4. EXERCISES:

Example 4.1: A 12 mm through hole needs to be drilled into a 6 mm thick plate of 316 Stainless Steel (316 S.S. is an austenitic steel); find the appropriate spindle rpm and the theoretical feed rate for this process.

Solution:

Find the cutting speed for a HSS tool from Table 1.1.1 and use the formula

$$\frac{(Cutting\ Speed\,[m/\min])\ x\ 320}{Tool\ \ Diameter\,[mm]} = Spindle\,Speed\,[rpm]$$

Using a middle value of 13.5 [m/min]

$$\frac{13.5\ [m/\min]\ \ x\ 320}{12\ [mm]} = 360\ [rpm]$$

Note though that you are drilling into stainless steel and need to consider using a harder tool (in general S.S. cannot be drilled with HSS tools).

For a carbide tool the spindle speeds need to be increased by a factor of 2 to 3. This would lead to an spindle speed of ~900 [rpm]

From Table 1.1.2 and converting units, a middle value of 0.18 [mm/rev] can be found for the feed rate.
Do not increase the feed rate by the same factor as the spindle speed; you may round it to 0.2 [mm/rev].

General Note #1: **Before drilling any hole, determine the location of the hole on the part and use a centre-punch (a pointed, chisel-like hand tool) and a hammer to mark (= centre-punch) the hole. This will prevent the drill bit from drifting.**

General Note #2: **Whenever drilling holes larger than ~1/4" or ~6 mm so-called pilot holes must be drilled, especially into harder materials; pilot holes are smaller holes with diameters <1/4" or <6 mm. In some instances a so-called centre-drill is used for this purpose. Having drilled the pilot hole, change the drill bit to the next size (in increments of ~1/4" or ~6 mm) until you arrive at the final hole diameter. Do not move the part with respect to the tool during this process. Watch your spindle speeds.**

General Note #3: **Always use cutting fluids to reduce friction and heat.**

Example 4.2: The hole from Example 4.1 needs to be reamed. Select a suitable spindle speed.

Solution:

Without changing the position of the part on the table, change the 12 mm drill bit to the appropriately sized reamer (probably 12.01 mm) and adjust the spindle speed to between 1/3 and 2/3 of the drilling speed.

Assuming a carbide reamer, the spindle speed for this operation should be between 300 and 600 [rpm]. Use a very slow feed rate and generous amounts of cutting fluid.

General Note #4: Assuming that this was the last operation for this part on the drill press, remove the part from the table, wipe it clean and carefully deburr all newly created edges.

Example 4.3: A part requires 4 counter-bored holes for ½" cap screws. On the production drawing the holes would be specified to have a diameter of 17/32" (this is called a clearance hole and allows the bolt to pass easily through the hole), a counter-bore diameter of 25/32" or 13/16" and a counter-bore depth of at least 0.500". Assuming that the part material is AISI-1020 plain carbon steel, 1.5" thick, with a Bhn of 140, select the appropriate tools and determine the appropriate speeds. Explain all the steps required to fabricate the holes.

Figure 4.3.1: Cap Screw

Example 4.3 continued:

Solution:

Step	***Process***	***Tool***	***Spindle rpm***	***Feed Rate***	***Comments***
1	*Layout of holes*				
2	*Centre punch*	*C-punch, hammer*			
3	*Set-up of part on table*				
4	*Pilot hole*	*1/4" HSS drill bit*	*~1600 [rpm]*	*0.16 [in/sec]*	*Table 1.1.1: ~100 [sfm]* *Table 1.1.2: ~0.006 [in/rev]*
5	*Required hole*	*17/32" HSS drill bit*	*~800 [rpm]*	*0.13 [in/sec]*	*Table 1.1.2: ~0.010 [in/rev]*
6	*Counter bore*	*Nominal ½" C-bore bit*	*~500 [rpm]*	*slower*	*Set depth gauge to >1/2"*
	Repeat from step 3 for remaining holes				

Example 4.4: Using a table similar to the one from Example 4.3, specify **all** fabrication steps to fabricate the part. Material of the bar: high strength bronze. Assume that 6 [mm] diameter brass rod is available as round bar stock in 1 [m] long pieces.

