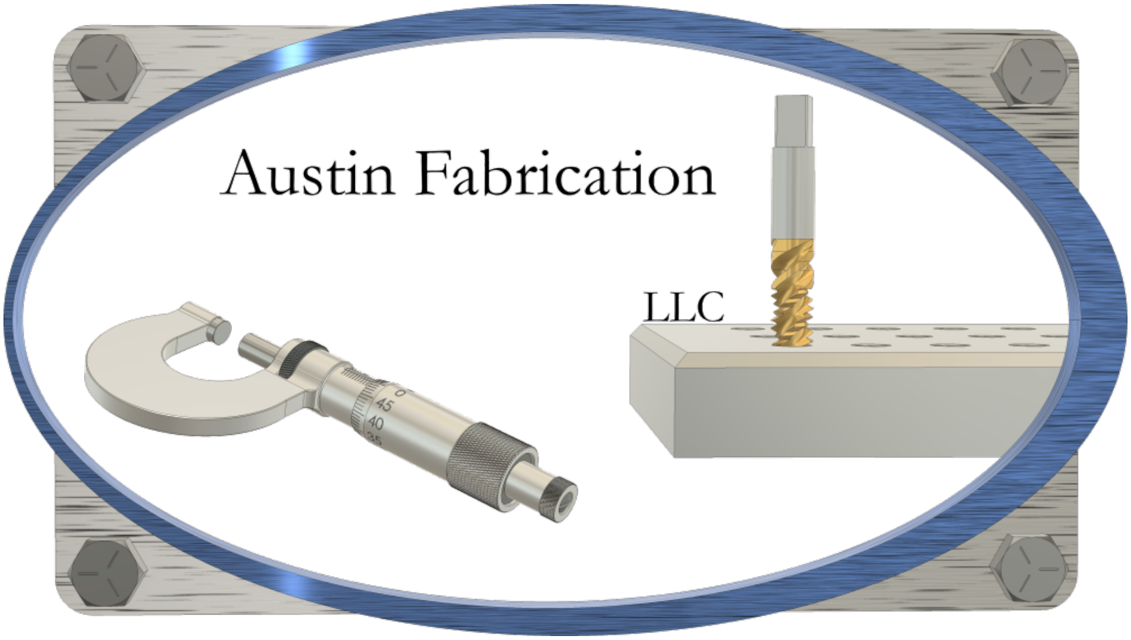
Quick Turnaround Best Practices

Recommendations to Reduce Cost and Lead Times

Austin Fabrication LLC



Contents

1 Intent of This Document

This document provides recommendations to reduce the lead time and cost for the parts or services you need. General limits of the shop/equipment are described also. Please reach out with any questions using the Contact section of the website.

Before requesting machining or CAD services from Austin Fabrication LLC, please read the Policies section and the DISCLAIMER and WAIVER OF LIABILITY.

Austin Fabrication LLC provides the following services:

- Affordable, quick turnaround custom one-off parts to move your project forward.
- Assistance with unique designs, modifications, mechanical and machining services that other shops won't halt production for. No job is too small! Specializing in clever solutions to "MacGyver" your way out of a bind.
- Manual lathe and milling machine work of most standard materials (plastics, aluminum, mild steel, stainless steel etc.). No woodworking services are offered. The lathe has a 14.5 in (370 mm) swing with a 2 in (51mm) spindle bore and 1 m (39 in) between centers. The max travel for the mill is $X = 26$ in and $Y = 12$ in with 22 in from spindle to table.
- Hole tapping, thread repair, broken bolt extraction and similar operations.
- Bandsaw cutting (maximum of 12 in width for plates and 7 in diameter for rods)
- CAD services for a flat rate to help bring your idea to life or reverse engineer a part to produce a .step file.
- Bead blast finish options available for small parts.
- Additional capabilities are 12 ton hydraulic press with 90 degree brake, metal shear (max bend width 12 in and 3 in at 3/8 in mild steel), manual hole punch, arbor press 1 ton).
- Standard metrology equipment: gage blocks, outside and depth micrometers, calipers and pin gages.
- No need to be local to the Space Coast! Parts are regularly shipped to and from Austin Fabrication LLC.

The business is neither a full service or full time shop. If the work required exceeds the capability of the shop, you'll be informed right away to avoid lost time for you. More information is available on the Policies tab on the website.

2 Tolerances and Material Stock Considerations

2.1 Design and Stock Selection

Holding tight tolerance requires more careful workholding setups and more cautious cuts and thus takes longer. If no tolerances are provided the parts are still machined as close to the target as possible while weighing cutting conditions, material and time. Note that material stock procured has dimensional variation. Consider the 12”x 12” aluminum plate with a thickness of 1/2” from McMaster-Carr shown in 1. The length and width can vary by ± 1/16”.

Easy-to-Machine MIC6 Cast Aluminum Sheet

1/2" Thick, 12" x 12"

86825K15

$102.34 each

Delivers Tuesday

Streamline your design process with our Solidworks Add-In

Available for Solidworks 2017 or newer.

