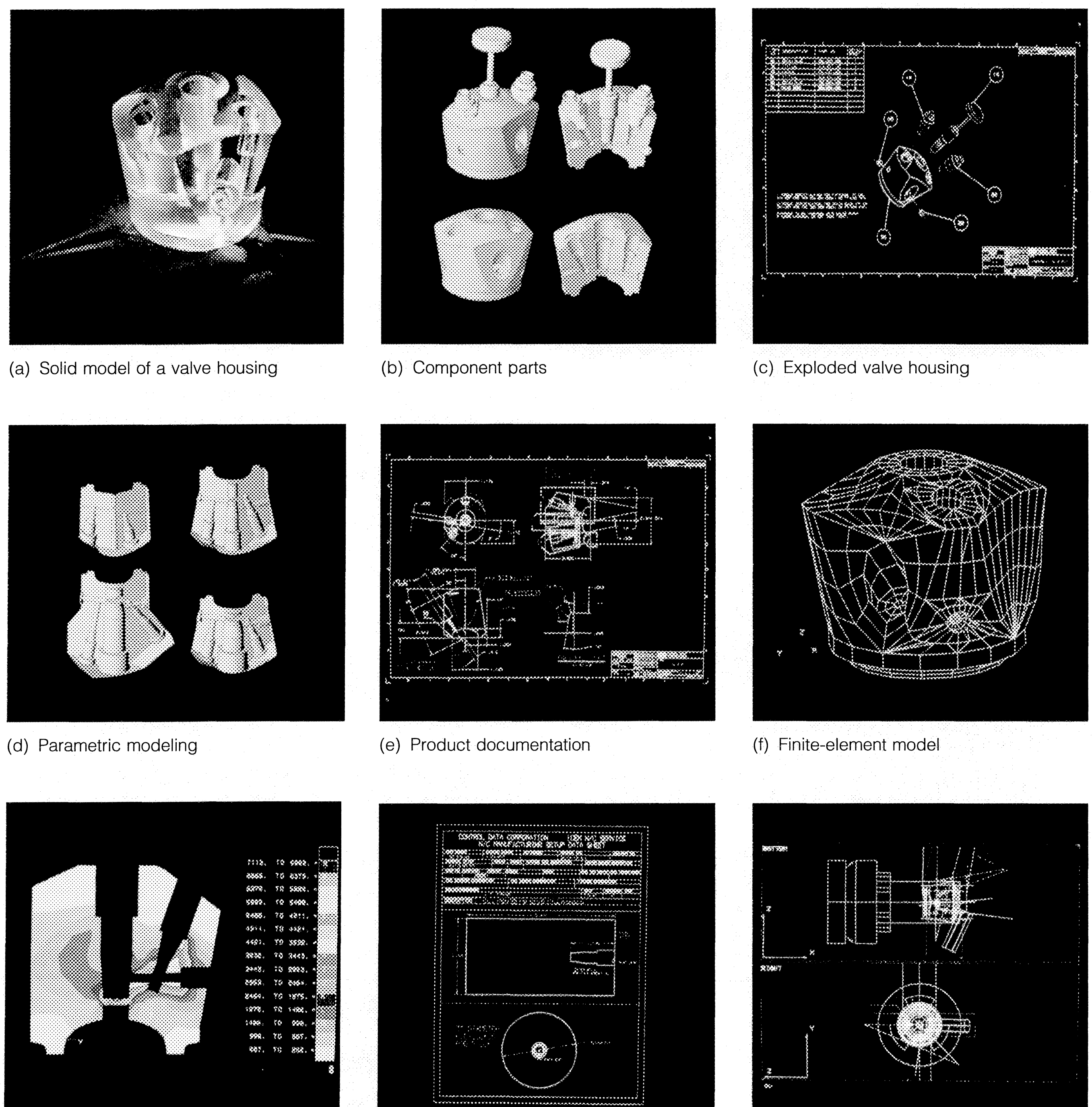
Chapetr 23 page 810

-Working Drawings-



(a) Solid model of a valve housing

(b) Component parts

(c) Exploded valve housing

(d) Parametric modeling

(e) Product documentation

(f) Finite-element model

(g) Color schemes for analysis

(h) NC generation

(i) Cutter paths before machining the part to obtain optimum cutter path design

FIGURE 23.1 Using CAD/CAM to Document and Manufacture a Part

- Inspection and control of product quality and reliability
- Assembly
- Testing and modeling
- Packaging, boxing, and shipping
- Determining cost
- Cataloging
- Installation and service
- Conducting final acceptance test
- Making alterations
- Recording for duplication, repair, or replacement

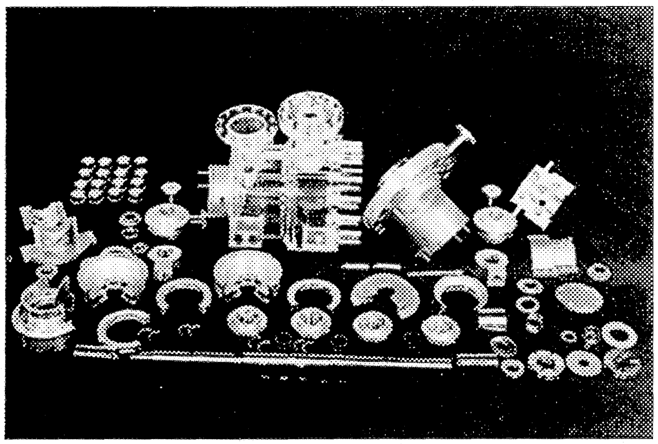


FIGURE 23.2 Disassembled Boiler Feed Pump

23.2 ASSEMBLY DESIGN CONSIDERATIONS

The way the product will be assembled must be considered early in the design phase. Perhaps as much as 80% of the cost to produce a part is set during the design phase. This is one of the many reasons why the design of the product and the design of the manufacturing process must occur simultaneously. **Design for manufacturability** (**DFM**) and **simultaneous** or **concurrent engineering** (**CE**) are keys to successful designs. The optimal method of assembly can be chosen early in the design process if the product and the production-assembly process are designed at the same time.

There are five basic types of assembly:

Manual assembly Performed manually, with the assistance of hand tools only.

Semiautomatic assembly A combination of manual and automated processes. The operator loads the product manually, the machine then performs one or more assembly operations, and the operator then unloads the product.

Adaptive assembly Programming the system to adapt itself automatically to certain variations based on sensors.

Automatic assembly All operations are performed automatically without human intervention or decisions; places constraints on the design for the assembly of the product as well as the orientation, presentation, and gripping of the parts.

Flexible assembly Uses flexible manufacturing equipment that builds families of related products or subassemblies on the same setup or with quick, automated setup changes built into the process.

23.3 CATEGORIES OF DRAWINGS

Engineering drawings can be classified into three broad categories:

- Layout and design study drawings
- Engineering and production detail drawings
- Assembly drawings

23.3.1 Layout and Design Study Drawings

Layout and design drawings depict proportions, dimensions, materials, and the relationship of parts in new or modified designs. Usually, they precede production drawings, are drawn to scale, and are useful in preparing and checking part drawings and assemblies.

Layout drawings, often effective in establishing patent claims, are completed by engineers and designers in conjunction with production personnel. When practical, layouts should be prepared in a form suitable for use in manufacturing, especially in situations where prototype or developmental models are required (Fig. 23.3).

23.3.2 Engineering and Production Detail Drawings

Detail drawings provide a complete engineering definition of the finished system, assembly, or part (Fig. 23.4). This includes design data references, laboratory instructions, and engineering specifications. These drawings are prepared for the shop floor and are used in the production of the desired product.

Individual parts of an assembly are machined separately during the manufacturing stage of the project. Parts are also called **workpieces**. A typical workpiece is detailed so the fixture designer can produce tools for holding and locating it during the machining operations.

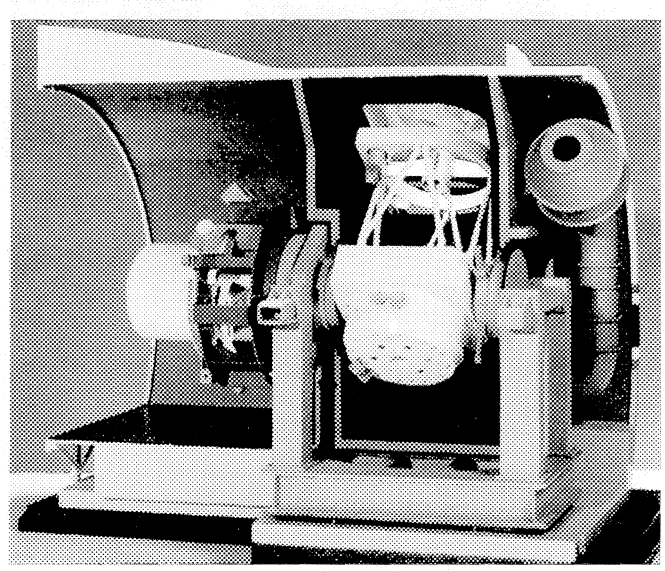


FIGURE 23.3 Model of an Infrared Telescope

Engineering production drawings are not created to accommodate a particular method of manufacture. They are meant to be used without additional explanation. If manufacturing or processing instructions are provided, this information is for reference only, unless such data is vital to the end definition and engineering control of the product.

23.3.3 Assembly Drawings

An **assembly drawing** defines the complete end-item requirements, and establishes item identification for the assembled configuration of two or more pieces, parts, subordinate assemblies, or any combination that are joined together to form an assembly (Fig. 23.5). Assembly drawings always include a **parts list** or a **bill of materials.** The sleeve valve in Figure 23.6 required an assembly drawing to show the proper relationship of the parts. During the design phase, a design layout was employed to establish and control the relationship between parts with regard to tolerancing and fits.

23.4 ASSEMBLY DRAWINGS AND THE DESIGN PROCESS

In most cases, the assembly is conceived as a whole (layout assembly) and broken into individual pieces (detail drawings) for manufacture and later assembly. This is called the **top-down** approach to design.

Assemblies are seldom designed from the bottom up, unless the individual parts are standard items that are to be assembled into a unit. **Bottom-up design** means that each component of a unit is designed separately and then put together in an assembly.

The sleeve valve in Figure 23.6 comes in four pieces. The design is *limited to the minimum number of parts*. Each piece fits inside the other. Since each piece was toleranced in relationship to the housing, the product was *designed as a unit*. The housing and end cap hold the other parts together. The bolts and nuts needed for assembly are not shown in the figure.

After the design and layout of a project are complete, a detail drafter pulls separate parts from the assembly and draws them on individual sheets. This process is called **detailing**. Details include appropriate views and dimensions.

Standard items that can be purchased off the shelf are shown on the assembly drawing and listed in the parts list. They do not require a separate detail, unless they are modified in some way. Standard parts include bolts, screws, nuts, retaining rings, dowels, pins, springs, gears, bearings, clamps, and purchased subassemblies.

As a product is designed and detailed, the manufacturing department begins to determine the tooling requirements. **Tool designers** create appropriate fixtures to hold the individual parts during manufacture. Machining involves the use of fixtures to locate, hold, and position the part for accurate, economical, and efficient production. Jigs and fixtures are also assemblies and are designed, laid out, and detailed in a manner similar to product assemblies.

23.4.1 Assembly Drawing Types

An **assembly drawing** may contain the detailed design requirements for one or more parts required in the assembly. It is prepared for each group of items that are to be joined together to form an assembly and reflects one of the following:

- A logical level in the assembly or disassembly sequence
- A functional unit
- A stocked, standard, off-the-shelf purchased item

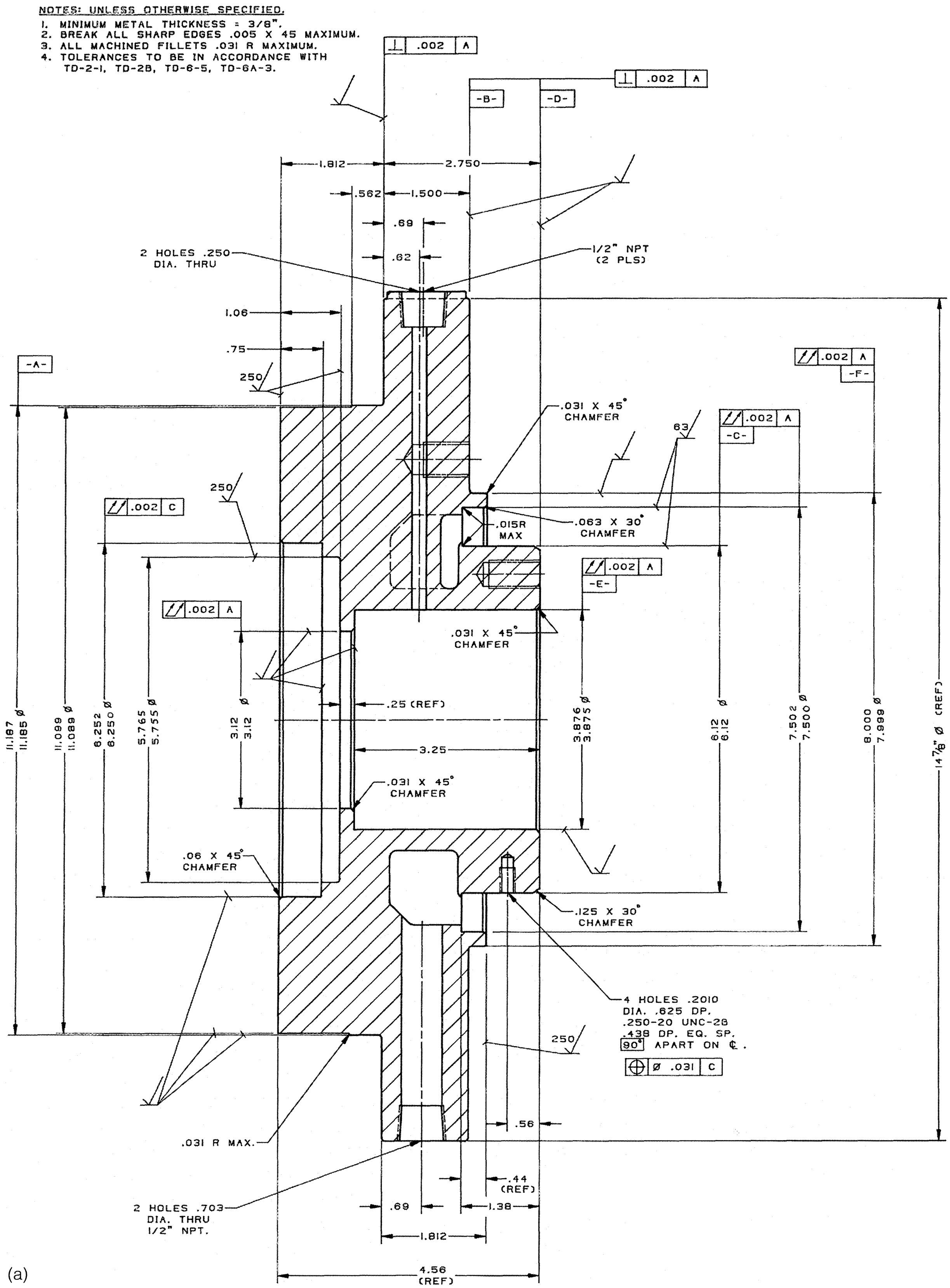
The assembly may be shown on the same drawing sheet on which the details appear or on a separate sheet. If the assembly is on a separate sheet, this will be sheet number 1, and the details will be shown on sheet number 2, etc. All sheets bear the same drawing number.

An assembly drawing may define either a **separable** or an **inseparable** assembly. **Welded assemblies** are inseparable assemblies. Figure 23.7 is an example of a welded assembly. The parts list is a **material stock list** and simply gives the stock steel sizes of each piece. On most weldments, the assembly is considered a detail of a part and each piece is dimensioned on the drawing. The detail of the weldment in Figure 23.8 is such a case. Here, even though the weldment is three separate pieces of aluminum joined together by welding, it is not considered an assembly. The drawing is simply a detail of the weldment. Each piece is not separately called out, ballooned, or listed in a material stock list. Instead, it is dimensioned on the sheet. (See Chapter 22 for more information concerning weldments and welded assemblies.)

23.4.2 Mechanical Assemblies and Drawing Requirements

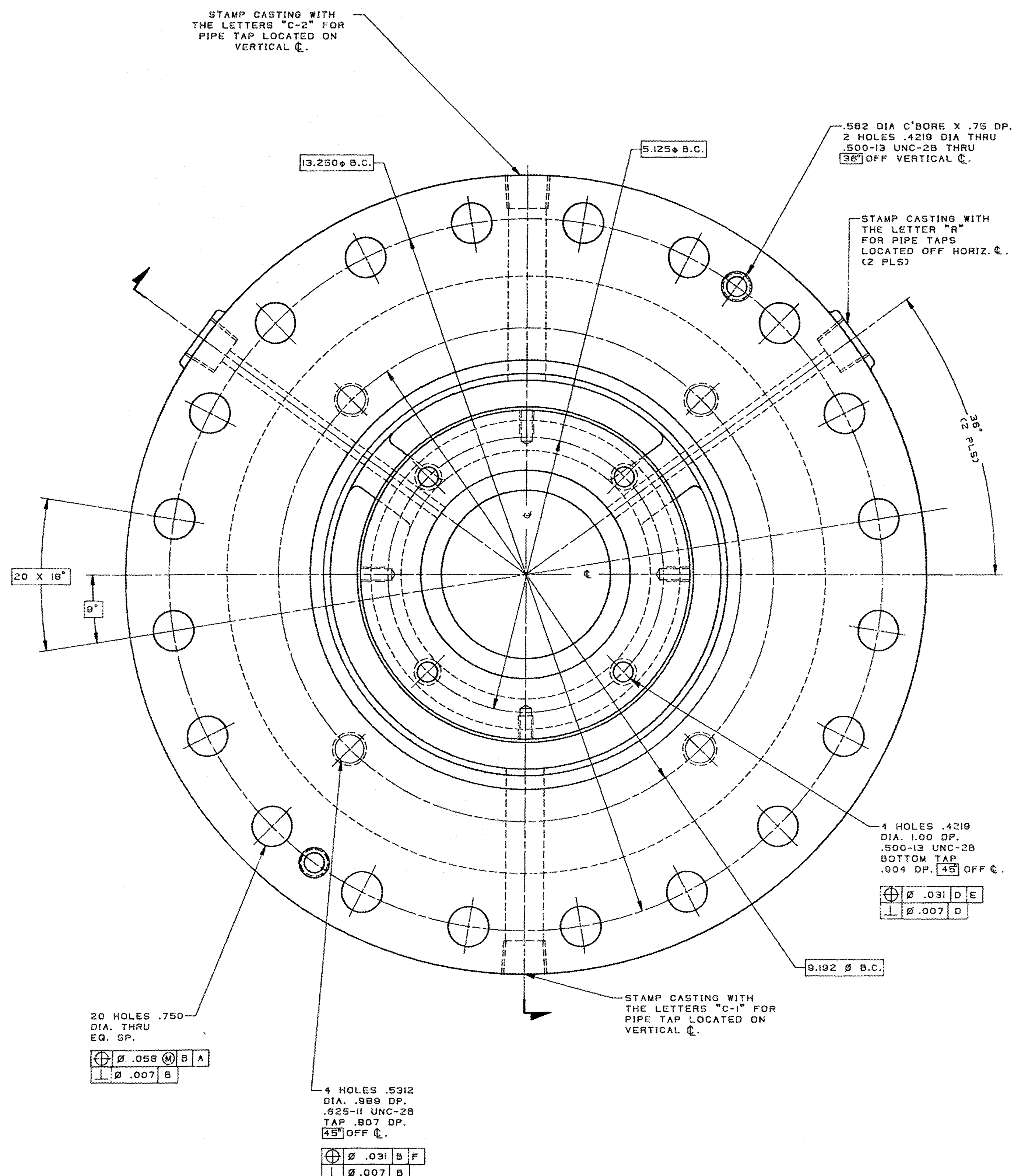
A mechanical assembly may have parts made from sand, permanent mold, or die castings (Fig. 23.9); rolled, extruded, or pressed-shape forgings; plates; bars; sheet metal; or a combination of any two or more of these. These parts may be assembled into a complete unit by welding, brazing, soldering, riveting, bolting, or other fastening methods. After assembly, additional work, such as a machining, may be necessary to complete the item.

The function of an assembly drawing is to provide a complete specification for joining together, in proper relationship, two or more detail parts or subassemblies to form an assembly. This type of drawing usually includes a graphic layout of component parts, necessary notes, and, if a separate list or bill of material is not involved, a tabulated list of parts. It should show the spatial relation of each part or



(a)

FIGURE 23.4 Cover Detail


STAMP CASTING WITH THE LETTERS "C-2" FOR PIPE TAP LOCATED ON VERTICAL ℄.
.562 DIA C'BORE X .75 DP. 2 HOLES .4219 DIA THRU .500-13 UNC-2B THRU 36° OFF VERTICAL ℄.
13.250⌀ B.C.
5.125⌀ B.C.
STAMP CASTING WITH THE LETTER "R" FOR PIPE TAPS LOCATED OFF HORIZ. ℄. (2 PLS)
36° (2 PLS)
20 X 18°
9°
℄
4 HOLES .4219 DIA. 1.00 DP. .500-13 UNC-2B BOTTOM TAP .904 DP. 45° OFF ℄.
⌀ .031 D E
⌀ .007 D
9.192 ⌀ B.C.
STAMP CASTING WITH THE LETTERS "C-1" FOR PIPE TAP LOCATED ON VERTICAL ℄.
20 HOLES .750 DIA. THRU EQ. SP.
⌀ .058 Ⓜ B A
⌀ .007 B
4 HOLES .5312 DIA. .989 DP. .625-11 UNC-2B TAP .807 DP. 45° OFF ℄.
⌀ .031 B F
⌀ .007 B

(b)

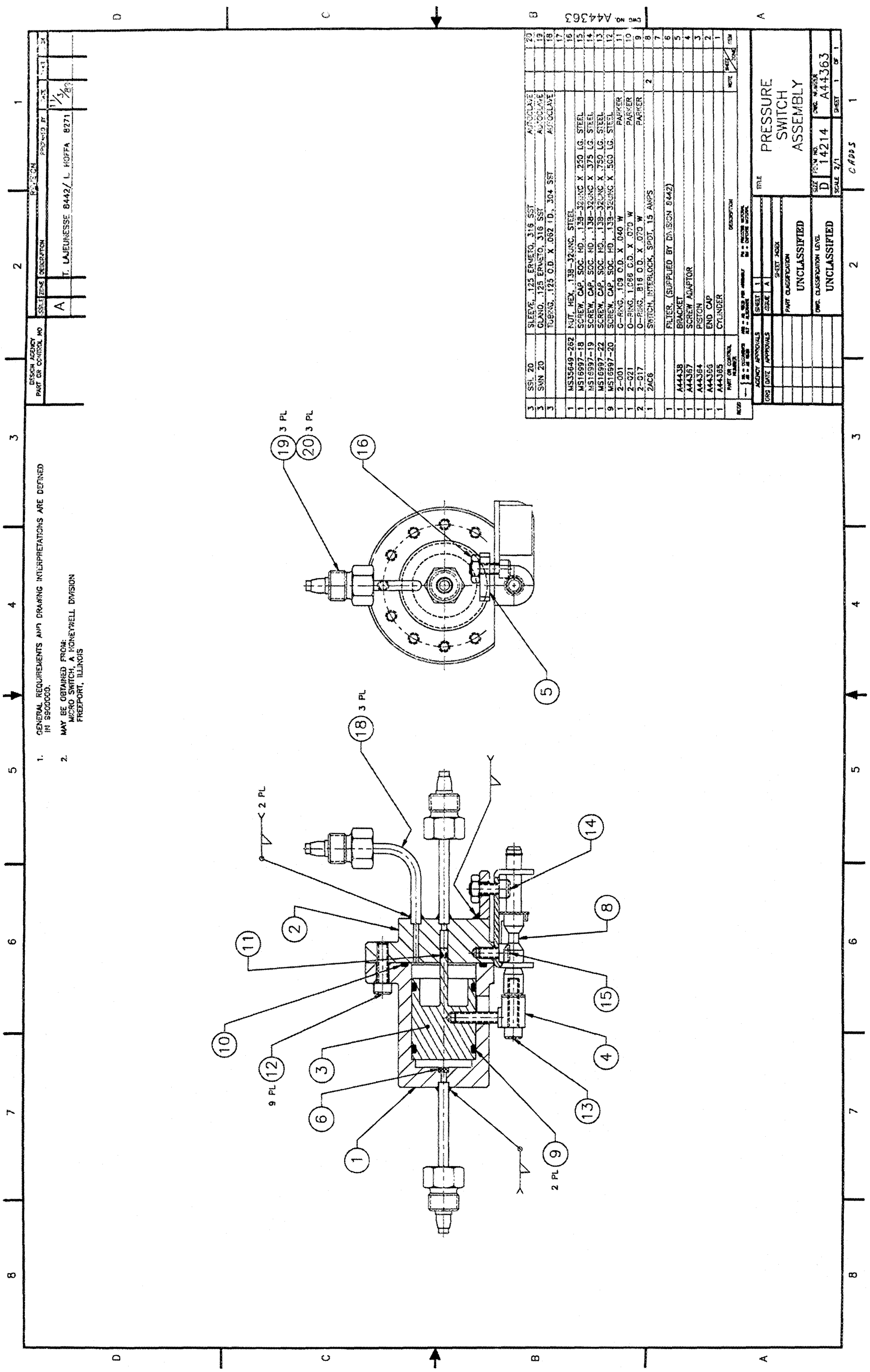


FIGURE 23.5 Assembly Drawing

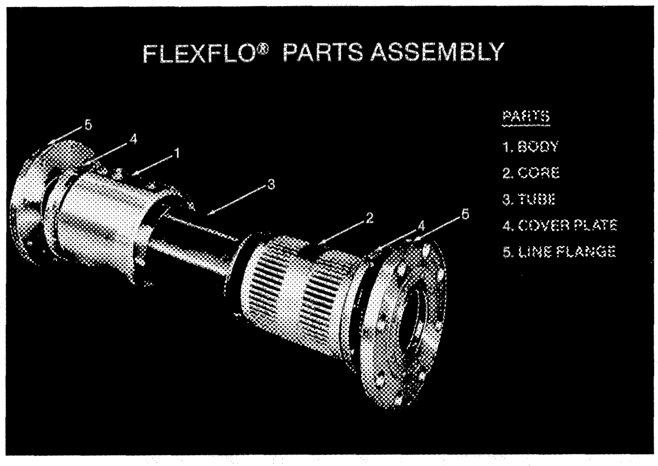


FIGURE 23.6 Disassembled Grove Flexflo Flexible Rubber Sleeve Valve

subassembly, the method of fastening, and the type of fasteners.

When necessary, the assembly drawing indicates subsequent operations to form the completed item. For example, heat treatment, machining dimensions, and finishes are specified here. If a mechanical assembly is made entirely from cut shapes with sufficient information for cutting each piece, the information to fasten them together and finish them might be given on a single drawing.

23.4.3 Detail Drawings

Detail drawings are fully dimensioned, accurately laid-out engineering drawings of individual parts (Fig. 23.10). All information needed to manufacture and produce the part is included. Adequate view description, correct dimensioning and tolerancing, accurate notes, and material designation are shown on the detail drawing. Components may appear on the assembly drawings or on separate details, or be established by written description.

23.4.4 Assembly Drawing Considerations

Assembly drawings for production parts may also be created from the detail drawings *after they have been approved by the checker*. This procedure gives a final check of the detail drawings for space clearances, limits, and satisfactory function in assembly.

Assembly drawings for jigs and fixtures are produced from the designer's sketches that are accurate layout drawings. The final assembly is broken into individual components that are detailed separately. The checker checks the assembly and the details in the final stage.

Product design, production volume, and facilities are the determining factors influencing the need for an assembly drawing. The quality of the finished product depends on the effective attachment methods, regardless of the quality of the individual parts.

Welded, soldered, or brazed parts that have characteristics requiring the parts to have a particular relation to one another and parts that are permanently assembled are shown in their assembled positions. Necessary dimensions and specifications are included for size control and other conditions. Parts that are pressed and line-reamed in place, parts that are secured together with pins, bushings, and similar assemblies, and parts that are machined after assembly require an assembly to show these operations and specifications for assembly control. Parts for which a surface finish must be applied after assembly may require an assembly drawing to ensure proper overall finish. The cam boxes in Figure 23.11 are examples of assembled devices.

The subassembly in Figure 23.12(a) shows the relationship and orientation of the components of a complicated mechanical assembly. This example illustrates the use of CAD in the design of a typical mechanical assembly. Assemblies, components, drawings, and animated working models of the project are generated by the engineer and designer directly on the system before any drawings are created. A solid model of a component in the assembly is shown in Figure 23.12(b). A model, the wireframe assembly, and a multiview drawing of the component document the part [Fig. 23.12(c)]. A solid model of the mechanical assembly animates the working conditions of the assembly [Fig. 23.12(d)].

23.4.5 View Selection and Dimensions on Assembly Drawings

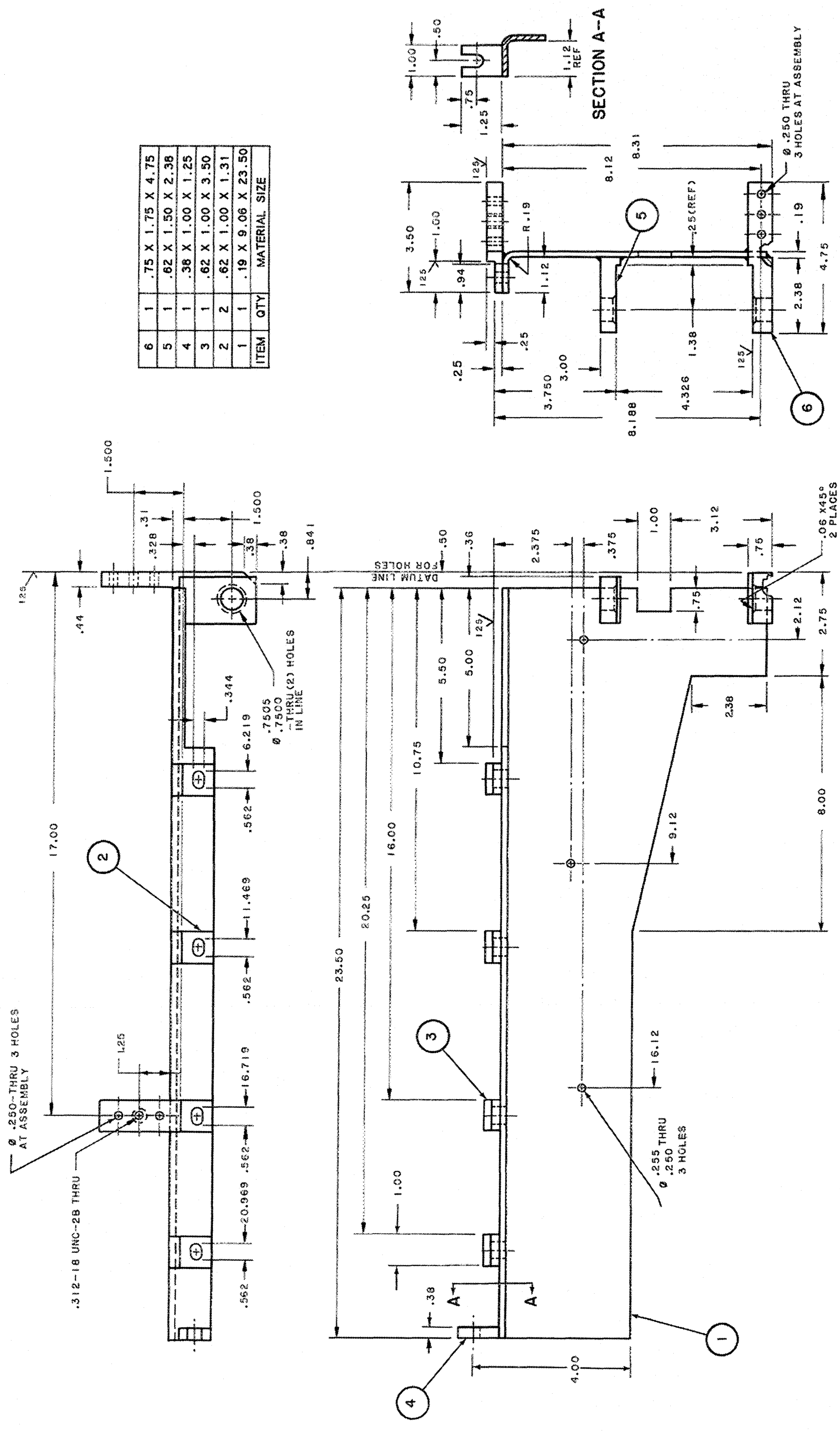
Usually, views are chosen to depict the assembly in its natural position in space, to define clearly how the parts fit together, and to describe the functional relationship of the parts. The minimum number of views needed to define the assembly should be given. Often, only one view is required.

Dimensions on assembly drawings are confined to setup dimensions, dimensions needed for assembly, dimensions required for machining after assembly, and clearance dimensions. Overall dimensions (height, width, and depth) may be included on the assembly for packaging assistance. When necessary, open and closed positions of movable parts on the assembly are given.

23.4.6 Hidden Lines, Crosshatching, and Phantom Lines on Assembly Drawings

Hidden lines should not be shown on assembly drawings, especially if they would confuse the reader. Instead, section views can show the relationship of internal parts (see later Figs. 23.30 and 23.31). Conventional section lining may be used. Material symbols are optional on assembly drawing sections. The section lines are drawn at angles to the object outlines and should be at a different angle for each adjacent part.