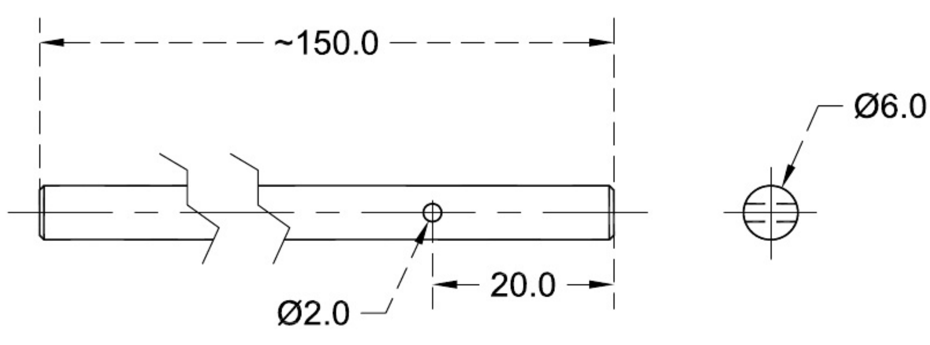

Figure 4.4.1: Rod with Radial Hole

Solution:

Step	**Process**	**Tool**	**Spindle rpm**	**Feed Rate**	**Comments**
1	*Cut part to length*	*Bandsaw*			
2	*Deburr ends*	*File*			
3	*Lay-out of hole and Centre punch*	*C-punch, hammer*			
4	*Set-up of part on table using V-blocks and clamps*	*V-blocks, Clamp(s)*			
5	*Drill hole*	*2 [mm] HSS drill*	*~3000 [rpm]*	*~2.5 [mm/sec]*	*Table 1.1.1: ~20 [m/min] Table 1.1.2: ~0.05 [mm/rev]*
6	*Deburr hole*				

Example 4.5: Plain carbon steel bar stock, 4" wide and 1" thick has been pre-cut (on the bandsaw) to 6" long pieces (= blanks). You are required to reduce the thickness to 0.925", producing a finished part of 6" x 4" x 0.925". Both, the top and bottom surfaces need to be machined and need to be parallel. The carbon steel has a Bhn <150.
Use a 2" face mill with 7 carbide cutting edges for the cutting operation. Calculate the required cutting speeds and feeds.
Also describe how you would set the part up and the steps required to produce the part.

Solution:

To find the appropriate spindle speed go to Table 1.2.1; when determining the feed (or chip load) Table 1.2.2 only provides HSS information. For the appropriate values for carbide tools go to the following site:

http://www.niagaracutter.com/techinfo/millhandbook/speedfeed/index.html

and select "Chart 1". There you will find the required information for carbide face mills.

Spindle speed:

recommended cutting speeds range from:

400 - 900 [sfm]	*from Table 1.2.1*	*or*
330 - 650 [sfm]	*from "Chart 1"*	

An average value would be: *500 [sfm]*

With

$$\frac{(Cutting\ Speed\,[sfm])\ x\ 4}{(Tool\ Diameter\,[in])} \cong Spindle\ Speed\,[rpm]$$

the required spindle speed would be 1000 [rpm]

Example 4.5 continued:

Feed rate:

recommended feeds from "Chart 1":

0.004 - 0.012 [in/tooth/rev] for cutting depths from 0.05" to 0.25"

Using $(Spindle\ RPM)\ x\ (Chip\ Load)\ x\ (\#\ of\ \ Teeth) = Feed\ Rate[in/\min]$

and a mean value of 0.008 [in/tooth/rev] will lead to a feed rate of 56 [in/min].

Set-up and machining:

Since both large faces of the part need to be machined and also need to be parallel to each other, but no information is available on the required quality of the surface finish, a single cut across each face could be enough to achieve the required reduction in height and the parallelism requirement. If a better surface finish is required, mill the part to appr. 0.940" and then use a surface grinder to remove the excess material and achieve the required surface finish.

Using a pair of parallels, ideally at least as long as the part, set the part up in a machine vise so that no more than ~1/4" of the part extends above the vise. This means that you need to select parallels of a suitable height. Ensure that the vise surfaces are clean and place the parallels approximately 1" from the edges of the part. Ensure that the part is in full contact with the parallels when tightening the vise and that the part is properly clamped.