Tolerance Rating	Tight
Material	MIC6 Aluminum
Thickness	1/2"
Thickness Tolerance	-0.005" to 0.005"
Width	12"
Width Tolerance	-1/16" to 1/16"
Length	12"
Length Tolerance	-0.063" to 0.063"
Flatness Tolerance	0.015"
Temper Designation	Not Rated
Appearance	Plain
Application	Precision Machining
Certificate Form	Digital
Certificate Type	Material Certificate with Traceable Lot Number
Comparable To	Alca 5, ATP 5, K 100-S, MIC6
Conductivity Percentage	36% IACS
Elongation	3%
Fabrication	Cast
Fixturing Plate Type	Jig, Tooling
Hardness	Brinell 65

Figure 1: Example of dimensional variation (tolerance) on available stock.

If you need a square plate of dimension $12 \pm .005$ in and the stock arrives oversize, it can be machined down. If the plate arrives undersized, the desired dimensions can’t be met. The following are important points to consider:

- Please assess if the nominal dimension (.5 in for example) is acceptable. Please specify tight tolerance only when necessary.
- For a flat plate similar to that of Fig. 1, a minimum of .005 in is usually required to clean up the surface to leave a nice finish.
- Be cautious when referencing the distance to holes and features from the edge of the work piece rather than from the center of the work piece or dimensions between features for the reasons mentioned above.

- If you need the part held to tight tolerance, there are two options:
    1. Purchase tight tolerance or precision ground stock (much more expensive)
    2. Purchase the next larger (or thicker) size and machine down to final dimensions (more expensive stock and more machining time).
- Cleaning up the surface goes the same for round stock on a lathe. An example is shown in Fig. 2.

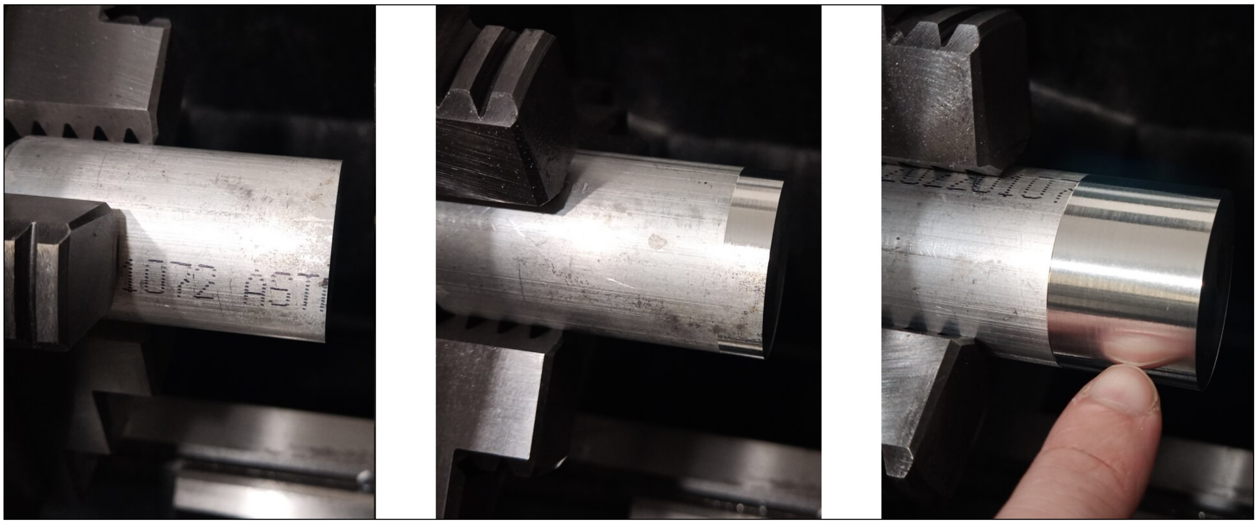


Figure 2: A 1.5 *in* aluminum bar is turned by taking .005 *in* off the diameter. Notice this doesn't quite clean up the surface (center image). This bar required taking .012 *in* off the surface to leave a nice finish. If a nice surface finish is preferred at 1.5 *in*, the next larger stock size is required and results in higher material costs and more machining time.

2.2 Tolerances

The owner is neither a professional machinist nor is the equipment of industrial size. Part and tool deflection always leave uncertainty and any documented tolerance specifications on the part will be considered before work begins. Generally, tolerances in the range of ISO 2768 medium are easy to achieve. Tolerances on diameters turned on the lathe, for example, cannot be reliably hit less than ± .0005 *in*. Austin Fabrication LLC offers quick turnaround functional parts and is not a high precision machine shop.

