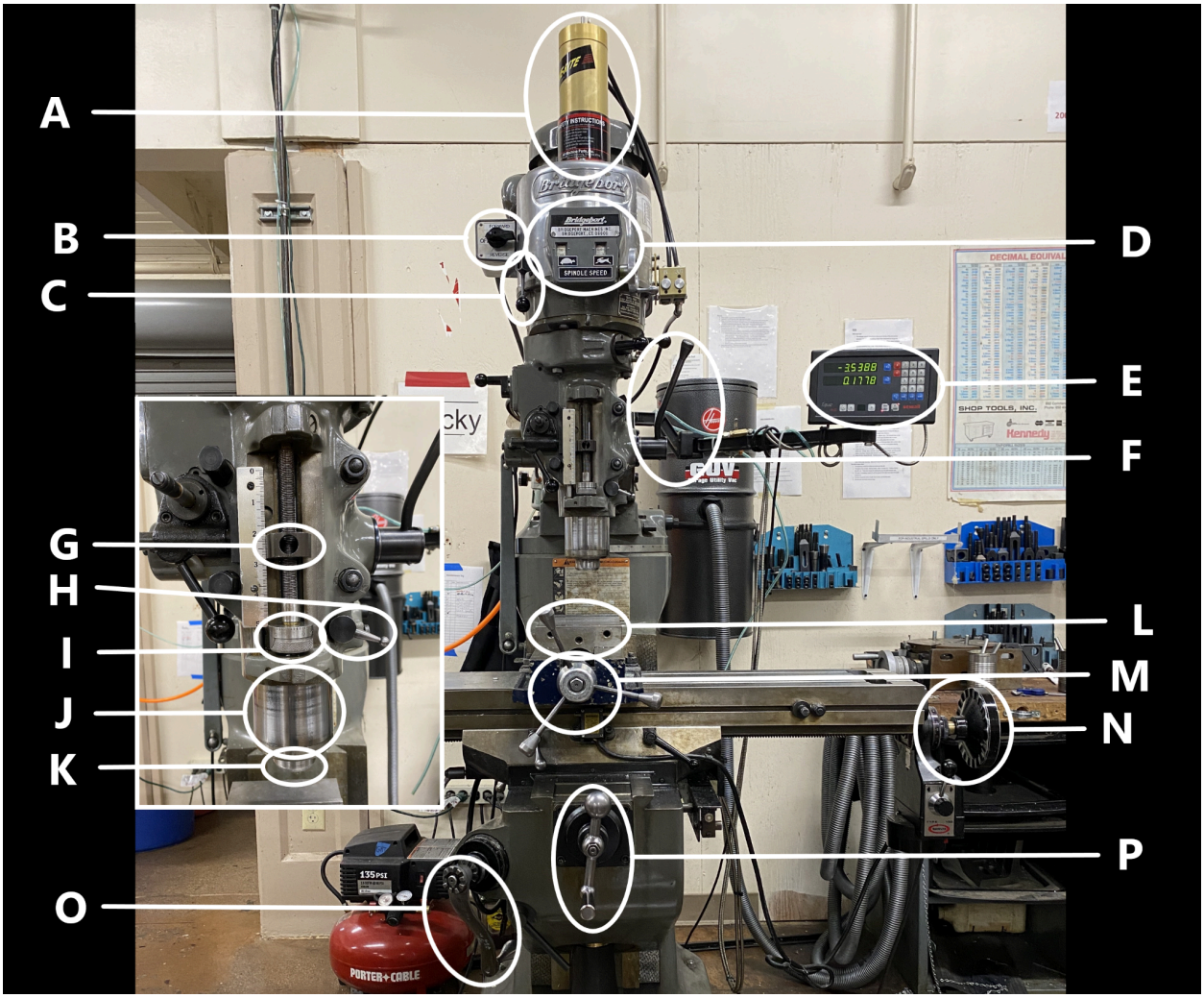
Mill Lead: Aiden Man

Identify the mill parts name and function using the diagram below:



A. Pneumatic Drawbar, Tightens or loosens tools in the spindle (eg collets, morse taper drill chuck)

B. Power Switch, Turns machine on/off in either forward or reverse direction

C. Spindle Brake, Stops the spindle, should only be used after the motor is turned off

D. Speed Change Handwheel (for Rocky only), Adjusts the machine's RPM by handwheel turn, should only be spun when the motor is on

E. DRO, Shows where the spindle (along with the tool) is relative to a user-defined zero

F. Quill Feed Handle, Raises and lowers the quill (along with the spindle) along the z-axis

G. Quill Stop, Moves up and down along with the spindle and quill

H. Quill Feed Lock, Locks the quill in place

I. Micrometer Nut, Used to stop the quill at a certain depth when lowered

J. Quill, holds the spindle, is able to raise and lower

K. Spindle, holds and rotates the collet+tool

L. Vise, Clamps stock in place

M. Vise Handle, Used to tighten/untighten the vise, should be removed or flipped when a part is actively being clamped

N. X-axis Handwheel, Moves table in the x direction

O. Knee Crank, Moves table in the z direction

P. Y-axis Handwheel, Moves table in the y direction

Describe the function of each tool:

Collet: Holds tools of specific diameters (we mainly use ½" collet)
Parallels: Raises the stock/part so it can be held above the vice edges
End Mill: Used for facing and slotting operations, cuts material from its sides
Edge Finder: Used to zero
Drill Chuck: Used to hold drill bits (<½")
Drill Bits: Drills holes
Center Drill: Creates divot, marks where holes will be drilled, prevents walking while drilling
Tap Guide: Pushes the tap handle and tap into the part, keeps the tool centered and perpendicular
Tap Handle: Holds the tap, allows user to manually turn the tap
Tap: Cuts threads into holes

Describe the drill bit types and what each is used for:

Regular: Same diameter throughout, usually smaller and held in the chuck
Silver and Deming: Cutting edges has a larger diameter than the shank, shank is typically ½" and goes directly into ½" collet, never the chuck

Operations Speeds:

Facing: 1400 rpm
Edge Finding: 1000 rpm
Center Drilling: 1200 rpm
Drilling: Consult chart next to the mill

Describe the following procedures:

Inserting tools into the mill:

1. Wait until the pressure gauge reaches 80 psi (after the air compressor has been turned on)
2. Push the quill to its highest position and lock it in place
3. Clean and inspect the tools and the spindle
4. Insert the collet with the tool, holding the tool in place while pushing the collet upward (using the hand guide); never clamp on flutes
5. Announce “loud noise” and click “in” with short, hard clicks until you hear the “clicking sound”or “pitch drop”

Removing tools from the mill:

1. Wait until the pressure gauge reads 80 psi (after the air compressor has been turned on)
2. Push the quill to its highest position and lock it in place
3. Place your hand underneath the tool
4. Announce “loud noise” and click “out” until the tool comes out and catch the tool
5. Set the tool aside and place your hand underneath the collet
6. Announce “loud noise” and click “out” until the collet comes out and catch the collet

Stopping the mill:

1. Turn the power switch to “off”
2. Apply the spindle brake

Clamping stock:

1. Clean off the part, vice, and parallels
2. Place parallels up against the jaws of the vise and place the part on top (surfaces should be flat)
3. Tighten the vise; apply pressure downward so the part sits flat; clamp as much surface area of the part as possible

Speed change for rocky:

1. Turn the power switch to “on”
2. Adjust the speed change wheel (only when the machine is on)

Clean up:

1. Wipe down the tools, part, and vise with a clean rag
2. Put parts and tools away
3. Clean the workspace; vacuum the chassis, inside the slots, and the areas behind/below it

Describe the following operations:

Facing:

1. Push the part barely into the endmill using the x axis handwheel (stop when you see/hear cutting)
2. Make the pass using the y axis and conventional milling first
3. Bring the tool to the opposite side of the part with climb milling
4. Zero the x axis
5. Use the DRO to measure next pass (maximum 50 thousandths (0.05”))
6. Make passes until the edge is fully faced (indicated with silver surface+lines)

Edge Finding:

1. Offset the magnetic head and lower the quill where the head is able to contact the edge of the part
2. When the machine is on, push the edge towards the edge finder
3. When the magnetic head goes from centered to slightly offset, set the corresponding axis zero (X_0 or Y_0)
4. Repeat step 3 at the same location until values are within 0.003” of each other
5. Offset by radius of the edge finder (.25”) to position the zero over the edge

Drilling:

1. Use the center drill to make a small divot at the desired location (no oil when center drilling)
2. Apply oil
3. Adjust the speed (based on drill bit size + material), then drill while pecking
4. If the hole over ¼” then drill steps up to the desired hole diameter

Tapping:

1. Oil the already drilled hole
2. Place the tap + tap guide in the hole and align the tap guide spring with the tap handle
3. Lower the quill, pushing the spring against the tap handle so only a sliver of the spring is showing
4. Oil the tap
5. ½ turn clockwise, ¼ turn ccw
6. Repeat until the hole is tapped (indicated with no more resistance)

Describe the difference between Conventional and Climb Milling:

Conventional: The part is moving against the rotation of the end mill blades. Use this for most operations because it is safer for the machinist.

Climb: The part and the rotation of the end mill blades are moving in the same direction.Only used as a finishing pass because flutes pull on the part.

Describe the process of facing stock to a specific length:

1. Face one edge
2. Flip the part over and face the opposite edge
3. Measure and determine the next pass towards desired length (always undercut)
4. Make the pass, and repeat 3+4 until the measurement is within tolerance