Next decide how much material you are going to remove from the first face: if you are planning on making a single cut across each face, then make each cut approximately 0.0375" deep. After the first face has been machined, remove the part from the vise, and using a micrometer take several measurements of the thickness of the part. You will now be able to determine how much material to remove when milling the other face.

Now place the part back in the vise, machined face on the parallels, and clamp it properly. Lower the cutter head the required amount and mill the part to the required height.

Example 4.6: The part shown below needs to be machined out of cast aluminum with a Bhn of ~100. The length and width of the blank do not need machining; it's height, as shipped, is 32 [mm].

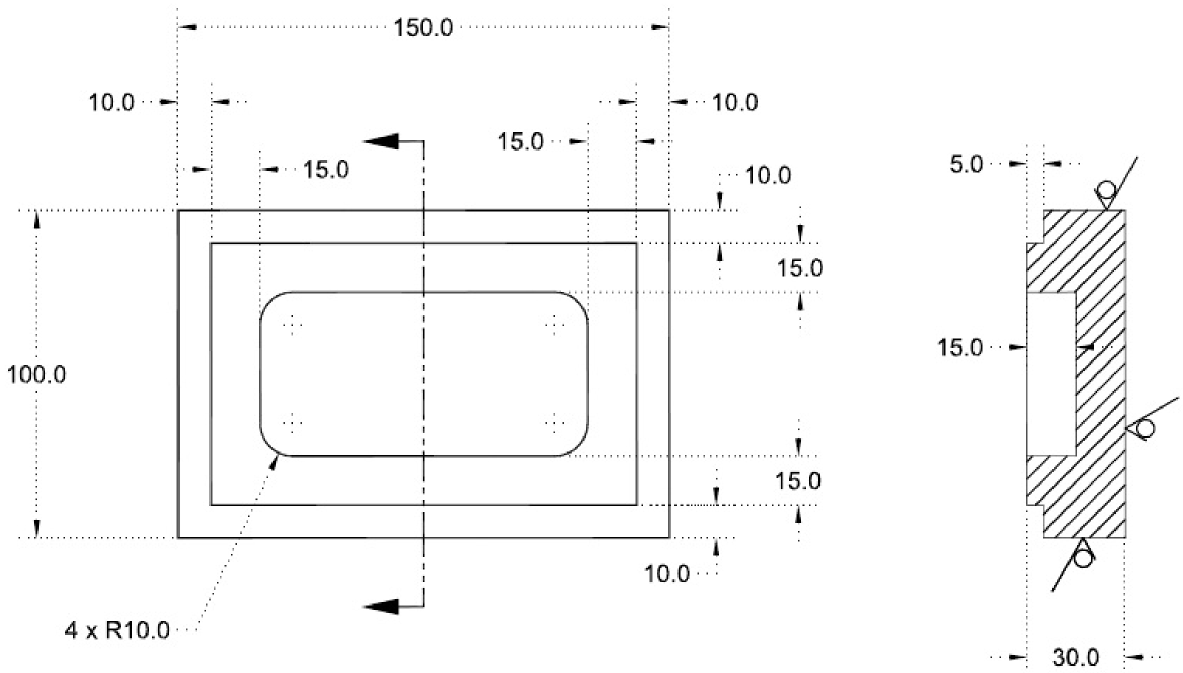

Figure 4.6.1: Cast Aluminum Cover

Specify suitable HSS tools for the fabrication of this part and determine the speeds and feeds for those tools/operations.

Solution:

Larger diameter tools are preferred over small diameter tools for their rigidity and also because of shortened "production" times. One limiting factor in this case is the specified fillet radius of 10 [mm]; the largest tool diameter to machine the "pocket" would be 20 [mm]. The step around the outside edge can be cut with any tool with a diameter considerably larger than 10 [mm]. Use a face-mill to reduce the height of the part to 30 [mm].

Therefore: use a 20 [mm] diameter, centre-cutting, end-mill (at least 4 flutes) and a face mill. Use Table 1.2.1 or Table 1.2.2 or "Chart 1" (from Example 4.5) to find the cutting speeds and feeds.