3 Thread Tapping

3.1 A Quick Primer on Threads

Threads can be tapped in any standard material and several important considerations are needed to prevent tapping difficulties. The following are the standard questions:

1. What size drill bit is required?
2. How deep should the hole be tapped?

Many tap/drill charts and tap/drill combo sets are available for purchase. These charts often have a fatal flaw: they size the hole for 75% thread engagement. If you head to the hardware store and purchase a 1/4-20 tap and attempt to tap a hole through 3/4 *in* of stainless steel the tap will break. The physics and engineering of threaded fasteners and bolted joints is stupendously complex and still an active field of research. This section is a minuscule overview of important concepts to help fabricate the parts you need while minimizing cost and time by avoiding broken taps. Figure 3 shows two different hole sizes on a CAD model of a 1/2-13 bolt.

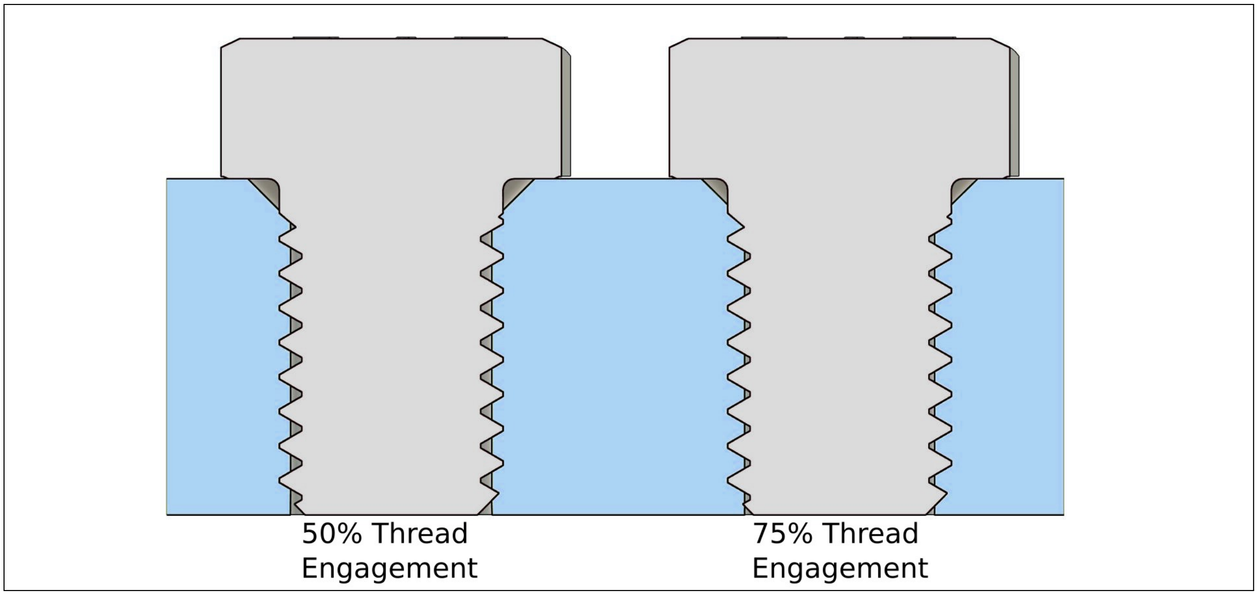


Figure 3: Two different starting hole sizes show the difference in percent thread engagement from 50% (left, larger starting hole) to 75% (right, smaller starting hole).

Notice there is more "meat" (blue section) for the hole tapped to 75%. There is minimal practical benefit in terms of thread strength for tapping greater than 75% thread with the exception for very thin items (panel nuts). Tapping torque increases rapidly with percent thread engagement.

There is a trade off between ease of tapping, required internal thread strength, percentage of thread engagement, and length of engagement (how deep the hole is tapped).

- Deeper holes result in higher stress on the tap, increasing the likelihood of breakage.
- Higher percent thread engagement (smaller starting hole) increases the stress on the tap.
- Harder materials increase the stress on the tap, particularly materials that work harden easily (304 SS, for example).
- Reducing the percentage of thread engagement (larger starting hole) is the most effective way to prevent tap breakage when considering tapping depth and material. Often, 60% to 65% is sufficient, particularly for non critical bolted joints.

Common practice is to tap to a depth such that the internal threads (nut/joint) can support the full breaking strength (tensile) of the bolt (external threads). This design practice is implemented since a broken bolt is easy to detect but partially damaged/stripped internal threads are not easy to detect. The first 3 - 5 threads support most of the load generated by the bolt and each thread does not share the load uniformly. Thus we should tap threads deep enough so the bolt fails before the internal threads do.

