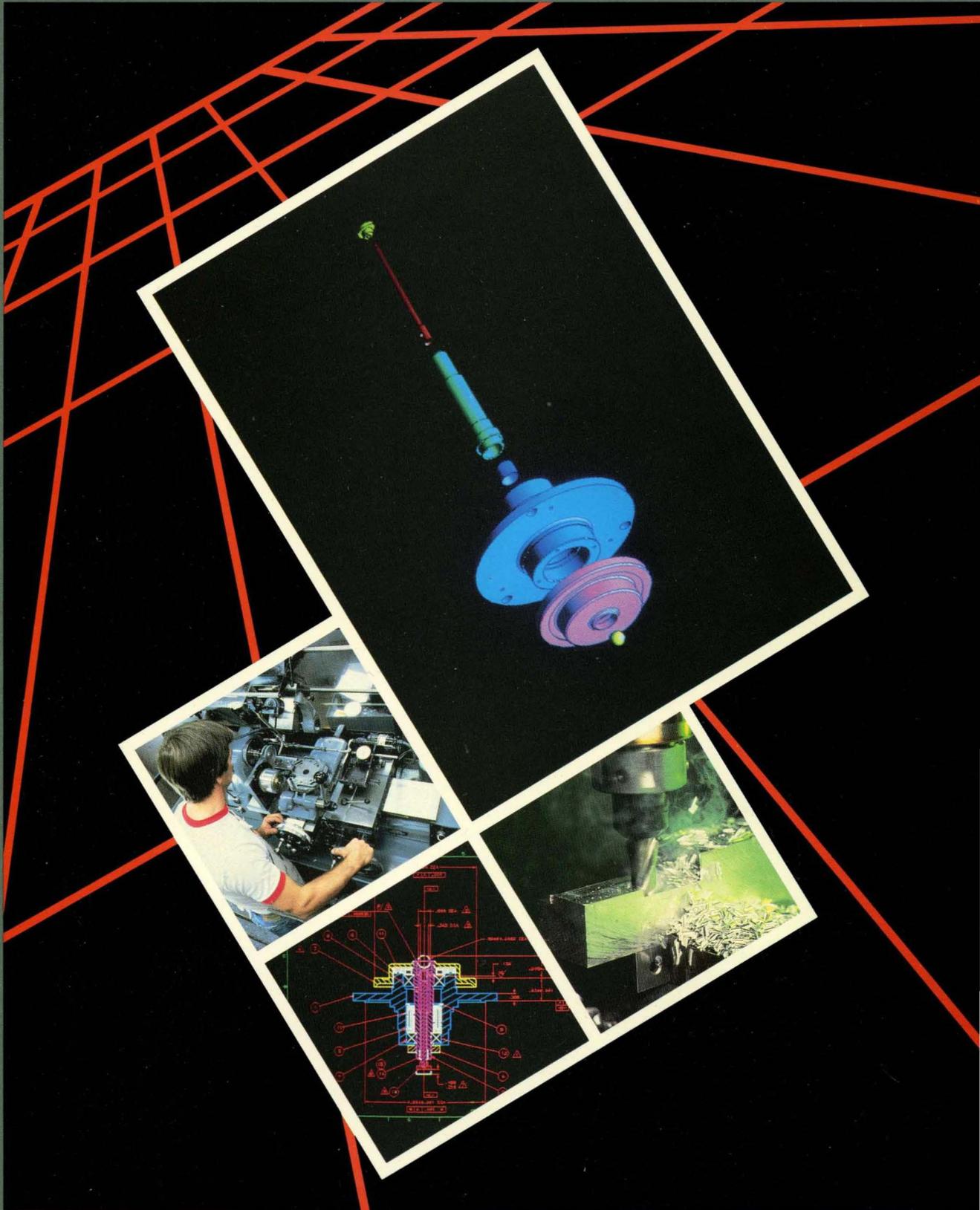


ICEM Design/Drafting User's Guide for NOS


CONTROL
DATA



Tutorial

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ICEM Design/Drafting User's Guide for NOS

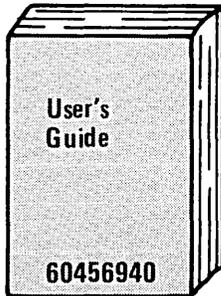
Tutorial

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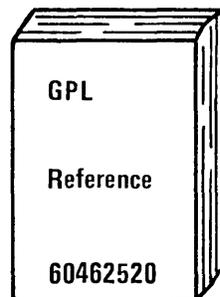
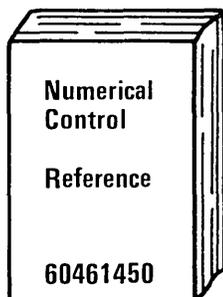
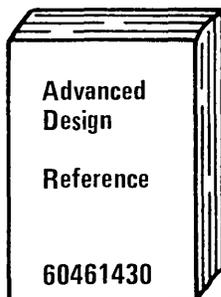
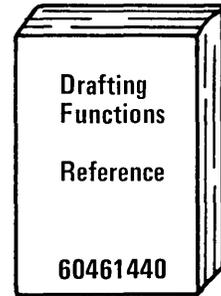
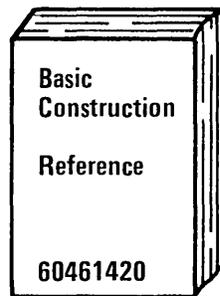
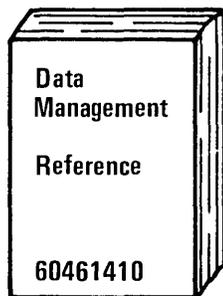
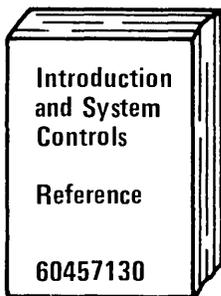
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Related Manuals

