Lab 2: Machining
BMEn 2151 “Introductory Medical Device Prototyping”
Prof. Steven S. Saliterman

The following exercises are meant to familiarize you with the capabilities of the Earl Bakken Medical Devices Center (MDC) and Student Machine Shop (ME 176). Some of the exercises are small group instruction, while others allow for independent skill development in using shop equipment. You should only participate in exercises that you are comfortable doing. Ask for direct supervision while performing something new until you feel you can proceed safely alone. Supervisors Cara, Steven and Aaron are in room G217-05 in the MDC. Evan, Rachel and Isaac have an office in the Student Machine Shop (612-626-0342). Be sure to have the supervisor inspect and initial with a marker pen each part you make.

Exercise 2.1: Safety Training
You must complete the MDC online safety course before using the facilities. The Anderson Innovation Labs (Anderson Labs) and Student Machine Shop (Student Machine Shop) have their policies and forms.

Exercise 2.2: Familiarization with Materials
Objective: To introduce you to materials that you will be using in the shop for machining practice. Designated stock is available in the MDC and Student Machine Shop for your use in these exercises.

A. Aluminum, 2” x ¼” flat plate, 6061.

B. Aluminum, 1” round bar 6061, A1.

C. Steel, 5/8” square bar (12L14 low carbon steel).
Exercise 2.3: Medical Devices Center (MDC) Small Group Instruction
Objective: Instruction in various hand and machine tools found in the MDC machine shop. The MDC supervisors will be instructing.

Notes:

Hand Tools

Drill Press & Vertical Bandsaw

Belt Sander

Height Gage on Granite Plate and Dykem Blue
Exercise 2.4: Sawing, Drilling, & Tapping Holes

Objective: Cutting a piece of aluminum flat to size, locating holes, drilling, and making threads with a tap. Knowing how to manually layout your workpiece is essential because digital readouts (DROs) are not on every mill and lathe, and many drilling operations are done on a drill press. (Dial indicators and handle gages are also useful when no DRO is present on a mill or lathe.) You may complete this exercise in the MDC Mechanical Lab.

Equipment and hand tools needed for this exercise:

- Band saw
- Belt sander
- Drill press
- Machinist square
- Dykem blue and remover
- Height gage and granite surface
- Center punch (spring loaded)
- Center/spotting drill
- #29 drill bit
- 8-32 tap and tap wrench.
- Countersink (82 degree chamfer) bit for drill press.

The following pictures illustrate each step necessary to complete the part. While only one hole is completed (drilled and tapped) here, you will complete all four holes in your workpiece. A complete list of steps in written form follows the pictures.

6061 aluminum stock - 2” x 12” ¼”

Toolbox for this exercise. Similar tools can be found elsewhere in the MDC Mechanical lab.
A. Marking the 2” cutting line.

B. Lubricating the saw blade (perform on both sides). You will need to temporarily lift and lock the blade guide.

C. Be sure to lower the blade guide to just above your workpiece.

D. Checking clearance of the workpiece under the blade guide.

E. Notice the rough cut and lack of squareness after cutting with the band saw.

F. Use a disk sander to square the rough cut. Slowly move the workpiece back and forth – but only in the front half of the disc.
K. Center punching the crosshairs.

L. Center drilling after center punching. Only a tiny starter hole is necessary. This keeps the drill bit later from “wandering.”

G. Check squareness with a machinist square.

H. Applying Dykem Blue.

I. Making the 1.625” marks.

J. Making the .375” marks.
M. The crank on the right lifts and lowers the drill table. The handle on the left locks it in place.

N. Workpiece secured in a vice for drilling. It is not necessary to clamp the vise to the table. Instead, firmly grasp the vise with your hand while you drill. Peck drill – in and out.

O. Drilling with the #29 drill bit.

P. Examples of center punched, center drilled, and drilled holes (x2) (cc).

Q. 8-32 tap and tap wrench. Place into the drilled hole and with a little downward pressure turn clockwise. Occasionally make a ¼ turn ccw to “break” chips.