We can calculate the minimum length of engagement (tapped thread depth) assuming a steel bolt into aluminum. For further reading and references to the detailed equations on threaded fasteners and joint design fundamentals, see the sources below (in order of thoroughness of information):

1. An Introduction to the Design and Behavior of Bolted Joints
2. Guide to Threads, Threading and Threaded Fasteners: Modern Machine Shop
3. Machinery's Handbook

For this example, we can take a grade 8, 1/4-20 bolt of class 2A and tap into 6061-T6 aluminum (most common type). Using standard values for the smallest allowable bolt tolerance, the largest allowable nut tolerance for the pitch diameter, standard ASTM tensile strength values (in *psi*) for a grade 8 bolt and 6061-T6 aluminum, and a tensile/shear stress ratio of 2 for the bolt, gives a tapping depth of approximately .4 *in*. This is 1.6 times the diameter of the bolt. This assumes all ideal geometries, material properties, and perfect machining which doesn't exist in any real scenario. There is a bit of extra safety in this calculation for how the equations utilize measured relationships between tensile and shear stress (see the texts above for more information). Applying the same calculation for a steel bolt into a steel joint of similar material strength, the length of engagement for a 1/4" bolt is about .19 inches. This is less than the diameter of the bolt.

This is where the rules of thumb for tapped threads comes from (and of course experimental verification). For non critical joints, the minimum recommended tapping depth for a steel bolt of diameter D into a steel joint is $1.0D$ and for a steel bolt into aluminum is $1.5D$. For extra assurance, you can tap aluminum to $2D$ and for tapping into steel, $1.5D$ (assuming steel bolts). These recommendations are not for critical joints or safety related hardware which require more thorough analysis.

A quick note on thread classes for Unified / ISO metric threads (60 degree threads, most nuts and bolts you'll work with) is provided here. The classes of threads determine the tolerance of the thread profile from ideal. In general, the bolt is made slightly smaller than the nut (internal threads) and are designed to ensure fit, compatibility, manufacturability and strength. For the classes, A refers to the external threads (bolt) and B refers to internal threads (nut/joint). Class 1 hardware are the nuts and bolts supplied with a swing set or bookshelf. They assemble quickly, have larger tolerance and are easier/cheaper to produce. Class 2 are the nuts and bolts you'll get at the hardware store. Class 3 are held to tighter tolerances and are used for more critical applications. Metric hardware uses a different tolerance rating with classes comparable to the 1-2-3 class ratings of Unified imperial threads.

When a bolt is tightened, it produces preload in the joint and physically stretches like a stiff spring. Torque specs are commonly developed so that the bolt is stretched to a fraction of its yield strength, often about 70%. The tensile strength is the value (units of pressure, force divided by cross sectional area) at which the bolt/material will break. Yield strength is the point at which the bolt no longer retracts/relaxes back to its original length and is plastically deformed, thus damaged (like an over-stretched metal slinky). Some bolts, like many engine head bolts, are torque-to-yield and should not be reused.

Unless explicitly stated otherwise, manufacturer provided torque specs are for **non-lubricated** fasteners, either under the head or on the threads. The clamping force from a bolt under identical torque can increase by over 30% when lubricated compared to the same bolt/nut that is not lubricated. Thus torquing a bolt to specs with lubrication can damage the bolt and become a safety concern.

When properly torqued, bolts in standard joints are well under their breaking strength. For the case of non critical bolts/joints that are properly torqued not near their failure point, the tapping depth rules of thumb commonly used are safe with respect to internal thread failure. This makes no mention of loosening due to vibration, thermal fluctuations etc.

To summarize this section, please consider your design: Do you actually need that 1/4-20 bolt tapped at 75% thread engagement a full inch through 304 SS? The proper answer is almost certainly no. Please keep the rules of thumb described in this section in mind. Beyond those lengths of engagement, there is no practical benefit to the strength of the threads to tap deeper as Austin Fabrication LLC does not produce products or services for structural or safety critical systems.

3.2 Tapping Recommendations

Many types of taps available for tapping various geometries. Attention must be paid to the difference between the depth of the hole and the depth of threads. Most taps have one of three taper geometries: taper, plug, and bottoming. Figure 4 shows the difference between how far threads are created before the tap bottoms out. Often if a single tapped hole is needed, the tapping is done by hand. Power tapping is a service offered which greatly speeds up the process. However, poor design practice can greatly increase machining time.

