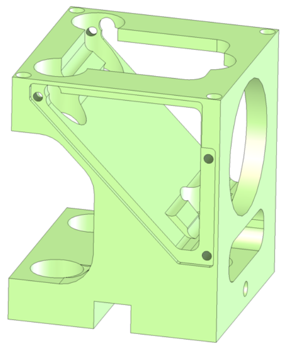


Machining Techniques

Dimensions, Tolerance, and Measurement
Available Tools

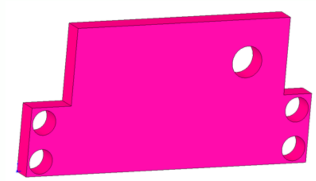
Why Machine Stuff?

- Research is by definition “off-road”
  - frontier work into the unknown
- You can’t just *buy* all the parts
  - mounting adapter between laser and telescope
  - first-ever cryogenic image slicer
- Although there are some exceptions
  - optical work often uses “tinker toy” mounts, optical components, and lasers
  - cryogenics often use a standard array of parts
  - biology, chemistry tend to use standardized lab equipment
- Often you have to design and manufacture your own custom parts
  - learning about the capabilities will inform your design
  - otherwise designs may be impractical or expensive

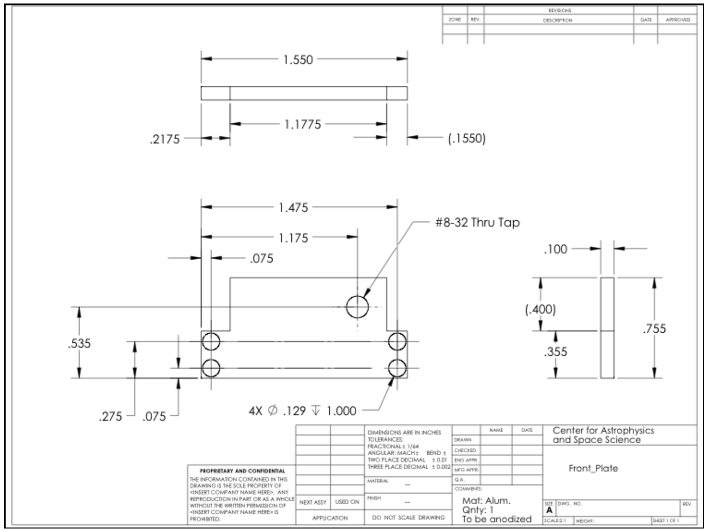
Critical Information

- If you ask a machinist to make you a widget, they’ll ask:
  - what are the dimensions?
  - what are the tolerances?
    - huge impact on time/cost
  - what is the material?
    - impacts ease of machining
  - how many do you want?
  - when do you need it/them?
  - what budget does this go on?
    - at $50 to $80 an hour, you’d best be prepared to pay!
- We’ll focus on the first two items

Dimensions



- You want to make a part that looks like the one above
- How many dimensions need to be specified?
  - each linear dimension
  - each hole diameter (or thread type)
  - 2-d location of each hole
  - total: 22 numbers (7 linear, 5 holes, 10 hole positions)

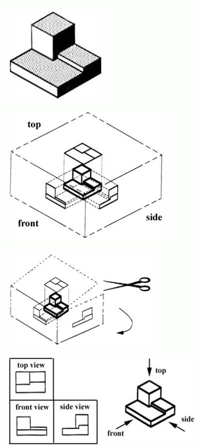


Notes on previous drawing

- Some economy is used in dimensioning
  - repeated Φ0.129 diameter holes use 4× to denote 4 places
    - 4 → 1
  - connected center marks on holes allow single dimension
    - 8 → 4
- Numbers in parentheses are for reference (redundant)
- Dimension count:
  - 16 numbers on page
    - 6 linear plus 2 reference (don't count)
    - 6 hole position, representing 10
    - 2 hole descriptors, representing 5
  - total information: 21 numbers (equal height of tabs implied)
- note: depth mark on 0.129 holes is senseless
  - artifact of the way it was made in SolidWorks

Standard views

- In American (ANSI) standard, each view relates to the others on the page such that:
  - pick the "main" view
  - a view presented on the right of the main view is what that part would look like if you looked at the part from the right side of the main view
  - a view above the main view is how the part looks from above the main view
  - etc.
- The examples on the right come from a good page:
  - http://pergatory.mit.edu/2.007/Resources/drawings/
- The international (ISO) standard *is exactly opposite!!*