Example 4.7: The part shown below needs to be fabricated out of 4" x 1" plain carbon steel (Bhn <150) bar stock. The bar stock has been pre-cut to 8" long blanks. Specify all fabrication steps to produce the part and the tools required.

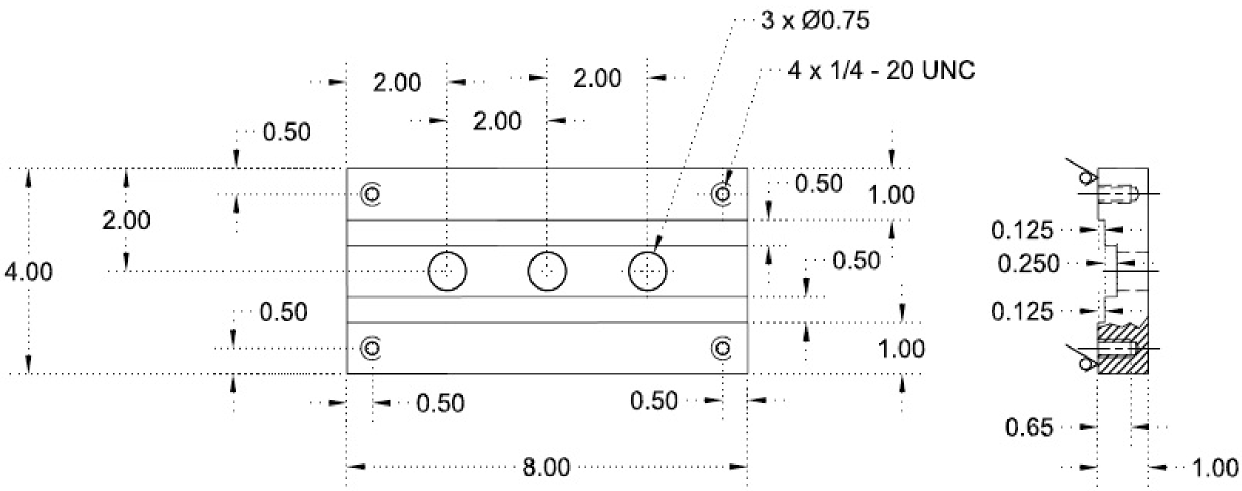

Figure 4.7.1: Base Plate

Solution:

Before proceeding with the solution, study the part and try to determine which machining operations will have to be performed and which machine(s) will be most suitable to perform those operations.
By looking at the drawing you determine that you need to produce:
- *-3 through-holes with a diameter of 0.75"*
- *-4 threaded (blind) holes with a 1/4 - 20 UNC thread*
- *-2 "steps" of 0.125" and 0.25" (relative) depth of specific (2.0" and 1.0") width.*

You should also note that the top surface of the part must not be machined.

Can these operations be performed on one machine, with one set-up, or do you need to use more than one machine?

All the required operations can be performed on a milling machine, using a single set-up. Since only 4 holes need to be threaded (= tapped), the actual tapping will be done manually.

Example 4.7 continued:

Tools required:

0.75" diameter through holes:	*either a 0.750" diameter, centre-cutting end mill, or a set of drills (0.250", 0.500" and 0.750" diameter)*
1/4 - 20 UNC holes:	*a #7 drill (0.2010" diameter) and a 1/4 - 20 flat bottom tap*
Milling operations of "steps":	*the narrower "slot" is 1.0" wide and, theoretically, could be machined with a 1.0" diameter tool; the preferred approach is to use a smaller diameter tool, so that the full width is achieved on the second pass. Use a 0.750" diameter end mill. The same tool can be used for the "wider" slot.*

Set-up:

The part will have to be set up on 2 parallels, ideally at least 8" long, ensuring that they are placed clear of the 3 through holes. Since the top surface of the part does not need to be machined only little, if any, of the blank should extend above the jaws of the vise. Ensure that the blank is in full contact with the parallels and clamp securely.

Machining:

Based on the dimensioning scheme used in Figure 4.3.4, you need to use the 4 corners of the blank as reference points and offset the tools from those 4 points. The sequence of machining operations, in this example, is largely based on personal preference. If a 0.750" diameter end mill is used to machine the 3 through holes, then it is better to machine the slots first, so that the through holes are "shorter". Since the threaded part of the 4 holes must be 0.65" deep, the drilled holes must be ~0.80" deep (otherwise the tap cannot cut to the required depth of 0.65").