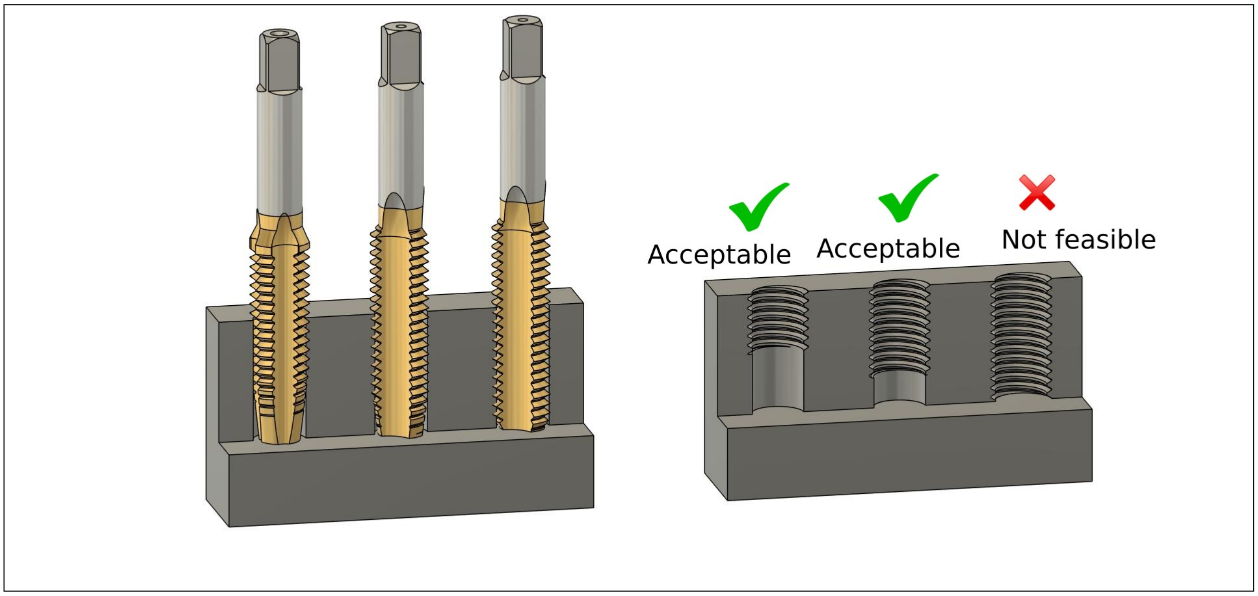


Figure 4: Three straight flute taps are shown from left to right: Taper, plug, and bottoming. The amount of lead in (taper) on the threads affects how close to the bottom of the hole they can tap threads. A taper tap is used for tapping by hand because the taper gradually walks/aligns the tap to the hole. The tapered cutting faces gradually take larger cuts, which reduces the input torque required.

Power tapping is often done with spiral flute taps for blind holes (different from the straight flute taps depicted in Fig. 4 where a plug tap provides balance between a taper tap (shallow depth, but low tapping torque/risk) and a bottoming tap (deep depth, but high tapping torque/risk). The hole in Fig. 4 with the red “X” shows a poor design choice. Bottoming taps often aren’t used to start a thread due to their high torque which means multiple taps are required for the same hole. More importantly, the threads shown with the red “X” go right to the bottom which isn’t possible with a bottoming tap that still has about 2 threads of taper/lead in. In odd cases, the bottom of the tap is ground flat and used to tap right to the bottom. This requires buying a new tap and is a slow process. If possible, design the part with a hole deeper than the required thread depth to provide sufficient room for tapping. With a bottoming tap, if needed right to the bottom of a hole, a power tapping arm with digital torque limiting and a mechanical torque clutch will still shatter a tap if it bottoms out.

There are no hard rules for reducing percentage of thread engagement to reduce tap breakage. The same alloy of metal can provide substantially different behavior based on heat treatment (annealed, quenched and tempered) and whether the material is hot rolled or cold rolled. For identical conditions, Fig. 6 shows the likelihood of tab breakage without reducing thread percentage or length of engagement.

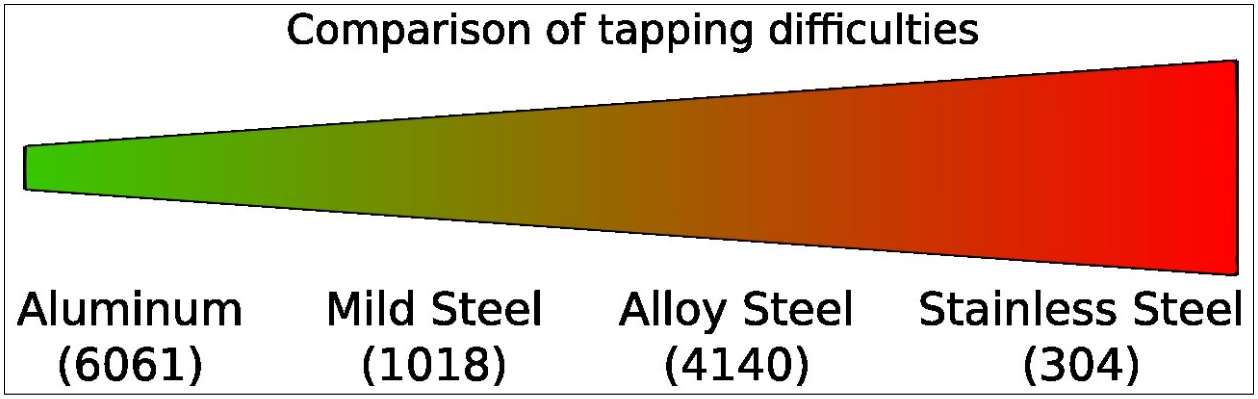


Figure 5: Conceptual comparison of tapping difficulties for different common materials assuming equal conditions.

If you prefer to let Austin Fabrication LLC select percentage of thread engagement to avoid tapping issues, that's no problem! If the parts you specify may lead to tapping troubles, you'll be contacted to discuss potential changes.