If there is some doubt, the note **FRONT**, to indicate the forward operating position, should be added to the detail



ITEM	QTY	MATERIAL SIZE
6	1	.75 X 1.75 X 4.75
5	1	.62 X 1.50 X 2.38
4	1	.38 X 1.00 X 1.25
3	1	.62 X 1.00 X 3.50
2	2	.62 X 1.00 X 1.31
1	1	.19 X 9.06 X 23.50

FIGURE 23.7 Welded Assembly

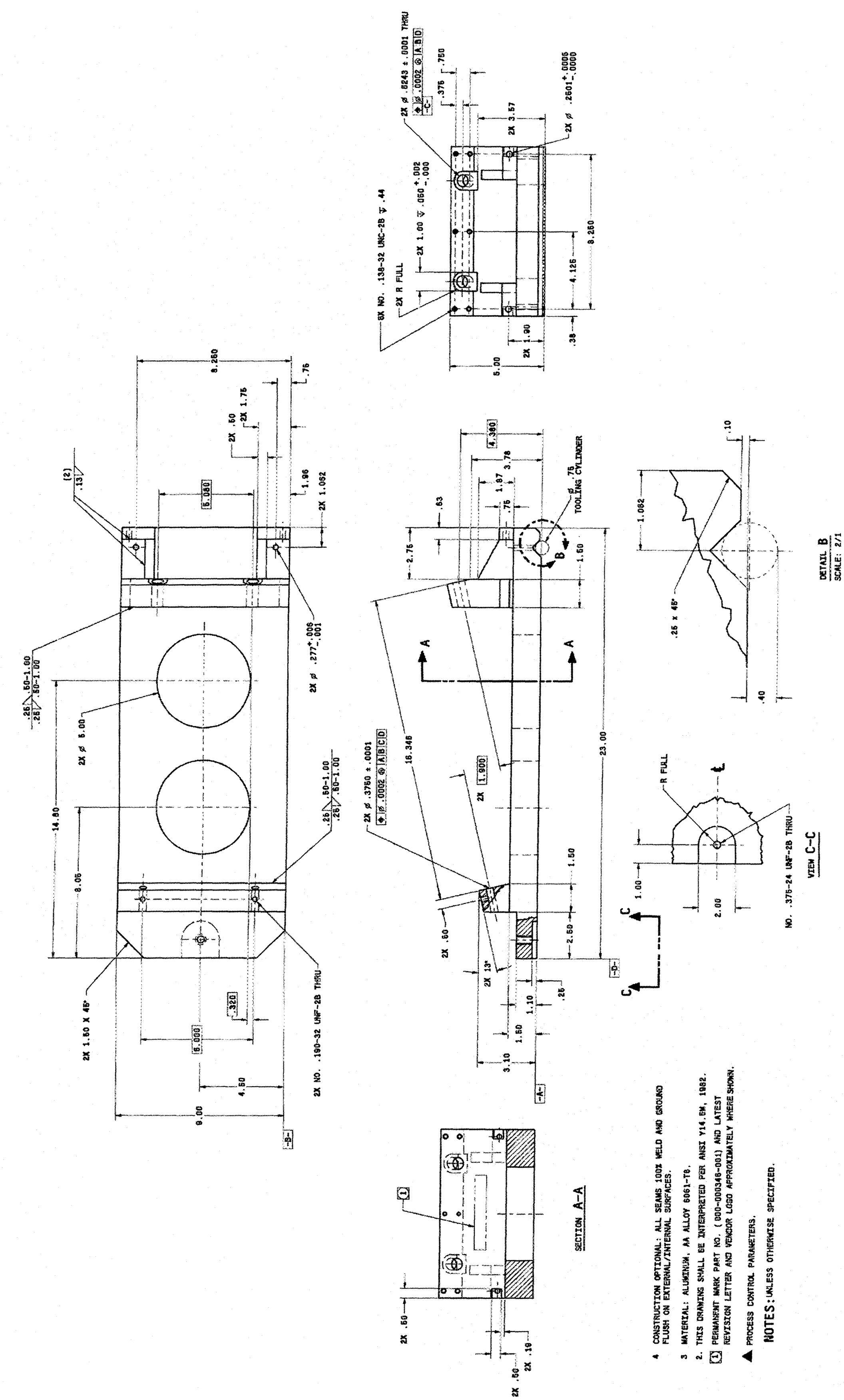


FIGURE 23.8 Welded Assembly Detail

Focus On . . .

AMERICA'S FIRST AUTOMOBILE

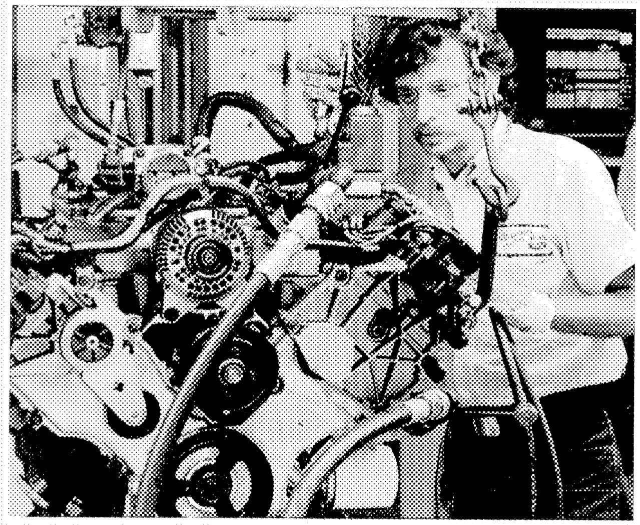
Who produced America's first automobile? One might be tempted to name Henry Ford, or maybe even Thomas Edison because of his electric car. However, the first car manufactured in the United States was the creation of Charles Duryea in Springfield, Massachusetts, in 1895.

Duryea was born on a farm in 1861 at a time when people relied on mechanical devices to accomplish their farm work. At seventeen, he began to cultivate his mechanical aptitude by assembling discarded farm parts into bicycles. Later on, he sold bicycles built from parts manufactured to his specifications.

He first saw a gasoline engine while he was displaying bicycles at the Ohio State Fair. The engine was much larger than could possibly be used in an automobile, but he knew that smaller engines were possible. He also knew that a German, Karl Benz, had recently patented the first automobile. He decided to build and patent the first American automobile.

After many years of thinking about his horseless carriage, Duryea and his brother finally built their first car in 1892. It had a gasoline-powered internal combustion engine with an electric ignition. The engine was attached to a converted horse buggy.

In 1893, the Duryeas produced a prototype called the *buggaut* and established themselves as the makers of the first successful American automobile. By 1895, they offered an improved 700-pound version for $2000.



Modern internal combustion engine.

Charles Duryea continued work on his dream machine and obtained nineteen patents, one of which was the first automobile patent issued to an American manufacturer. In 1896, the Duryea Motor Wagon Company produced thirteen cars of the same design. They were the first manufacturers to produce many copies of a single design. The car won several races against domestic and foreign automobiles.

drawing of a part to indicate the position of the part in the assembly. It is also acceptable to show a part before an assembly process operation with phantom lines and after the operation with object lines.

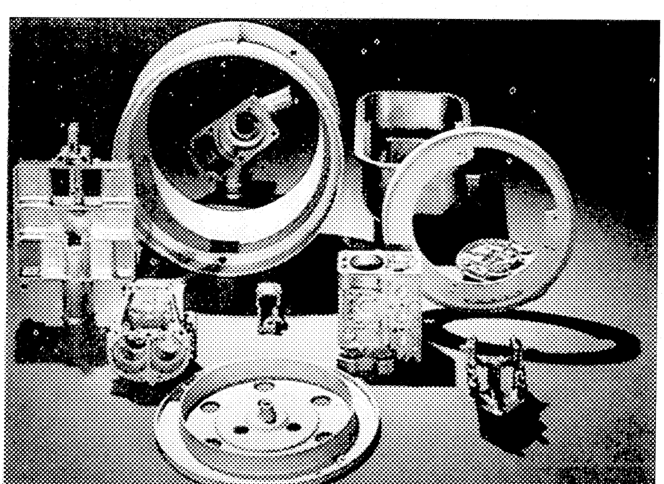


FIGURE 23.9 Die Cast Parts

23.5 ASSEMBLY DRAWINGS AND PARTS LISTS

An **assembly** is a combination of two or more parts joined together in one working unit. A **subassembly** is an assembly of parts that aid in producing a larger assembly. The purpose of an assembly drawing is to show the spatial relation of each part to the others and to identify all parts in the assembly by a number for each unique part. Assembly drawings include a list of all parts of the assembly, called a **parts list**. The parts list must be keyed to the drawing so that individual parts are clearly identified on the assembly. This is done by *ballooning* the drawing (see Fig. 23.5 and later Figs. 23.30 and 23.31).

Assembly drawings consist of two parts: an assembly delineation drawing, and a parts list, either integral or separate. The separate parts list provides the greatest overall economy and flexibility. A parts list is included with each assembly drawing to furnish information needed to order standard parts and to manufacture stock materials for non-standard parts.

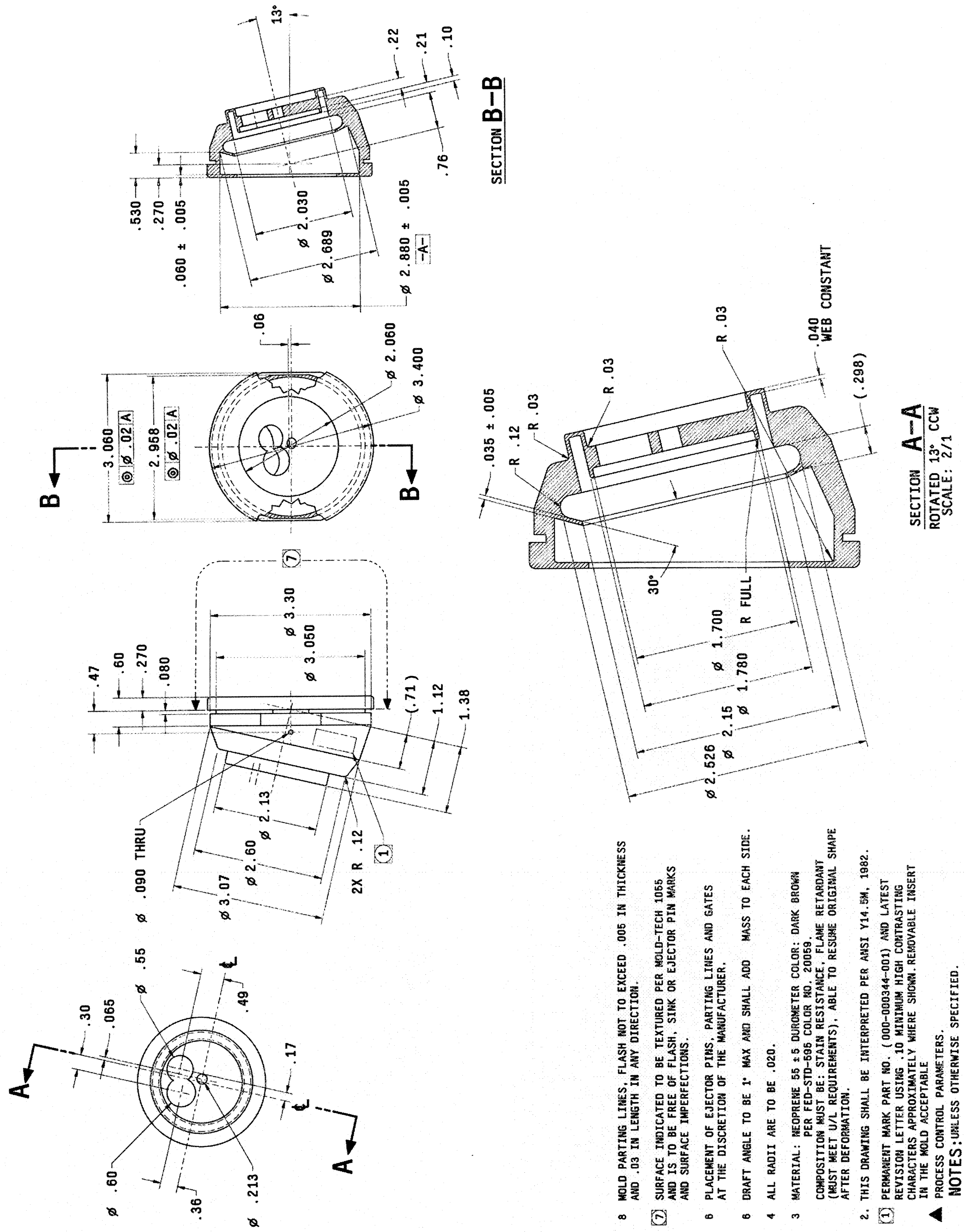


FIGURE 23.10 Acoustic Microphone Cup

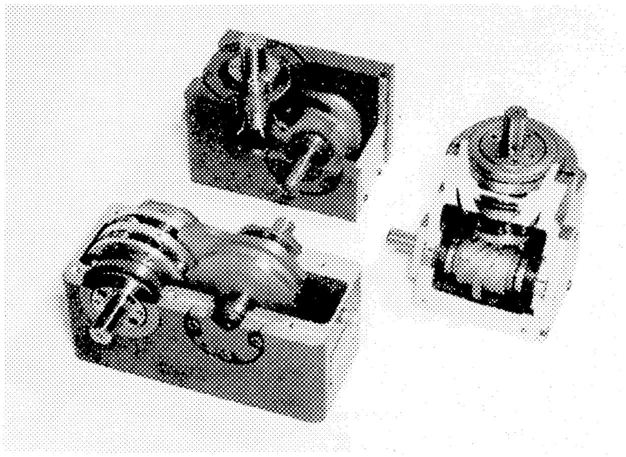
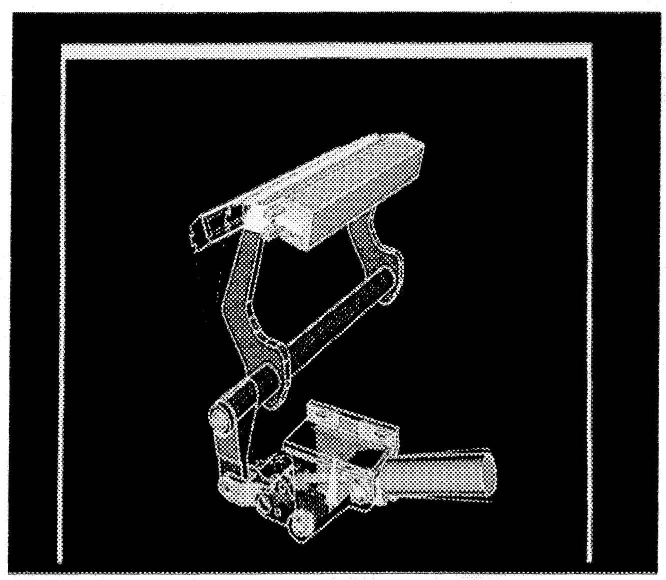


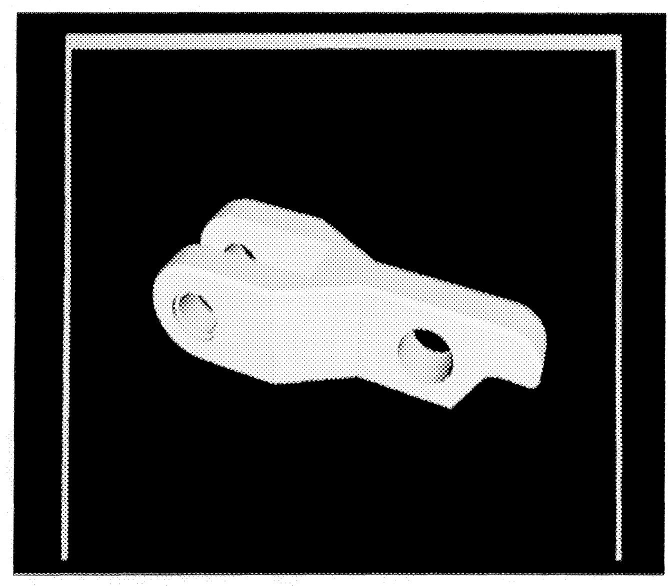
FIGURE 23.11 Cam Boxes

23.5.1 Parts Identification on an Assembly

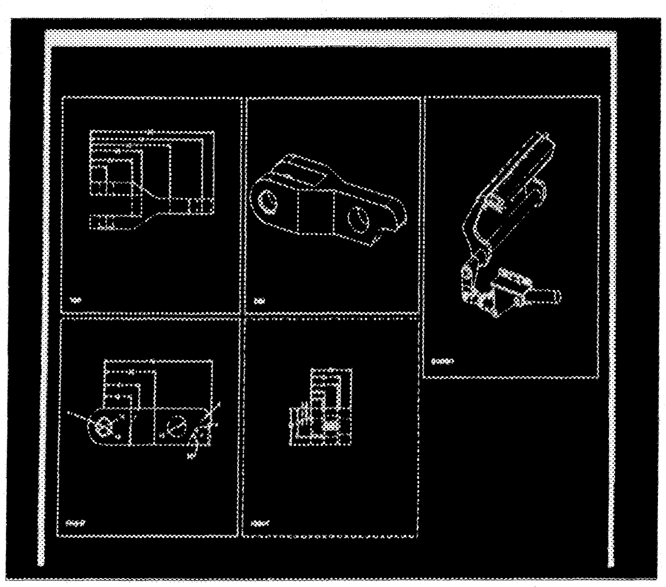
Ballooning is the process of identifying each part in the assembly. Each drawn part has a circle with a number inside it and a leader extending from the balloon to the piece and ending with an arrowhead. Balloons either are placed in a line along the middle of the drawing (horizontally or vertically) or are scattered throughout the drawing. The choice of method depends on the complexity of the assembly. Leader lines from balloons should not cross and can be straight lines or curved. Balloon circles are drawn anywhere from .5 to .75 inches in diameter (12 mm to 20 mm) with a template on manually drawn assemblies or with a command such as **CIRCLE** on a CAD system.



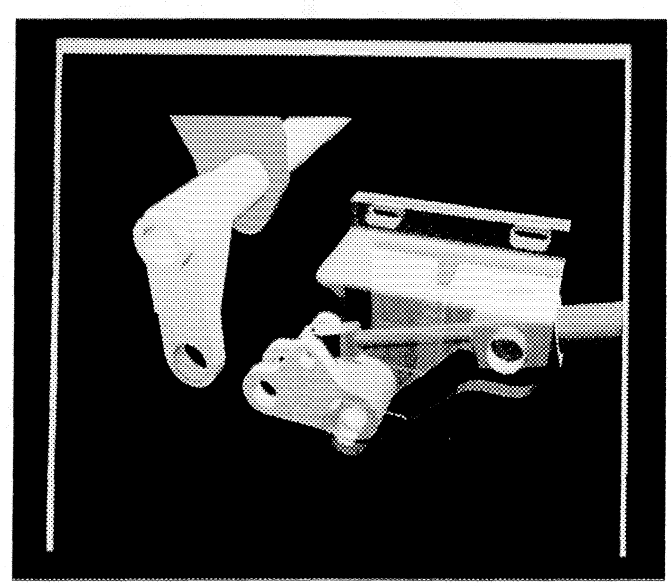
(a) Mechanical assembly



(b) Solid model of component



(d) Assembly animation



(c) Component documentation

FIGURE 23.12 CAD and Design

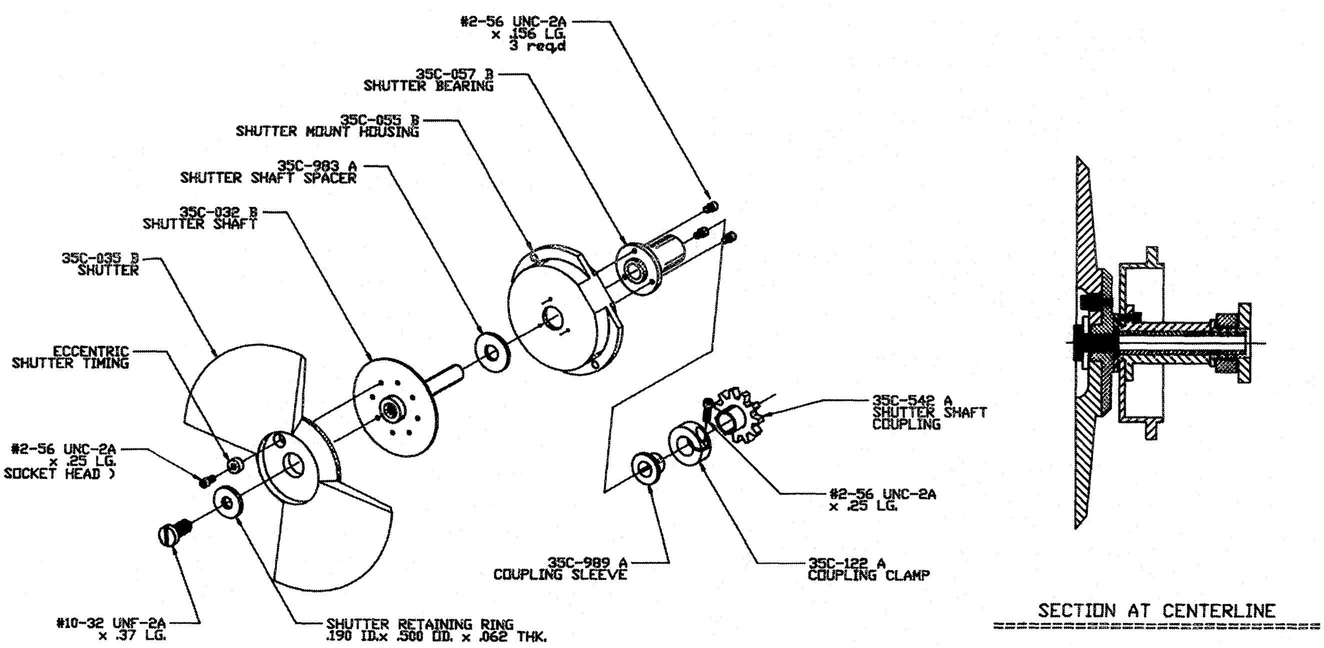


FIGURE 23.13 UltraCam Shutter Assembly

Many CAD systems allow you to enter a command that will insert a balloon with the appropriate text within the circle. The **BALLOON** command is picked, the text is entered via keyboard, the part edge is selected, and the balloon is positioned on the drawing. The system draws the balloon, the leader, the arrowhead, and the text. Some systems have automatic sequence number capability.

In some cases, the parts of an assembly are called out on the drawing without the aid of numbered balloons. In Figure 23.13, the shutter assembly has been displayed pictorially. Each of the parts is described in notes that have leaders pointing to the part.

23.5.2 Parts Lists

The assembly drawing must have a complete **parts list**. Each parts list includes individual part numbers, the name and a description of each part, and the material and quantity of all items required for one complete assembly (Fig. 23.14).

On tool and die drawings, the parts list is sometimes called a **stock list**. Here, material allowance is added for purchase information. Die drawings also have a stock list on the assembly drawing sheet.

The assembly parts list is placed above the title block when on the same sheet as the drawing. The part numbers are arranged to read upward so that new parts can be added if needed. The precise method of listing parts varies among companies. The parts or stock list sometimes uses the vertical line divisions of the revision record (block at top right of the drawing form). Spacing of horizontal divisions is uniform in both revision blocks and the parts list.

Parts List Heading Arrangement When an integral parts list is included on a drawing sheet, the heading **PARTS LIST** is placed on the bottom of the list and the part numbers read upward. Figure 23.15 is an example of a company title block with a parts list. The quantity, item number (balloon number), part number, and description are shown here.

When the parts list is separate from the part drawing, the heading **PARTS LIST** is at the top and the list is constructed from the top down. The following describes the four basic columns on a parts list (Fig. 23.16):

QUANTITY REQUIRED The number entered in this column denotes the quantity, volume, length, or other unit of measure required to complete one of the items to which the column applies. When this number applies to something other than quantity, the unit of measure is entered in this column or in an optional unit of measure column.

FSCM The Federal Supply Code for Manufacturer's number assigned to the originating design activity whose part or identifying number appears in column 3 is shown in this column. Many company title blocks exclude this entry.

PART OR IDENTIFYING NUMBER The identifying number for each item on the parts list is shown in this column.

NOMENCLATURE OR DESCRIPTION The assigned noun or name of the item whose identifying number is in the part number column appears in this column.

ITEM	QTY	PART NO.	DESCRIPTION
55	1	664-359239	CABLE, ASSY - MONOCHRONOMETER, YEL
54	1	664-359238	CABLE, ASSY - MONOCHRONOMETER, GRN
53	1	664-359237	CABLE, ASSY - MONOCHRONOMETER, RED
52			
51			
50	1	693-349584	BOARD, P.W. - U.V. SCANNER
49	1	301-961283	SPRING, CPRSN-.063 OD X 1 LG CRES
48			
47			
46	12	165-359506	WASHER, FLT .127 TFL-
45			
44	1	125-362041	PIN, DOWEL -.1553 DIA X .450L CRES
43	4	125-811591	PIN, DOWEL -.1251 DIA X .375L CRES
42	4	125-824305	PIN, DOWEL -.0626 DIA X .312L CRES
41			
40			
39	3	130-961281	RING, RETAINING - EXT, .073 ID
38	2	105-828447	NUT, HEX 2-56 S-BK
37			
36			
35	6	101-827620	SCREW, CAP 4-40 X 1" S-HXSO
34	2	101-961013	SCREW, CAP 2-56 X 1-3/8 S-HXSO
33	2	101-803947	SCREW, CAP 2-56 X .875 S-HXSO
32	1	101-961282	SCREW, CAP 2-56 X .750 S-HXSO
31	2	101-961201	SCREW, MACH 2-56 X 1" FL-S-SL
30	2	101-828409	SCREW, MACH 2-56 X .375 P-S-BK-SL
29	3	101-828408	SCREW, MACH 2-56 X .25 P-S-BK-SL
28			
27			
26	2	201-361974	GEAR, (MODIFIED) 42 TEETH
25	1	201-359899	GEAR, (MODIFIED) 84 TEETH
24	3	201-349033	GEAR, (MODIFIED) 132 TEETH
23			
22			
21	1	150-359263	BUSHING, MIRROR HOUSING
20	6	145-863274	BEARING
19			
18			
17	1	201-356344	RACK, GEAR U.V. MODIFICATION
16	1	520-348987	GRATING
15	1	333-349032	PLATFORM, GRATING
14	1	223-356345	SHAFT, CROSSOVER U.V. MODIFICATION
13	4	223-349019	SHAFT, GEAR
12			
11	1	548-361969	ASSY. FILTER
10			
9	1	499-348983	SPUD
8	1	105-348982	NUT, RETAINER
7	1	178-348984	EXTENSION TUBE #1, U.V. SCANNER
6	1	178-349027	EXTENSION TUBE #2, U.V. SCANNER
5	1	178-347010	EXTENSION TUBE #3, U.V. SCANNER
4	1	299-348981	HOUSING, MIRROR
3			
2	1	299-349023	DIRECTION CHAMBER U.V. LIGHT (LEFT)
1	1	299-348989	DIRECTION CHAMBER U.V. LIGHT (RIGHT)

E SIZE 223-356367 DWG. NO.

	BY	DATE
DR	D.M.DUARTE	1/18/90
CHK		
DSGN		
ENGR		

BECKMAN — BECKMAN INSTRUMENTS, INC.

TITLE MONOCHRONOMETER ASSEMBLY

E SIZE | CODE IDENT NO. 07978 | DWG NO. 223-356367

MOD LIO-A | SCALE 1/2 | 1st USE 355899 | SHEET 1 OF 1

FIGURE 23.14 Monochrometer Assembly Parts List

23.5.3 Drawing Sheets

Drafting sheets are polyester film, vellum, bond, or other type of paper preprinted with the border, title block (Fig. 23.17), and revision block. You must know what each aspect of the drawing sheet means, and what information needs to be added before the project is complete. This understanding is also important when reading existing drawings. The following list describes each part of a typical drawing sheet as shown in Figure 23.18(a) and (b):

1. Ancillary drawing number. Permits the engineer to file print copies so that, when folded correctly, all drawing numbers will appear in the upper left corner.
2. Sheet number for multiple sheet drawings
3. Ancillary revision identification
4. Revision identification symbol
5. Description of the revision or the identification of the change authorization document
6. Issue date of the revised drawing
7. Required approval signature for revisions
8. Microfilm alignment arrowheads
9. DSJ—distribution key or code, if used
10. Company name and address. Must agree with FSCM number for companies with multidivisions and departments.
11. Drawing title
12. Assigned drawing number
13. Weight record. Should indicate whether it is actual, estimated, or calculated, when required, and if it is gross (before machining) or net (after machining).
14. FSCM number, if required for identification of the company or design activity whose drawing number is used
15. Predominant scale of the drawing
16. Drawing size letter designation
17. Signature of the drafter and the date the drawing was started
18. Signature of the responsible person who checked the drawing and the date of signing
19. Signature of the responsible engineer, to signify approval of the design by engineering, and the date of signing
20. Signature of the responsible issuing person and the initial date of issue
21. Notes
22. Approval by an activity other than those described above
23. The appropriate surface texture designation that applies
24. The general tolerances that apply to the overall document
25. The appropriate material specification, which should include type, grade, class, or other classifications as applicable
26. Zones: letters vertically (bottom up, starting with A) and numbers horizontally (right to left, starting with 1)

23.6 NOTES

Notes on drawings (Fig. 23.19) supply information that cannot be presented in any other descriptive way. A standard method of applying, placing, and revising notes on engineer-

FIGURE 23.15 Parts List

-001	ITEM NO.	VERSATEC PART NO.	DESCRIPTION
2	13	140-862005-606	STUD,SELF-CLINCH-FH, NO. 10-32 X.750 LG
2	12	140-862005-206	STUD,SELF-CLINCH-FH, NO.4-40 X .750
4	11	104-044042-014	FASTENER,SELF-CLINCHING, NO. 10-32
26	10	140-862005-404	STUD SELF-CLINCH-FH, NO.8-32 X .500
22	9	104-044042-002	FASTENER,SELF-CLINCHING, NO. 8-32
4	8	106-044316-006	WASHER EXTERNAL TOOTH, NO. 6
8	7	102-044729-003	LOCKNUT, HEX, NO. 2-56
2	6	140-017322-011	FASTENER,SELF-CLINCHING SS NO. 4-40
8	5	102-044629-001	NUT, SADDLE
7	4	104-045364-001	NUT, HEX JAM, NO. 1/4-20
6	3	104-044356-003	INSERT,THREADED STAINLESS STEEL,NO. 8-32 X.248 LG
3	2	140-021009-001	NUT-HEX, NO. 8 LIGHT
1	1	000-012345-051	COVER,CONTROL,FREQUENCY PANEL (13)

QUANTITY PER VERSION | PARTS LIST

PROPRIETARY
The contents of this document are PROPRIETARY TO VERSATEC INC. and are not to be disclosed to others or used for purposes other than intended without the written approval of Versatec

MATERIAL:
SEE ABOVE P/L
DATA BASE B.O.M
AVAILABLE.

UNLESS OTHERWISE SPECIFIED
DIMENSIONS ARE IN INCHES. ALL PARTS TO BE DEBURRED AND EDGES BROKEN .010 MAXIMUM

TOL.	1PLC.	2PLC.	3PLC.	ANG.
±	.1	.03	.010	1°—

THIRD ANGLE PROJECTION

FINISH: (7)

SIGNATURE	DATE
DRN. VALENZUELA	9-10-90
CHK.	- -
APPV.	- -
APPV.	- -
APPV.	- -

DO NOT SCALE DRAWING

VERSATEC A XEROX COMPANY — SANTA CLARA CALIFORNIA 95051

TITLE: COVER,CONTROL, FREQUENCY PANEL

SIZE	CODE IDENT.	DRAWING NO.	REV.
E	50804	000-012345	17

SCALE: 1/2 , AS NOTED SHEET 1 OF 1

A

2 1

QTY REQD	FSCM	PART OR IDENTIFYING NO.	NOMENCLATURE OR DESCRIPTION
PARTS LIST			

(a) Columnar arrangement for integral parts list

PARTS LIST			
QTY REQD	FSCM	PART OR IDENTIFYING NO.	NOMENCLATURE OR DESCRIPTION

(b) Columnar arrangement for separate parts list

FIGURE 23.16 Parts List Arrangement

ing drawings is used, to maintain company-wide uniformity. Although there are standard formats, placements, and sequences for notes on the drawing, each company will have its own **company standards**.

General notes (Fig. 23.19) are those that apply to the total drawing and, if placed on the drawing at each point of application, would be repetitive and time consuming to apply. **Local notes** are those that apply to a specific portion, surface, or dimension on a drawing. The following rules can serve as a guide when putting notes on a drawing.