R. The 8-32 tap should be tapered and pointed at the bottom for tapping holes that go through the workpiece. Advance the tap well through the hole, then backout.
W. Completed chamfer and tap.

X. Tapped hole with an 8-32-3/8 flathead screw in place.

S. Manually turn the 8-32 tap with the wrench. Keep advancing until most of the tap threads have gone into the hole.

T. Chamfer bit

U. Chamfer drilling.

V. Dykem Blue cleaner
**Summary of Steps:**

1) Begin with a section of aluminum flat plate, 2” wide and ¼” thick. Measure a length of 2” and cut with the band saw (leave a little margin for sanding the edge smooth).

2) Using a disc or belt sander, lightly sand the sawed edges so that they are square with the factory finished edges. The part should measure about 2” x 2” when you are done. (If precision squaring and sizing were required, you would complete this operation using a mill instead.)

3) Layout the pattern below on the workpiece using Dykem blue and a vertical height gage (use a common datum as shown in the drawing below).

4) Center punch the holes first, then make precise starter holes with the center/spotting drill on the drill press, and finally drill through the workpiece (x4) with a #29 drill bit (based on an 8-32 tap). When drilling, hold your workpiece in a vise and be sure there is clearance below for the drill bit. Use a drop of lubricant.

5) Next obtain an 8-32 tap and tap wrench, and proceed to manually tap the holes. Holding the wrench and tap perpendicular to the workpiece, apply downward pressure and slowly turn the wrench clockwise so that it will cut the threads. Once cutting, after ~every one turn in, make a ¼ turn counterclockwise out to “break” chips that form. A drop of lubricant will help.

6) Using a countersink, add a large chamfer on top for the screws, and tiny chamfer on the bottom to “deburr” each hole (removes sharp edges).

7) Test the holes with an 8-32-3/8” Phillips flat head machine screw (leave one in place).

8) Remove the Dykem blue with the Dykem cleaner (wear gloves). Return the cleaner to the designated inflammables storage cabinet.

Notes:
Exercise 2.5: Turning a Part on the Lathe

Objective: Learn to prepare stock, face, turn to a diameter, create a shoulder, drill, and part a workpiece. Instruction will include familiarization with the features and components of a lathe, use of the chuck to hold a workpiece, selecting and setting up tools, and performing various operations. You will do this exercise in the Student Machine Shop located in ME176.

Evan, Rachel and Isaac are the supervisors, and will be training you. Peter is the shop foreman. You will learn how to operate a lathe and use the digital readout (DRO). Be sure to study and read everything below, including the notes at the end of this section before doing any work. Thank you Evan for contributing to the following instruction!

It will be helpful if you come to the shop with other team members. However, each person should make their own parts. Schedule for two hours on the signup sheet attached to the cabinet in the Brainstorming Room. Schedule additional time later as needed to complete your work independently. Only three students can be in the shop at a time.
Hardinge Lathe

Components of a Lathe
1. Preparing stock.

A. Obtain a 1” diameter 6061 aluminum round rod from storage.

B. Lift the horizontal band saw and position your stock bar using a ruler so as to cut off a 2” piece. Tighten the vice.

C. Start the saw and let it automatically descend and cut through the work. It should automatically stop. The switch and controls are on the left. The red handle starts the saw. The small wheel will start the saw lowering.
D. Use a burnishing wheel or sander to deburr and chamfer your stock. Notice the slightly chamfered edge.

2. Chucking your workpiece, facing and zeroing the Z axis.

A. Select the proper size collet for the stock (1") and place both in the headstock. The handle is turned drawing the collet tightly into the spindle, and firmly holds the stock. Leave the stock sticking out about 1 ½" and tighten.
B. Select a tool (left hand cutting) and place it in the quick change tool post. Adjust the tool angle for a facing operation (to clean the front end of the stock for a smooth finish) by loosening the nut and swiveling the tool holder. You will use this surface to zero the digital readout (DRO), giving you a reference point for locating positions along the Z axis. (See the axis diagram below.)
C. Note that the face is zero, and \(-Z\) goes into the part. You locate \(Z=0\) by leaving your tool in position after facing, and zeroing the DRO Z value. Think of \(X\) as being simply the diameter of the part. For a lathe, \(Y\) is a reference for where your tool tip is. Generally it is at the middle of the part \((Y=0)\) and preset by the tool holder. Subtle adjustments are necessary from time to time.
More about cutting tools (these are HSS ground tools) and jargon:

Rather than using ground tools, we are using steel (or carbide) tool inserts held in a tool bit insert holder. This is held by an adjustable tool holder. This in turn is placed onto the quick change tool post.