For tough to tap materials with small taps, cold form taps are preferred to prevent breakage. These taps require different size starting holes compared to cutting taps. Job requests with large amounts of small diameter taps (6-32 or smaller, for example) in tough materials may be rejected. Some CNC machines may be more appropriate for certain jobs as a mechanical tapping arm creates lateral forces that aren't an issue for larger taps. Note that cold form taps develop what appears to be a cross thread (or a double thread) but are a normal part of material being deformed from two directions, see Fig. 6.

Figure 6: Double thread appearance from cold form taps. This crest recess decreases with higher percent engagement.

4 Hole Drilling and Counterbore Operations

4.1 Nominal Hole Sizes

A drill bit drilling through metal naturally drills slightly oversize. Manufacturing imperfections, spindle runout, and buckling due to the axial force creates an oversize hole. For most applications (even thread tapping) this is not an issue. Hole sizes specified are drilled with the closest drill bit from the following options:

- Fractional drill bits from 1/64th - 1/2 by the 64th
- Fractional drill bits from 9/16 - 1 *in* by 1/16 ths.
- Number drill bits 1 - 60
- Letter drill bits A - Z
- Metric drill bits from 1 - 10 *mm* by .1 *mm*.

If a specialty size hole is required (metric diameters other than multiples of .1 *mm*) the cost of the drill bit may be included in the total. When drilling plastic, the resulting hole is usually smaller than the diameter of the drill bit. Low thermal conductivity causes the plastic in the immediate region of the drill bit to thermally expand. Thus you are drilling through an already expanded material. When the plastic cools, thermal contraction reduces the hole diameter. If tight tolerances on holes drilled through plastic are required, please specify.

4.2 Reamers

Section 4.1 describes the diameter variations for holes produced by drill bits. If tight tolerance is required for a hole, reamers are the standard approach. These are available in just about any size imaginable and are purchased as needed. The cost of the reamer may be included in the total for the work performed.

Figure 7: Standard reamer used for finish holes to accurate diameters with a smooth finish.

4.3 Counterbores

Counterbores for socket head cap screws are done with pilot counterbore tools but can also be done with end mills. For a counterbore tool, the clearance diameter for the bolt must allow the pilot diameter of the tool to enter.

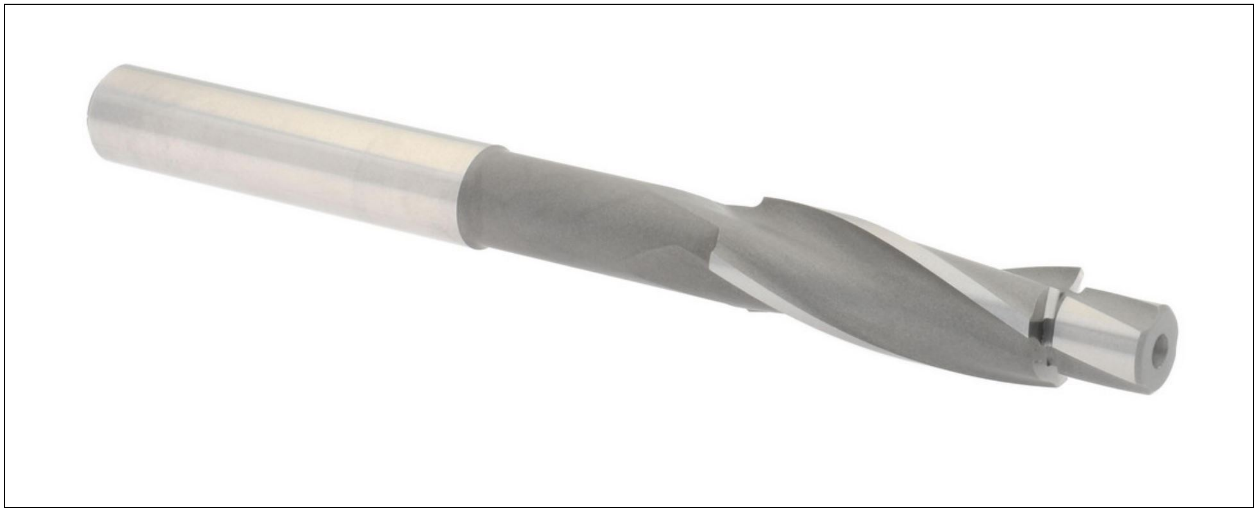


Figure 8: Pilot counterbore commonly used for creating the counterbore for socket head cap screws.

For designs where less clearance (free play) for the bolt is desired, the counter bore tool may not work. End mills can also be used to produce the counterbore but often have a face clearance angle that leaves a tapered bottom when plunging (Fig. 9).