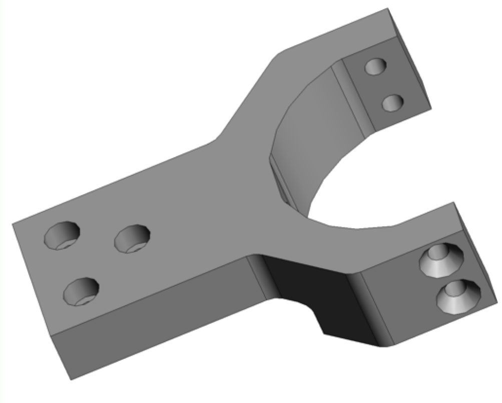


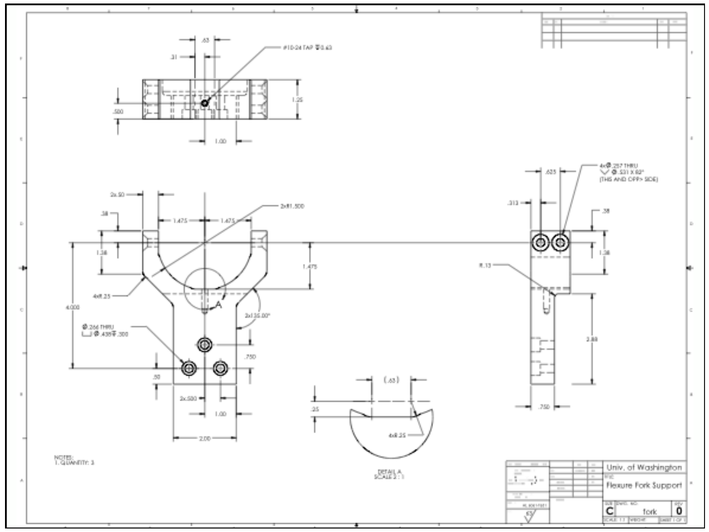
Tolerances

		NAME	DATE
DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ± 1/64 ANGULAR: MACH ± BEND ± TWO PLACE DECIMAL ± 0.01 THREE PLACE DECIMAL ± 0.002	DRAWN		
	CHECKED		
	ENG APPR.		
	MFG APPR.		
MATERIAL --	Q.A.		
FINISH --	COMMENTS: Mat: Alum. Qnty: 1 To be anodized		
DO NOT SCALE DRAWING			

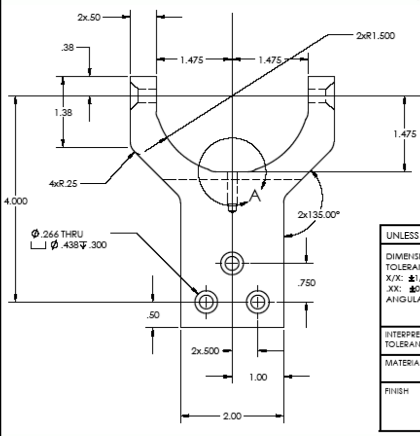
- From the previous drawing, we see useful information in the title block
  - made from aluminum, only one, to be anodized (relevant for threads)
  - dimensions in inches; trust numbers, not drawing scale
  - .XX values held to 0.010 inches
  - .XXX values held to 0.002 inches

Another example





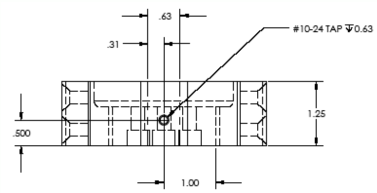
A closer look; #1

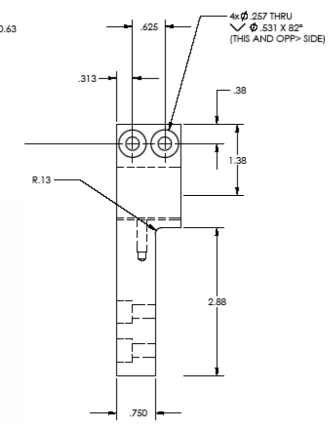


- New features:
  - radius spec.
  - angular spec.
  - counterbore spec.
  - detail (A) notation
- Note different tolerances, alloy specification

UNLESS OTHERWISE SPECIFIED:		DATE	NAME
DIMENSIONS ARE IN INCHES TOLERANCES: X/X: ±1/64 .X: ±0.05 .XX: ±0.010 .XXX: ±0.003 ANGULAR: MACH±1°	DRAWN	4/30/03	TWM
	CHECKED		LNC
	ENG APPR.		LNC
	MFG APPR.		
INTERPRET GEOMETRIC TOLERANCING PER:	Q.A.		
MATERIAL AL 6061-T651	COMMENTS:		
FINISH 63			