Use Notes To:

- Clarify features that can be defined more accurately through words than by graphical delineation and dimensions

ITEM	QTY	PART NO.	DESCRIPTION
4	2	141-356123	BEARING (K6)
3	1	171-349020	OIL PICKUP
2	2	130-349008	END RING
1	1	223-349005	ARMATURE SHAFT ASSY (GRINDING)

MATERIAL

FINISH

UNLESS OTHERWISE SPECIFIED
DIMENSIONS ARE IN INCHES
TOLERANCES
.X ±.050
.XX ±.020
.XXX ±.005
ANGULAR ±0° 30'
MACH. SURF. √

INTERNAL THD HEIGHT 55% MIN
THREADS: CLASS 2A OR 2B
REMOVE BURRS & SHARP EDGES .020 MAX.
MACH. FILLET RADIUS .020 MAX.
MACH. SURF. FLAT WITHIN .001 IN./IN.
OTHER SURF. FLAT WITHIN .005 IN./IN.
CONCENTRICITY MACH. SURF.
T.I.R. WITHIN 1/2 SUM OF DIAS. TOLS., .001 MIN.
DO NOT SCALE DRAWING

	BY	DATE
DR	D.M.DUARTE	5/8/89
CHK		
DSGN		
ENGR		

MOD L10

BECKMAN — BECKMAN INSTRUMENTS, INC. SPINCO DIVISION 1050 PAGE MILL ROAD PALO ALTO, CALIFORNIA 94304

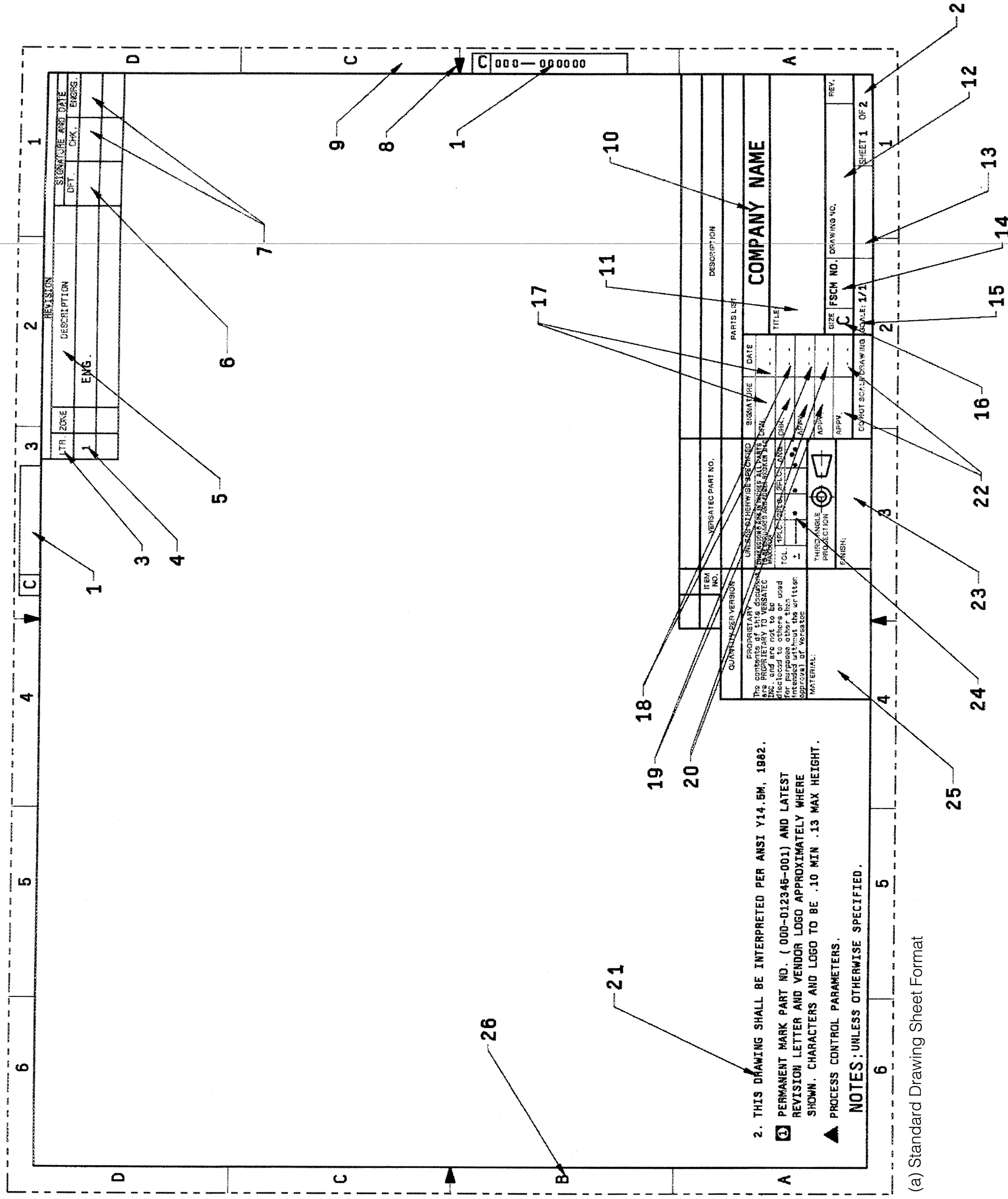
TITLE ARMATURE AND END RING ASSEMBLY (BALANCING)

SIZE	CODE IDENT NO.	DWG NO.
C	07978	223-349006

SCALE 2X | 1ST USE 349001 | SHEET 1 OF 1

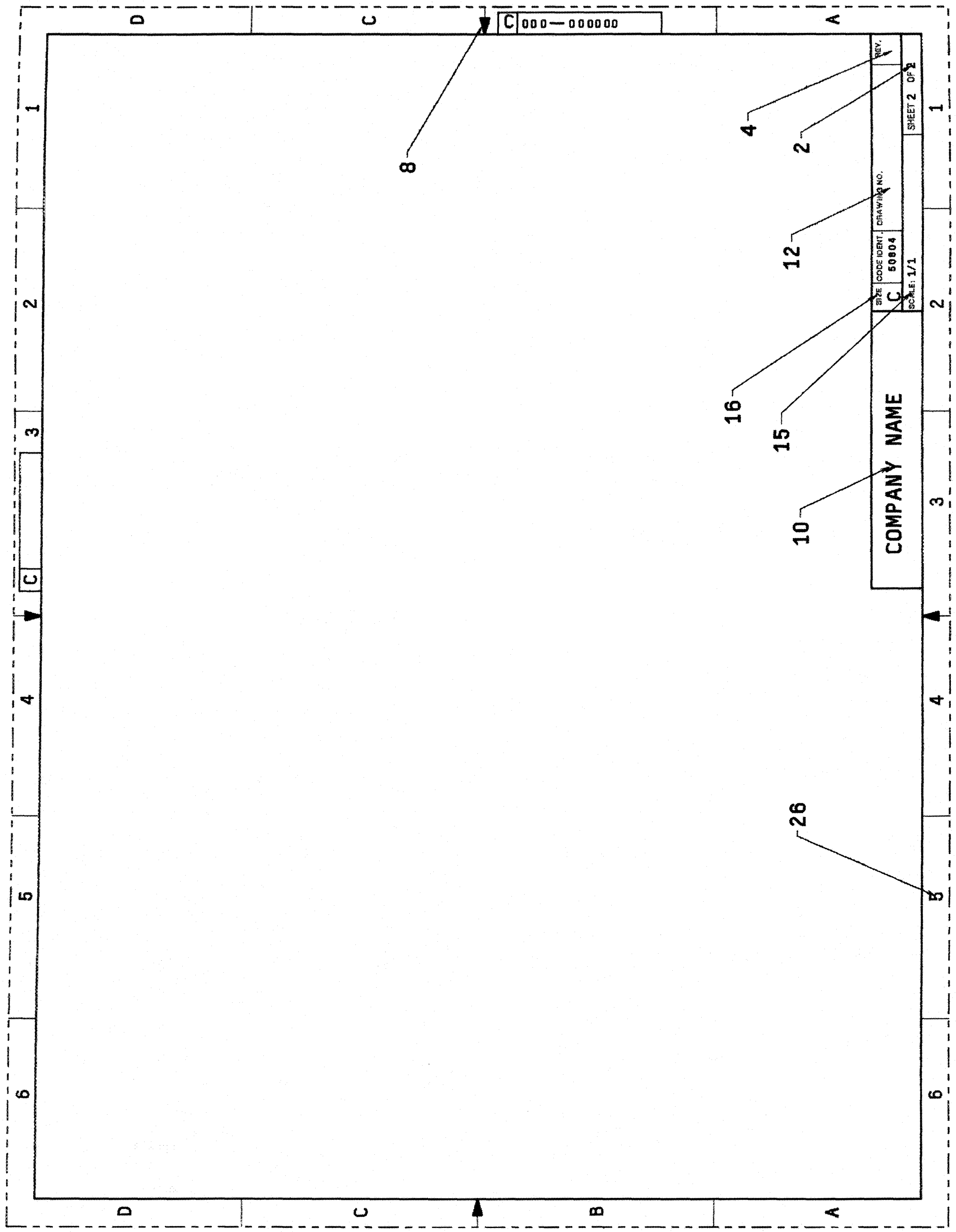
PART: 04.ME.2235.349006 DRAW: A 05-08-89 15:41:13

FIGURE 23.17 Armature and End Ring Assembly Title Block and Parts List



(a) Standard Drawing Sheet Format

FIGURE 23.18 Standard Drawing Sheet Format



(b) Standard Drawing Sheet Format—*Continued*

FIGURE 23.18 Standard Drawing Sheet Format—*Continued*

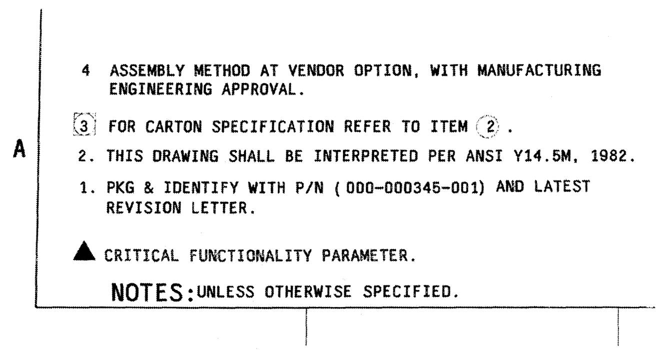


FIGURE 23.19 General Notes

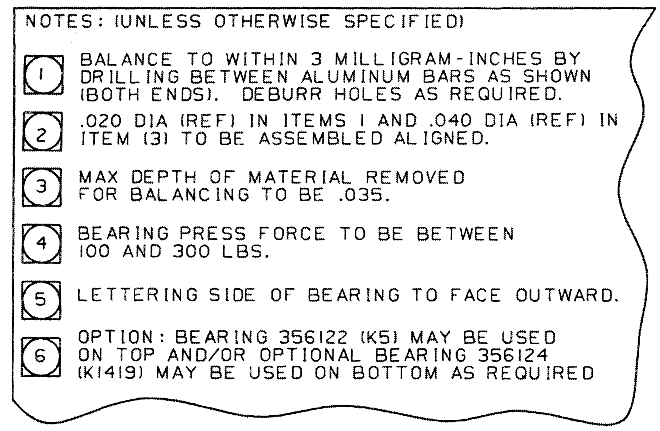


FIGURE 23.21 Notes Listed from the Top Down

- Give instructions for applying special treatments
- Give instructions for utilization of specific processes
- Describe instructions to supplement standard symbols
- Provide additional information to the drawing document or for its use
- Add clarifying notes so the part can be made correctly the first time

Notes Should Be:

- Clear and concise
- In the present tense
- Positioned parallel to the bottom edge of the drawing
- Carefully composed to relay one message; capable of one interpretation
- Preprinted on appliqués or, if the drawings are computerized, entered in a standard library for repeated use

Notes Should Not:

- Be underlined on the drawing
- Contain abbreviations other than the most commonly understood shop trade terms
- Duplicate information on an associated parts list or shop practices reference document
- Contain dimensions that are already documented elsewhere on the drawing
- Describe complex processes (which should instead be documented either in a specification or a process document)
- Reference information that is given elsewhere in the product documentation

23.6.1 General Notes

General notes on drawing sizes "B" through "F" are placed in the upper or lower left-hand corner of the sheet. Some companies construct their note sequence from the bottom up, as in Figure 23.20; others list theirs from the top down (Fig. 23.21). Notes at the top of a sheet are numbered from the top down; those at the bottom of a sheet are numbered from the bottom up. The width of the general note column should not exceed 6 to 8 in. (150 to 200 mm).

When the drawing is completed on a CAD system, it is possible to reuse notes that are common to a number of situations and designs. Such notes are saved as a **text file** and inserted as blocks on the drawing. Because text files can be edited, variations and changes can be incorporated before they are inserted into a drawing.

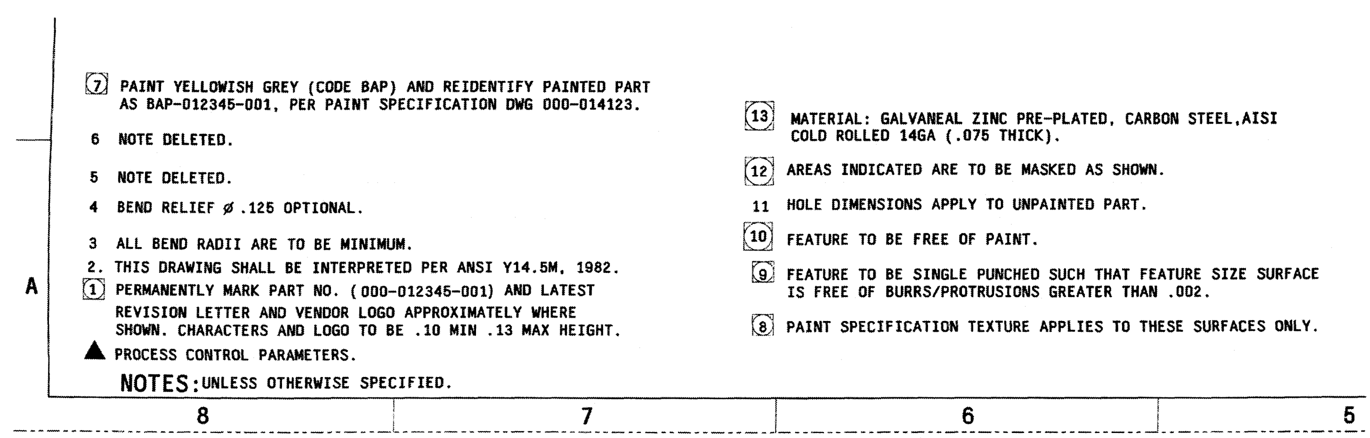


FIGURE 23.20 Notes Listed from the Bottom Up

23.6.2 Local Notes

Local notes on the drawing must be placed outside the outline of the part and close to the item that is being referenced.

Fabrication operations such as **BEND**, **DRILL**, **TAP**, **PUNCH**, and **BORE** are *not* shown on the drawing. This permits manufacturing to determine the type of operation required to produce the part within the required tolerances. Features such as **SPOTFACE**, **COUNTERBORE**, **COUNTERSINK**, **UNDERCUT**, and **THREAD** may be given in notes.

23.7 REVISION OF ENGINEERING DRAWINGS

The **revision block** (Fig. 23.22), located in the upper right-hand corner of the drawing, is for recording changes to the drawing. Because revisions initiate a substantial number of change documents in all functions of a business, the need for accuracy and completeness in the revision process should not be underestimated.

23.7.1 Revision Terminology

The following terms describe the process of revising drawings and are found in revision blocks on drawings.

Added A new feature or view introduced to the document

2 1

REVISION					
LTR.	ZONE	DESCRIPTION	SIGNATURE AND DATE DFT.	CHK.	ENGRG.
1		ENGRG.	VICTOR.V 2/22/89		
2	E5	ADDED ITEM 5 THRU 7 AND HOLES MFG.	VICTOR.V 3/07/89		
3	B2	ADDED HOLE CHART & PAGE 2 INPUT MFG.	VICTOR.V 3/12/89		
4	C7	REVISED SHT 1 & 2 PER ENG. CHANGES	VICTOR.V 3/26/89		
5	D3	ADDED BOTTOM VIEW & DETAIL F REV TOP VIEW LOCATION AND ADDED V15 THRU V22 ON HOLE CHART	VICTOR.V 4/03/89		
6	A5	REVISED LOCATION U1 ON CHART AND SHT 2 REV RADII WAS:.313 IS: .375	VICTOR.V 4/07/89		
7	F4	ADDED N5, N6, R2, R3, Y3 AND Y4	VICTOR.V 9/18/89		
8	C2	INCORP PROTO CHANGES PILOT RELEASE	VICTOR.V 9/18/89		
9	D6	REVISED PER ACO NO. 147	VICTOR.V 11/01/89		
10	E5	REVISED DIMENSIONS WAS: 3.50 & 6.25 IS: 4.00 & 6.00	VICTOR.V 11/16/89		
11	G8	ADDED ITEMS 2 & 3	VICTOR.V 12/06/89		
12	H3	REVISED ITEM 4 & 5 PER DETAILED PART	VICTOR.V 1/26/90		
13	F7	REVISED QTY OF ITEMS 7 & 9	VICTOR.V 3/08/90		
14	A6	ADDED SECTION A-A AND DETAIL B	VICTOR.V 6/01/90		
15	C5	REVISED NOTES 4, 7 AND 9	VICTOR.V 6/25/90		
16	E6	DELETED NOTES 5 AND 6	VICTOR.V 8/06/90		
17	G4	REVISED PER ACO NO. 353	VICTOR.V 9/10/90		

H

G

FIGURE 23.22 Revision Block

Approval An endorsement attesting to a revision made on a drawing or a parts list

Change A specific alteration made as part of a revision on a drawing. A revision may include one or more changes.

Deleted A feature or view removed from the document

Obsolete (inactive, canceled) A condition in which the drawing has been discontinued by the design activity. The word *inactive* or *canceled* may be used.

Redrawn A new original drawing with the same drawing number that has been substituted for a previous drawing

Revision (revised) One or more changes to a drawing, made after distribution or release, according to an established revision procedure

Revision designation Alphabetic, numeric, or alphanumeric characters that identify a revision

23.7.2 Revising Drawings

Revisions are made by erasure, crossing out, addition, redrawing, or, for the case of CAD-generated drawings, editing. When picking the method to be used to revise a drawing, first consider achieving and maintaining the best possible quality, legibility, and reproducibility by the most economical means. Unless otherwise specified, use the most recently approved graphics symbols, designations and letter symbols, abbreviations, and drawing practices. The exception is the use of geometric and position tolerance symbols that may be different from the latest issue of ANSI Y14.5. If the latest symbol is desirable, an explanatory note should be provided on the drawing. Superseded symbology on the drawing should remain unchanged, provided the interpretation is clear.

23.7.3 Incorporating Changes

Dimensional changes entered on a drawing are made to the same scale as the portion of the drawing undergoing revision. If the drawing is not to scale, and the pictorial portion of the drawing is made to proportion, all dimensional changes are made to the proportions of the delineation affected.

When information is added to a drawing, the additions must match the lettering style and line weight of the existing drawing as closely as possible.

23.7.4 Simplifying the Design Process and Saving Time

Saving time on a project may mean bringing it to market ahead of the competition. Overdrawn and detailed designs add time and cost to a project. The following list can help you check for simplicity:

1. Use text description wherever possible to eliminate drawing completely.

2. Use text description wherever practical to eliminate projected views.
3. Eliminate views where the shape can be given by description, e.g., **HEX**, **SQ**, **DIA**.
4. Show partial views of symmetrical objects.
5. Avoid elaborate, pictorial or repetitive detail.
6. When necessary to detail threads, do not show them over the full length of the stud, bolt, or tapped hole.
7. Eliminate detail of nuts, bolt heads, and other standard hardware. Show outlines when it is necessary to show position.
8. Reduce detail of parts on assembly drawings. Simply show the part position.
9. Avoid unnecessary hidden lines that add no clarification.
10. Use sectioning only when it is necessary for the clarity of the drawing.
11. Simplify graphics for holes and tapped holes by use of symbols.
12. Omit views with no dimensional or written instruction.
13. Within limits, a small drawing is usually easier and quicker to make than a large one.
14. When two parts are only slightly different, complete graphical representation of both parts is not required. The note **SAME AS EXCEPT** _______ or **OTHERWISE SAME AS** _______ may be given.
15. Drawings made to modify stock or commercial parts should be as plain as possible. Avoid detail.
16. Use standard abbreviations wherever possible.
17. When necessary, enlarge small details on larger parts for clarity.
18. Draw small parts large enough to avoid crowding so that they may be easily read, but not unnecessarily large so as to waste space on the drawing.
19. Do not duplicate dimensions.
20. Substitute recognized standard symbols, to simplify greatly the drawing of common objects.
21. Eliminate repetitive data by use of general notes.
22. When drafting, do as much free-hand drawing as the work permits, in preference to using instruments.
23. Where practical, use geometric symbols instead of notes.
24. Where acceptable, give rectangular coordinate or tabular dimensioning instead of dimension lines.

23.8 A CHECKLIST FOR ENGINEERS AND DESIGNERS

A drawing should be checked after it has been completed. Compare it against the following.

Readability

1. Is the drawing easy to read?
2. Are the part outlines distinct from dimension lines?
3. Is the lettering neat and clear?
4. Is all of the information on the drawing?
5. Will the drawing make a good print?
6. Have all the rules of standard drafting practice been followed?
7. Is the nomenclature correct? Will everyone understand it the same way?
8. Is the drawing title truly descriptive?

Completeness

9. Are all necessary views given?
10. Are some views unnecessary?

Notes

11. Are the general notes properly located?
12. Are any exceptions to the general notes clearly pointed out?
13. Are any notes crowded or hard to find?
14. Could any of the notes be misunderstood?
15. If a specially purchased item is required, is procurement information given?
16. If special procedures are required in making or assembling, have they been noted on the drawing?

Parts List

17. Does the parts list agree with the drawing?
18. Have overall dimensions been given?
19. Are standard parts specified correctly?

Dimensioning

20. Are out-of-scale dimensions (if any) clearly marked?
21. Is it necessary to leave a dimension out of scale?
22. Are all dimensions given?
23. Are there any duplicate dimensions?
24. Are dimensions kept well away from the outline of the part?
25. Is the scale designated?

Tolerances

26. Have all tolerances given been carefully considered?
27. Are all tolerances to the maximum possible?
28. Are any tolerances too large? too small?
29. Has the drawing been checked for possible tolerance stackups?

Finishes

30. Are all machine finishes given, and do they conform to applicable specifications?
31. Are all paint and plating finishes specified?

Processes

32. Is heat treatment needed?
33. Have standard manufacturing processes been followed?
34. Can the part be produced simply and economically?

Materials and Parts

35. Are standard or purchased parts used to the maximum extent?
36. Are all special or reworked parts noted?

Assembly

37. Are there no mechanical interferences?
38. Will parts assemble without difficulty?
39. Does the work agree with associated mechanisms?
40. Are all parts properly numbered or designated?

Cost

41. Could the function have been accomplished at less expense with the same results?
42. Could the design have contained fewer parts?
43. Have you given thought as to how this would be built?

Reliability

44. Have you checked the design for possible failure?
45. Have you considered safety factors?

23.9 REPRODUCTION AND STORAGE OF DRAWINGS

The last step in the design-detailing process is outputting the drawings of the project. Reproduction of manual or CAD drawings involves a process called *whiteprinting*. Before a CAD drawing can be printed, it must be plotted on one of many types of plotters available.

23.9.1 Whiteprinting

Whiteprinters (Fig. 23.23) make copies of drawings. The whiteprinter is still referred to as a "blueprint machine" by many people. As an engineer you can expect to run prints of drawing projects.

Regardless of the type of "paper," the whiteprinter makes a "positive" image of the drawing on whiteprint paper. The process depends on the transmission of light through the drawing paper and onto the developing surface. The lines, lettering, or other graphics block the light in order for a positive image to be developed on the print. Because they must block light, the lines must be high quality, crisp, and dark. Use H or HB grade lead, and press sufficiently hard to create a dark, crisp drawing. CAD-plotted drawings employ black ink, so the drawing's lines and lettering block light well.

FIGURE 23.23 Blueprint Duplicating Machines

23.9.2 Plotting

Output devices include printers, plotters, and photocopy equipment. The **plotter** allows you to produce drawings on paper, vellum, or drafting film in a multitude of colors. Some plotters are limited by the size of the plotting surface. Others can plot drawings of any length, although they are limited to standard paper widths. Pen plotters use ballpoint pens, felt-tip pens, or liquid-ink pens. When plotting a drawing, you have many options for scaling the drawing, rotating it, selecting colors, and selecting different line widths.

After the drawing is plotted on vellum or polyester film, a whiteprinter can make copies. Multiple copies can be made on the plotter, but plotting is slow and tedious. Elesctrostatic plotters are much faster than pen plotters and can plot a drawing without the mess of wet-ink technical pens. Figure 23.24 shows COM units, one of the latest innovations in plotting technology. A laser plotter is faster than an electrostatic plotter, and is as accurate as a pen plotter.

A CAD plotter can output as many high-quality originals as required. However, having more than one original could cause considerable document control problems! The "old" original should be removed from circulation or destroyed.

FIGURE 23.24 COM Units

Applying Parametric Design . . .

ASSEMBLIES, EXPLODED ASSEMBLY MODELS, AND REPORTS (BOM)

The **Assembly mode** allows you to place together component parts and subassemblies to form assemblies (Fig. A). These assemblies can then be modified, documented (Fig. B), analyzed, or reoriented. **Assembly mode** is used for the following functions:

- Placing components into assemblies
- Exploding views of assemblies
- Part modification, including feature construction
- Analysis

With Pro/ENGINEER You Can:

- Assemble Place together component parts and subassemblies to form assemblies
- Delete or replace Remove or replace assembly components
- Modify assembly placement offsets, create and modify assembly datum planes, coordinate systems and cross sections
- Modify parts directly in assembly mode
- Get assembly engineering information, perform viewing and layer operations, create reference dimensions, and work with interfaces

With Pro/ASSEMBLY You Can:

- Create new parts in assembly mode
- Create sheet metal parts in assembly mode
- Mirror parts in assembly mode (create a new part)
- Replace components automatically by creating interchangeability groups.
- Create assembly features, existing only in assembly mode and intersecting several components
- Create families of assemblies, using the family table
- Simply the assembly representation
- **Move** and **Multiply** commands for assembly components
- Use Pro/PROGRAM to create design programs that allow user entries to program prompts to alter the design model

Creating an assembly is accomplished by adding components (parts) to a base component (parent part) using a variety of constraints. A **placement constraint** specifies the relative position of a pair of surfaces on two components. The **mate, align, insert**, and **orient** commands and their variations can accomplish this task.

General Principles for Constraint Placement

- The two surfaces must be of the same type (for example, plane–plane, revolved–revolved). The term "revolved surface" means a surface created by revolving a section or by extruding an arc or a circle. Only the following surfaces are allowed: plane, cylinder, core, torus, sphere.
- If you put a placement constraint on a datum plane, you should specify which side of it you are going to use, yellow or red.
- When using **Mate Offset** or **Align Offset** and entering a value, you will be given the offset direction. If you need an offset in the opposite direction, make the offset value negative.
- When a surface on one window is selected, another window may become hidden. For a description of how to bring the hidden window to the front, see the appropriate manual for the hardware used.
- Add constraints one at a time. For example, it is not possible to align two different holes in one part with two different holes in another part via a single align command; two different align constraints must be defined.

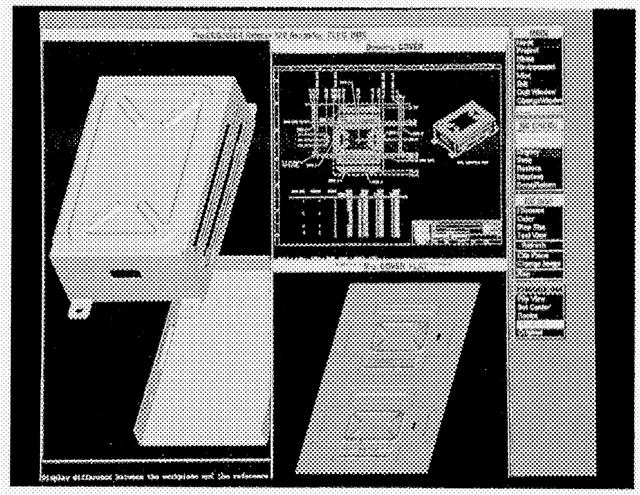


FIGURE A Electronic Product Assembly, Including Printed Circuit Board and Package

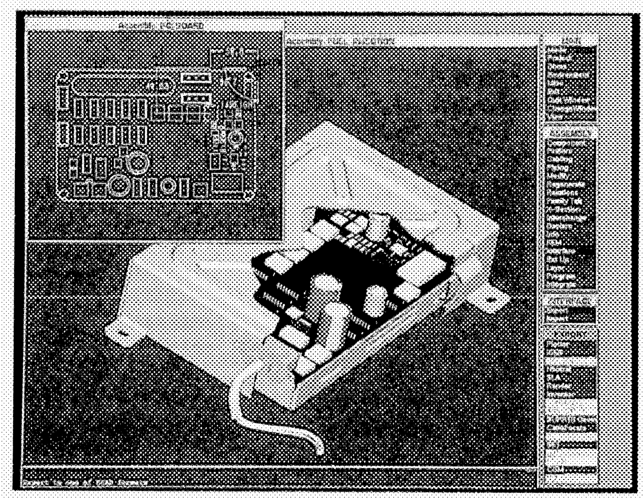


FIGURE B Assembly Package and Documentation

- Placement constraints are used in combinations in order to specify completely placement and orientation. For example, one pair of surfaces may be constrained to mate, another pair to insert, and a third pair to orient.

Mate (Fig. C) makes two surfaces touch one another: coincident and facing each other. When using datums, this means that two yellow sides, or two red sides, will face each other. **Mate Offset** (Fig. D) makes two planar surfaces parallel and facing each other. The offset value determines the distance between the two surfaces.

The **Align** command (Fig. E) makes two planes coplanar: coincident and facing in the same direction; or aligns revolved surfaces or axes to be coaxial. You can also align two datum points, vertices, or curve ends; selections on both parts must be of the same type (that is, if a datum point is selected on one part, only a datum point on another part can be selected). The **Align Offset** command (Fig. F) aligns two planar surfaces at an offset: parallel and facing in the same direction.

The **Insert** command inserts a "male" revolved surface into a "female" revolved surface, aligning axes.

The **Orient** command (Fig. G) orients two planar surfaces to be parallel and facing in the same direction; offset is not specified.

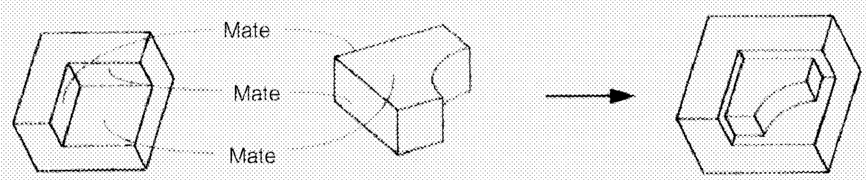


FIGURE C **Using Mate to Constrain the Part in the Assembly**

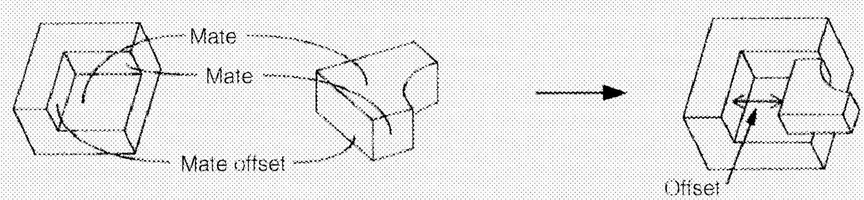


FIGURE D **Mate Offset Constraint**

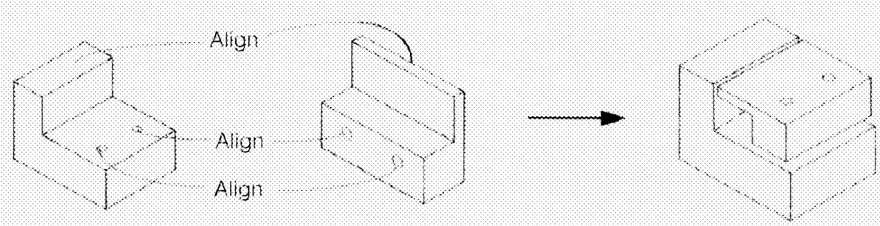


FIGURE E **Align Used to Constrain the Part**

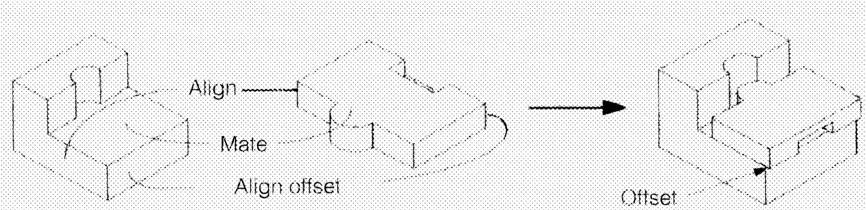


FIGURE F **Align Offset Used to Constrain the Part**

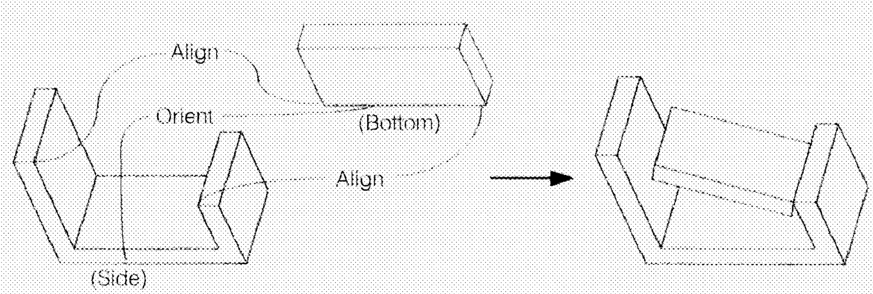


FIGURE G **Using Orient**

The **Coord Sys** command (Fig. H) places a component into the assembly by aligning its coordinate system with a coordinate system in the assembly (both assembly or part coordinate systems can be used). Both coordinate systems must exist before starting the assembly process. Coordinate systems can be picked or selected by name from namelist menus. The components will be assembled by aligning X, Y, and Z axes of the selected coordinate system.

The **Tangent, Pnt On Surf,** and **Edge On Surf** commands control the contact of two surfaces at their tangency, at a point, or at an edge (Fig. I). An example of use of these placement commands is the contact surface or point between a cam and its actuator.

In most cases, a combination of constraints will be required. **Mate, mate,** and **insert** constrain the two parts shown in Figure J. **Mate, insert,** and **orient** are also a possibility, depending on the parts (Fig. K).

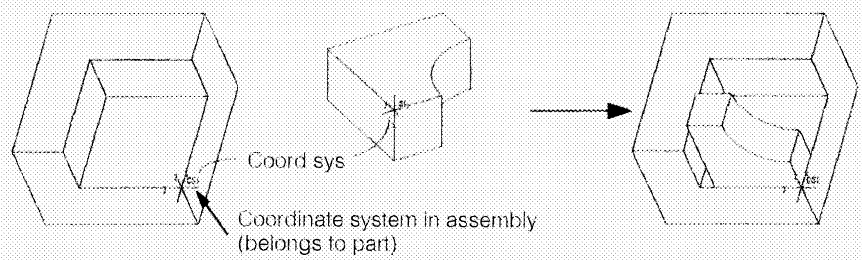


FIGURE H **Using Coord Sys to Constrain the Part**

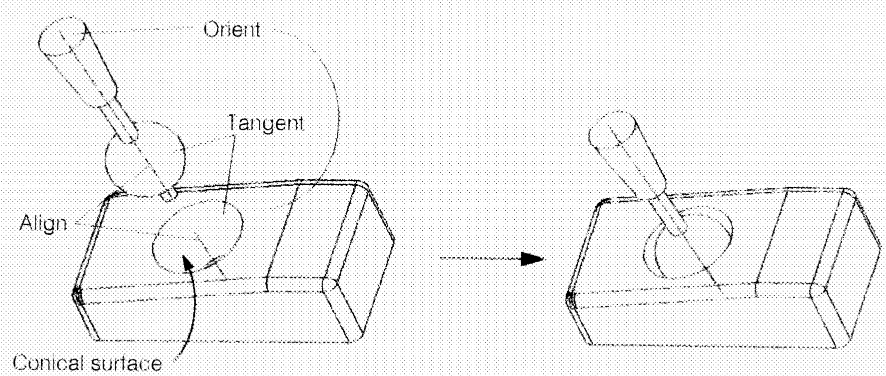


FIGURE I **Using Tangent to Constrain the Part**

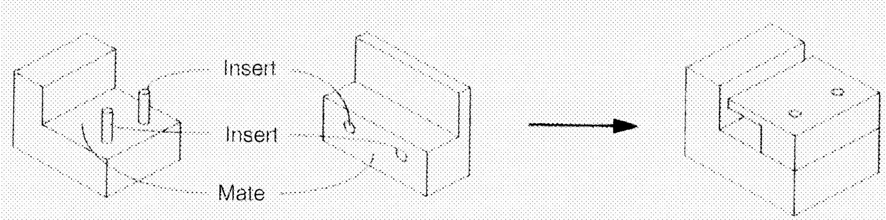


FIGURE J **Insert, Insert, and Mate Used to Constrain the Part**

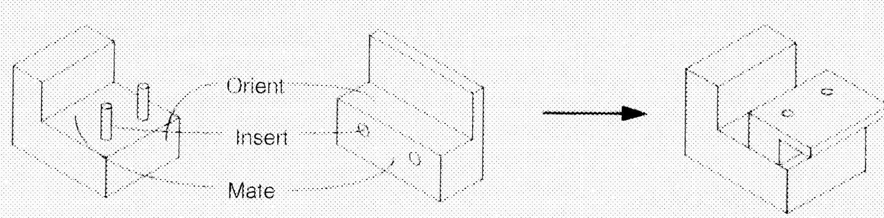


FIGURE K **Mate, Insert, and Orient Constraining a Part**

Assemblies (Fig. L) can also be displayed exploded (Fig. M). The explode distance can be modified to any value.