Next determine the required speeds and feeds.

Example 4.8: The part shown below needs to be fabricated from 2.5" AISI 1030 Q&T steel, Bhn 400. Determine the type of tool(s) required and the speeds and feeds for the required operations.

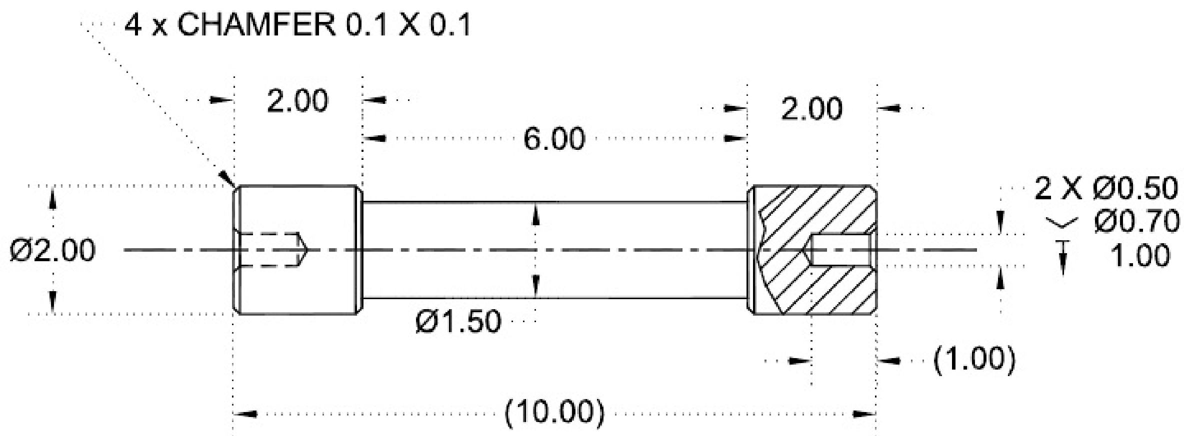

Figure 4.8.1: Cylindrical Spacer

Solution:
The part will have to be machined to length, turned to the required diameters and two 0.50" diameter holes will have to be drilled and counter-sunk.
Machining to length (assuming that the part was pre-cut on the bandsaw to ~10.25" length) and turning the required diameters and chamfers can be done with standard turning tools. But because of the very high Bhn of 400, carbide tools will be required. The holes in both ends will be fabricated by using carbide drills and counter-sinks mounted in a chuck in the tail-stock.
Using the appropriate tables the speeds and feeds for the turning and drilling/counter-sinking operations can be determined.
Based on a dimeter of 2.5" and from Table 1.3.1 the cutting speed for carbide tools can be found to be 280 [rpm]. From Table 1.3.2 the roughing and finishing feeds are .008 - .030 [in/rev] for the rough cut and .006 - .010 [in/rev] for the finishing cut (use .020 and .008 [in/rev] for roughing and finishing respectively).
Turning:
Cutting speed: 280 [rpm], roughing feed rate: 5.6 [in/min], finishing feed rate: ~2.2 [in/min].
Drilling:
Cutting speeds: 1/4" drill: 800 [rpm], ½" drill and c-sink: 400 [rpm], Feed: manual (.004 - .010 [in/rev]).

Example 4.9: The part shown below needs to be fabricated from typical aluminum round bar stock. Determine the type of tool(s) required and the speeds and feeds for the required operations.