A closer look; #2



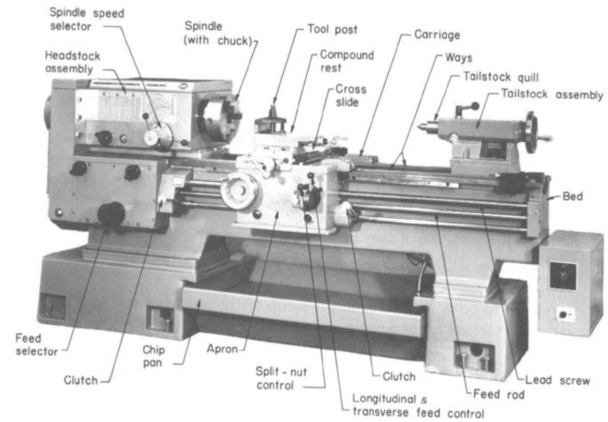


- New features:
  - depth spec. for tapped hole
  - countersink spec.
  - hidden lines (dashed)

Machining

- The primary tools in a machine shop:
  - lathe: for cylindrically symmetric parts
    - part rotates, tool on *x-y* stage
  - milling machine (or mill): rectangular parts, hole patterns
    - spindle rotates tool, part on *x-y-z* stage
  - drill press: for low precision or chasing pilot holes
    - like a mill, except no fine motion control, thus no side-cut capability (a matter of holding strength as well as motion)
  - bandsaw: for roughing out stock
    - circular band of a saw blade makes for a continuous "hack saw"
  - sandpaper, files, granite block
  - grinding wheel (make lathe tools, diamond pins)
  - measurement equipment

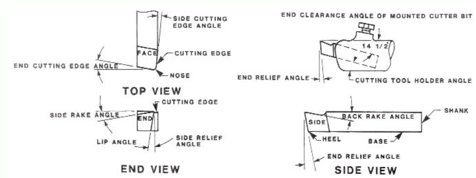
The Lathe



Lathe Capabilities

- Precision outer diameter
- Precision inner diameter
- Stepped and angled transitions
  - can drive tool at angle other than 90°
  - with numerical control, arbitrary profiles possible
- Facing off
  - flat, or even conical
- Threading (though complicated, advanced skill)
  - outer thread
  - inner thread
  - complete control over pitch, multi-thread, etc.
- Boring
  - usually with drill bit (possibly followed by reamer) in tail stock
  - but can use boring bar to make larger holes

Lathe Tools



MATERIAL	SIDE RELIEF	END RELIEF	BACK RAKE	SIDE RAKE
LOW CARBON STEEL	10 to 12	8	8 to 16	12 to 14
MEDIUM CARBON STEEL	10 to 12	8	12 to 16	12 to 22
HIGH CARBON STEEL	10	8	8	12
HIGH SPEED STEELS	10	8	8	12
ALLOY TOOL STEELS	10	8	8	12
CARBON TOOL STEELS	10	8	8	12
ALLOY STEELS	10	8	8 to 12	12 to 15
STAINLESS STEEL	10	8	8 to 10	14 to 20
ALUMINUM	12	8	35	15
BRASS	10	8	0	1 to 5
BRONZE	10	8	0	0 to -4
COPPER	14	12	16	20
GREY CAST IRON	10	8	5	12
NICKEL	15	13	8	12
MONEL	15	13	8	14
WOOD	20	20	30	30

(WHEN GRINDING TOOLS TO BE USED IN STANDARD ANGLED TOOL HOLDERS, REMEMBER THE 14 1/2 DEGREE BUILT-IN ANGLE AND GRIND ACCORDINGLY)

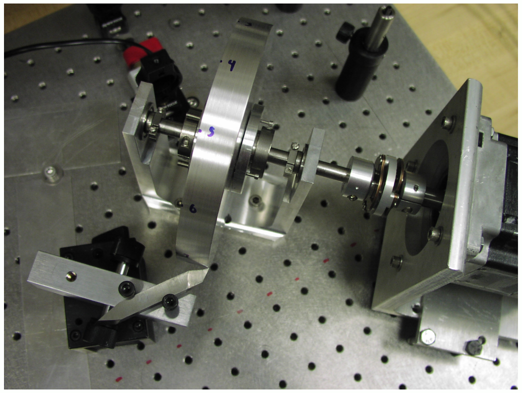
The canonical lathe tool: dimensions depend on material being worked