The angle, screw, and sleeve assembly shown in Figure N is an example of the type of capabilities available. The angle part is the base part (Fig. O). The screw is constrained via **insert** and **mate** (Fig. P), and the sleeve is constrained by **mate, insert,** and **orient** (Fig. Q). The completed assembly (Fig. R) can be modified by redefining the constraints. An exploded cosmetic view of the assembly can be displayed with the reference planes, coordinate system, and hidden lines (Fig. S) or without them (Fig. T). The last stage of the project involves putting the assembly in the draw mode and displaying the appropriate views (Fig. U).

After the assembly is complete, an assembly drawing is created. With Pro/REPORT you can then generate a bill of

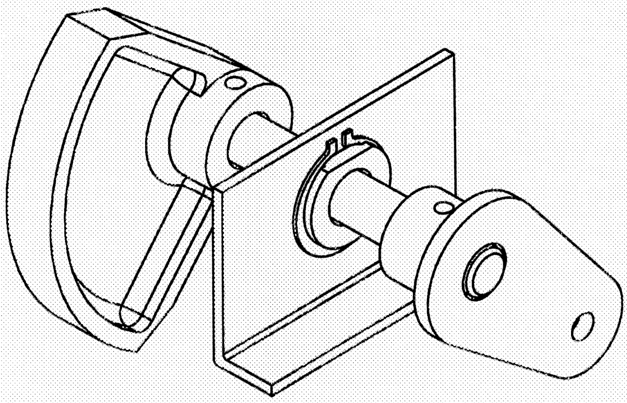


FIGURE L Assembly

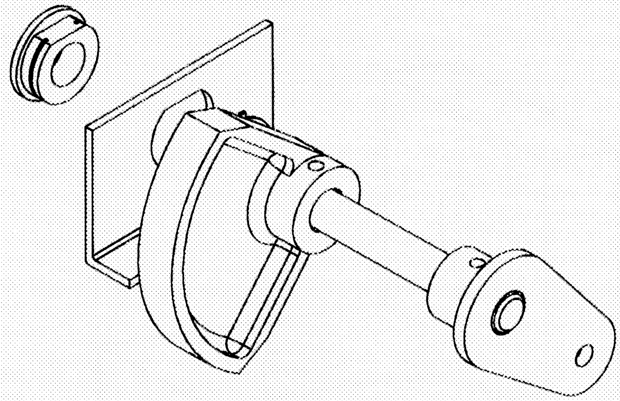


FIGURE M Exploded Assembly

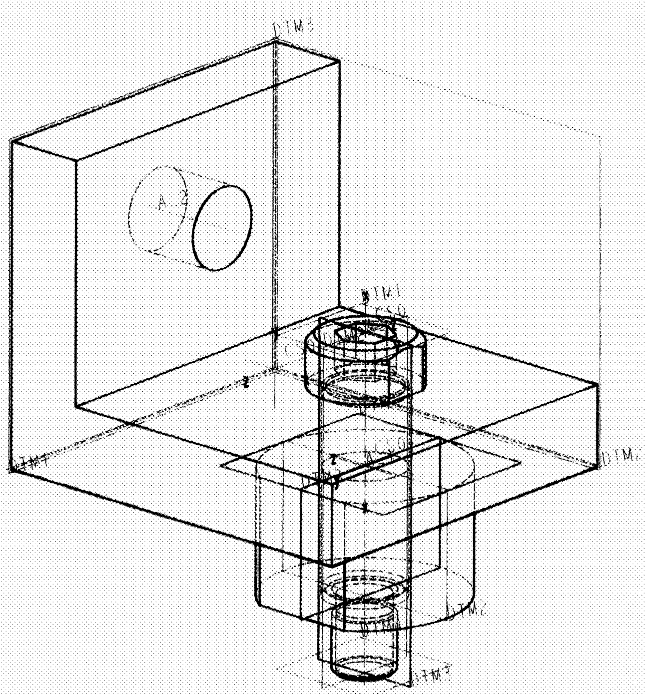


FIGURE N Angle, Socket-Head Shoulder Screw, and Sleeve Assembly

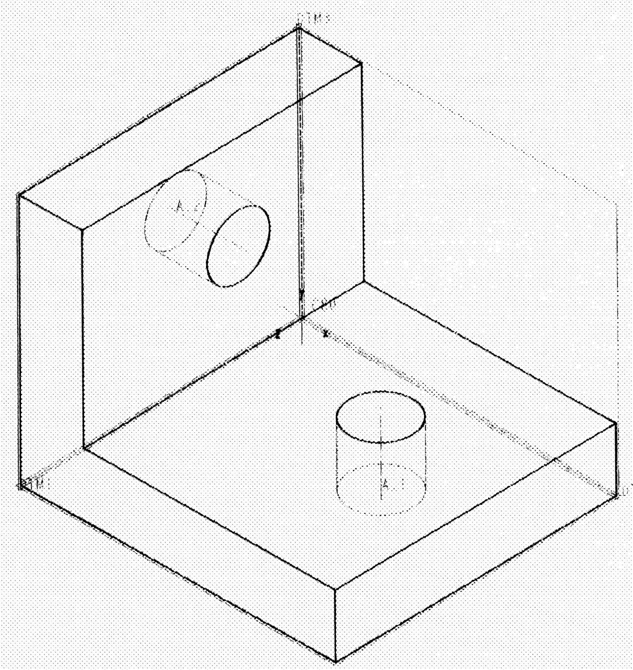


FIGURE O Angle Part

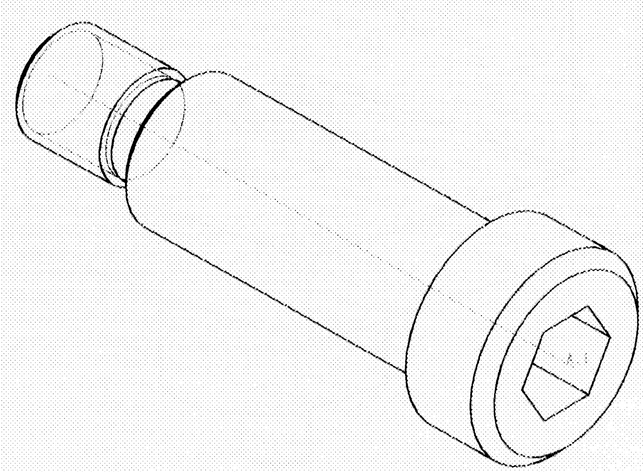


FIGURE P Socket-Head Shoulder Screw

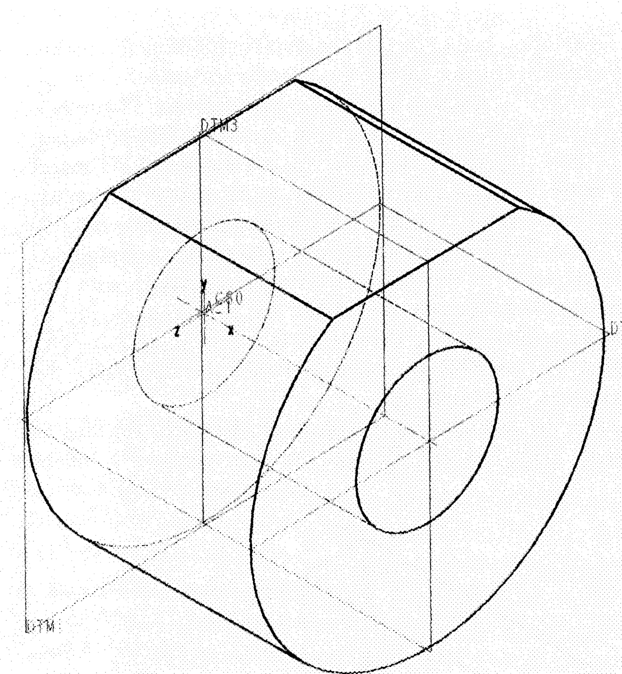


FIGURE Q **Sleeve**

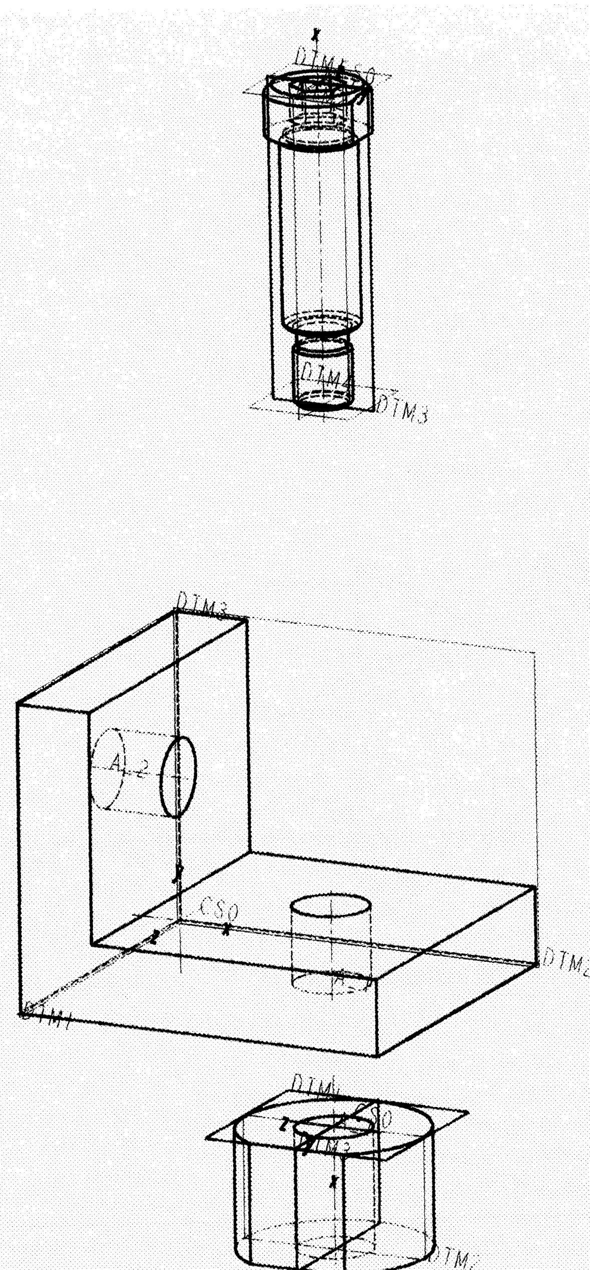


FIGURE S **Exploded Assembly with Datums and Coordinate System Displayed**

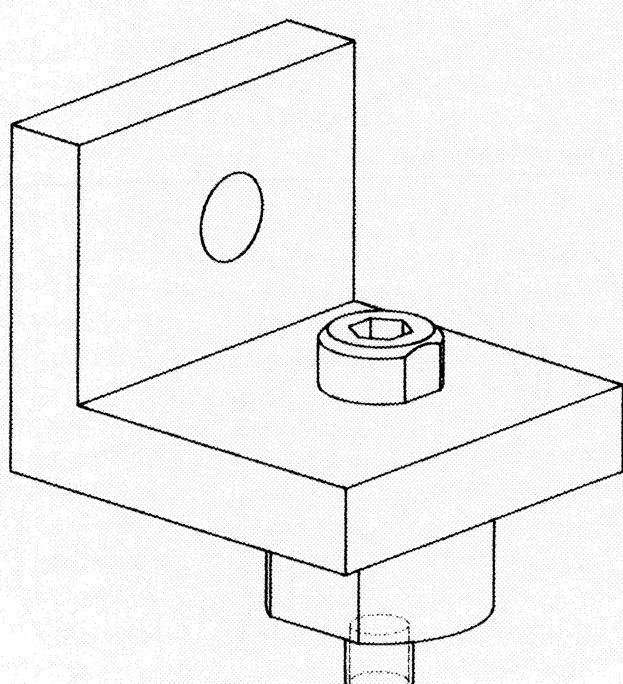


FIGURE R **Constrained Parts**

materials or other tabular data as required for the project. **Pro/REPORT** introduces a formatting environment where text, graphics, tables, and data may be combined to create a dynamic report. Specific tools enable you to generate customized **bills of materials** (BOMs), family tables, and other associative reports, including the following.

- Dynamic, customized reports with drawing views and graphics can be created (Fig. V).
- User-defined or predefined model data can be listed on reports, drawing tables, or layout tables. This reported data can be sorted by any individual requested data type display.

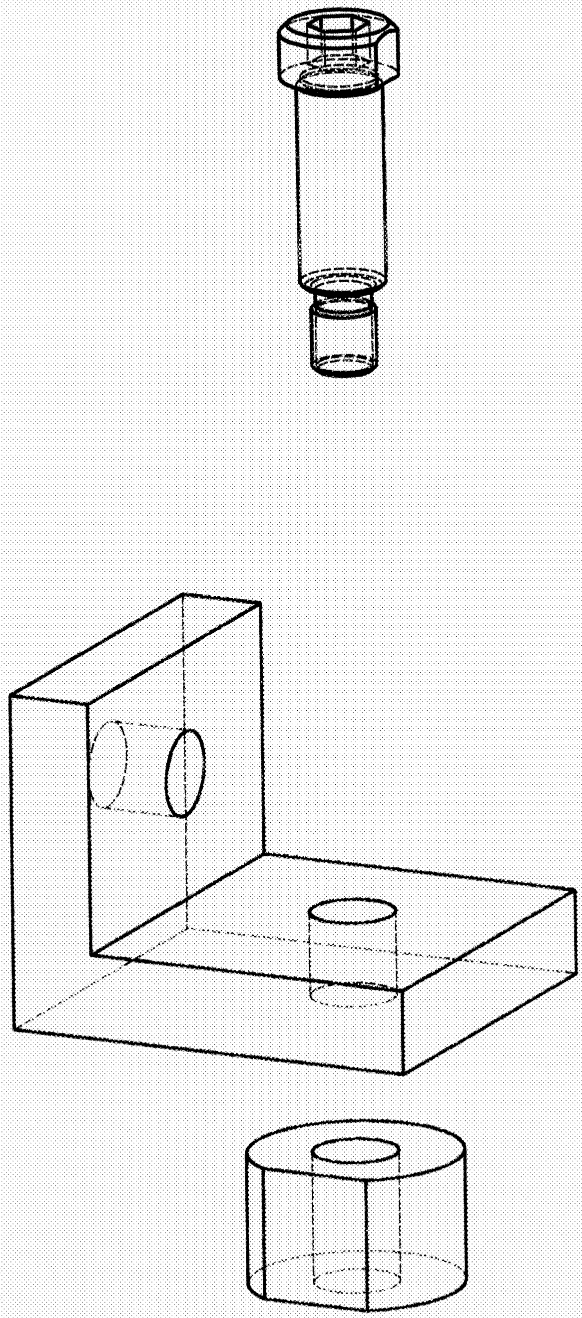


FIGURE T Exploded Assembly with Hidden Lines Only

- Regions in drawing tables, report tables, and layout tables can be defined to expand and shrink automatically with the available model information that has been requested to display.
- Filters can be added to eliminate specific types of data from displaying in reports, drawing tables, or layout tables.
- Recursive or top-level assembly data can be searched for display.

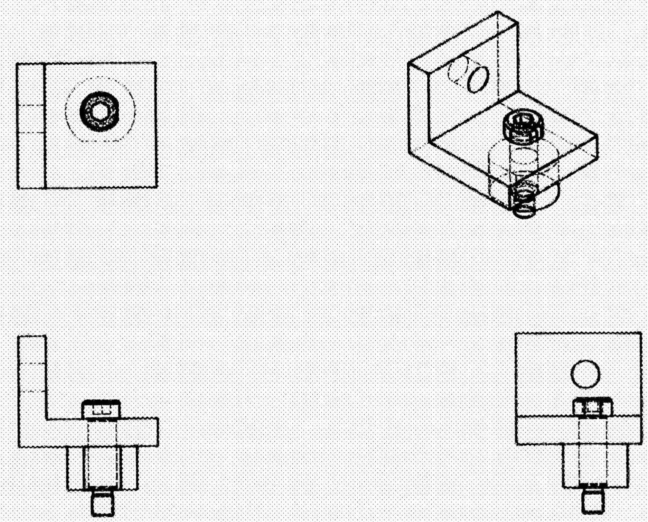


FIGURE U Assembly in Drawing Format

- Duplicate occurrences of model data can be listed individually or as a group in a report, drawing table, or layout table.
- Assembly component balloons can be linked directly to a customized **BOM** (Fig. W) and updated automatically when assembly modifications are made.

In **Report Mode**, data can be displayed in a tabular form on reports just as it is in drawing tables. The data reported on the tables is taken directly from a selected model and updates automatically when the model is modified or changed. A common example of a report is a bill of materials report or a generic part table (Fig. X).

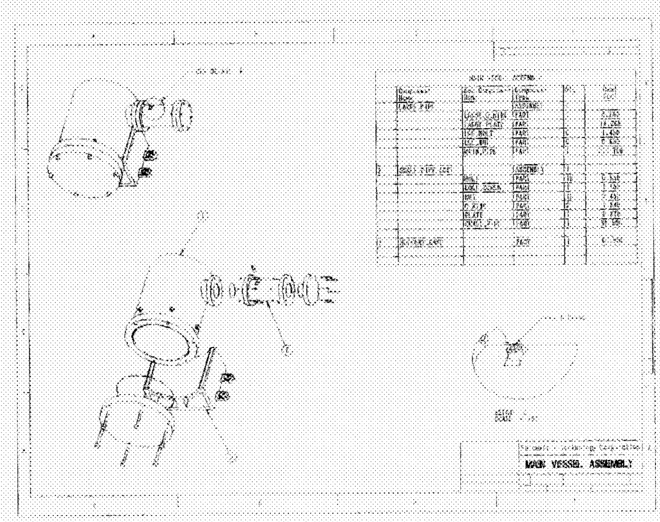


FIGURE V Assembly, Exploded Assembly, and Bill of Materials

(Continued)

Repeat Region row.

Top Level Assembly BOM		
Index	Component Name	Quantity

Report table BEFORE update for model data.

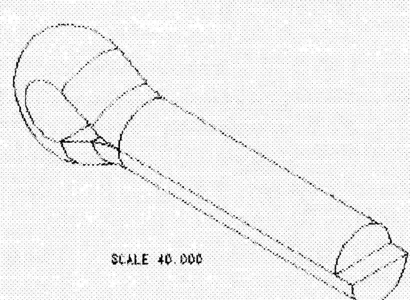
Expanded Repeat Region.

Top Level Assembly BOM		
Index	Component Name	Quantity
1	LARGE_PIPE	1
2	SMALL_PIPE_ASSY	1

Report table AFTER model added.

FIGURE W Creating a Bill of Material

GENERIC PART NAME: *PINA*				
	MODEL PARAMETERS			
INSTANCE NAME	*a*	*head_dia*	*NOM.SIZE*	*prong*
PINAD1	0.032	0.060	.0312	0.010
PINAD2	0.048	0.090	.0468	0.020
PINAD3	0.060	0.120	.0625	0.030
PINAD4	0.076	0.160	.0781	0.040
PINAD5	0.090	0.190	.0937	0.040
PINAD6	0.104	0.220	.1093	0.050
PINAD7	0.120	0.250	.125	0.060
PINAD8	0.134	0.280	.1406	0.060
PINAD9	0.150	0.310	.1562	0.070



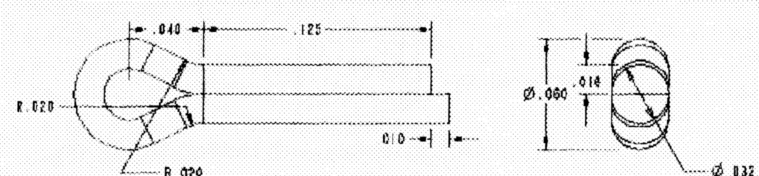


FIGURE X Drawing Table

23.9.3 Computer-Output Microfilm (COM) Units

Computer-output microfilm (**COM**) units provide fast and accurate plotting of drawings. The microfilm is usually mounted on an aperture card. The CAD-generated database is transferred from the system to a graphics controller (Fig. 23.26), and the laser film plotter creates the aperture card directly from a variety of data formats (Fig. 23.25). The microfilm can be enlarged.

FIGURE 23.25 Aperture Card Laser Plotter

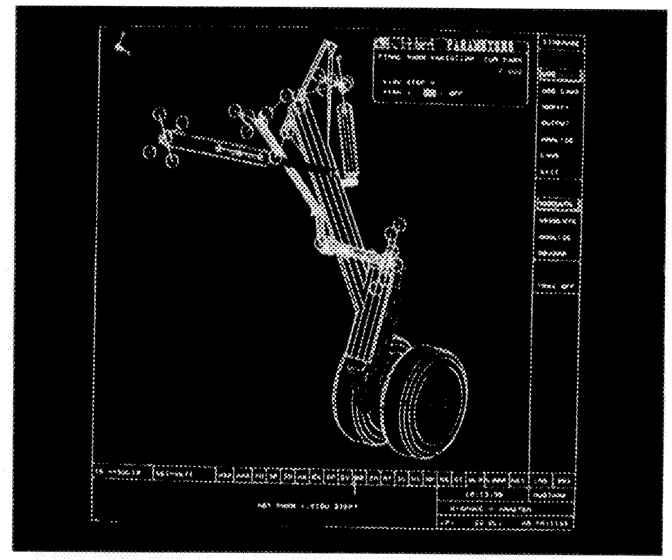


FIGURE 23.27 CAD Assembly

23.10 PREPARING ASSEMBLIES AND DETAILS WITH A CAD SYSTEM

A 3D CAD system makes it easier to check the spatial relationship between parts in an assembly, and the parameters in the model may be varied to produce different-size components (Fig. 23.27). The assembly can be rotated to view it from any angle to examine it for interferences (Fig. 23.28). With a 2D system, as with manual drafting, complex projections are required to achieve the same results. Since the 3D system can view a part from any angle, isometric and perspective views can be created with little extra effort.

Exploded views of the assembly are easy to create from a 3D assembly model (Fig. 23.29). In Figure 23.30 the assembly was modeled on a CAD system and plotted on an electrostatic plotter. The assembly was displayed pictorially, which enhances the assembly because each part is realistically depicted in its assembled position. Since the pictorial illustration clearly displays the parts of the assembly and each part is ballooned appropriately, separate views were not required. A separate section (**VIEW A–A**) is provided in Figure 23.30.

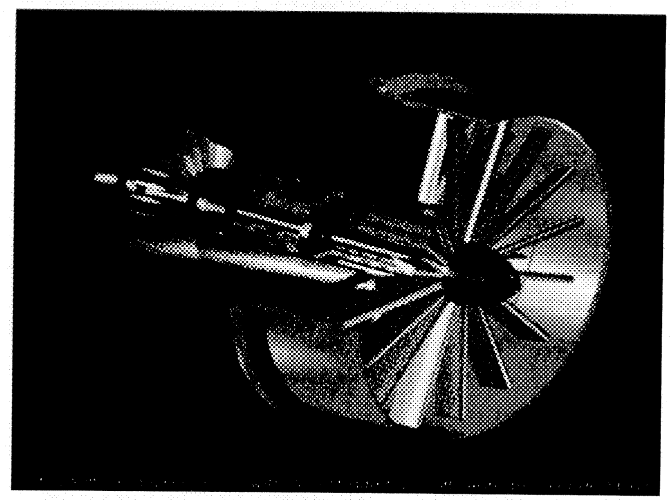


FIGURE 23.28 Turbine Assembly

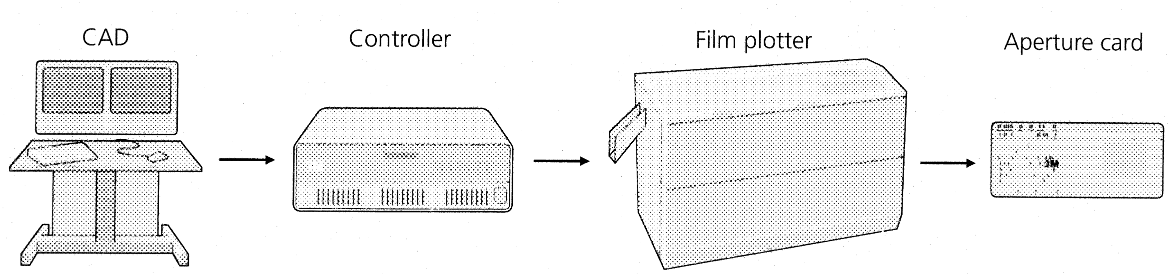


FIGURE 23.26 Aperture Card Output

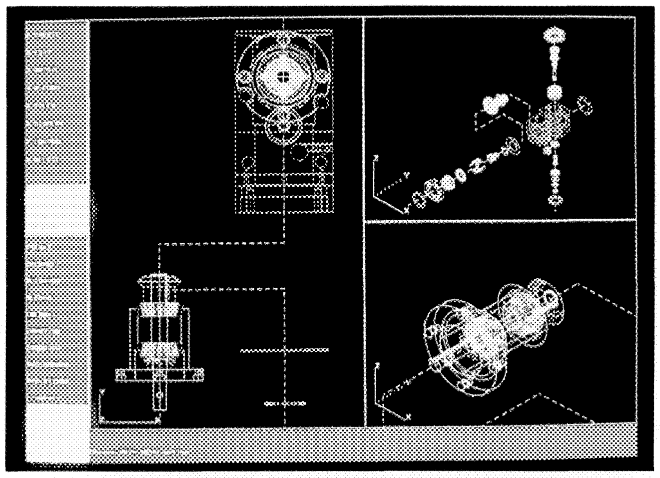


FIGURE 23.29 Exploded Mechanical Assembly

The assembly in Figure 23.31 was also created with a CAD system. Here, the traditional method of providing appropriate views of the assembly and ballooning is illustrated. The notes are in the lower left corner, the parts list is above the title block, and the revision information is at the upper right of the page.

Since a CAD system was involved, the title block, parts list format, revision block, and note format were all added to the drawing as subparts or drawings, or what some systems call **blocks** (AutoCAD). Because these company (and ANSI) standard formats are used on every drawing for this company, they are predefined formats that are retrieved and placed on any detail or assembly without reconstructing them each time a project is completed. Predefined formats are also accompanied by standard sheet sizes.

With a CAD system, hidden lines can be removed automatically from the assembly. For solid modeling systems, the combined parts can be displayed with proper visibility, as shown in Figure 23.32. Figure 23.33 shows a jig box that was designed on a 3D CAD system. The hidden line removal command (**HIDE**) was used to show visibility on this assembly model.

After the model of the assembly is completed, a designer separates the parts and files them individually. Each part may now be recalled and detailed. The model can serve to generate any views required by the engineer to detail the part properly.

23.10.1 Jigs, Fixtures, Dies, and Tooling

In order to manufacture a part, it is necessary to establish the tools. Tools are created to produce a part or product efficiently. Previously, tools were created after the part was engineered and designed. **Concurrent tooling development** involves considering tooling requirements as the part is designed, not after the fact. Jigs, fixtures, dies, and other tooling require intense design work as much as, if not more than, the part itself.

Changes in production methods, the development of new manufacturing products, and, in particular, the perfecting of economical production methods involve many design problems with tooling, dies, jigs, and fixtures. Making drawings requires special experience, and is done either by a company division maintained for that purpose or by independent tool specialists.

In general, a **tool** is a piece of equipment that helps create a finished part. It may be anything that must be designed and/or made in order to manufacture the part. The following is a list of tools found in industry.

Molds Used to form a variety of parts for consumer, industrial, and medical applications (Fig. 23.34)

Dies Used to forge, cast, extrude, and stamp materials in various physical states (solid through fluid)

Tooling The individual component of a mold or a die; might include a cavity, nest, core, punch, bushing, slide, or sleeve

Fixtures Used to hold and locate parts of assemblies during machining or other manufacturing operations (Fig. 23.35). The accuracy of the product being produced determines the precision with which the fixture is designed. Figure 23.36 shows a fixture for holding a part while machining.

To design and manufacture a finished part efficiently, product design engineers work with tool and fixture designers as well as manufacturing engineers. CAD systems promote this interaction by providing a common source of information for the product design and the associated tool/fixture design, manufacture, and production. When designing a tool or fixture, the 3D model of the part to be produced is retrieved to determine how the tool or fixture should be built to produce the finished product. The tool or fixture is then designed directly on the system (Fig. 23.35).

Since duplicating the design for CAM-related purposes is eliminated, time is saved. This also helps eliminate errors caused by misinterpreting design information. A CAD system handles large amounts of information that the engineer or designer needs to determine complex relationships between the tool/fixture and the part. The visual representation of the tool/fixture on the display, as it relates to the part, provides an important link between engineering and manufacturing. This eliminates the tedious work of interpreting the detailed part drawing and then manually calculating individual fits and tolerances of the tool or fixture required to produce the finished part. If product design changes are necessary, the tool/fixture design is modified and updated.

Once the tool/fixture is designed, a detail drawing to provide a geometric description of each part of the fixture is prepared. Detail drawings, an integral part of all steps in the design-through-manufacturing process, can represent the tool/fixture design in any view and include dimensions, surfaces, hidden lines, and other appearance control features. These drawings may serve for marketing, design,