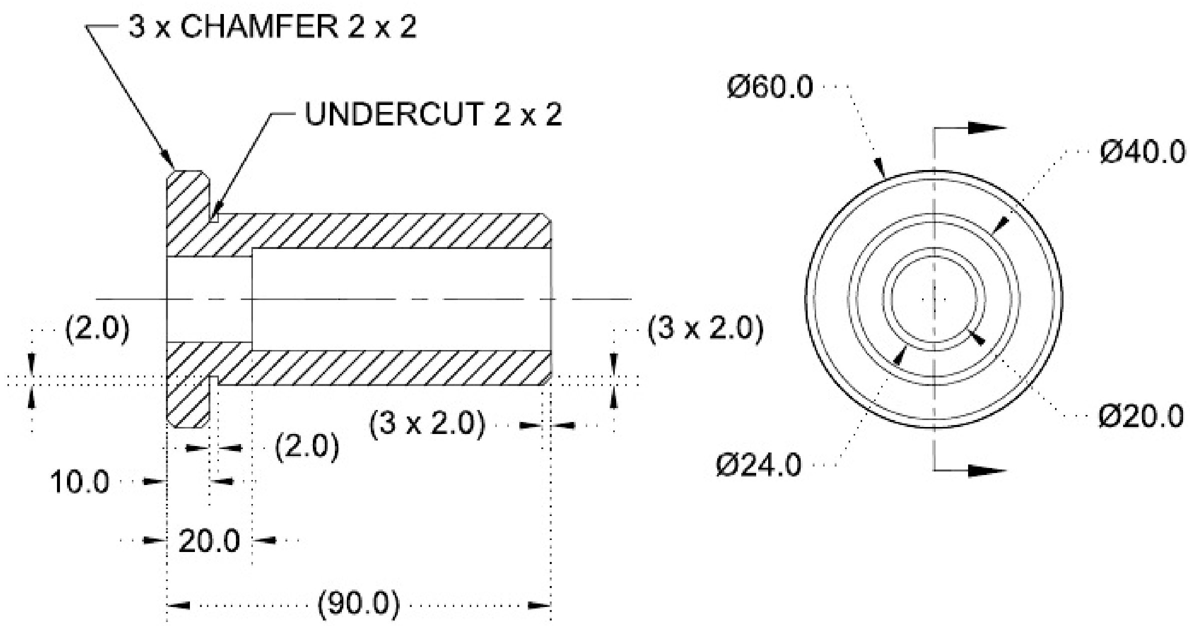

Figure 4.9.1: Sleeve

Note 1: a 2 x 2 chamfer call-out specifies that the chamfer has to be produced by removing 2 [mm] of material in the x direction and 2 [mm] in the y direction, producing, in this case, a face inclined at 45 degrees to the vertical or horizontal.

Note 2: a 2 x 2 undercut call-out specifies a groove which is 2 [mm] wide and 2 [mm] deep. Undercuts are required when the part has to mate perfectly with another part when assembled.

Solution Example 4.9:

Tools: *standard HSS turning tools for outside diameters and chamfers, cut-off or parting tool for the undercut and the actual parting-off of the part from the stock. For the inside bores: a centre-drill and a set of drill bits ending with a 20 [mm] diameter bit and a 24 [mm] end mill or a 24 [mm] boring bar (Note: a 24 [mm] drill bit would produce a chamfered transition between the 20 [mm] diameter bore and the 24 [mm] diameter bore and can therefore not be used).*

Speeds and feeds:

Turning the outside diameters:	*between 450 and 700 [rpm] and ~300 [mm/min] roughing and ~125 [mm/min] finishing.*
Inside bores:	*between 800 and 2000 [rpm] using pecking motion to clear the chips. Manual, slow feed.*
Undercut and parting-off:	*<450 [rpm] Manual, slow feed.*

Use generous amounts of cutting fluid for inside bores and cut-off operations.

Example 4.10: The part shown below needs to be fabricated from typical mild steel, AISI 1020, Bhn 150, round bar stock. Determine how to fabricate this part and specify all tools, speeds and feeds. Use a manual tap to produce the thread.