Lathe Tools, continued

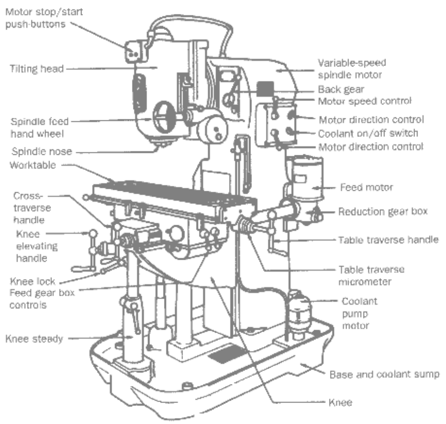
lathe tools are usually shaped by the machinist using a grinding wheel

a "boring bar" lets you get deep inside a part for making an inner diameter (for holes larger than available drill bits & reamers)

"A Rudimentary Lathe"



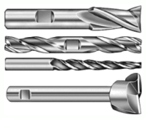
The Milling Machine



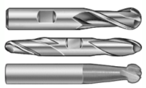
Milling Machine Capabilities

- Surfacing/Shaping
  - fly cutting; facing edges
- Pockets
  - tightness of corners depends on diameter of bit
- Slots
- Hole Patterns
  - with table encoders, easily get to 0.001 inch (25 microns)
- With numerical control, arbitrary shapes/cutouts
  - gets around etch-a-sketch problem: can draw circles, etc.!
- Simple and Complex Angles
- Boring (can use boring bar here, too)

Mill Bits

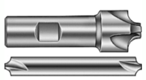


square end-mills are the workhorse bits:
- pockets
- slots
- edge trim
- facing



ball-end mills make rounded pockets or spherical pockets; also fillets

This device holds a lathe-like tool bit to become a fly-cutter, for surfacing large flat faces

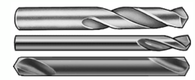


corner-rounders form rounded corners!

conical end-mill for chamfers

Drills and Reamers

standard “jobber” drill: will flex/walk, follow pilot



stub drill for less walk/greater rigidity

center drill establishes hole position with *no walk*

reamers (straight or spiral)
finish off hole (last several thousandths)
precise hole diameter
for insertion of dowel pins, bearings, etc.
plunge while spinning, extract still

countersink: for screw heads & deburring hole

Drilling Practices

- Drills come in fractional inches, metric, and a standard wire gauge index
  - wire gauge index is most common in U.S.: most finely graded
  - see http://www.carbidedepot.com/formulas-drillsize.htm
- Drills walk when pushed into unbroken surface
  - must use a punch to establish a conical defect for drill to find
  - or use a center drill (no walk) to get the hole started
  - stub drills better than jobber, but not as good as center drill
- Use pilot hole for larger holes
  - especially if precision important: use several steps so drills primarily working on walls

Taps and Dies: making threads

T-HANDLE

- Taps thread holes, after pre-drilling to the specified diameter
- Taper tap for most applications
- Plug tap for getting more thread in bottomed hole
  - preferably after taper already run
- Bottom tap for getting as many threads as possible in bottomed hole
  - preferably after plug already run
- Dies for outside thread: seldom used
  - *buy* your screws & threaded rod!!

Example Procedure

- Final part outer dimensions are 1.550×0.755×0.100
  - so find 1/8-inch aluminum stock and cut on bandsaw to something bigger than 1.625×0.8125 (1 5/8 by 13/16)
  - de-burr edges with file or belt sander
- Establish outer dimensions
  - get 0.755 dimension
    - put in mill table vice on parallels, part sticking about 0.1 inches above jaws

Procedure, cont.

- end-mill exposed (up-facing) face until all low spots gone, taking multiple passes at about 0.010 inches per pass
- de-burr and rotate 180° in jaw about horizontal axis
- end-mill new side (opposite first) until low spots gone
- de-burr and measure; figure out amount remaining to cut
- place back in vice, either finished side up
- bring up knee until end-mill just touches and set knee dial to zero
- make successive passes, bringing up knee until the prescribed amount has been removed
- measure to make sure

– get 1.55 dimension
- place in jaw with large face up, rough edge extending beyond jaw side
- use side of end-mill to shave edge; traveling in direction of cut (conventional cut)
- once low spots done, cut opposite direction for smooth finish (climb cut)
- de-burr, and rotate 180° about vertical axis, rough edge sticking out
- smooth out this surface, measure (maybe even in place), and do final trims to bring it into spec.; de-burr

Procedure, cont.

