. Do not place dimensions on the views



YES!!

NO!!

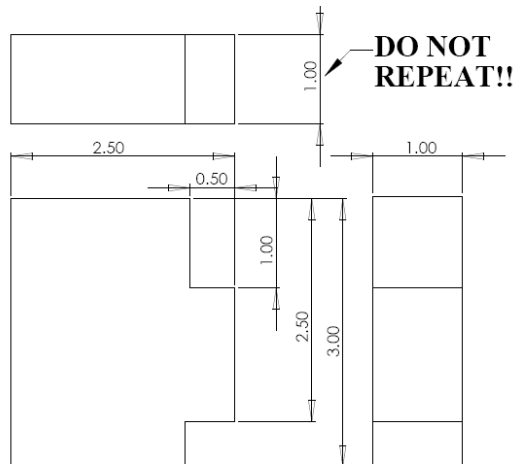
6. Dimension the shortest features first and move away from the view



YES!!

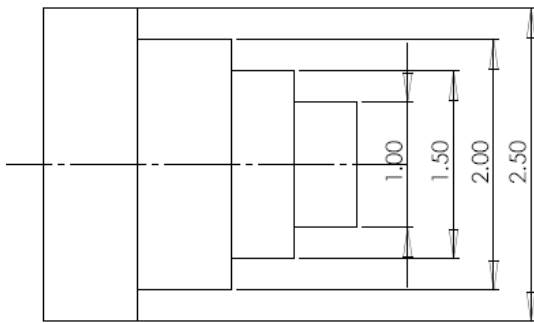
NO!!

7. Do not repeat Dimensions

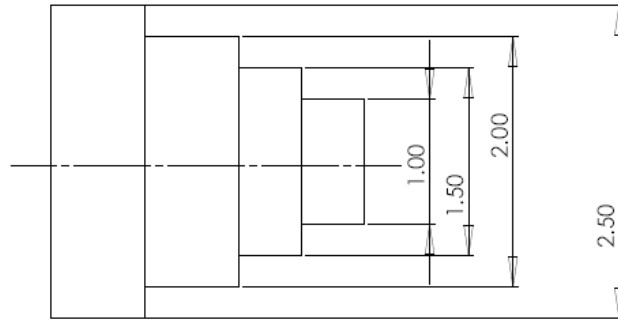


DO NOT REPEAT!!

8. Stagger dimensions to make the part easier to read

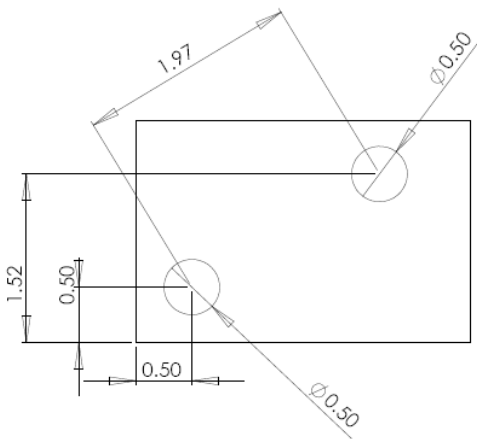


NO!!

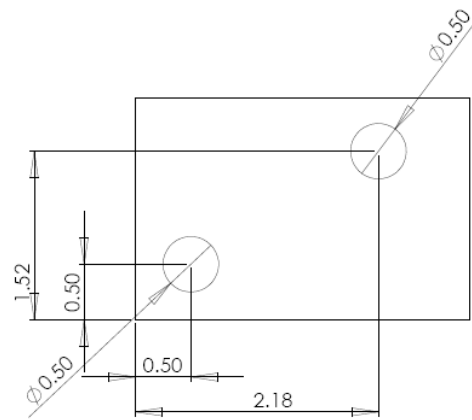


YES!!

9. Dimension along a given coordinate system rather than using center to center dimensions



NO!!

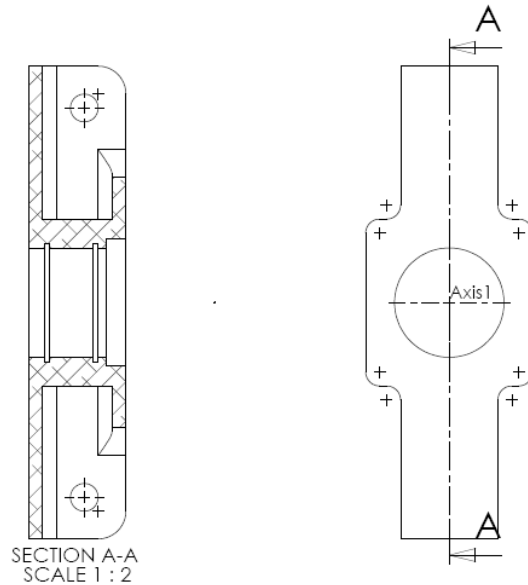


YES!!

10. Use the same number of significant figures for all dimensions with the same tolerance

See Tolerancing Below

11. Use cut-away views to show internal features



In addition to these best practices, it is helpful to consider how the part will be made. When using a mill, we generally define an origin either from one corner or a hole, if you dimension from different edges, it requires resetting the origin and can lead to alignment problems. When using a lathe, we generally define our coordinates from one face and the diameter of the part. In addition, as the lathe produces axisymmetric parts the most useful view is the view where the part appears as a rectangle.

Tolerancing:

When dimensioning parts, it's very important to consider how precisely the part must be made. Many teams choose to have tolerance guidelines. An example of one of these conventions is 1.234 means $\pm 0.0005''$, 1.23 means $\pm 0.005''$, 1.2 means $\pm 0.05''$, etc. . .

Using just one tolerance level may not be the best choice. Let us consider a jig plate. The distances and locations of the holes may need to be accurate to $\pm .0005''$, but the external dimensions of the plate may be able to vary up to several inches. Having lower tolerances which do not affect the quality of the part can speed manufacture.

Also, you can use one-sided tolerances; these are common when there's a maximum or minimum size. Let us consider making a pair of stacking boxes, one with an inner length of 3'' and one with an outer length of 3'' nominal, but each box will be made by a different person. To make sure that the boxes will fit together no matter what: the smaller box could be dimensioned to $2.995 \pm .005$ and the larger box to $3.005 \pm .005$. However, then we find that if everything is produced to specification we'll end up with a gap of $.010''$. Instead to keep reasonable tolerances we can make the smaller box $3.000'' +0/-0.005$ and the larger box $3.000'' +.005/-0$.

Finally, avoid accumulating tolerances. If you dimension off of one feature rather than dimensioning off of subsequent features it is possible to avoid large tolerances on the overall part. However, you should still keep important dimensions and always consider what

dimensions on a part are critical. For example, if 3 holes need to be +/- .0005" from each other, they should be dimensioned off of each other, but the far edge should not be dimensioned off the 3rd hole to keep the overall part dimensions within tolerance.