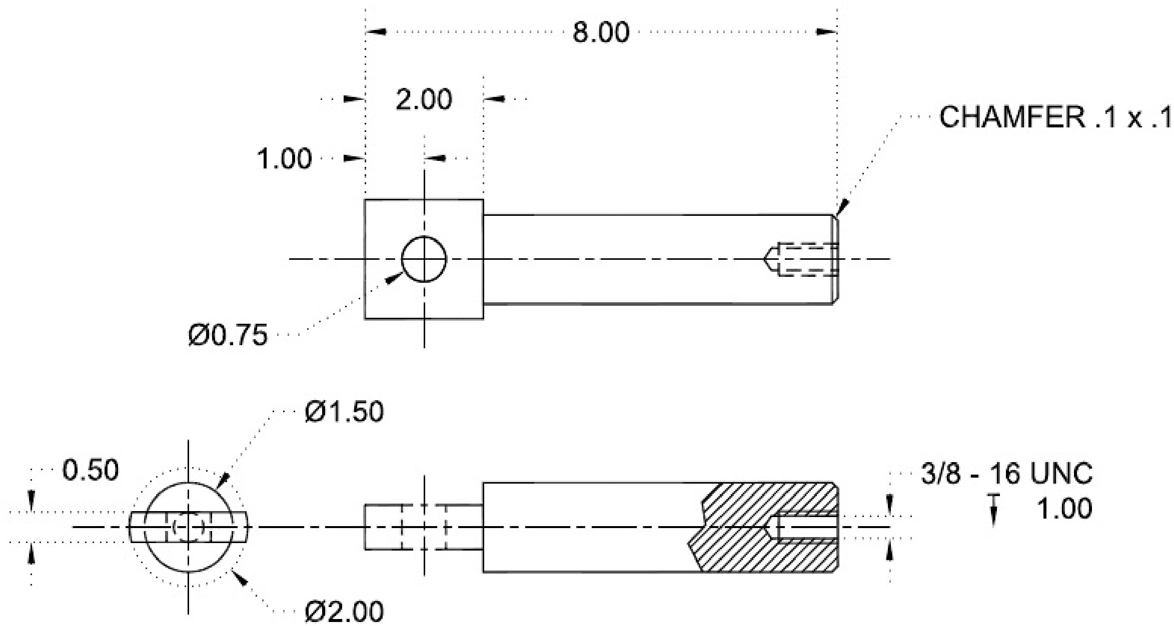

Figure 4.10.1: Lug-Pin

Solution:

Machines and tools:

To produce this part, both, a lathe and a mill will have to be used. Assuming that the part will be produced from bar stock of appropriate length and diameter, a centre-hole will be drilled at the right end of the part and then a 5/16" hole for the 3/8" threaded hole will be drilled to a depth of ~1.5" (this will prevent the tapping tool from bottoming out when producing the 1" deep thread).Then the part will be supported there by a centre in the tail-stock. Now turn the 2.0" and 1.50" diameters and the chamfer. Using a cut-off tool turn the part to the required length.

Now move the part to a milling machine: securely mount the part vertically in V-blocks, using the 1.5" diameter of the part, and a machine vise. Using an end mill, diameter 0.5", for example, begin to remove the required material, to produce the two flat faces, in several passes.

Now mount the part horizontally, using the same V-blocks and drill the hole perpendicularly to the just machined faces. You may have to use a dial indicator to confirm the proper orientation of the part. Then drill the 0.75" diameter hole in 2 or 3 stages.

Finally, manually tap the 3/8" hole. Mount the part (using the flat faces) in the machine vise.

Speeds and feeds: *for carbide tools*

Drilling: *speed theoretically ~3500 [rpm], but since the hole is relatively deep compared to its diameter I would not use a speed higher than ~2000 [rpm]; use manual feed and generous amounts of cutting fluid*

Turning: *speed between 800 and 1000 [rpm] and feed rates of ~18 [in/min] for roughing and ~7 [in/min] for finishing. Lower speeds and manual feed for parting-off.*

Milling faces:

speed at 600 [sfm] ~4000 [rpm] and based on a chip load of 0.004 [in/rev] a feed rate of theoretically 64 [in/min]. Due to the relatively poor mounting arrangement of the part, I would reduce the feed to about half the theoretical value. Use plenty of cutting fluid.