  - get 0.100 dimension
    - center in jaw, with guaranteed > 0.030 above jaw: machining into vice is very bad: NEVER let the tool touch the jaw!
    - use large-ish end-mill or even fly-cutter to take down surface by 0.010; take out and de-burr
    - flip part to remove other side (skin) by an additional 0.015, measuring before final cut (in place, if possible)
- Establish hole pattern
  - leaving in place, establish coordinate origin
    - use edge-finder to get edge positions, resetting encoders to zero at edge-finder jump
    - remember to account for 0.100 edge-finder radius (need to re-zero at 0.100 in appropriate direction)
  - center drill each hole position
    - use small center drill, in collet if possible (rather than chuck)
    - at each coordinate pair, run in center drill as far as you can without exceeding final hole size

Procedure, cont.

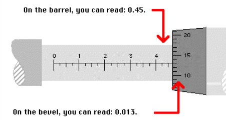
  - drill holes
    - use #30 drill on four holes
    - use #29 drill for 8-32 pre-tap
      - see http://www3.telus.net/public/aschoepp/tapdrill.html
  - take part out and de-burr holes (with countersink in hand)
- Cut two notches out
  - place part in vice so that the tab that will remain is completely free of vice jaws
    - use edge-finder to establish left-right origin
    - measure end-mill diameter carefully (maximum extent of teeth)
    - work out x-positions corresponding to full cut on both sides
    - bring up knee to touch material, set to zero
    - with end-mill off to side, bring up knee 0.400 inches (usu. 4.00 turns of crank)

Procedure, cont.

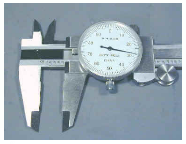
- begin swiping 0.020 at a time off of edges until you are 0.005 from designated stopping points
- move end-mill to side so that final travel will be against blade direction for best finish (climb cut)
- bring up knee by final 0.005
- go final 0.005 in x-direction for final cut
- make final cut, then walk away in x to finish bottom cut
- end-mills cannot be plunged unless material at center of end-mill is already cleared out: they aren't drills

- Tap 8-32 hole with taper tap
- Final de-burr, final measurement check
- Clean part, check fit to mating piece(s)

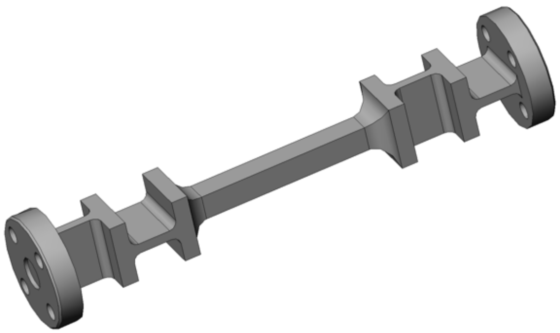
Measurement Tools



- General Purpose Caliper
- Micrometer
  - reading a micrometer:
    - http://feh.eng.ohio-state.edu/Tutorials/micrometer/reading.html
- Dial Indicator
- Depth Micrometers
- Cleaning is a very important part of measurement



Intro to SolidWorks



SolidWorks Overview

- SolidWorks is a totally fantastic design package that allows:
  - full 3-D "virtual" construction/machining
  - excellent visualization: rendering and rotation
  - feedback on when enough dimensions are established
  - parameters such as volume, mass, etc.
  - conversion from 3-D to 2-D machine drawings
  - assembly of individual parts into full assemblies
  - warnings on interferences between parts in assemblies
- Typical sequence:
  - 2-D sketch in some reference plane, with dimensions
  - extrude sketch into 3-D
  - sketches on surface, followed by extrude or cut, etc.

Our Exposure to SolidWorks

- Computers in lab have SolidWorks on them
- Pick a machining piece you want to model
  - or find/dream-up your own, but be careful to pick appropriate difficulty level
  - if it's your own creation, you must describe its purpose
- Measure relevant dimensions of piece to model
- Go through SW online tutorials until you have enough knowledge to make your 3-D model
- Make 3-D model, and turn this into 2-D machine drawing
  - with dimensions in "design" units and appropriate tolerances

Assignments

- Reading from Chapter 1:
  - (black = 3rd ed.; red = 4th)
  - sec. 1.1 except 1.1.8; sec. 1.1 except 1.1.8
  - sec. 1.2; secs 1.2, 1.3
  - secs, 1.3.4–1.3.8; secs 1.4.1–1.4.4, 1.4.8
  - sec. 1.4;sec 1.5
- SolidWorks Tutorial & part emulation, including:
  - 3-D part, matching measurements
  - 2-D drawing a machinist would enjoy
  - description of part function, if not a pre-made part
  - brief write-up including difficulties overcome, estimated mass (from SolidWorks model), and a brief description of how one would make the part—roughly at level of second indentation (dash) in lecture detail of the example part
  - see website for (definitive) lab instructions/details