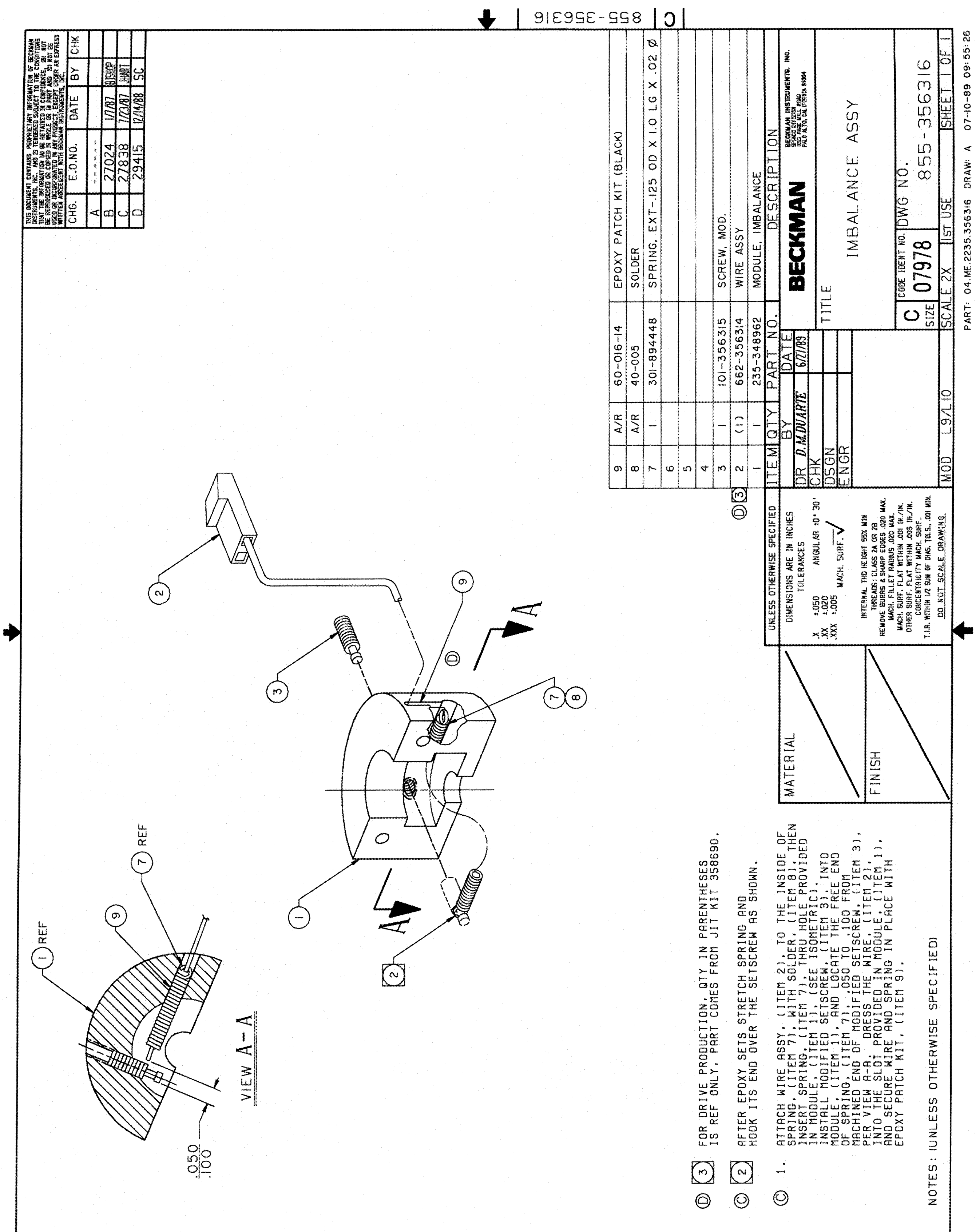


FIGURE 23.30 Imbalance Assembly

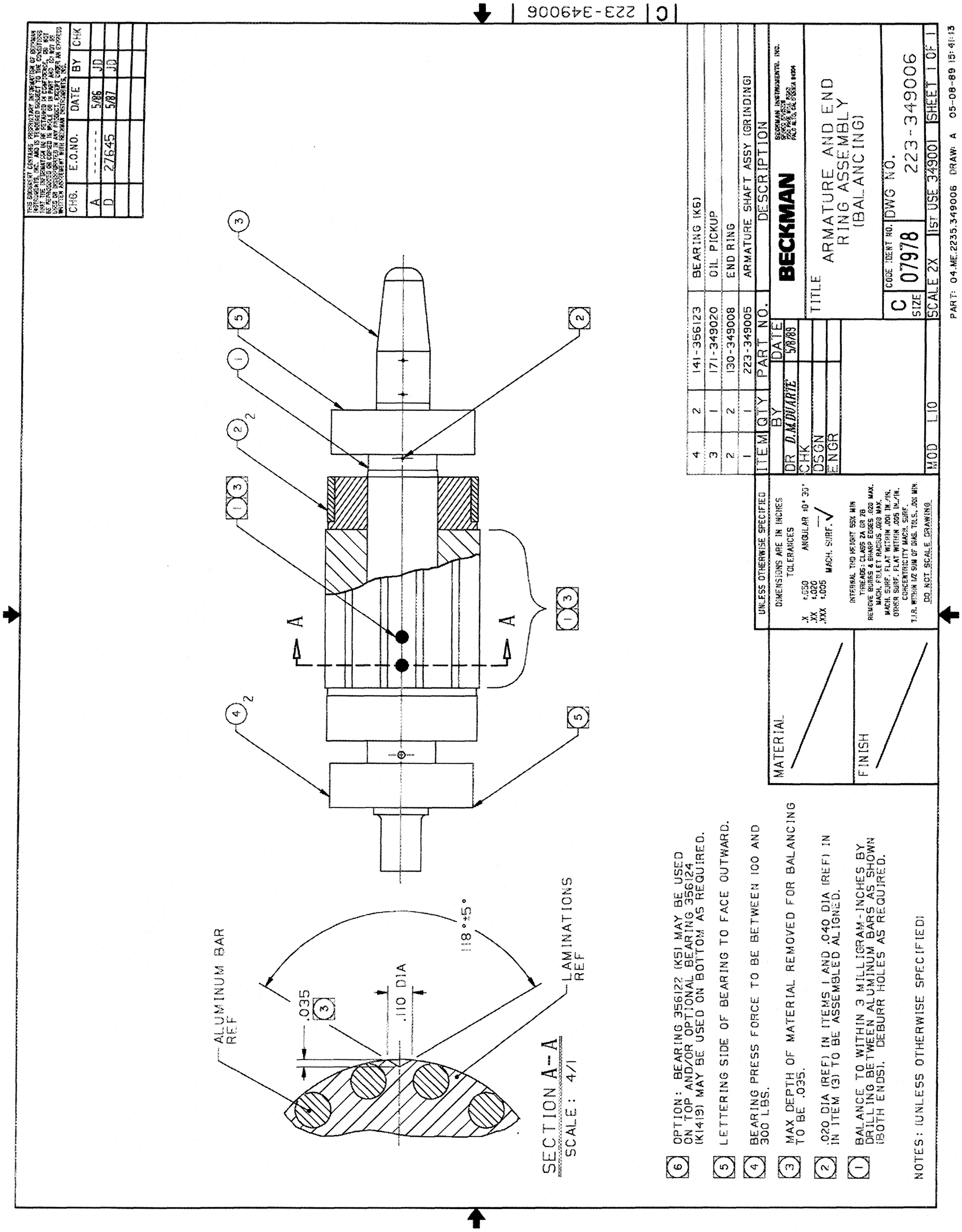


FIGURE 23.31 Armature Assembly

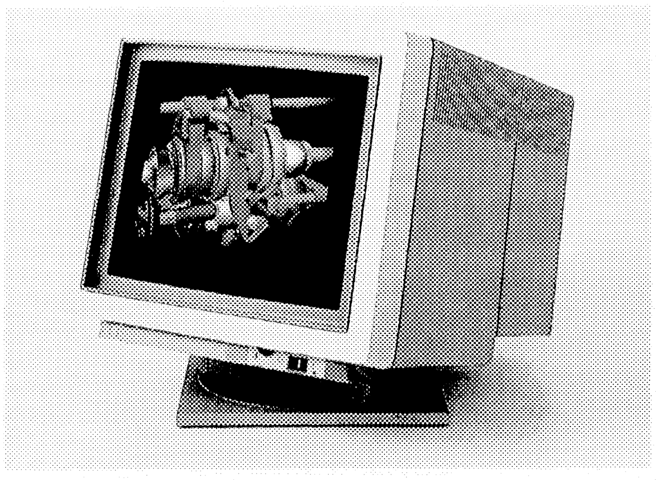


FIGURE 23.32 Solid Model Assembly

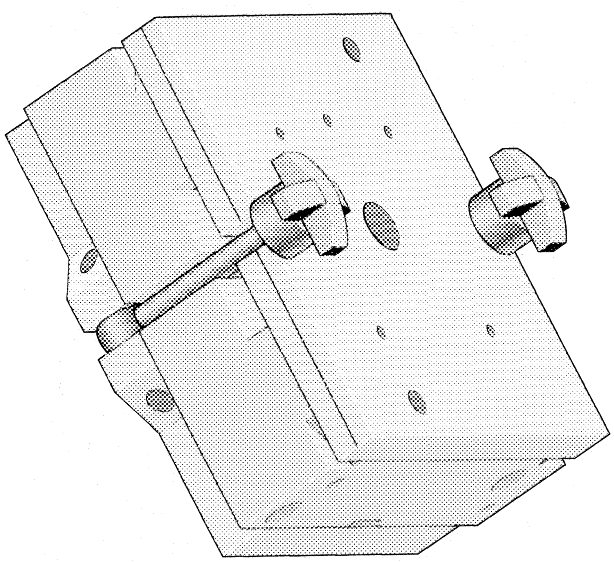


FIGURE 23.33 Jig Box Assembly with Hidden Lines Removed

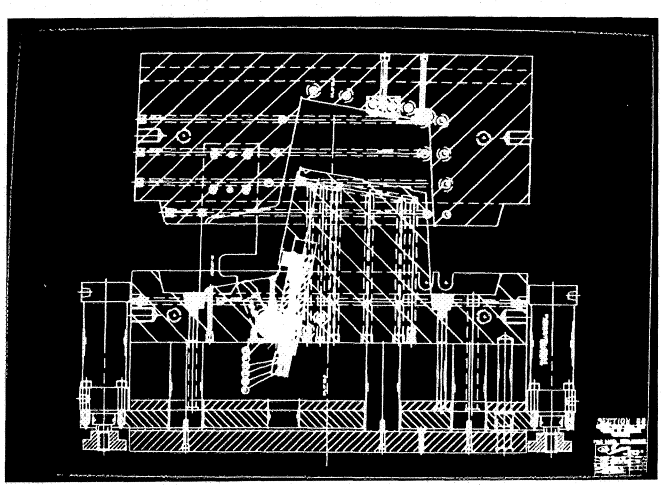


FIGURE 23.34 Mold Design Using a CAD System

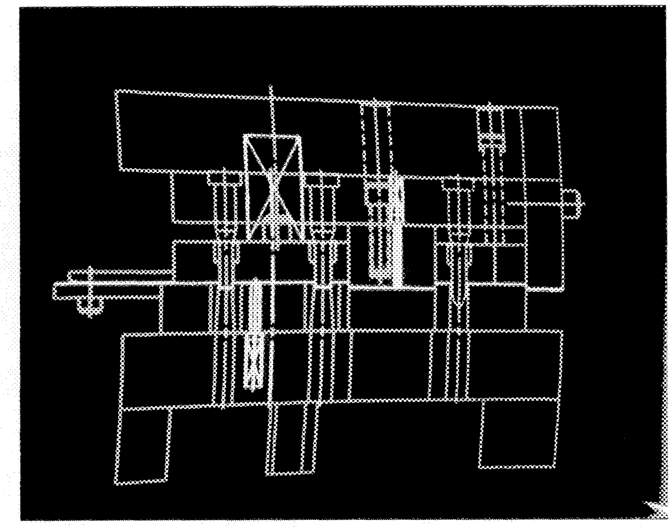


FIGURE 23.35 Tooling Fixture Designed on a CAD System

review, and manufacturing approval, or as input to the documentation, purchasing, and production departments.

To list the parts and materials needed to produce a tool/fixture designed on the system, CAD systems can automatically output a bill of materials for the planning and purchasing departments.

23.10.2 Visual Simulation to Verify Tool and Fixture Design

To provide clearances between the tool/fixture and the part, the designer can check and verify the minimum clearance needed by enlarging the view on the graphics display with the **ZOOM** and **MEASURE** commands. This ensures optimum use of materials. Color can help discriminate components for ease of viewing and highlighting.

To evaluate a tool design, a CNC programmer generates a toolpath (cutter path) and visually simulates the movement of a cutting tool around a part. The simulated cutter moves on the display screen to verify the result of the toolpath definition. The toolpath is plotted on a drawing along with the part and the fixture. This reduces costs for test machining, machine setup, and prototype creation by eliminating reruns. Process planners can then use this information to create process instructions and plans for the same part (Fig. 23.36).

23.10.3 Tool Design Using Standard Library Parts and CAD

Fixture design is an important application for computer-aided design. Tool designers can often benefit more than anyone from CAD/CAM. Since part size and shape are key factors in deciding how to locate and clamp, one obvious advantage of a CAD system for tool design is that workpiece geometry is already stored on the system. The ability to use

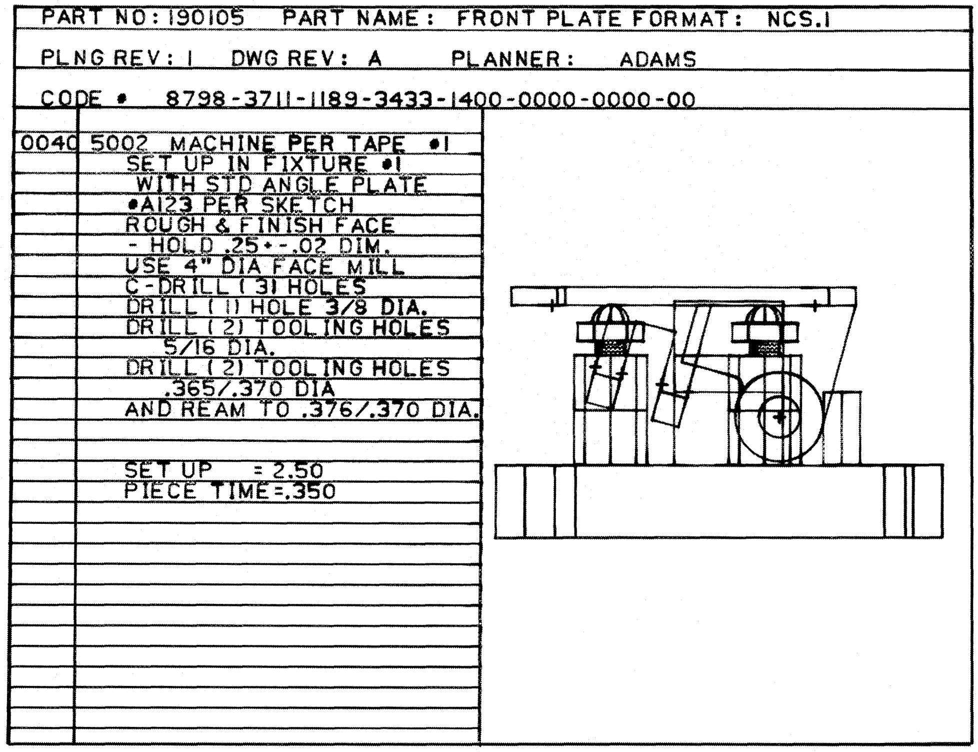


FIGURE 23.36 Planning Machine Operations on a CAD System

and reuse a **standard library** of tooling components, made up of standard components plus in-company standards, is important to fixture layout and design.

Figure 23.37 demonstrates the steps taken to design a fixture out of standard components from a tooling library of parts. In the following example, CARR LANE components are used. The CARR LANE library of parts has more than 5000 standard items available in 2D and 3D versions.

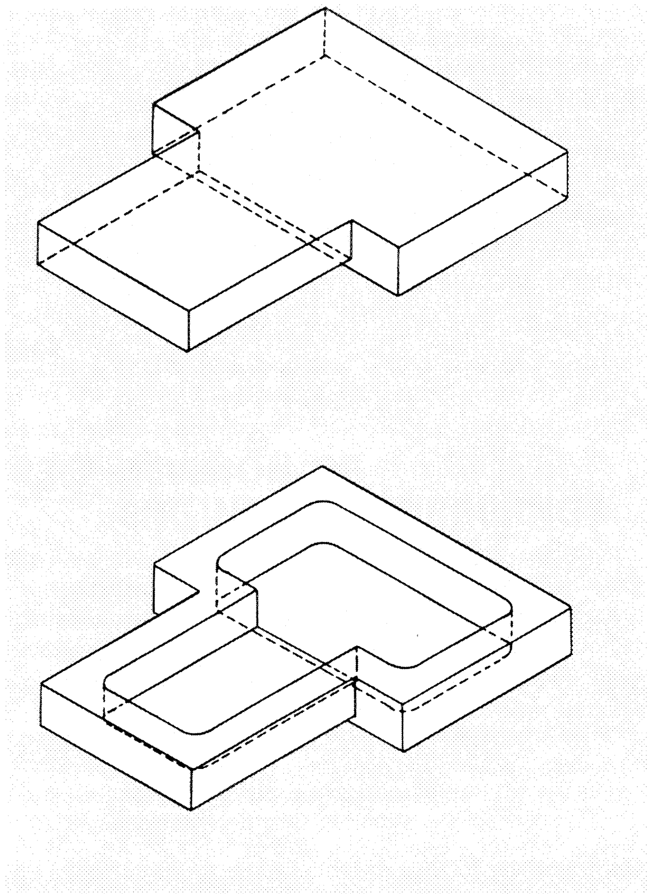
CARR LANE provides schools with the library of parts free of charge. CARR LANE supports micro-based CAD systems, including AutoCAD, CADKEY, VersaCad, and Personal Designer, as well as workstation systems such as ComputerVision and Anvil. The following describes a set of steps in the design of a fixture from standard components already modeled on the CAD system and saved as a parts library.

1. Workpiece Geometry [Fig. 23.37(a)] Part geometry, before and after machining, is the key to deciding how to hold a part. Our sample part will be pocket milled, then drilled. When parts are designed on a CAD system and represented graphically, there is an extra advantage. After retrieving the workpiece, drawing layers containing unneeded dimensions and notes are turned off. Parts are normally shown in phantom lines and/or different colors. Parts are not a component of the fixture itself. Before proceeding further, tool sizes and cutting forces must be considered.

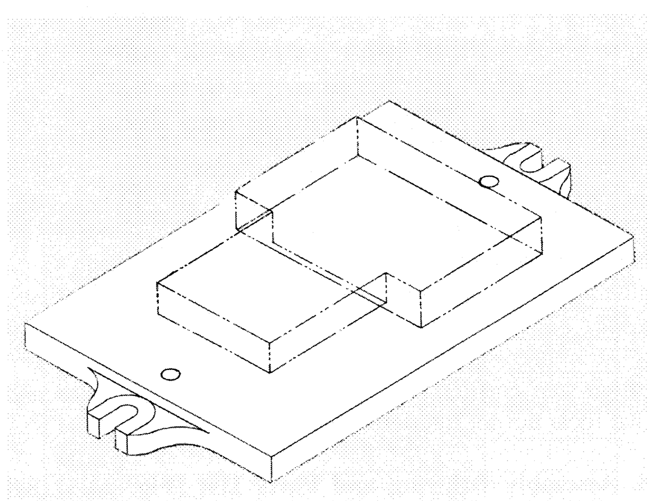
2. Fixture Base [Fig. 23.37(b)] Our example involves a standard, off-the-shelf *mill fixture base*. Whether you use a premachined base or make your own tooling plate, it must be big enough to hold the workpiece and any clamps you may need. Usually, you should allow some gap under the part for supports. The base must be thick enough to resist distortion from clamping forces and vibration from cutting forces. Standard cast angles, cubes, and other sections are also available off-the-shelf, and easily can be modified for fixtures. For cylindrical parts, the "fixture base" may be a set of blank chuck jaws, custom-shaped to fit the part.

3. Supports [Fig. 23.37(c)] For proper part location, designers commonly pick multiple fixed support points and occasionally some adjustable supports. In this example, three *rest buttons*, all underneath the workpiece, are the fixed supports. Geometric tolerancing considerations may dictate where the rest buttons are placed. Although you would usually place the supports by eye, you could also rely on your CAD system's center-of-mass function for heavy or complicated parts to distribute weight evenly. If the parts are thin and require additional backup against machining forces, you could add adjustable supports (hydraulically operated if you want the greatest accuracy).

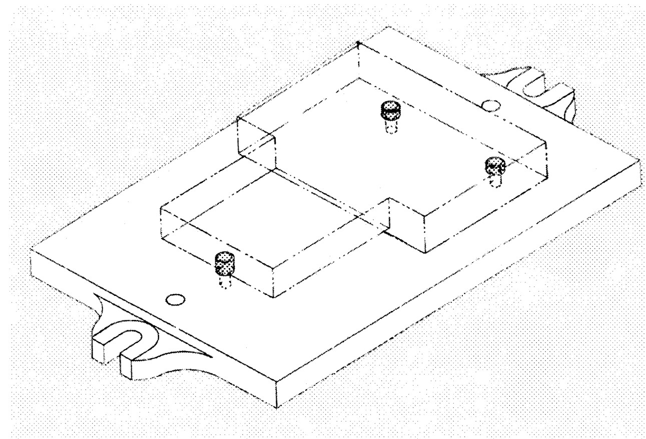
4. Locators [Fig. 23.37(d)] Next, we locate our part laterally. In the example, we bring in three *round pins*, tangent to the part's side reference surfaces, to orient it correctly. We choose locators strong enough to resist anticipated lateral cutting forces. Three *spring stop buttons* hold the workpiece firmly against the round pins while we clamp.



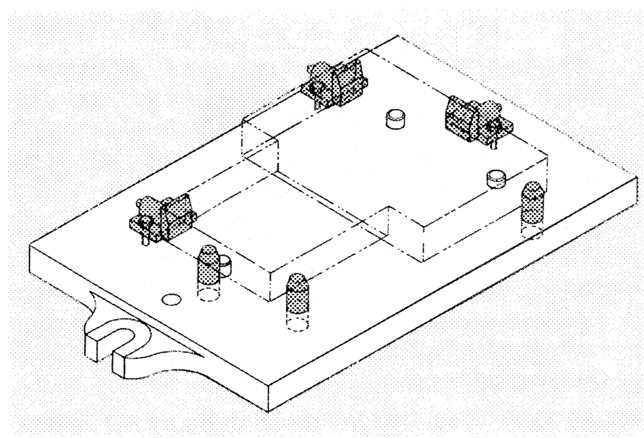
(a) Workpiece geometry



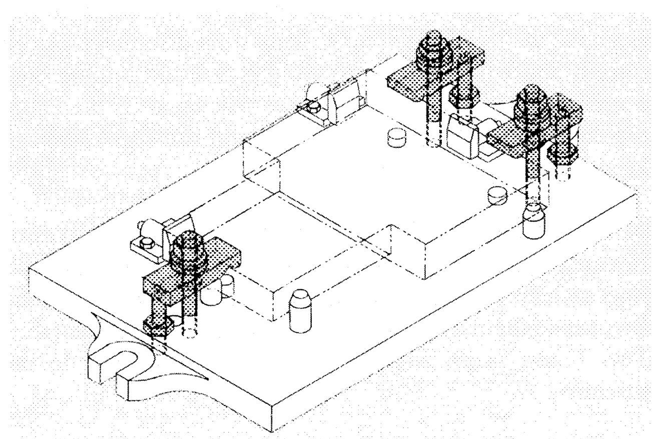
(b) Fixture base



(c) Supports

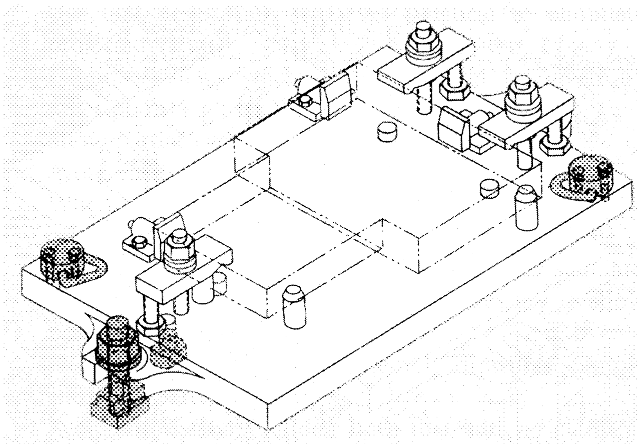


(d) Locators

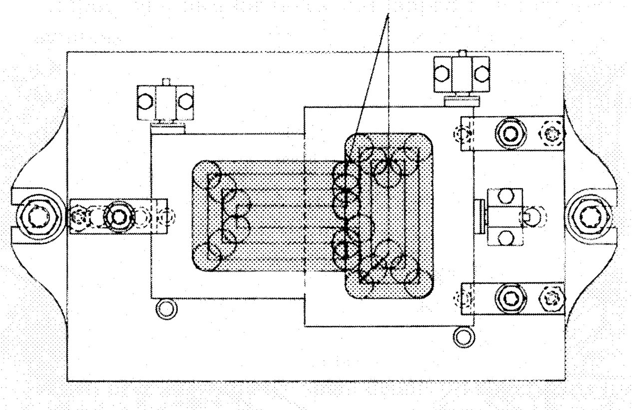


(e) Clamps

FIGURE 23.37 Tool Design Process Using a CAD System and a Component Library of Parts



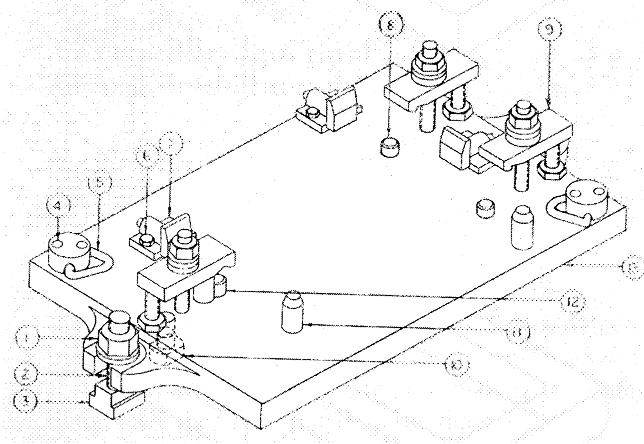
(f) Fixture accessories



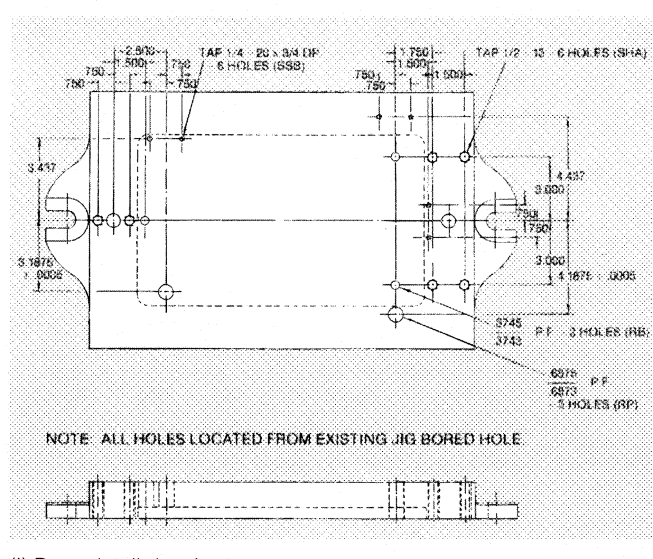
(g) Checking tool clearance

FIGURE 23.37 Tool Design Process Using a CAD System and a Component Library of Parts—*Continued*

ITEM	QTY	PART NO	DESCRIPTION
1	2	CL-4-FN	FLANGE NUT
2	2	CL-5/8-11X2.50	STUD
3	2	CL-10-TN	T-NUT
4	4	46233	SOCKET HEAD CAP SCREW
5	2	CL-20-HR	HOIST RING
6	6	S3896	HEX HEAD CAP SCREW
7	3	CL-20C-SSB	SPRING STOP BUTTON
8	2	CL-6-RB	REST BUTTON
9	3	CL-12-SHA-1	CLAMP STRAP ASSEMBLY
10	2	CL-687-SLFK	FIXTURE KEY
11	3	CL-5-RP	ROUND PINS
12	1	CL-7-RB	REST BUTTON
13	1	CL-4-MFB	MILL FIXTURE BASE



(h) Assembly drawing and parts list



(i) Base detail drawing

5. Clamps [Fig. 23.37(e)] Cutter forces are considered when choosing the type, size, and number of clamps. In this example, we use three simple *clamp strap assemblies,* placed directly above the supports to avoid part distortion. We keep our clamps clear of the areas to be machined. If the entire top surface were to be milled, we could use *edge clamps* to keep the top totally clear.

6. Fixture Accessories [Fig. 23.37(f)] Fixture accessories are the final components we select. We choose two *hoist rings* to help lift our fixture into place. Two *sure-lock fixture keys* locate the fixture base accurately on the machine table. Two *T-nuts*, studs, and *flange nuts* fasten the base to the machine.

7. Checking Tool Clearance [Fig. 23.37(g)] After tentatively placing all fixture components, you can check tool clearance. Using the CNC graphics capabilities of your CAD/CAM system, you can determine tool paths, then, ultimately, generate CNC programs. This machining verification step ensures that the part is entirely machined and that tools do not interfere with the fixture.

8. Assembly Drawing and Parts List [Fig. 23.37(h)] The first step in documenting the fixture design is an assembly drawing. Assembly drawings are produced easily by turning off the drawing layer containing the workpiece. You can then use your system's capabilities to add item callouts and create a bill of material.

9. Base Detail Drawing [Fig. 23.37(i)] A detail drawing of the fixture base is also easy to produce by turning off the drawing layers containing other components. Here, two views, top and front, were selected to add machining dimensions. As a final step, we could generate a CNC program to machine the base.

You May Complete Exercises 23.1 Through 23.4 at This Time

QUIZ

True or False

1. The revision block is one of the most important things you should look at when reading an existing drawing for the first time.
2. Dimensions are given on all assemblies to locate and define the parts geometry.
3. Sections are used on assemblies to provide a convenient way of displaying clearly the unit's geometry.
4. A parts list and a bill of materials are two distinct and different aspects of an assembly drawing.
5. Since they will normally differ for each project, general notes are seldom standardized.
6. Hidden lines are shown whenever possible on the assembly drawing to clarify each part's geometry.
7. A parts list that is on the assembly drawing is listed from the top down and is placed below the revision block.
8. Ballooning is the process of calling out each part of an assembly by providing a circle attached to a leader that points to the piece on the drawing.

Fill in the Blanks

9. __________ provides prints of a drawing that have white lines with a blue background.
10. __________ are used to hold and locate a part during machining operations.
11. Layout drawings help to establish and depict __________, __________, and the relationship of __________.
12. __________ is the process of taking individual pieces of an assembly and redrawing them with sufficient views and __________.
13. Working drawings serve to order __________, plan operations, determine __________, and assemble the unit.
14. Welded assemblies are considered __________ assemblies.
15. Assemblies are fastened by one or more of the processes including __________, __________, __________, __________, and __________.
16. __________ on assemblies are provided for setup, assembly, and clearance.

Answer the Following

17. What are jigs, fixtures, and dies?
18. Why are notes required for most drawings? What is the difference between local and general notes?
19. What is the function of the assembly drawing?
20. Explain the difference between the concept of bottom-up design versus the top-down design approach.
21. Describe a parts list and each of its major headings.
22. How are views selected for an assembly drawing?
23. Explain the difference between a separable and an inseparable assembly.
24. Name three ways that you could simplify a drawing.

EXERCISES

Exercises may be assigned as sketching, instrument, or CAD projects. Transfer the given information to an "A"-size sheet of .25 in. grid paper. Exercises that are not assigned by the instructor can be sketched in the text to provide practice and to enhance understanding of the preceding instructional material. Draw the drawing format, title block, and other standard information for the exercise. If using AutoCAD or another system that provides standard formats, use them instead of the one provided here or in the worksheets.

After Reading the Chapter Through Section 23.11.1, You May Complete the Following Exercises

Exercise 23.1 Do a complete parts list for Problem 23.27 or Problem 23.22.

Exercise 23.2 Do a complete parts list for Problem 23.28 or Problem 23.23.

Exercise 23.3 Do a complete parts list for Problem 23.29 or Problem 23.24.

Exercise 23.4 Do a complete parts list for Problem 23.30 or Problem 23.25.

ITEM	QTY	PART NO.	DESCRIPTION

EXERCISE 23.1

ITEM	QTY	PART NO.	DESCRIPTION

EXERCISE 23.2

ITEM	QTY	PART NO.	DESCRIPTION

EXERCISE 23.3

ITEM	QTY	PART NO.	DESCRIPTION

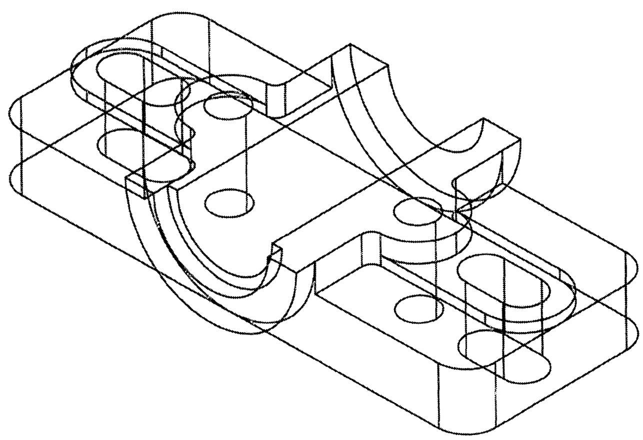
EXERCISE 23.4

PROBLEMS

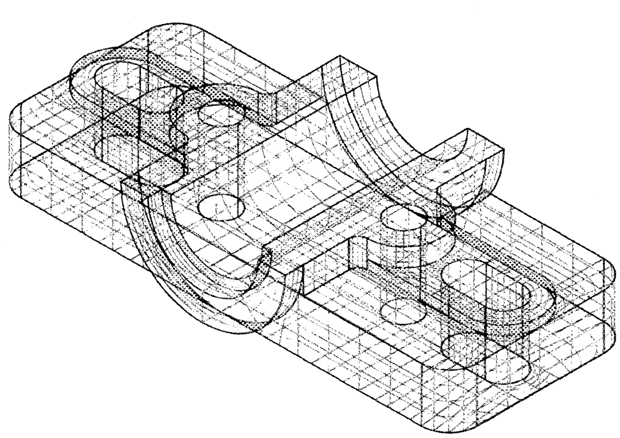
Jig and Fixture Assembly Design Projects

Design a jig and fixture to hold the following parts while machining. Do a layout of the fixture, a finalized assembly, a parts list, and details of each nonstandard item in the assembly. Show the workpiece part in phantom lines, or plot in a second color. Use Chapters 17, Threads and Fasteners, Chapter 18, Springs, and Appendix C (and CARR LANE parts if available) for standard parts.

Figure 23.38 shows an example of a student-designed fixture. The journal bearing housing provided in Problem 23.7 was used as the part to be machined. A wireframe model was completed first [Fig. 23.38(a)]. A surface model of the part was completed next [Fig. 23.38(b)]. Figure 23.38(c) shows a dimensioned detail of the journal bearing housing. An assembly was designed from standard CARR LANE parts [Fig. 23.38(d)], and an assembled detail was completed along with a parts list, as shown in Figure 23.38(e).