Example 4.11: Based on the image below, answer the following questions:

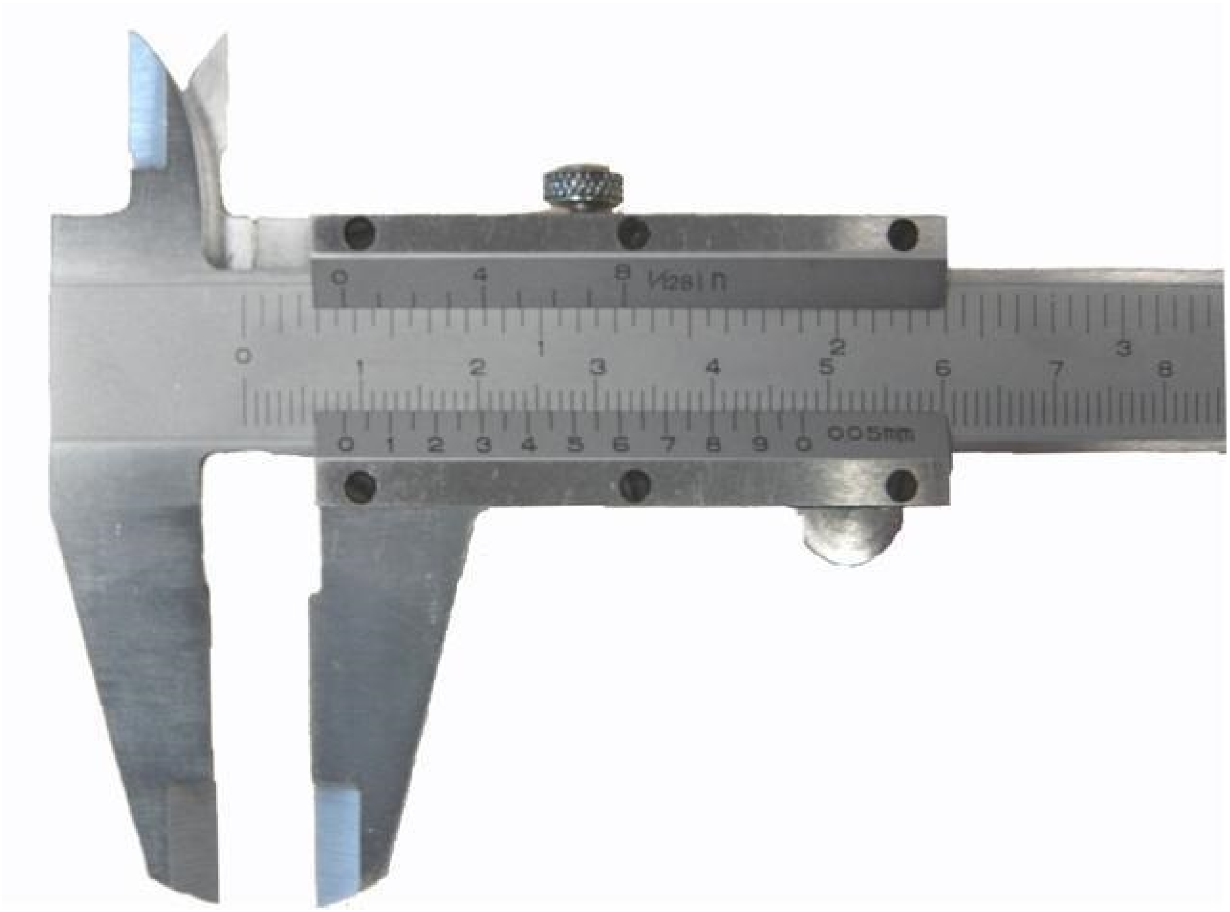

Figure 4.11.1: Vernier Caliper

What is the reading in [mm]?

What is the discrimination of the main-inch scale?

What is the reading in fractional inches?

What is the reading in decimal inches?

Is the Vernier inch scale more or less precise than a typical inch-based Vernier?

What is the precision of the inch based scale?

Solution: *8.65 [mm]*
1/16"
11/32"
0.3438" *(based on conversion from fractional to decimal, BUT NOT based on the discrimination of the instrument).*
Less precise; its precision is 1/128" which is equivalent to 0.0078"
1/128"

Example 4.12: Shown below is a standard micrometer; what is the reading of the instrument?

Figure 4.12.1: Inch Micrometer

Solution: *The zero line on the thimble appears to be right on the axial line on the sleeve, indicating a measurement of exactly 0.250", that also means that there are no values from the thimble (multiples of 0.001") to add to the 0.250"; but when looking at the Vernier scale we can see that the first line on the thimble matches a line on the Vernier scale's "1" line; therefore the final reading is 0.2501".*
(0.25" + 0.000" + 0.0001" = 0.2501")