These guidelines can be applied to most projects. Ask an instrument maker questions if you are unsure of anything, and before machining complex parts. Never guess!

If possible, machine the inside of a part before machining the outside profile. This makes for a more rigid set up that will improve the life of the tooling, and create a better part finish.

Use the surface feet per minute (SFM), or inches per minute, provided by the tooling manufacturers/instrument maker. They have done extensive research, use it to your advantage. This will also improve the life of the tooling, and create a better part finish. SFM will factor in the feedrate and diameter of the material, adjusting the RPM to maintain the SFM. RPM is constant regardless of the diameter.

When manually machining with DO ONE events, remove the excess material before attempting to machine fillets/tapers/radii.

The tip of the tool needs to be located at the point closest to the end of the part (Z=0) before entering the DO ONE event.

When doing a large fillet/taper/radius, you may need to make several passes, stepping closer to the finish position each pass, instead of going to the finish position immediately.

Keep the amount of your stock material sticking out of the chuck/collet as short as possible, while still leaving room for machining without damaging the chuck/collet. If possible, do not exceed 1.5 times the diameter of the stock. If you need to machine a longer piece, use a live center.

All RPMs must be less than 2200 for the chuck, less than 4000 for the collet closer.

Use the proper nose radius when describing the tooling:

We have 2 sizes for the nose radius, .015” and .031”. Look and compare the inserts to tell which size you are using.

Grooving and parting tools have .008” radius on the corners. Measure the tool for proper width.

Use the tailstock to drill all holes, NEVER cnc drill.

Use the correct post (Southwest) and tool library (COE Student Shop).

Right and Left hand 55° turning tools, 1” diameter stock:

* RPM…..1200 RPMs
* Roughing Depth of Pass…..0.050”
* Feed per Revolution…..0.015” per revolution
* Finish Cut Depth of Pass…..0.010”
* Finish RPM…..1500 RPMs
* Finish Feed per Revolution…..0.003” per revolution

Grooving tool, 1” diameter stock:

* RPM…..1000 RPMs
* Roughing Feed per Revolution…..0.003” per revolution
* Finish Cut…..0.001”
* Finish RPM…..1000 RPMs
* Finish Feed per Revolution…..0.003” per revolution

Threading:

* RPM…..155 RPMs
* Depth of Pass…..0.005”
* Pitch…..1/Threads per Inch (example: ¼-20 threads = 1/20=.050”)
* Spring Passes…..2
* Number of Starts…..1
* Plunge Angle…..29.5

Boring:

* RPM…..800 RPMs
* Roughing Depth of Pass…..0.030”
* Feed per Revolution…..0.005” per revolution
* Finish Cut Depth of Pass…..0.010”
* Finish RPM…..1000 RPMs
* Finish Feed per Revolution…..0.003” per revolution