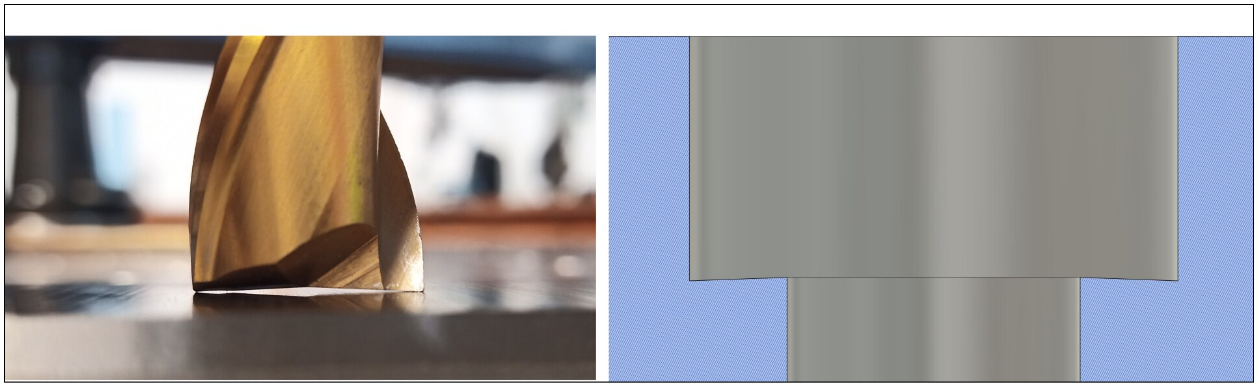


Figure 9: Tapered bottom of counterbore left by using an endmill.

When a socket head bolt is tightened, the tapered bottom shown in Fig. 9 increases the stress concentration on the bolt's head. This is rarely a problem for non critical bolted joints but is noted here for awareness. Unless requested, it will be assumed that an end mill produced counterbore for socket head bolts is acceptable.

5 Bead Blast Finish

5.1 What is Bead Blasting?

Sand blasting (or bead blasting) is a common technique of cleaning up the surface of a part by removing rust, oxidation, and surface appearance irregularities. An air compressor forces air through a nozzle that siphons blast media and ejects at high speeds to impact the surface. Often, sandblasting is the generic name and media other than sand is used (thus the name bead blasting, used here interchangeably). These range from very delicate baking soda, to crushed walnut shells, glass beads, and the more aggressive silicon carbide for heavy rust removal. All varieties come in different particle size ranges (mesh) from course to fine. Figure 10 shows a before and after comparison using fine glass beads.

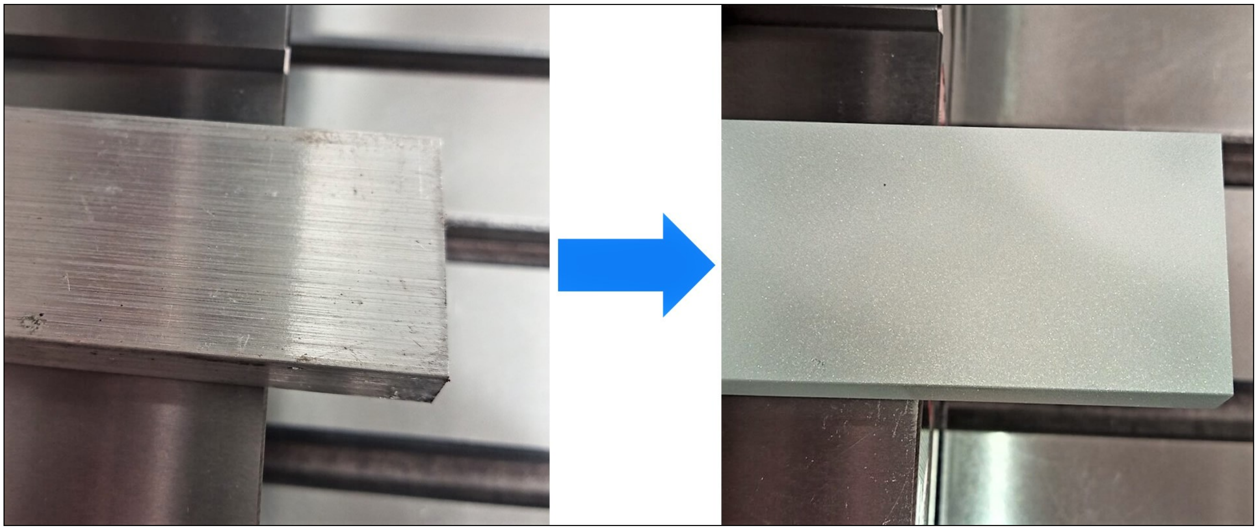


Figure 10: Example of surface finish comparison from raw aluminum flat bar to the bead blast finish. The bead blasted surface is more uniform than shown here due to shop lighting.

5.2 Bead Blast Finish for Machined Parts

A bead blast finish (walnut shell) for parts less than 16" x 12" is available for an extra fee. Objects larger can fit but leaves little room for the nozzle to angle and reach all directions. The finish provides no benefit other than creating a uniform surface finish with a satin/matte, micro pitted appearance. The bead blasting can remove rust effectively with more aggressive blast media. In general, Austin Fabrication LLC is not a sand/bead blasting service and does not accept large bead blasting/rust removal requests.