3. Determining your X-position, turning to a diameter, and creating a shoulder.

A. Adjust your tool bit angle for turning to a diameter as shown above. Loosen the post with an Allen wrench, and swivel into position. Touch off the side of the workpiece and make a full pass. Turn the lathe off and measure the diameter with a caliper. Return the tool to the workpiece surface (if you moved away), and enter the diameter for the X axis on the DRO.
Steps:

1) Turn the larger diameter for the full length of the part, plus an additional amount for eventually parting (cutting the part from the stock being held in the collet). Make repeated .05” passes until down to 0.9” (look at your handle gage or the DRO). With each pass, bring the tool closer to the workpiece only when you are farthest to the right, off the workpiece. When done turn the lathe off and double check the diameter with a caliper.

2) Next turn the smaller diameter and create the square shoulder. If you position your tool post so that the tool bit edge is parallel with the face, you can create the shoulder as you turn the diameter (if you are careful with each pass). For a deep shoulder, an alternative method is to undersize your passes in the z direction by ~5 thousandths, and make the shoulder as a separate step. Create the shoulder by coming into the workpiece in the X axis at the precise Z location of the shoulder. You can turn the spindle speed up to 1700 rpm for the finishing passes.

B. Periodically lubricate with a brush.

C. Turn to a square shoulder (see below).
D. Drilling a hole through the part.

A. Extend the tailstock spindle out an inch, and insert the tapered end of the drill chuck snuggly into it. (Later remove the chuck by retracting the spindle under an inch – it pops out.) Insert the center drill into the chuck and tighten. Always remove your chuck key. Center drill at 1000 RPM. Use the peck technique.

B. Set your speed, and turn the tailstock handle to make a small dimple in the face of the part. Use lubricant. Replace the center drill with a drill bit and drill to depth (-Z value).

C. Take the left-handed facing tool off and replace it with a parting (cutoff) tool. Reduce speed to 400 rpm. Make sure the cutoff tool is perfectly square on the lathe. Account for tool width when locating the parting tool along the Z axis. It should also be close to Y=0 (center of the part).
Advance the tool into the workpiece slowly to cut the piece off of the stock. Lubricate frequently with a brush while the spindle is turning. Parting puts a lot of stress on the part and lathe, and the lateral force could pop the workpiece out of the spindle if not properly secured.

D. Wipe your tools clean and return them. Clean the lathe with short air bursts so that it is ready for the next user. Sweep up. Deburr your part and you’re done!

**Important notes**

A. “Touching off” is where you remove a very small amount of material so that you know that the cutting tool is exactly aligned with the face of the part. Use this strategy to set zero points or define other dimensions. For example, if you touch off a part and remove some material off the face, then measure the height to be .597”, you know that your tool is at a that height, and then you must move in .097” to reach a desired height of .500”. This strategy is used on the lathe and the mill.

B. Do not have a cutting tool contacting any work piece while it is not cutting. Examples of this include leaving and end mill spinning on in one spot on the steel or leaving a lathe bit stationary on a spinning part. Also do not have a cutting tool contacting the work piece during startup

C. Drilling: always center drill your holes before drilling. You only need to center drill deep enough to make a small indent for the drill bit to get started. With the drill bit, move the drill in for a couple of seconds, back it out slightly, and then advance the drill a little further. Repeat until the drill bit is through the piece. If on the mill, make sure you do not hit the parallels underneath the part

D. Tightening: in the machine shop, tight means TIGHT. This goes for vices, drill chucks, etc
E. Chucks are used for drill bits and center drills. Collets are used for end mills and edge finders. In a lathe, ether can be used to hold parts. The collets for the lathe and mill are different.