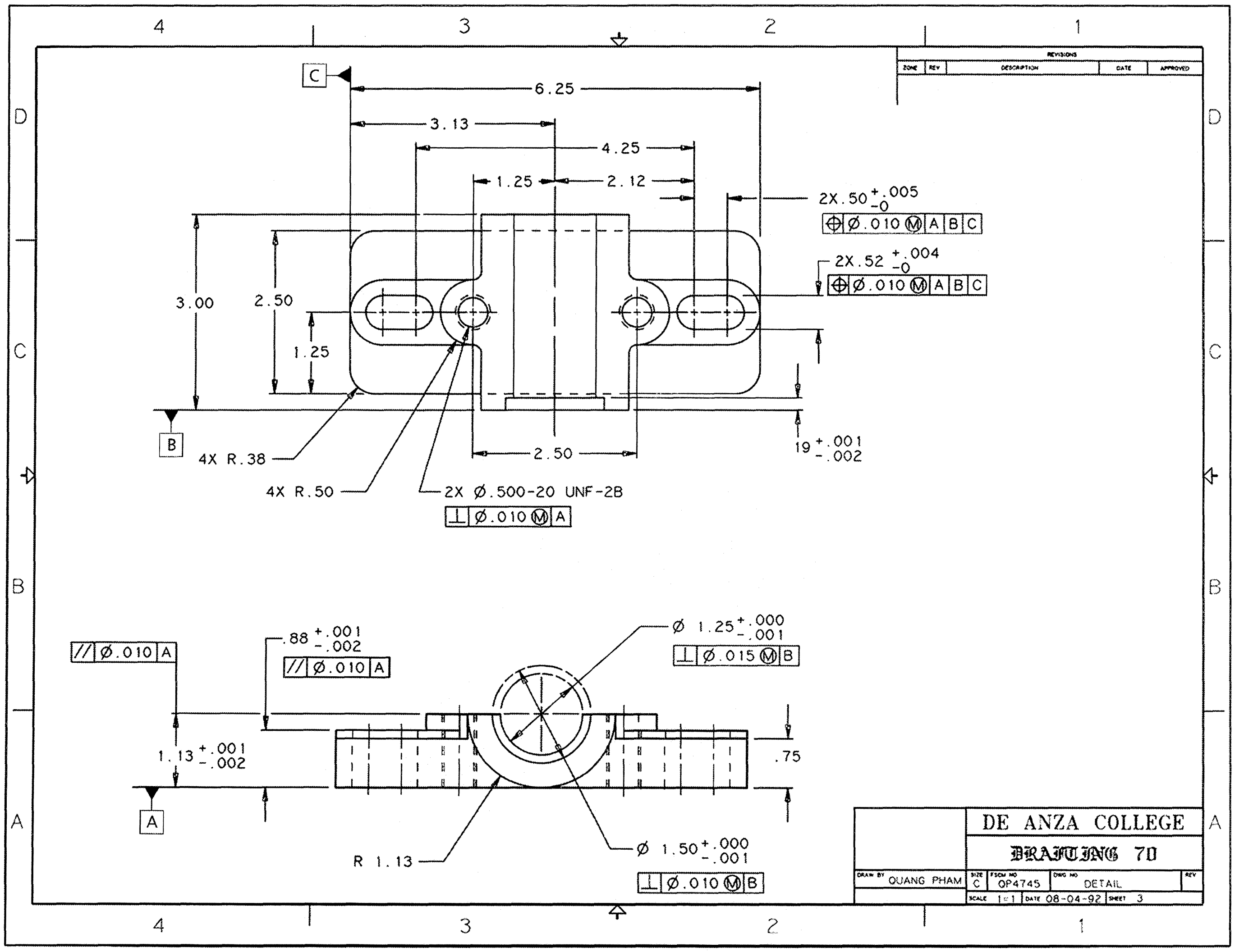


(a) Wireframe model of the journal bearing housing shown in Problem 23.7



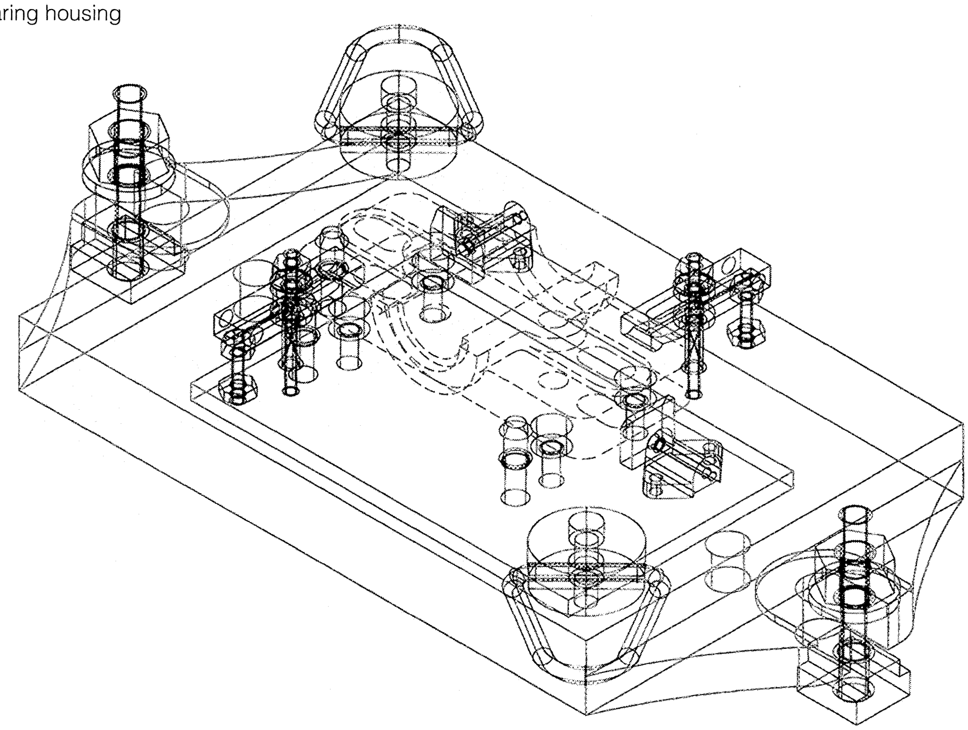
(b) Surface model of the journal bearing housing

FIGURE 23.38 Journal Bearing Fixture

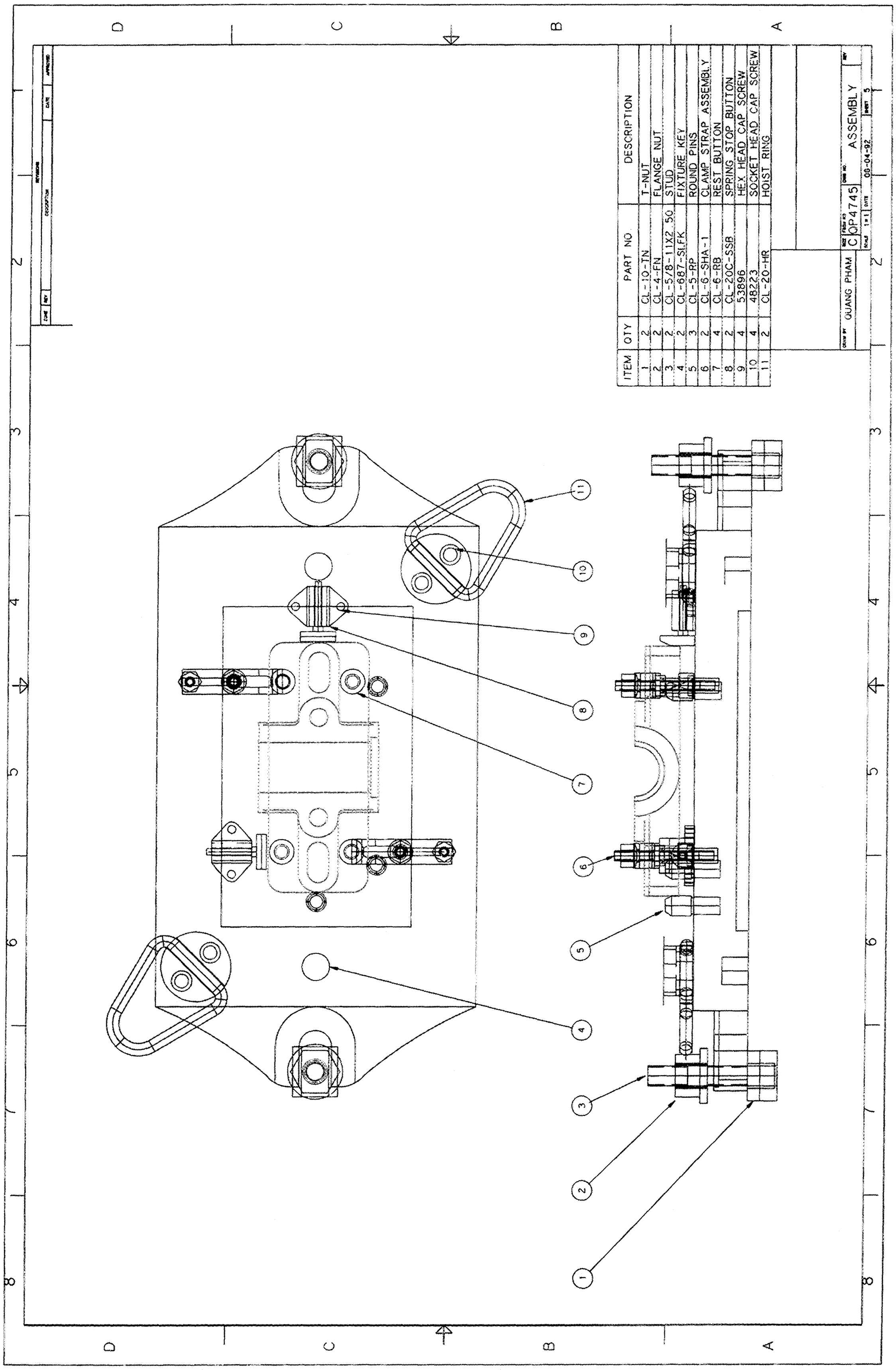


(c) Dimensioned detail of the journal bearing housing

FIGURE 23.38 Journal Bearing Fixture—*Continued*



(d) Machining fixture for the journal bearing housing



(e) Assembly drawing of the fixture for the journal bearing housing

FIGURE 23.38 Journal Bearing Fixture—*Continued*

Detail Drawings

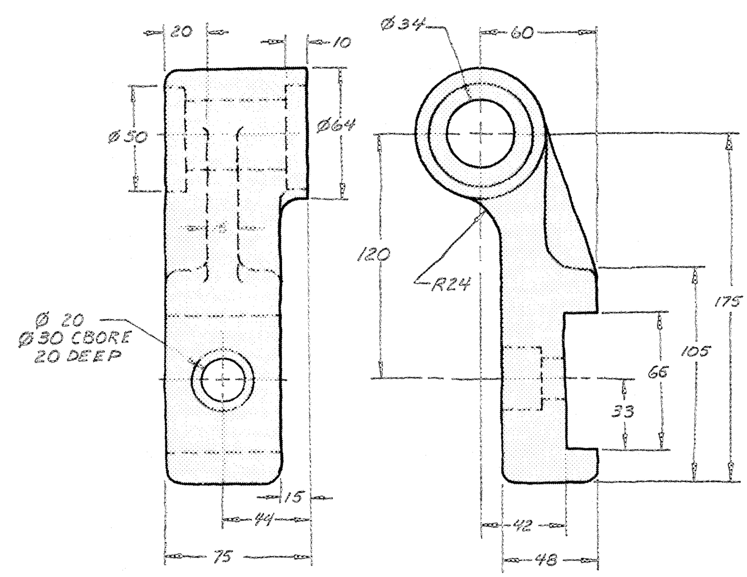
For detail drawings, use appropriate ANSI standards for dimensioning and tolerancing for all problems. Complete a rough sketch of the part before drawing it with instruments or a CAD system. Choose an appropriate-size drawing sheet for each project. If a 3D system is available, model the part first and then generate a dimensioned detail.

For all drawings, use ANSI or ISO standard sheet sizes, parts list format, revision block, and title blocks, as shown in this chapter. The dimensions given for individual parts are in most cases for construction of the part's geometry only. With the exception of a few of the projects shown in the chapter body, the problems shown here are by no means meant to represent the correct way of dimensioning the part. The given dimensions will enable you to draw the part. *It will be your responsibility to select the proper views and place the dimensions, notes, and other information on the drawing.* Show all finish marks and use symbology wherever possible.

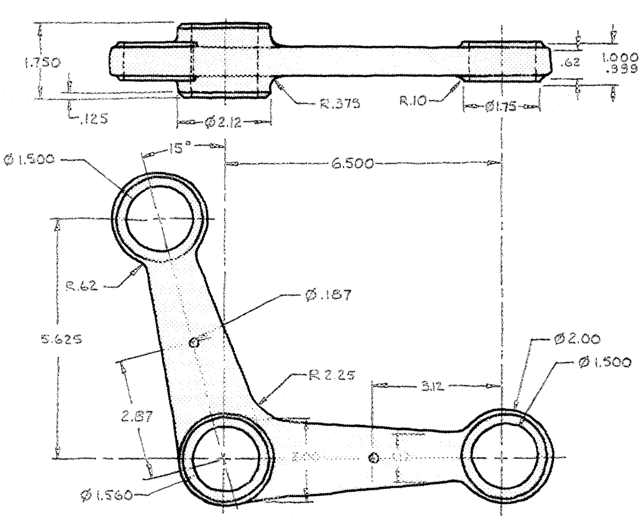
All pictorial drawings are to be converted to multiview details with appropriate view selection and proper dimensioning methods. In most cases, the drawings presented here as projects can be drawn full scale. If a project prohibits full-scale rendering, a reduced scale can be used. All projects done on a CAD system are to be modeled/drawn full scale and then plotted according to your instructor and the limitations of your plotter.

Decimal-inch projects and metric drawings are provided. You may convert any of the projects to the other measurement system. In many cases, you will find that converting the dimensions will give odd and inappropriate sizes. You may redesign any of the parts using even and logical sizes for that measurement system. For example, though 1.00 in. converts to 25.4 mm, it is acceptable to change the metric dimension to 25 mm or 24 mm. The same is true for standard parts such as screws, nuts, washers, and other off-the-shelf items; look up the closest standard size before ordering the item (placing on the parts list).

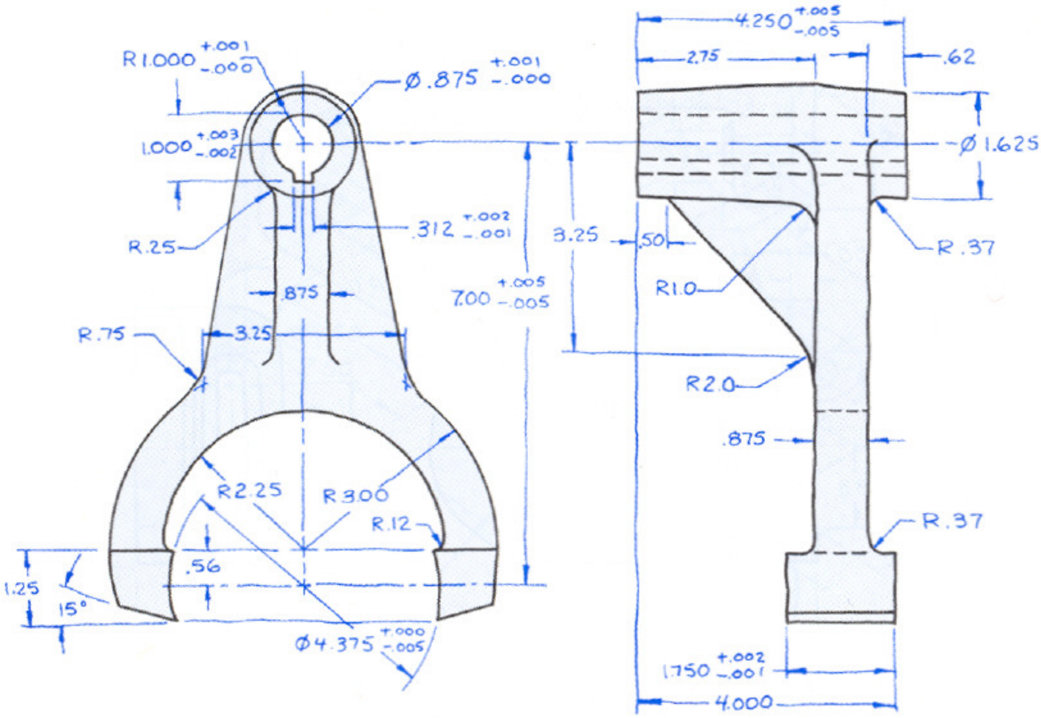
Problem 23.1 Draw and detail the adjustable guide.



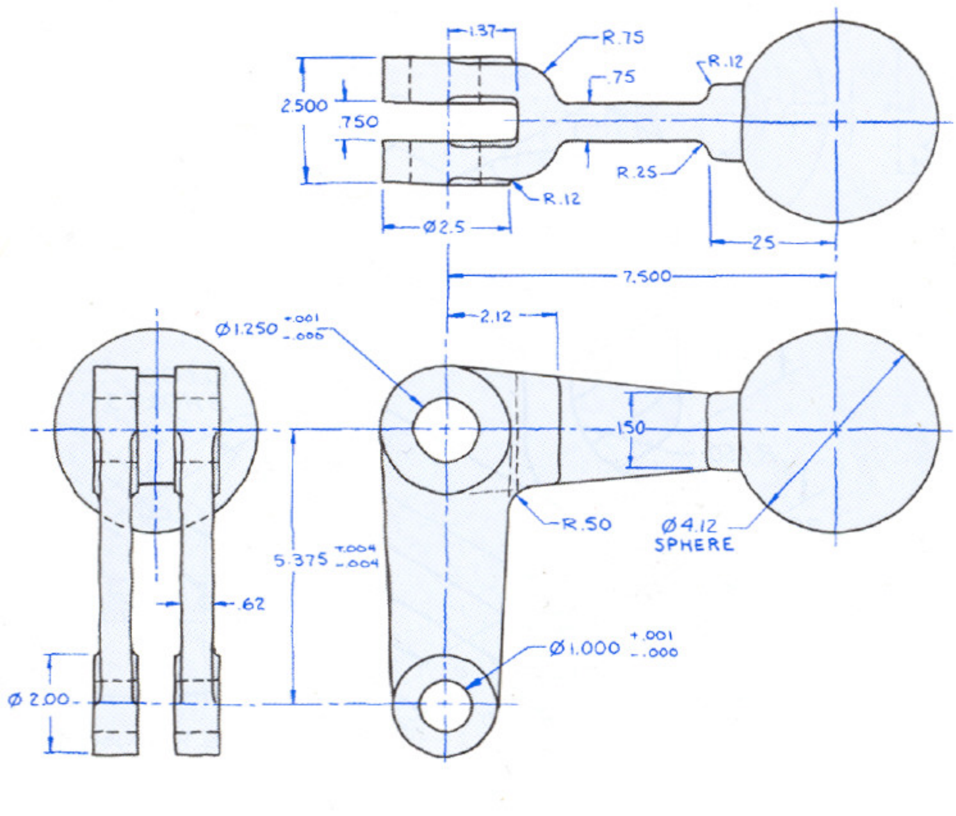
Problem 23.2 Do a detail drawing of the crank arm.



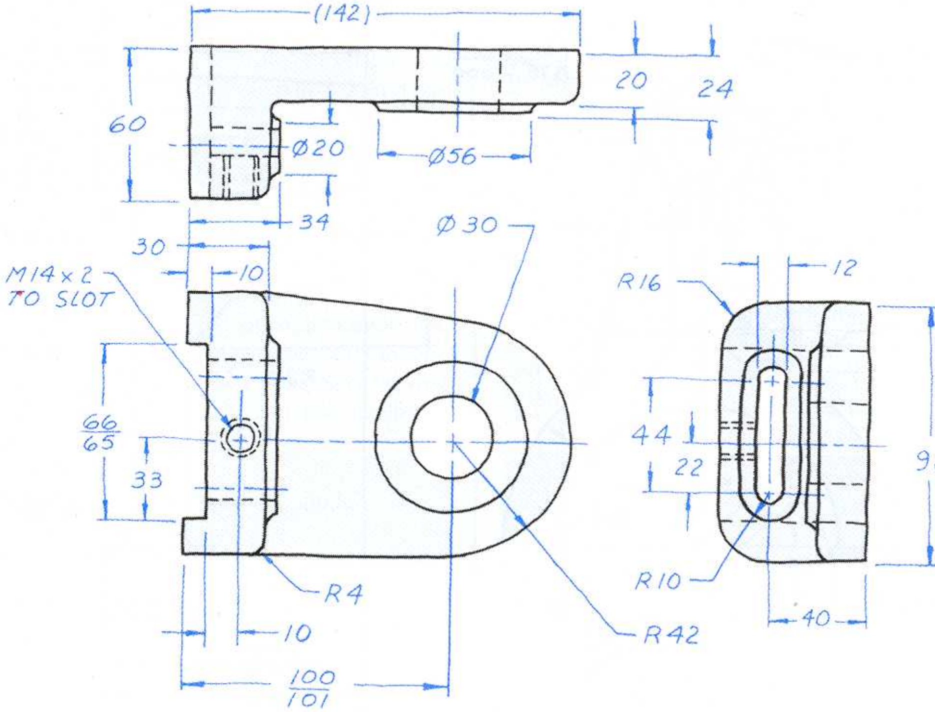
Problem 23.3 Draw and detail the shifter fork.



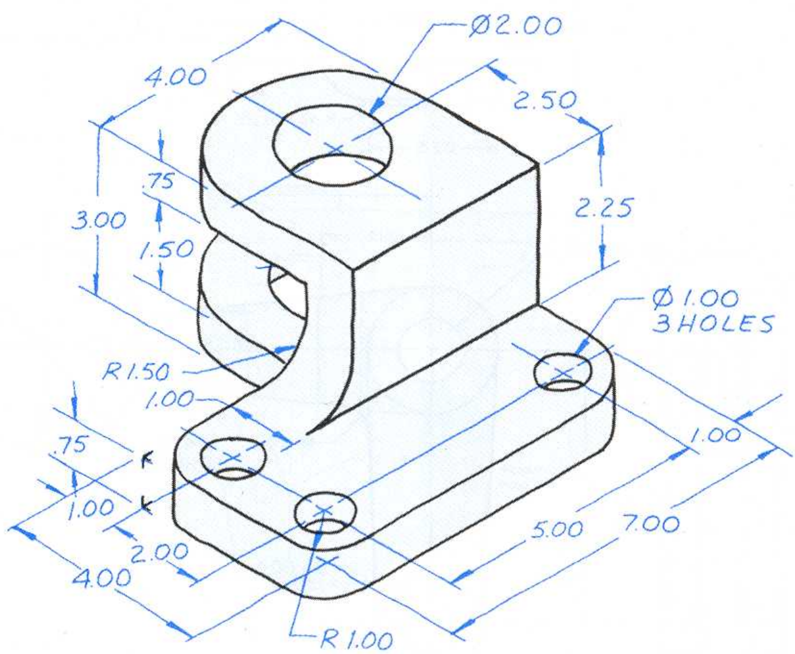
Problem 23.4 Draw and detail the flywheel.

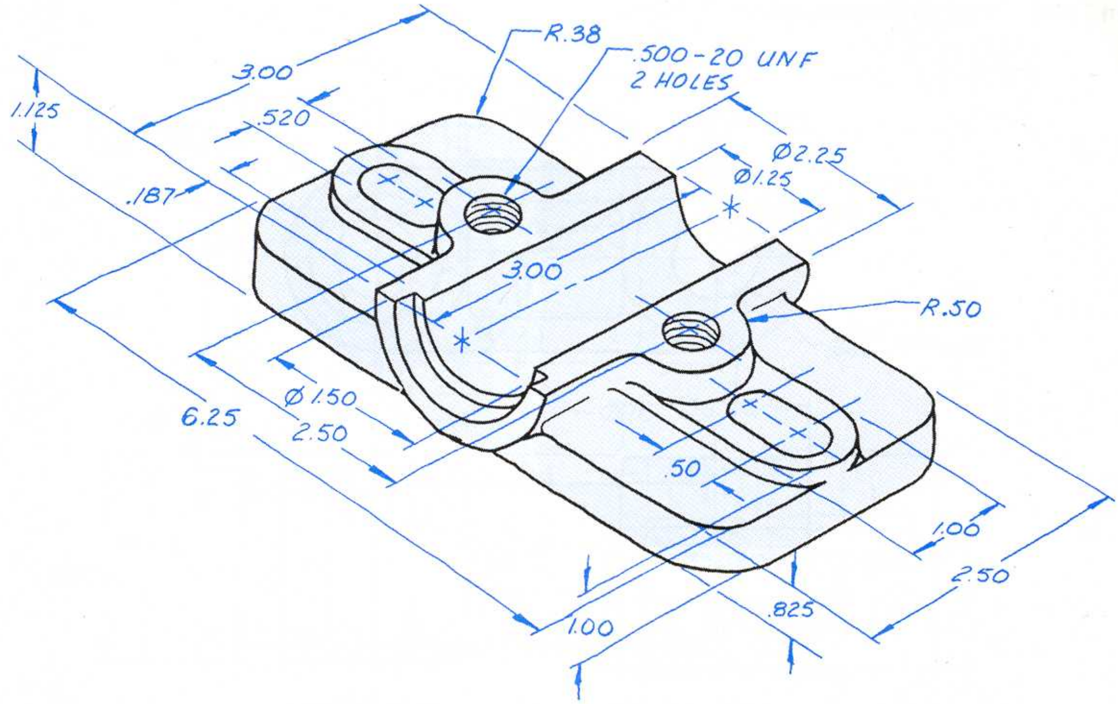


Problem 23.5 Do a detail drawing of the bearing adjustment.



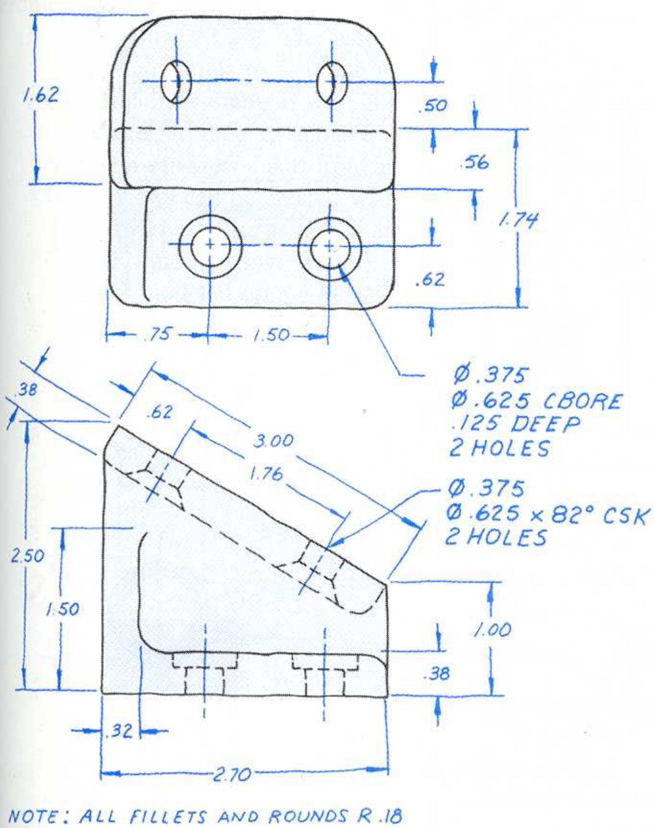
Problem 23.6 Draw and detail the guide bracket.



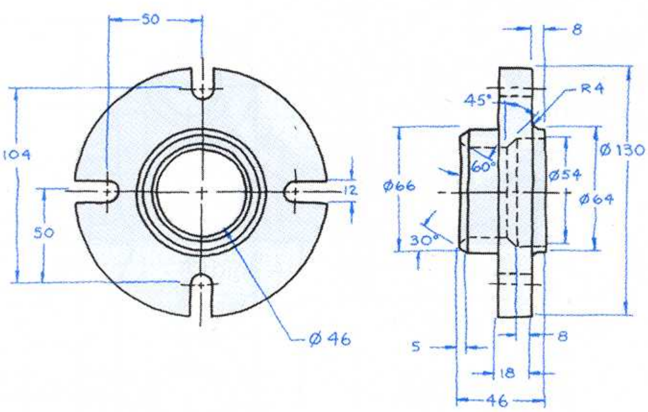


Problem 23.7 Detail the journal bearing housing.

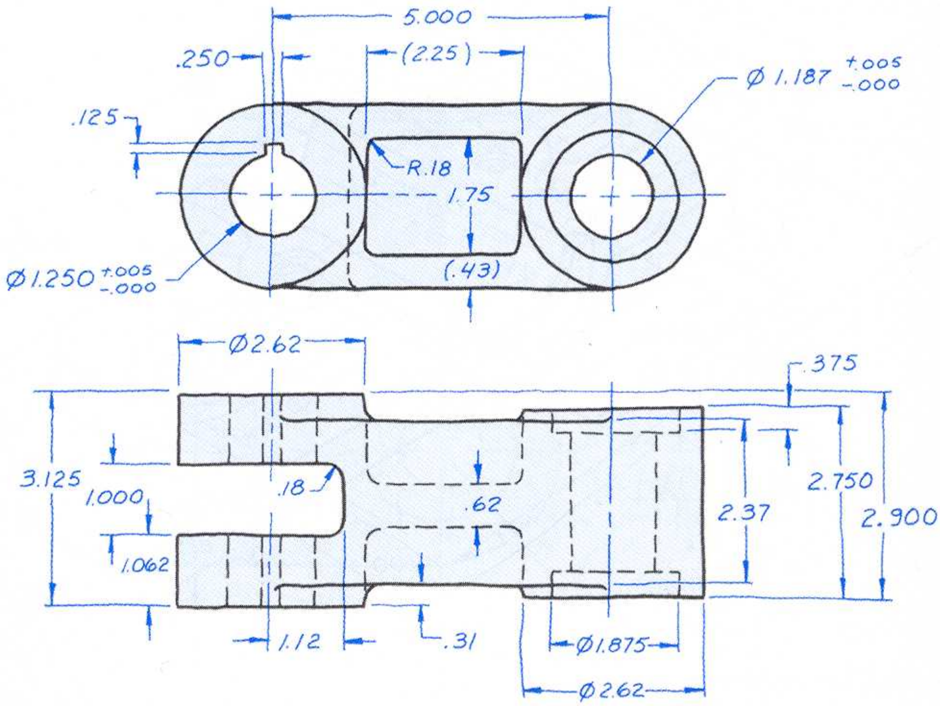
Problem 23.8 Draw and detail the offset bracket.



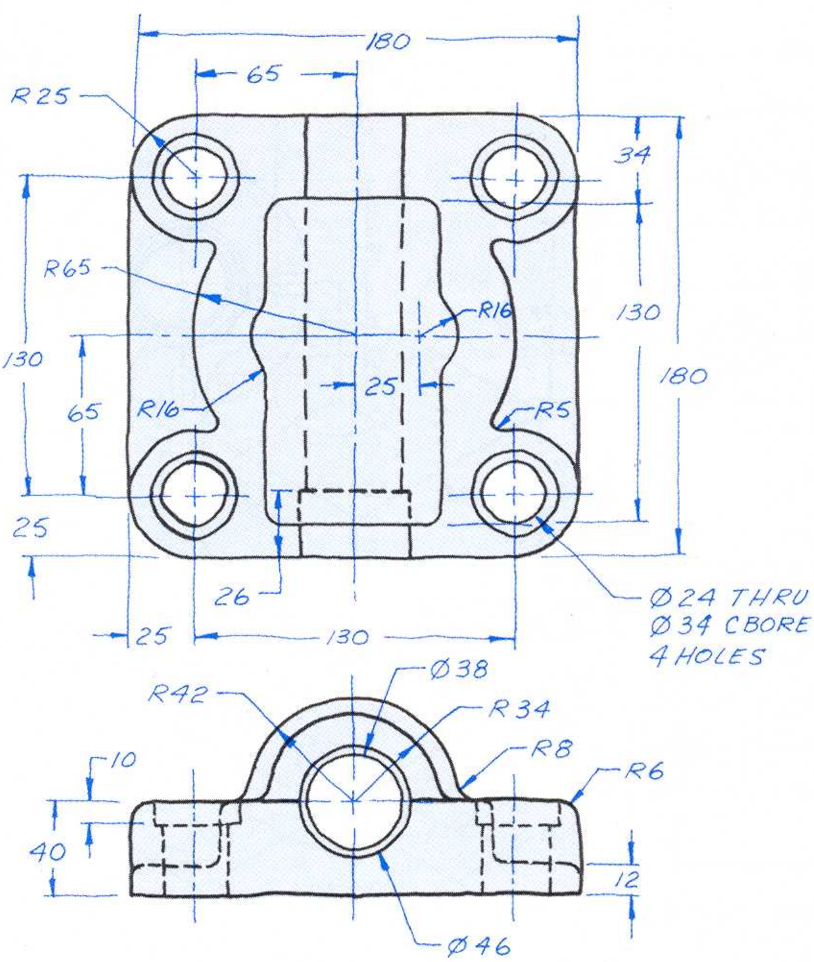
Problem 23.9 Complete a detail of the thrust bearing cap.



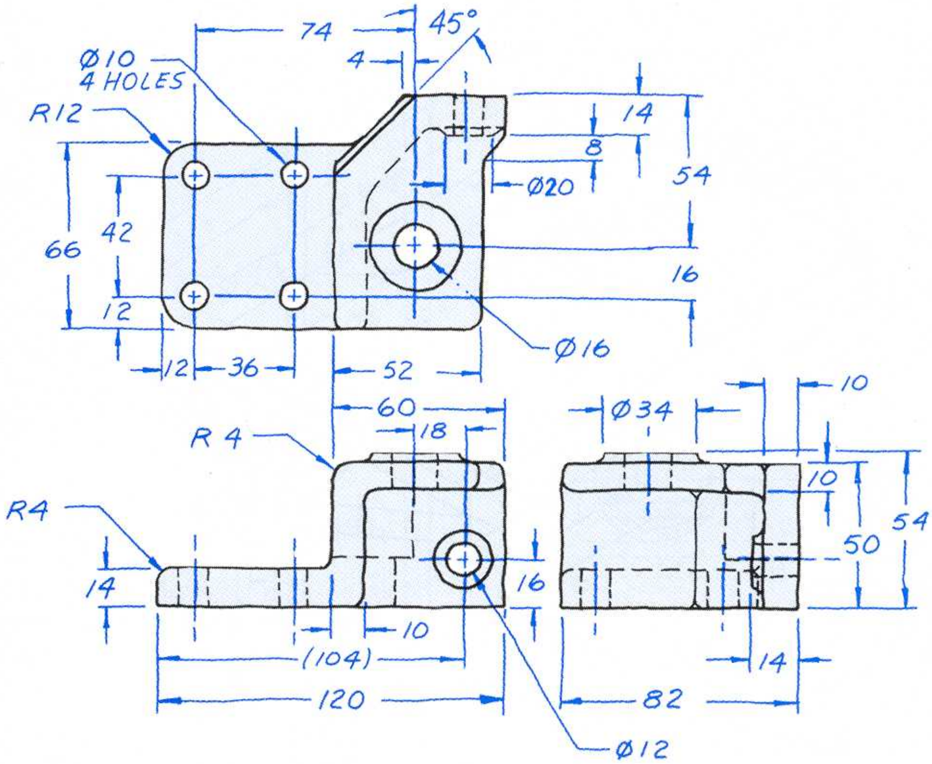
Problem 23.10 Detail the anchor bracket.



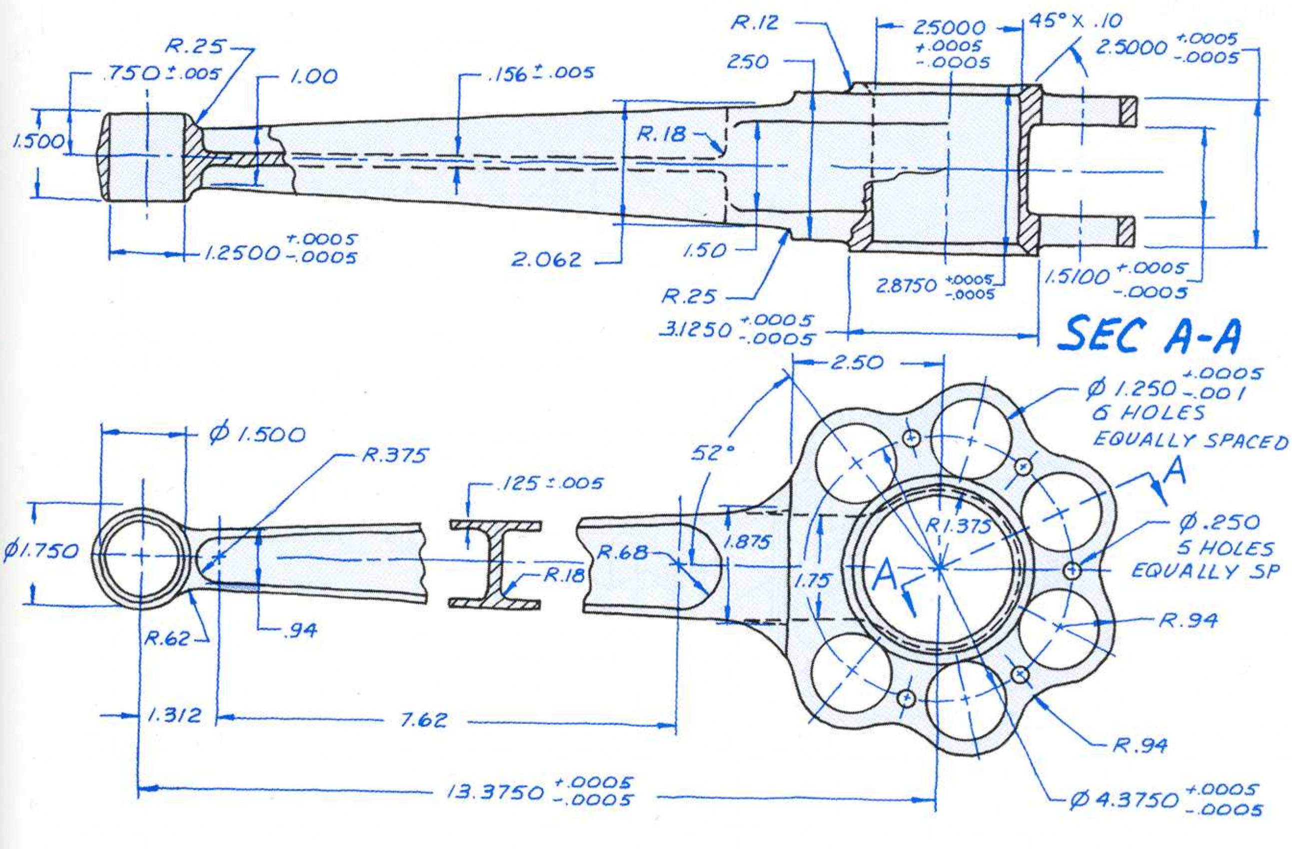
Problem 23.11 Draw and detail the bracket.