6 Computer Aided Design (CAD) Services

6.1 Scope of Capabilities

If you have some mechanical part or contraption you have sketched out and wish to have it in a formal CAD file for 3D viewing, technical drawing creation, or for sending out to manufacturing, that's no problem! If you need a simple part or assistance designing a tool or setup to solve your task at hand, we can work out a design to minimize cost and complexity.

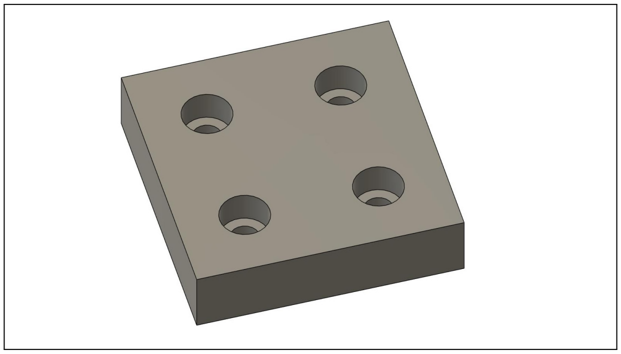


Figure 11: CAD/design assistance for simple parts.

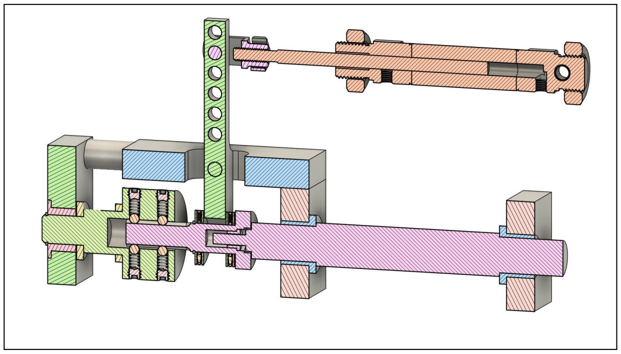


Figure 12: CAD/design assistance for unique solutions.

Please note the following:

- The owner is neither a professional machinist nor a mechanical engineer and does not provide engineering services. No engineering analysis, reliability quantification, or structural integrity is implied by any part or CAD file produced by Austin Fabrication LLC.
- The term "reverse engineer" as used on the website and related documents refers to the process of developing a CAD file from a physical item, and no formal engineering analysis is implied.
- The term "design" as used by Austin Fabrication LLC implies producing a CAD file or physical item that satisfies the customer's requirements and not an engineered product. Please see the DISCLAIMER and LIABILITY WAIVER.

6.2 File Types

Please upload CAD files using the .step format. This is a universally recognized format for all modern CAD software packages. Fusion by Autodesk is used and cannot import part files from other common CAD software suites like SolidWorks or Creo.

7 Additional Info

7.1 Comments on Time

Many machining videos exist online from hobbyists to multi-axis CNC machines producing aerospace components. In those quick 10 minute overviews, they give a vastly inaccurate depiction of how long the machining process actually takes. Austin Fabrication LLC offers manual machining only. The following may help in guiding expected costs:

- Parts held to tight tolerance require more setup time and more cautious machining.
- Parts with unique shapes require more setup time and occasionally machining unique fixtures to safety secure the part.
- Parts that require many different operations (milling, turning, tapping, knurling etc.) require many tooling changes, part orientation changes, and precision alignment with different tools). Figure 13 shows small parts with complex shapes and many operations. Figure 14 is a simple part that requires removing a 3 jaw chuck, swapping in a 4 jaw chuck, and then dialing the part in with low runout that could easily be 20 minutes of work, for perspective.

Figure 13: Complex parts requiring different machining operations.

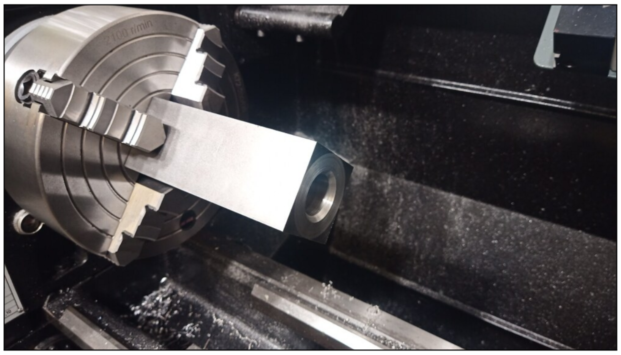


Figure 14: Example of a simple part with a slow setup time.

Requests for services may be rejected if the scale of the operation exceeds the safe operating limits of the machines.

7.2 Shop Updates for August 2026

By September 2026, Austin Fabrication LLC may offer the following services in addition to those listed on the website.

1. MIG and TIG welding services for non structural, non pressurized vessels components.
2. Wire EDM tap removal to remove broken taps that the usual finger tap extractors are hopeless against.
3. CNC plasma cut brackets less than 25” x 23”.

The website and this document are updated regularly.

Thanks for your support,
Austin