F. End mills are designed to cut sideways, not down (*plunging is possible with center cutting end mills*). They are not drills.

G. If you feel that you have messed up, or are unsure about something, stop and ask or stop and measure. You can always take off more material, but it is much harder to put material back.

H. You are always free to use an air gun (short bursts) to blow chips and debris away from your workpiece, but be careful of others.

I. On a lathe “X” is the diameter, “Z” is left/right and “Y” is the location of your tool tip up and down (see discussion in 2B above). On a mill, “X” is left/right, “Y” is front/back, “Z” is up/down.

J. Clean up: Blow all chips off of the mill and lathe and onto the floor. Use a broom to sweep up the chips and throw them away. Put all the tools back in the designated class-materials area.
Exercise 2-6: Milling

Objective: Machinist instruction in operating a mill and use the digital readout (DRO). This include familiarization with the features and components of a mill, aligning the mill vise with an edge-finder, setting up your work piece and establishing a zero reference, and perform various milling operations. You will do this exercise in the Student Machine Shop located in ME176.
Dill hole with #7 drill bit and tap with 1/4-20 UNC
1. Preparing stock.

Use the horizontal band saw to cutoff a suitably sized workpiece. You will be using 5/8" square steel bar (12L14 low carbon steel).

2. Gathering your tools, inserting an end mill into the spindle and using the Kurt vise.

A. Gather up the tools you will need. These are (left to right) a center drill, ½” end mill, angle block, parallel plates (2), deburring tool, mill collet with wobble edge finder, drill, steel stock and drill chuck for mill. You also need a rubber mallet.

B. Have a supervisor instruct you on how to insert a collet and end mill into the mill spindle, adjusting speed, quill stop, Z axis and motor speed.

C. Open the vise with the removal handle. The handle is taken off during milling. The part should be tapped firmly atop two parallel plates within the vise while you tighten it. This assures the part is level with bed.

D. Use a ½” end mill at 800 rpm, and face the top, bottom, and two ends of the rectangle. There is no need to face the front and back.
2. Locating Edges and Establishing X=0 and Y=0.

A. Watch these You Tube videos on how to use a wobbler edge finder to zero the X and Y axis on the DRO, and for locating the center of the workpiece: Setting a Work Zero and Work Locating on the Milling Machine. The latter clip talks about wobblers at 4:37 – skip ahead to this point.

B. Use calipers and measure the height and length of the part. Zero the Z axis by loosening the knurled wheel below the table.

C. Continue to face the top and end of the part using .040” passes until the exterior dimensions are met. For these passes, you can feed the end mill basically as fast as you can crank the wheel.
D. Edge find off the bottom left corner of the piece. Use the DRO to set a (0, 0) position, accounting for the diameter of the edge finder. (Set spindle speed set at ~800 rpm for a wobbler edge finder.)

1) Flick the wobbler with your finger so that it wobbles. Slowly bring it up to the workpiece edge until it stops wobbling. Advance slight, and stop when the wobble first starts to return. When the edge finder is up against the part, the two cylinders are concentric, meaning that the edges are against each other.

2) We care about the position of the center of the spindle and not the edge. The center of the spindle is 1 radius away from the edge, or 0.25” for a .5” wobbler.
3. Drill holes and milling the slot & angled flat.

A. Insert the drill chuck in place of the ½” end mill. Center drill the locations of the slot ends and the hole. Drill with a #7 drill bit.

B. Replace the drill chuck with collet and ¼” end mill, and make the slot at and RPM of 800. Use .040” passes until through the part. Always lower the Z axis when over one of the slot end holes, unless you are using a center cutting end mill. Clamp your work very tightly.
C. Insert the angle block and clamp the workpiece firmly above it. Locate the top edge of the workpiece and mill down an amount determined by the measurements given in the drawing (you need to calculate the side of the right angle!)

Deburr and you are finished. Use the hand tool and deburring wheel.