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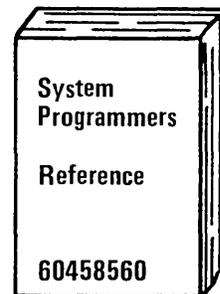
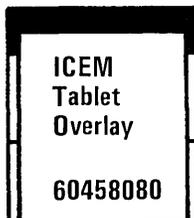
Manual Set:



Additional:



60457140



Manual History

Revision F documents ICEM DDN Version 1.6, printed in February 1986. This revision includes Selective View Blanking, Hidden Line Removal, and miscellaneous technical changes. The N/C Manufacturing chapters have been removed from the manual. New tablet art has also been added to reflect Version 1.6 of the ICEM tablet overlay. Because extensive changes have been made, change bars and dots are not used and all pages reflect the latest revision level. This edition obsoletes all previous editions.

Previous Revision	System Version	Date
A	1.0	July 1979
B	1.1	September 1980
C	1.3	September 1981
D	1.4	April 1983
E	1.53	December 1984

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About This Manual

This manual is a user's guide to the CONTROL DATA® Integrated Computer-aided Engineering and Manufacturing Design/Drafting/Numerical Control (ICEM DDN) software system. It is intended to give new users an understanding of the elements of ICEM DDN and to provide a base on which further experience can be built. Because this user's guide does not describe all of the functions of the ICEM DDN system, it should be used in conjunction with the other ICEM DDN manuals.

Audience

This guide is written for beginning ICEM DDN users. Some knowledge of drafting principles and 3-D geometry is assumed. Experience with computer graphics, graphics terminals, and interactive terminal usage is helpful but not necessary in using this manual.

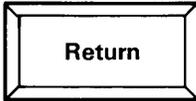
Organization

This manual is organized as follows:

- Chapter 1 deals with the Network Operating System (NOS), including logging in to NOS, gaining access to the ICEM DDN system, and manipulating files. This chapter also includes an overview of the features and capabilities of the ICEM DDN system.
- Chapter 2 introduces the most basic ICEM DDN functions.
- Chapter 3 describes the most frequently used drafting functions including section lining and dimensioning.
- Chapters 4 through 11 consist of practice sessions that give step-by-step instructions on how to create various parts. You should begin with section 4 and proceed through the sessions sequentially.
- Appendix A contains an alphabetical listing of terms and definitions.
- Appendix B contains a list of frequently used control card parameters.

Conventions

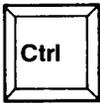
Throughout this guide, we use representation of actual keys to show you when you should press a key. For example,



is represented as

(RETURN)

and



is shown as

(CTRL)

Two keys appearing side by side means hold down the first key while pressing the second. For example,

(CTRL) P

means hold down **(CTRL)** and press the P key.

Sometimes you have to press more than one key in succession to execute a certain function. This is indicated with a plus sign. For example,

(CTRL) P + (RETURN)

means hold down **(CTRL)**, press the P key, and then release the **(CTRL)** key and press **(RETURN)**.

System displays are shown in uppercase letters and they are in computer font.

Any entries you make to the computer are shown in blue.

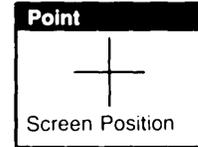
Keyboard and tablet selections are shown in chapters 2 through 11. The tablet squares represent the equivalent keyboard selections. For example, if you want to define a point by screen position, you can either select menu choice 9 from the main menu followed by menu choice 1 from the Point menu, or you can touch the SCREEN POSITION tablet square with your tablet pen to go directly to that menu selection:

Keyboard Selection

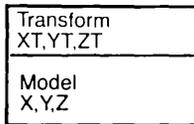
Select 9.POINT from the main menu.

Select 1.SCREEN POSITION from the Point menu.

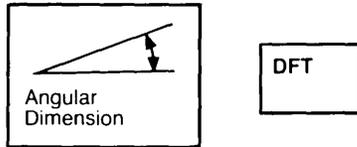
Tablet Selection



Sometimes the tablet square shown is divided into parts. In this case, your selection on that square is shown in blue:



Not all of the tablet squares fit on the three main overlays: the fixed overlay, the Basic Functions Upper (BFU) overlay, and the Basic Functions Lower (BFL) overlay. If the tablet square shown appears on an overlay other than the three main overlays, it is indicated to the right of the tablet square, using the abbreviation for that overlay.