Problem 23.12 Draw and detail the part.



Problem 23.13 Draw and detail the master connecting rod.



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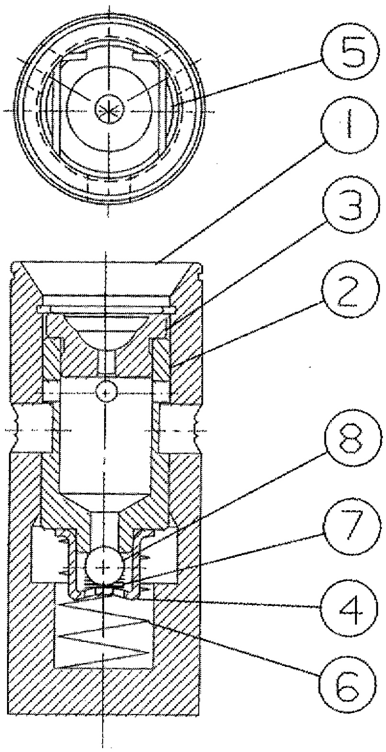
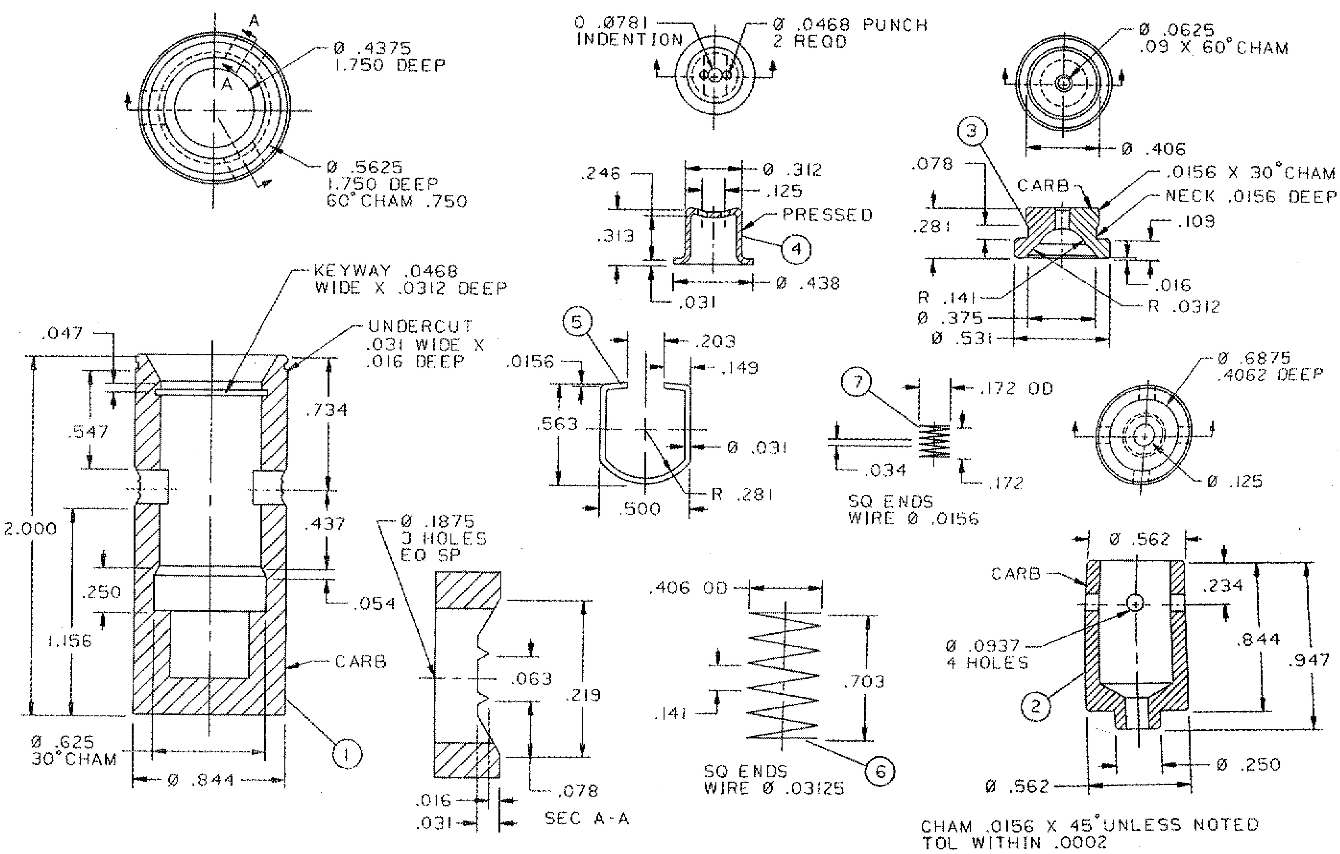
Assembly Drawing Projects

For assembly projects, prepare a layout assembly of the parts by blocking them in for each view required. Be sure to provide sufficient space on the sheet for the assembly and the parts list. The parts list can be generated on a word processor or a CAD system and printed on a separate sheet.

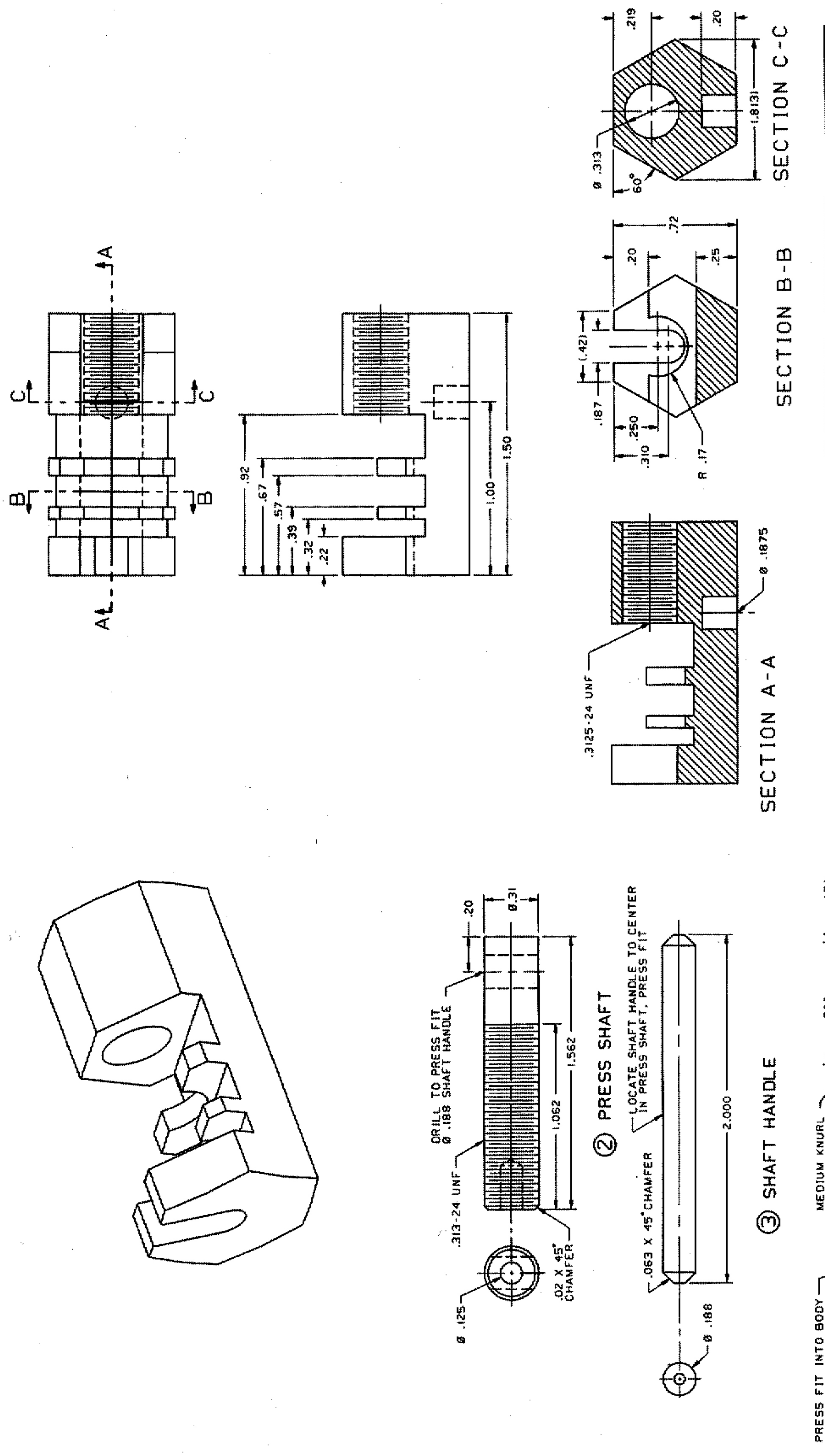
Problem 23.19(A) and (B) Do an assembly (A) and details (B) for the hydraulic valve assembly.



NO.	PART NAME	REQD	MATL
8	BALL BEARING Ø 3/16	1	STEEL
7	BALL CHECK VALVE SPRING	1	SPG STL
6	PLUNGER SPRING	1	SPG STL
5	RETAINER RING	1	SPG
4	BALL RETAINER	1	STL
3	PUSH ROD SEAT	1	STL
2	PLUNGER	1	STL
1	LIFTER BODY	1	STL



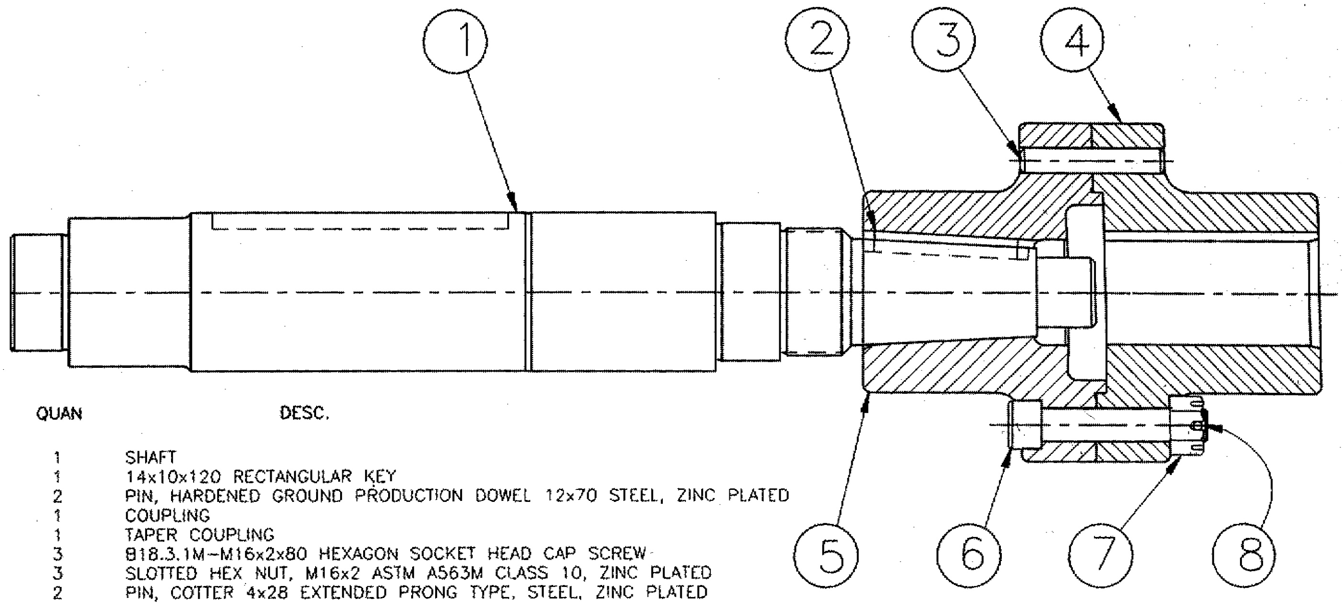
Problem 23.20 Do an assembly and details of the bike chain puller assembly.



PARTS LIST

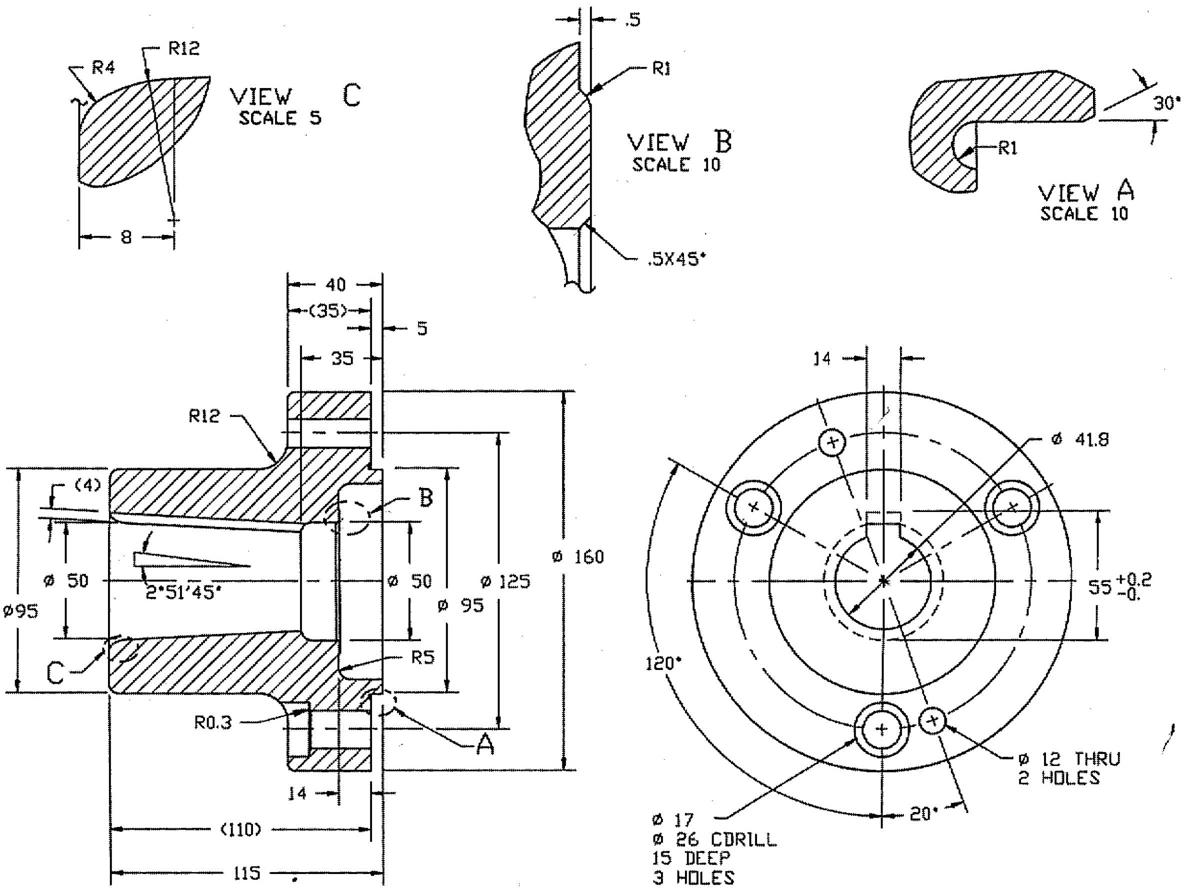
NO.	DESCRIPTION	MAT'RL	REQ'D.
5	TOOL BODY HANDLE	CRS	1
4	ROTATING PRESS POINT	CRS	1
3	SHAFT HANDLE	CRS	1
2	PRESS SHAFT	CRS	1
1	TOOL BODY	CRS	1

Problem 23.21(A) and (B) Draw and detail the assembly and details for the coupling.



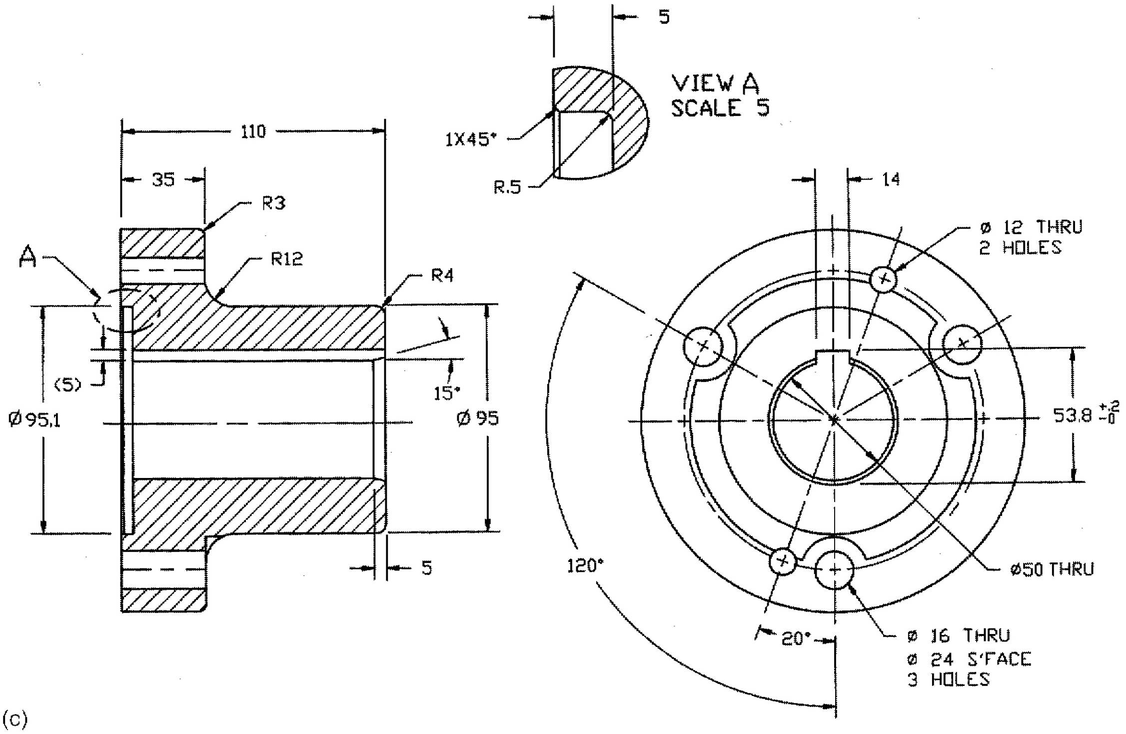
#	QUAN	DESC.
1	1	SHAFT
2	1	14x10x120 RECTANGULAR KEY
3	2	PIN, HARDENED GROUND PRODUCTION DOWEL 12x70 STEEL, ZINC PLATED
4	1	COUPLING
5	1	TAPER COUPLING
6	3	B18.3.1M–M16x2x80 HEXAGON SOCKET HEAD CAP SCREW
7	3	SLOTTED HEX NUT, M16x2 ASTM A563M CLASS 10, ZINC PLATED
8	2	PIN, COTTER 4x28 EXTENDED PRONG TYPE, STEEL, ZINC PLATED

(a)

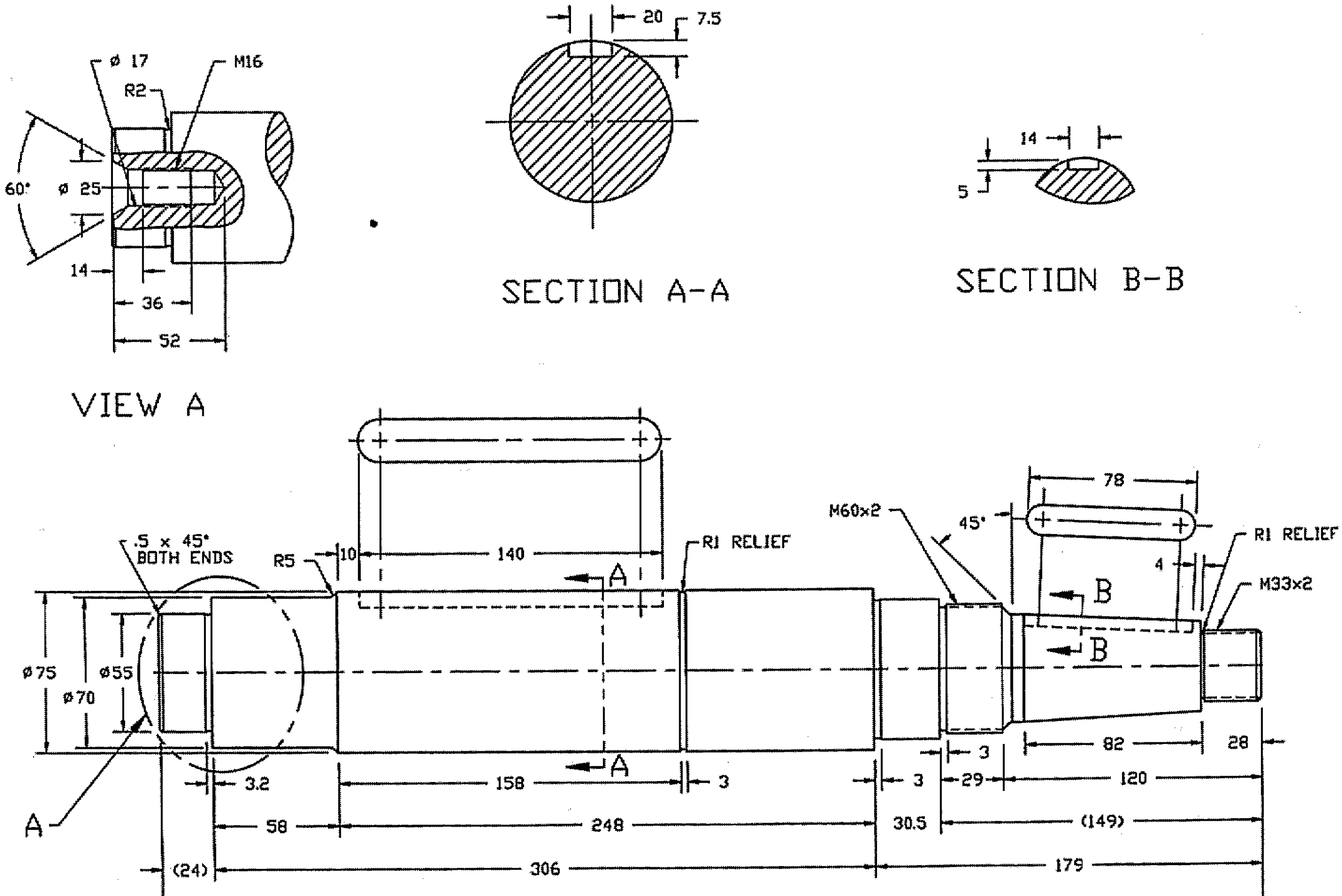


(b)

Problem 23.21(C) and (D) Draw and detail the assembly and details for the coupling.

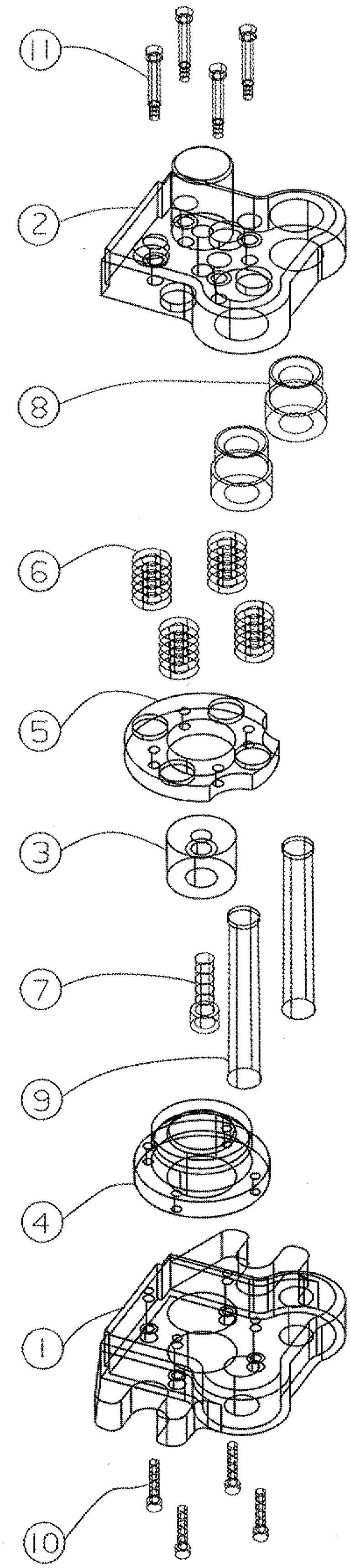


(c)

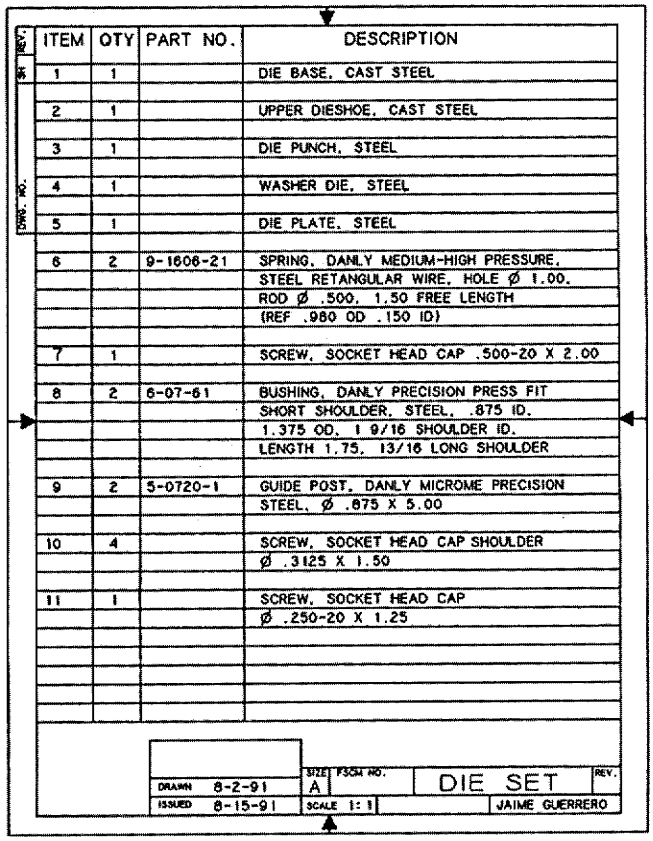


(d)

Problem 23.22(A) Through (I) Draw and detail the die set. Do a complete assembly and set of details.



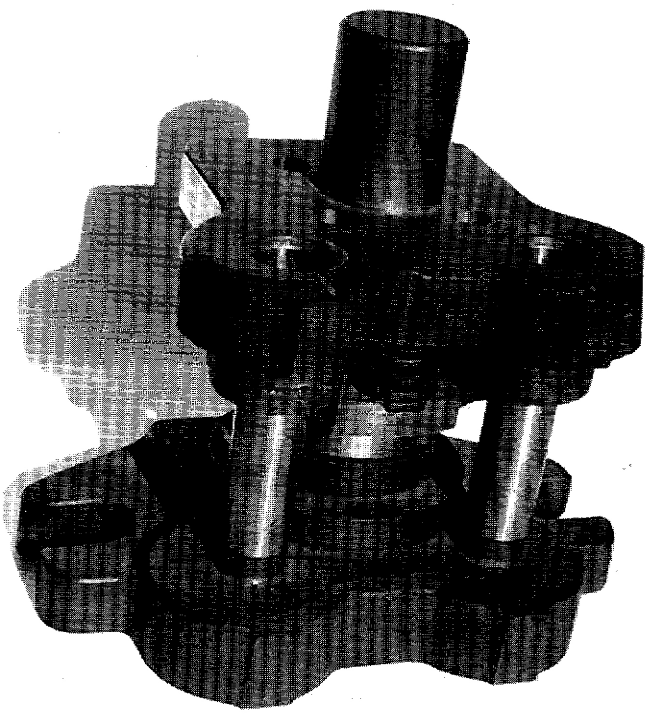
(a)



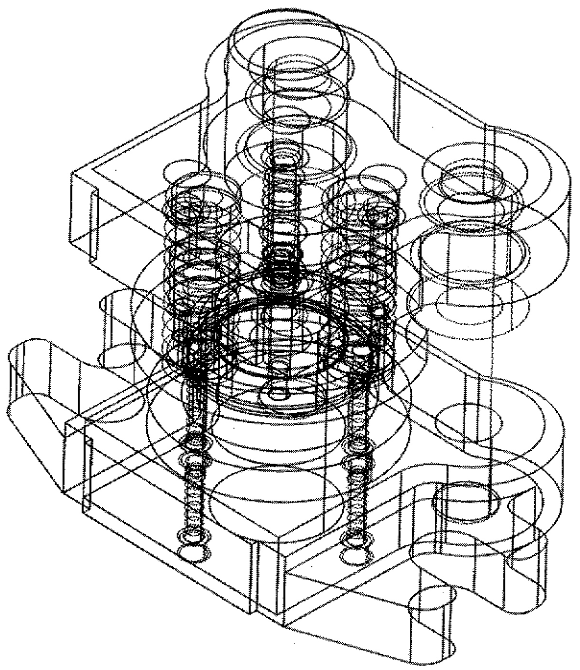
ITEM	QTY	PART NO.	DESCRIPTION
1	1		DIE BASE, CAST STEEL
2	1		UPPER DIESHOE, CAST STEEL
3	1		DIE PUNCH, STEEL
4	1		WASHER DIE, STEEL
5	1		DIE PLATE, STEEL
6	2	9-1606-21	SPRING, DANLY MEDIUM-HIGH PRESSURE, STEEL RETANGULAR WIRE, HOLE Ø 1.00, ROD Ø .500, 1.50 FREE LENGTH (REF .980 OD .150 ID)
7	1		SCREW, SOCKET HEAD CAP .500-20 X 2.00
8	2	6-07-61	BUSHING, DANLY PRECISION PRESS FIT SHORT SHOULDER, STEEL, .875 ID, 1.375 OD, 1 9/16 SHOULDER ID, LENGTH 1.75, 13/16 LONG SHOULDER
9	2	5-0720-1	GUIDE POST, DANLY MICROME PRECISION STEEL, Ø .875 X 5.00
10	4		SCREW, SOCKET HEAD CAP SHOULDER Ø .3125 X 1.50
11	1		SCREW, SOCKET HEAD CAP Ø .250-20 X 1.25

DRAWN 8-2-91	SIZE A	FSCM NO.	DIE SET	REV.
ISSUED 8-15-91	SCALE 1:1			JAIME GUERRERO

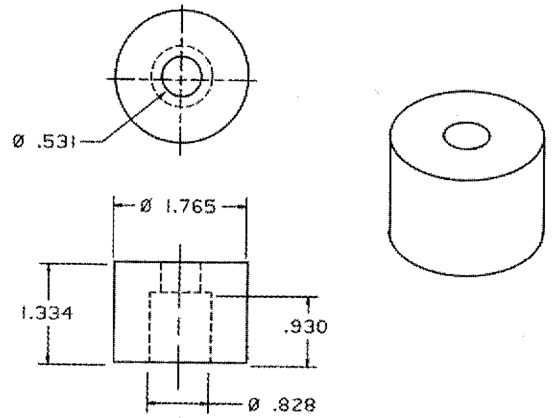
(b)



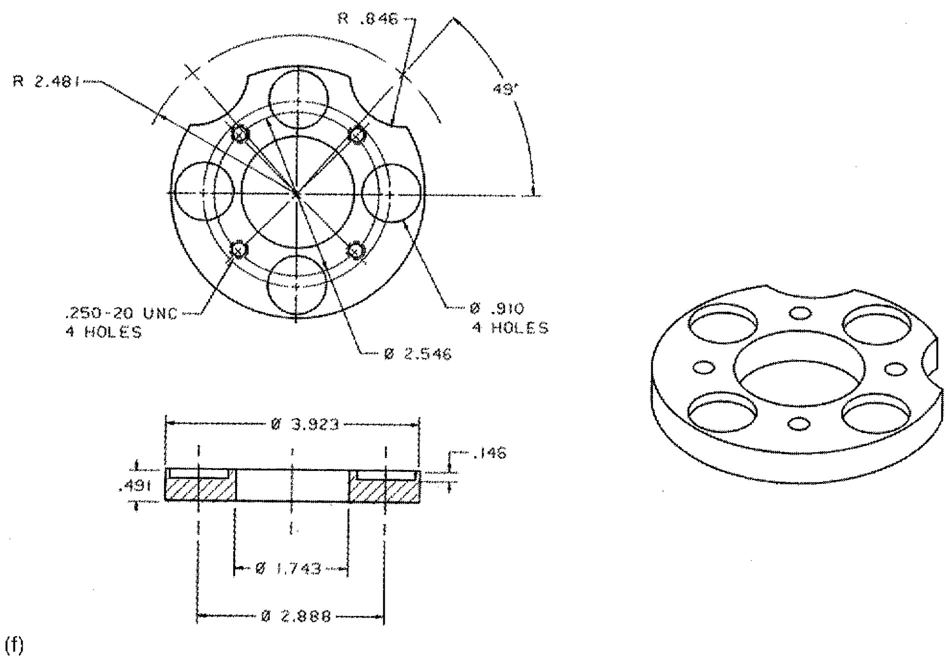
(c)



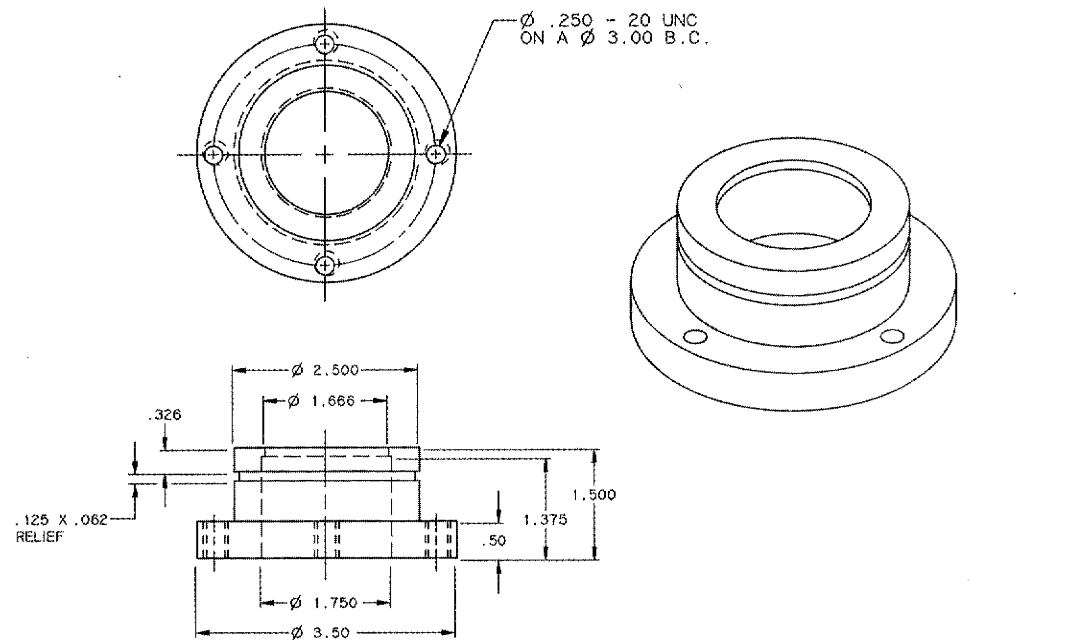
(d)


Ø .531
Ø 1.765
1.334
.930
Ø .828

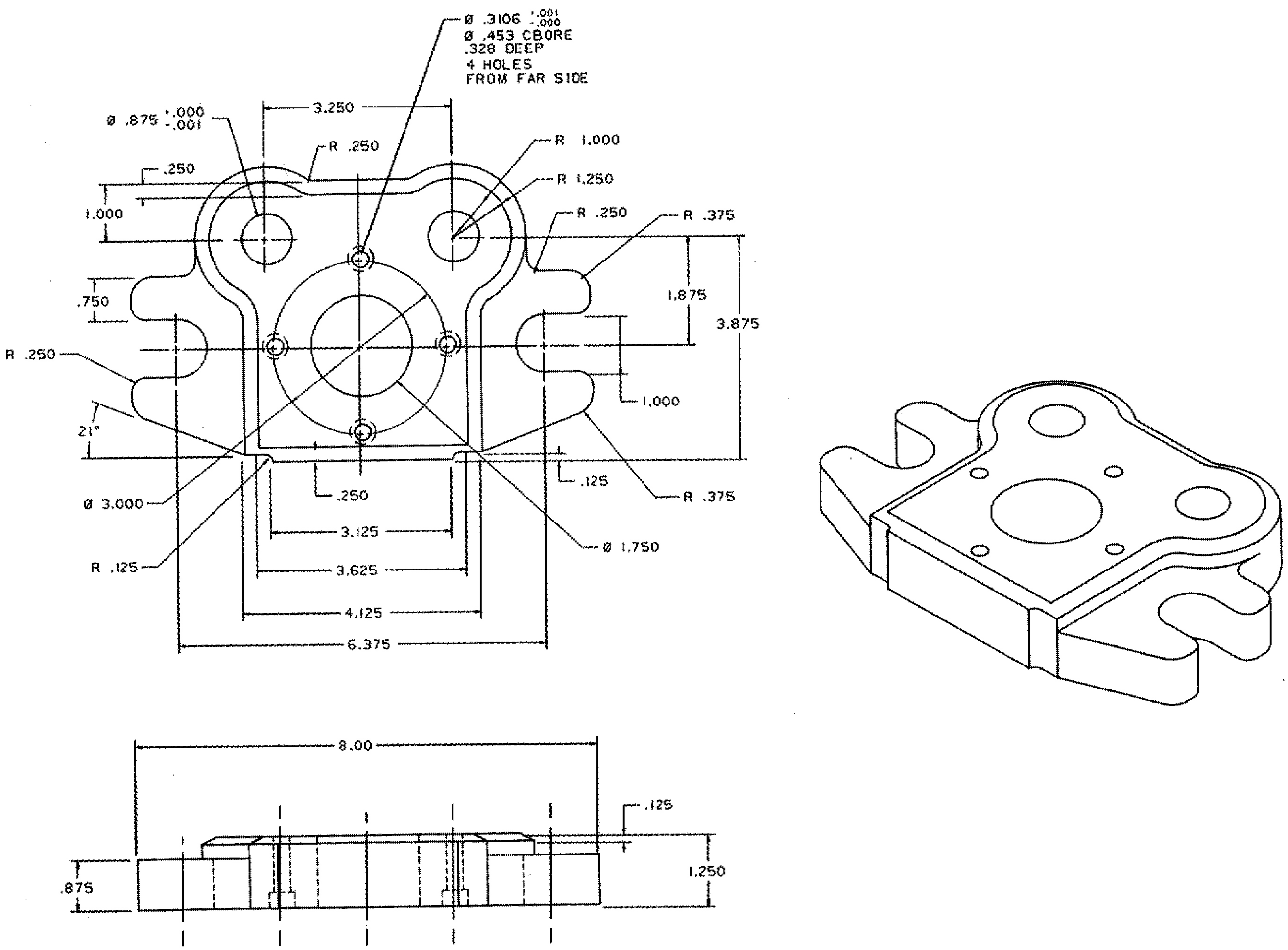
(e)


R .846
R 2.481
49°
.250-20 UNC
4 HOLES
Ø .910
4 HOLES
Ø 2.546
Ø 3.923
.146
.491
Ø 1.743
Ø 2.888

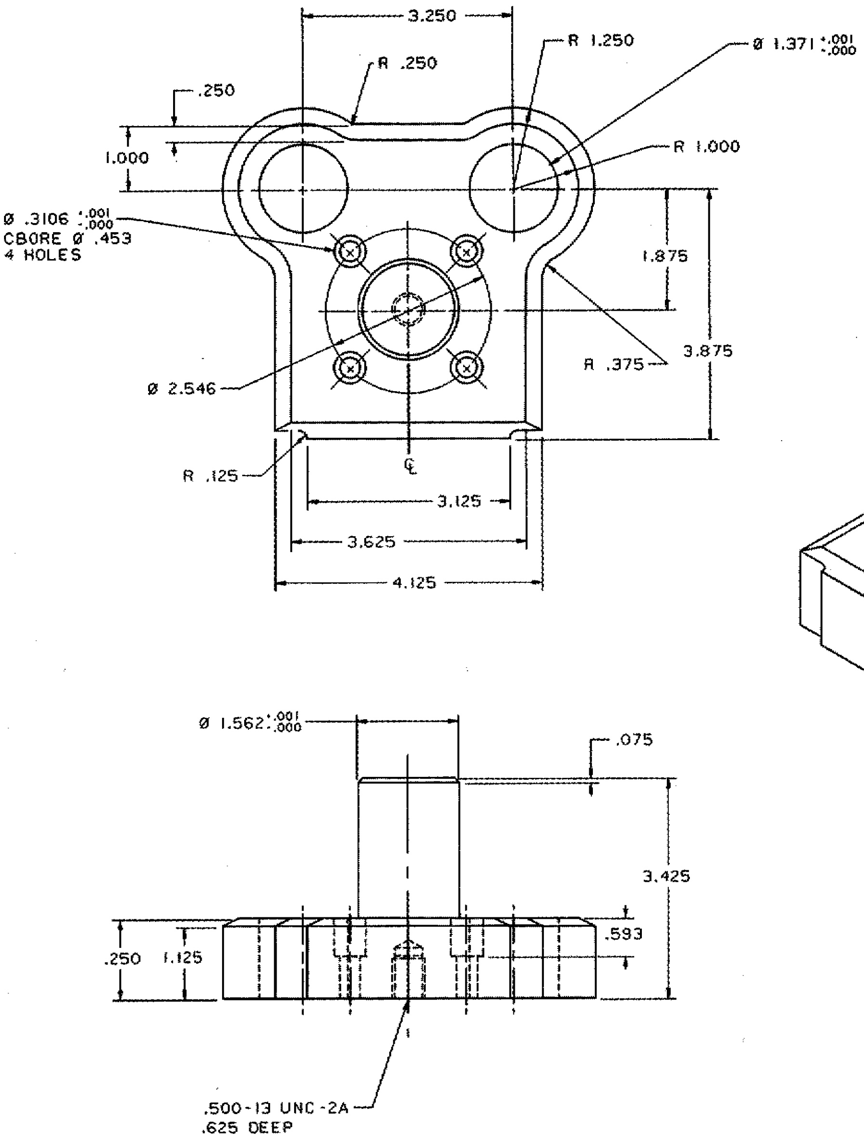
(f)

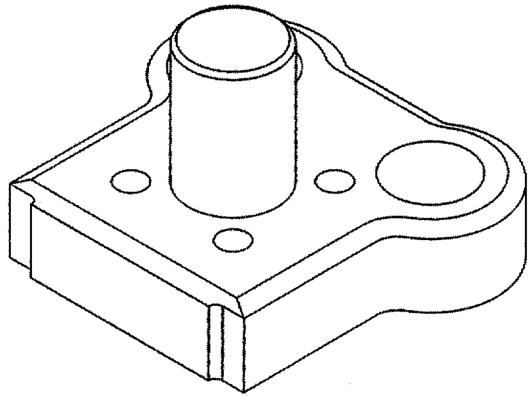

Ø .250 - 20 UNC
ON A Ø 3.00 B.C.
Ø 2.500
Ø 1.666
.326
.125 X .062
RELIEF
1.500
1.375
.50
Ø 1.750
Ø 3.50

(g)


Ø .3106 +.001 -.000
Ø .453 CBORE
.328 DEEP
4 HOLES
FROM FAR SIDE
Ø .875 +.000 -.001
3.250
R .250
R 1.000
R 1.250
.250
1.000
R .250
R .375
.750
1.875
3.875
R .250
1.000
21°
.125
Ø 3.000
.250
R .375
3.125
Ø 1.750
R .125
3.625
4.125
6.375
8.00
.125
1.250
.875

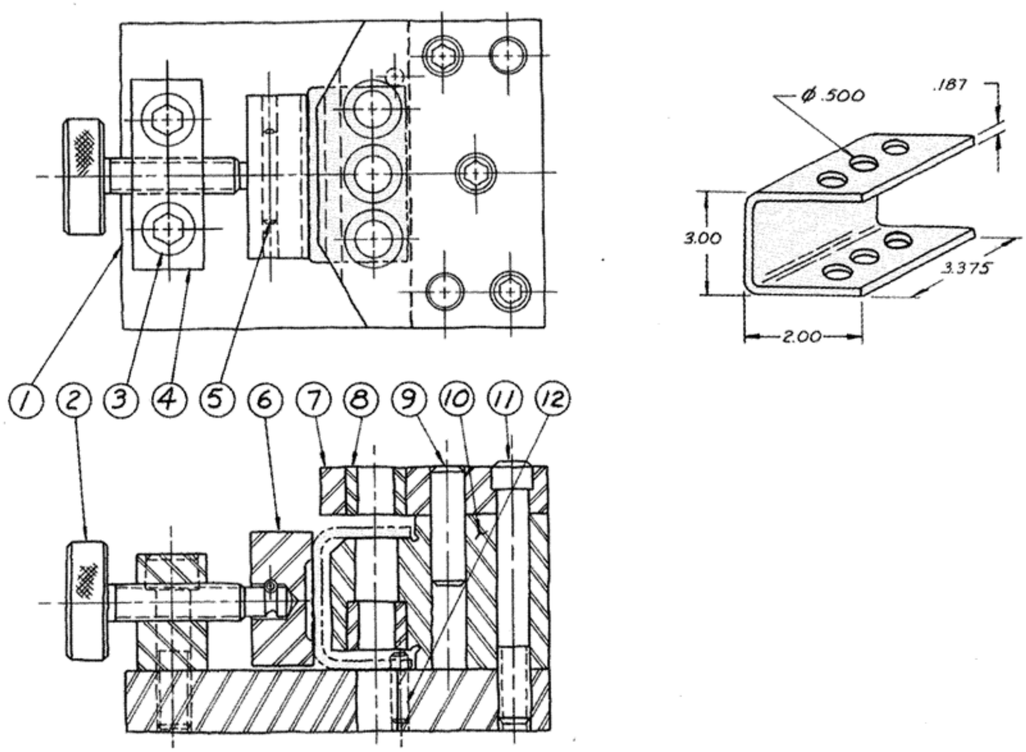
(h)


3.250
R 1.250
Ø 1.371 +.001 -.000
R .250
.250
1.000
R 1.000
Ø .3106 +.001 -.000
CBORE Ø .453
4 HOLES
1.875
3.875
R .375
Ø 2.546
R .125
℄
3.125
3.625
4.125
Ø 1.562 +.001 -.000
.075
3.425
.593
.250
1.125
.500-13 UNC-2A
.625 DEEP



(i)

Problem 23.23(A) Through (H) Draw and detail the jig and fixture assembly.

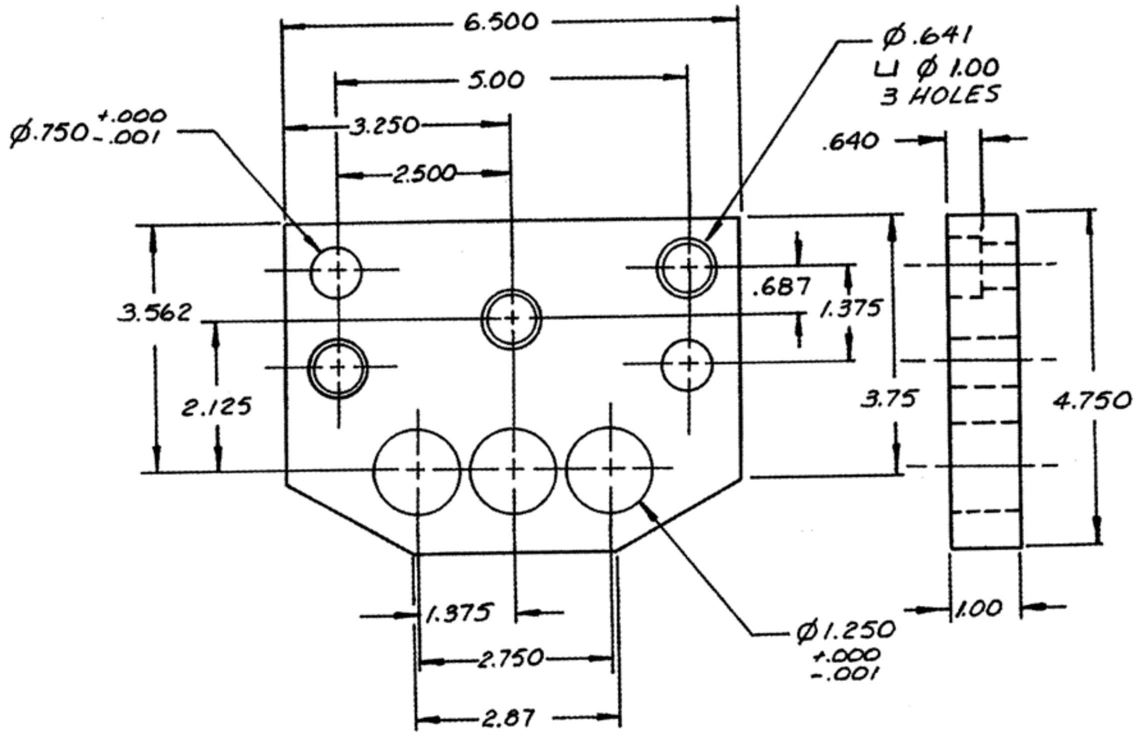


(a)

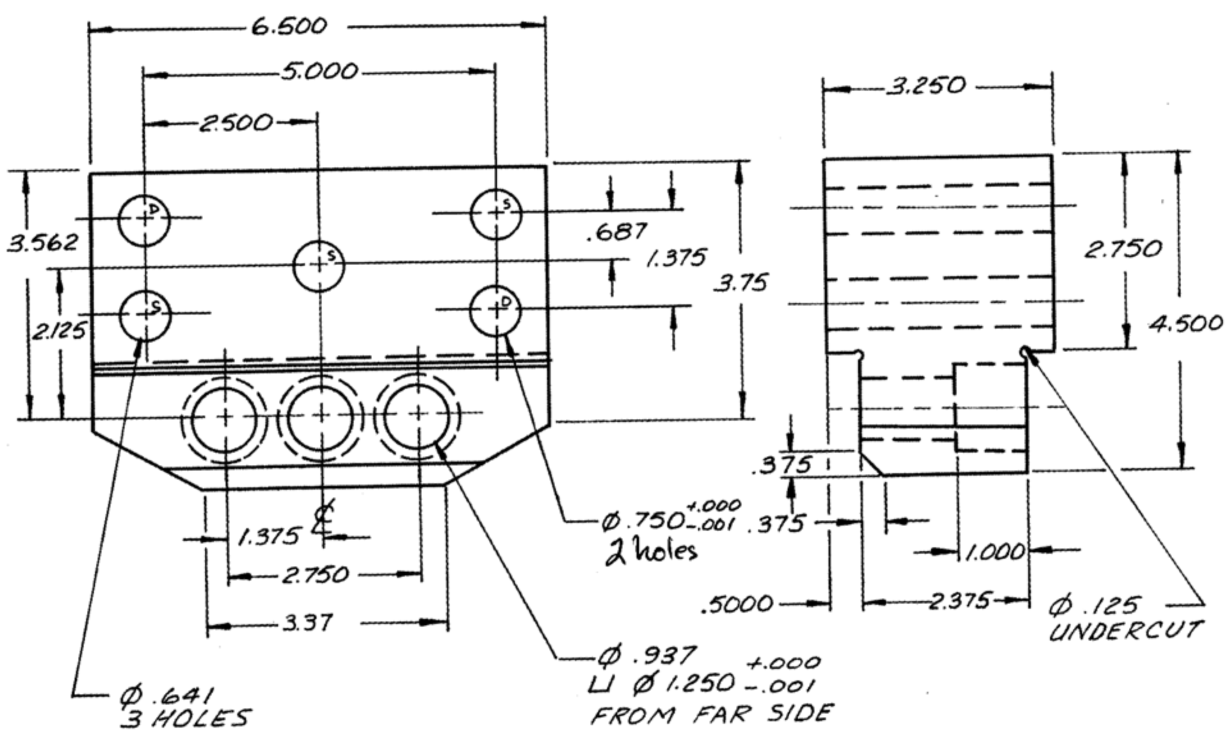
ITEM	QTY	PART NO.	DESCRIPTION
1	1		BASE PLATE, 1.375 X 6.50 X 9.00, STEEL
2	1		SCREW, KNURLED HEAD, STEEL
3	2		SCREW, HEX SOC HD CAP, .750-10 X 3.00
4	1		CLAMP BLOCK, 1.50 X 2.50 X 4.00, STEEL
5	1		PIN, MACHINE DOWEL, Ø .250 X 2.00
6	1		CLAMP, 1.375 X 3.00 X 3.50, STEEL
7	1		BUSHING PLATE, 1.00 X 4.75 X 6.50, STEEL
8	6		BUSHING, TYPE P HEADLESS PRESS FIT MODIFIED ID = .500, 1.25 OD X 1.00 LG
9	2		PIN, MACHINE DOWEL, Ø .750 X 2.50
10	1		LOCATOR BLOCK, 3.25 X 4.50 X 6.50
11	3		SCREW, HEX SOC HD CAP, .625-11 X 5.00
12	1		PIN, MACHINE DOWEL, Ø .375 X 1.50

DRAWN 3-4-91	SIZE A	FSCM NO.	PLATE JIG, DRILL	REV.
ISSUED 7-30-91	SCALE 1:1			

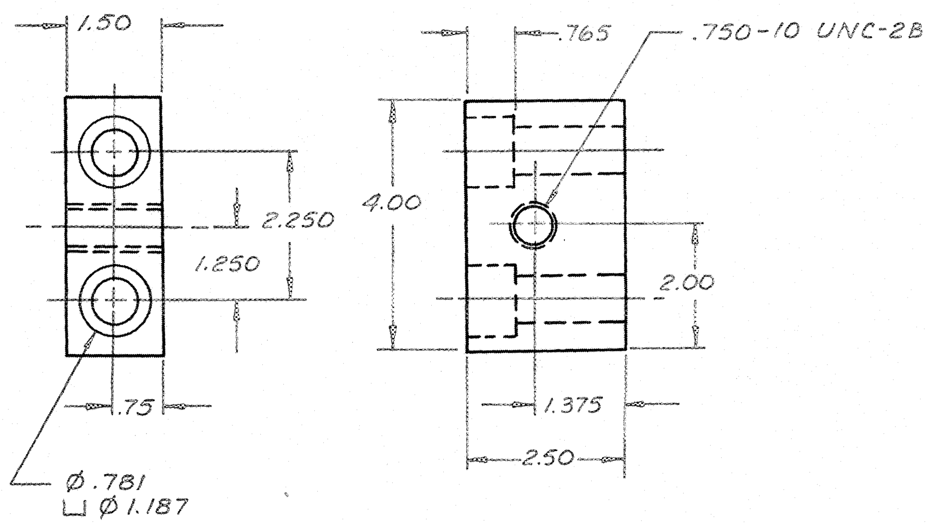
(b)


6.500
5.00
3.250
2.500
Ø.750 +.000 -.001
Ø.641
⊔ Ø 1.00
3 HOLES
.640
3.562
.687
1.375
2.125
3.75
4.750
1.375
2.750
2.87
Ø1.250 +.000 -.001
1.00

(c)

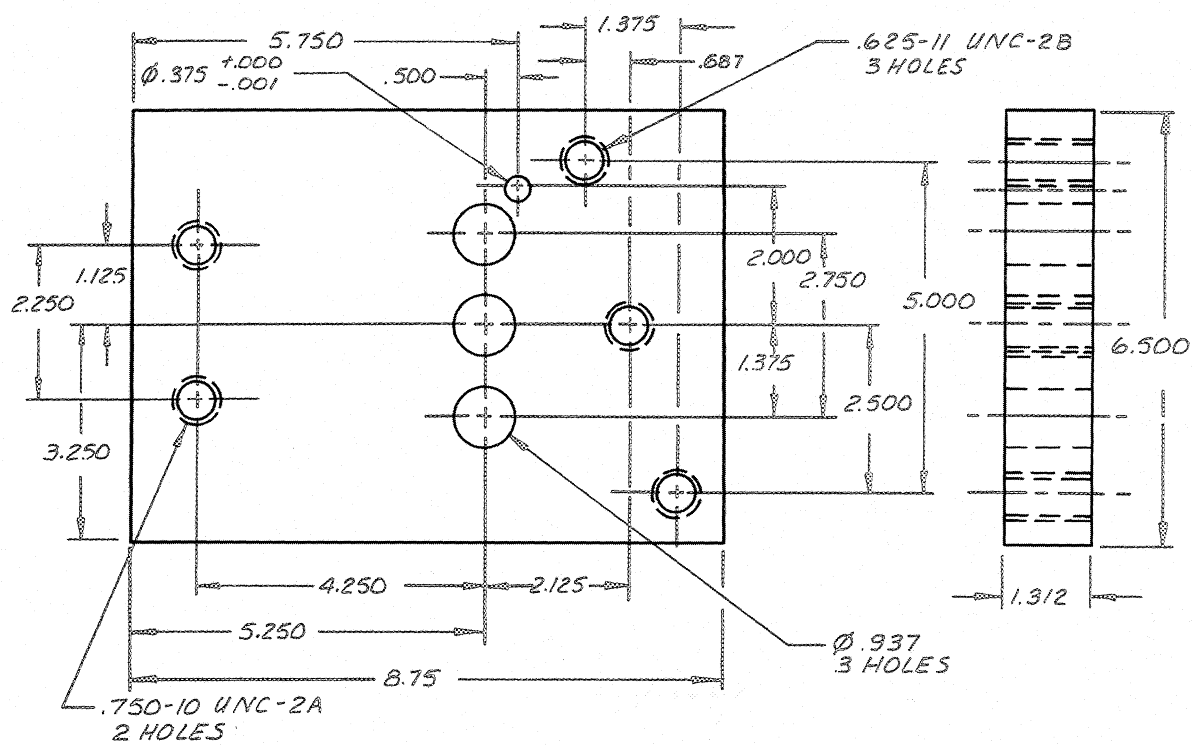

6.500
5.000
2.500
3.250
3.562
.687
1.375
3.75
2.750
4.500
2.125
.375
1.375
2.750
3.37
Ø.750 +.000 -.001
2 holes
.375
1.000
.5000
2.375
Ø .125
UNDERCUT
Ø .937 +.000
⊔ Ø 1.250 -.001
FROM FAR SIDE
Ø .641
3 HOLES

(d)



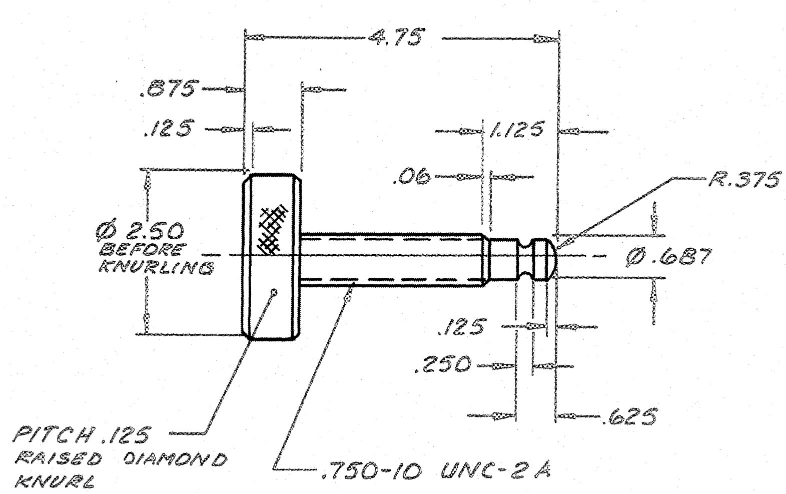
1.50
.765
.750-10 UNC-2B
4.00
2.250
1.250
2.00
.75
1.375
2.50
Ø .781
⌴ Ø 1.187

(e)



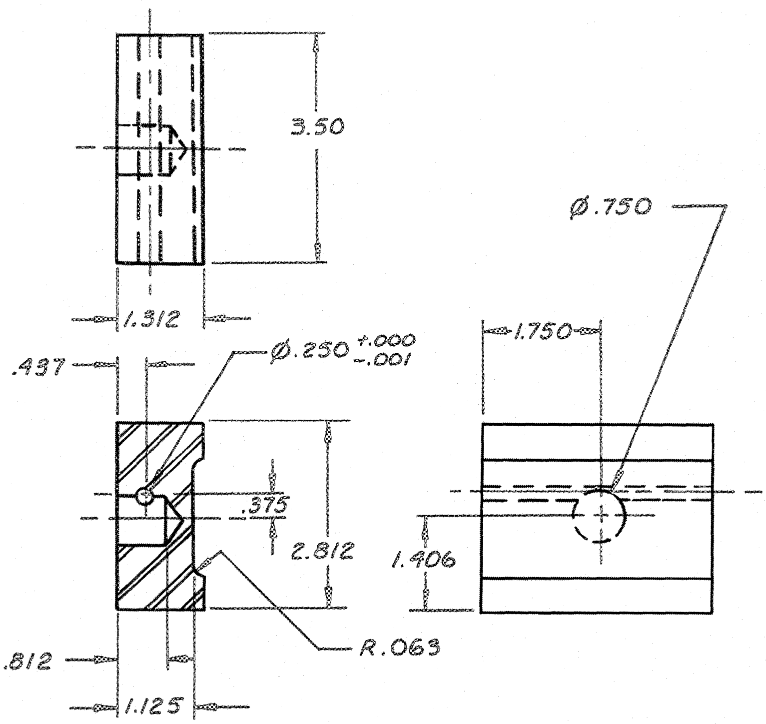
5.750
1.375
.687
.625-11 UNC-2B
3 HOLES
Ø.375 +.000 -.001
.500
2.000
2.750
5.000
1.125
2.250
1.375
2.500
6.500
3.250
4.250
2.125
5.250
8.75
1.312
Ø .937
3 HOLES
.750-10 UNC-2A
2 HOLES

(f)



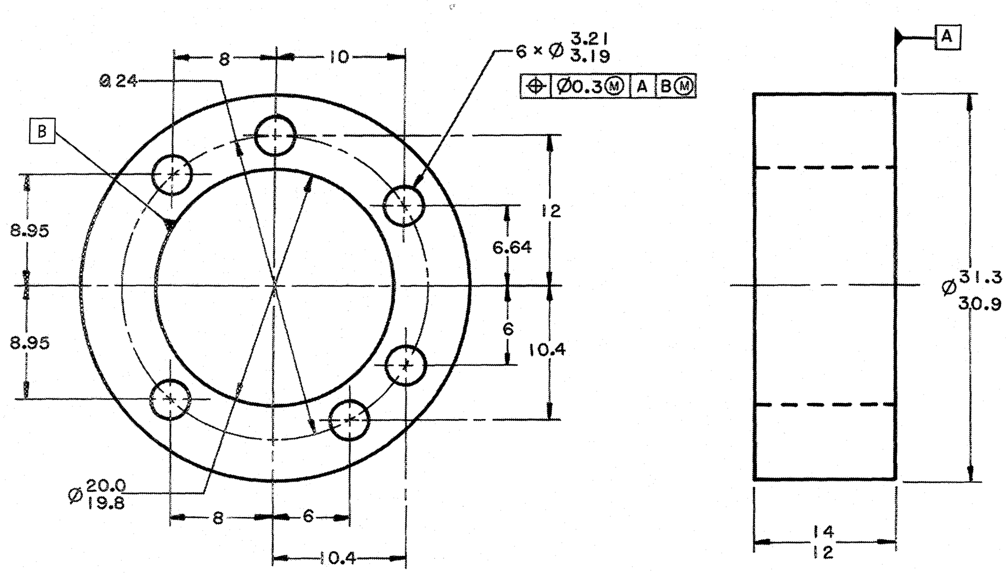
4.75
.875
.125
1.125
.06
R.375
Ø 2.50
BEFORE
KNURLING
Ø .687
.125
.250
.625
PITCH .125
RAISED DIAMOND
KNURL
.750-10 UNC-2A

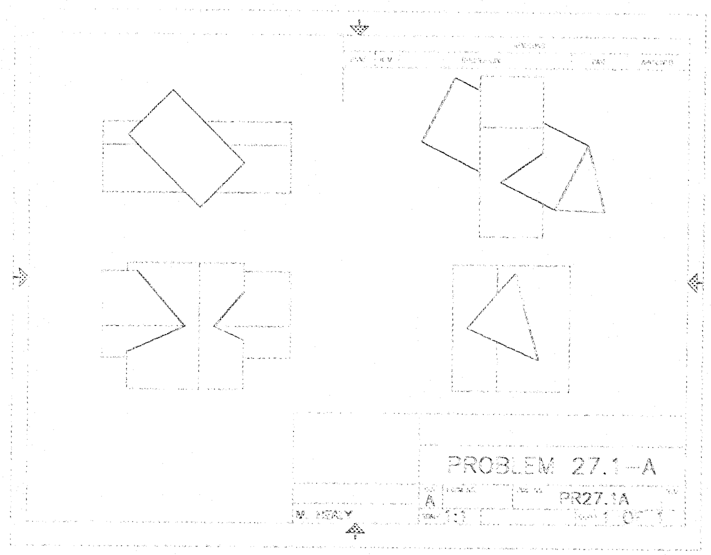
(g)

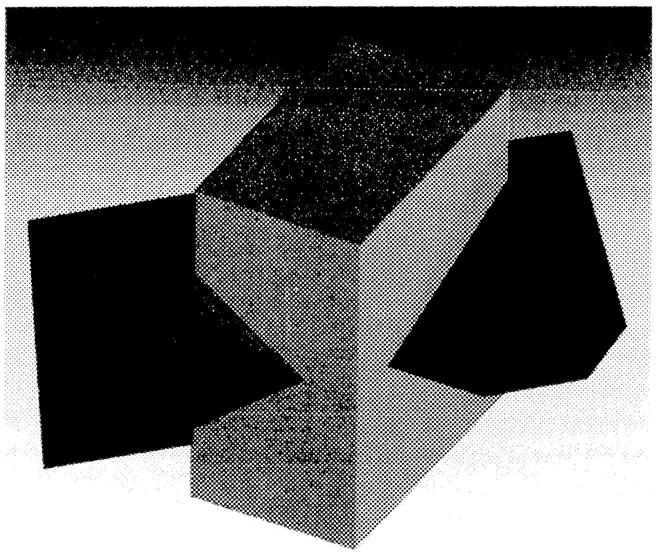


(h)

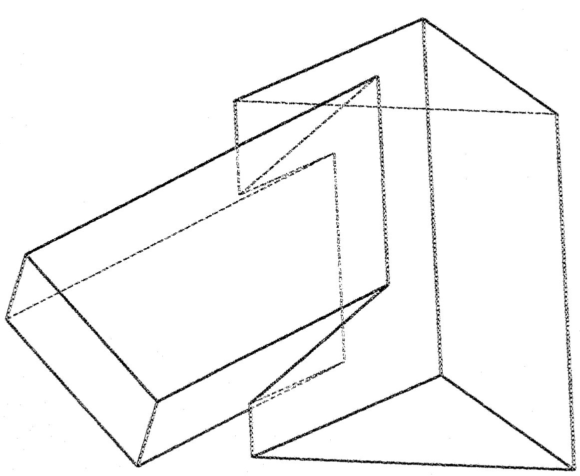
Problem 23.24 Design a fixture to hold the ring while machining the center hole and the six small holes.

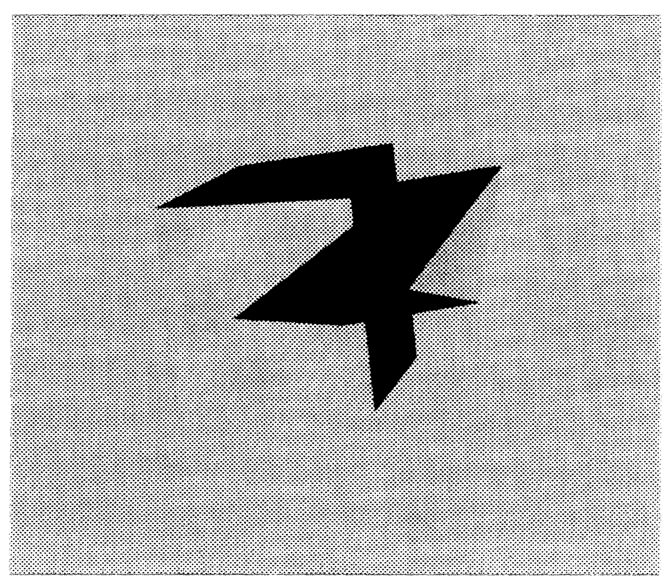






PROBLEM 27.1 Example Solution for Problem 27.1(A)





PROBLEM 27.1 Example Solution for Problem 27.1(B)