In this example, the angular dimension tablet square appears on the drafting overlay (abbreviated as DFT).

Additional Related Publications

The following is a list of Control Data publications related to the use of ICEM DDN:

Manual Title	Publication Number
ICEM Design/Drafting GRAPL Programming Language	60461460
ICEM Engineering Data Library Reference Manual	60459740
NOS 2 Reference Set, Volume 1 Introduction to Interactive Usage	60459660
NOS 2 Reference Set, Volume 2 Guide to System Usage	60459670

Ordering Manuals

Control Data manuals are available through Control Data sales offices or through:

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Submitting Comments

The last page of this manual is a comment sheet. Please use this comment sheet to give us your opinion of the manual's usability, to suggest specific improvements, and to report technical or typographical errors. If the comment sheet has already been used, you can mail your comments to:

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Publications and Graphics Division ARH219
4201 Lexington Avenue North
St. Paul, Minnesota 55126-6198

Please indicate whether or not you would like a written response.

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Using the Tablet	1-17
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Logging In

Logging in is the procedure that establishes communication between your graphics terminal and the computer. Because there are several graphics terminals available and many ways that your system could be modified, your login procedure may differ slightly from that described here. If you find that the system doesn't respond as expected, ask your computer center personnel for help.

Setting Up Your Terminal

You are probably using a Tektronix 4115 terminal (the plate on the terminal identifies the model). If you are using another type, check with your computer center personnel for any variations in the procedures that follow.

Before you can log in, you must prepare your terminal as follows:

1. Set the power switch of your terminal to the on position. It is located in the center of the panel above the keyboard. Allow a few minutes for the terminal to initialize. The terminal is ready when the lights on the keyboard go off.
2. Put your terminal on-line by pressing the LOCAL key located in the upper right-hand area of the keyboard. The terminal is on-line when the small light in this button is *not* illuminated. If the light is on, press the key to make the light go off.
3. Find out both the telephone number you will use in communicating with the computer and the baud rate (the speed at which the computer and terminal communicate) of that data line. The data processing personnel at your installation have this information.
4. Press the DIALOG key located in the upper right-hand area of the keyboard to make the dialog area of the terminal visible. The small light on the DIALOG key is illuminated when you are in dialog mode.
5. Set the terminal status by pressing the SETUP key until the small light on the key is illuminated. As soon you see an asterisk displayed on your terminal, type the STATUS command (abbreviated as STA) and press the RETURN key to list the current terminal status settings:

```
STA (RETURN)
```

Check to make sure that the terminal settings highlighted in figure 1-1 match your terminal settings. If any of the settings are incorrect, you can change them by typing the first three characters of the setting name, a space, the new setting, and the RETURN key. For example, if you want to change the baud rate setting to 9600, type:

```
BAU 9600 (RETURN)
```

Another asterisk displays.

NOTE

If you make a typing error while you are in the setup mode, use the RUBOUT key to correct your error. Do not use the backspace key.

6. To exit from setup mode, press the SETUP key again. The light on the key goes off.

```

                TERMINAL STATUS

                REPORT/INPUT
TBSTATUS.....OUT
TBHEADERCHARS.....CONTROL
TBSIZE.....SMALL
REOM.....1
GINAPERATURE.....8
GINPICKWINDOW.....0,0 4095,4095
RLINELENGTH.....0

                GENERAL
PAGEFULL.....NONE
EDITCHARS.....# Q ~
ECHO.....YES
LOCKKEYBOARD.....NO
DAENABLE.....YES
IGNOREDEL.....NO
LFCR.....NO
CRLF.....NO
SNOOPY.....NO
ERRORLEVEL.....2

                DIALOG
DABUFFER.....500
DACHARS.....80
DAINDEX.....2 0 0
DA2INDEX.....7
DALINES.....34
DAMODE.....REPLACE
DASURFACE.....1
DAVIS.....YES
DAPOSITION.....0,0

                COMMUNICATIONS
BAUDRATE.....9600 9600
XMTLIMIT.....9600
STOPBITS.....1
XMTDELAY.....0
PARITY.....NONE
PROMPTSTRING.....' '
PROMPTMODE.....NO
QUEUESIZE.....8100
FLAGGING.....INPUT
EOMCHARS.....D3 N
EOLSTRING.....' Q '
EOFSTRING.....' Ex '
BREAKTIME.....100
BYPASSCANCEL.....N
    
```

Figure 1-1. Terminal Status Settings for the Tektronix Model 4115 Terminal

Connecting Your Terminal

Some terminals are wired so that they are automatically connected to the system when they are turned on. These terminals are often called hardwired terminals. Other terminals are connected to the system using a telephone and phone lines. If there is no telephone near the terminal you are using, you can assume it is a hardwired terminal. In this case, you can skip the rest of this section and go on to Entering the System.

A terminal that requires a telephone uses a modem. There are two kinds of modems: an acoustic coupler modem or a data set modem. If you have an acoustic coupler (the receiver of the phone is placed in the rubber cups of an acoustic coupler), refer to step 1. If you have a data set, refer to step 2.

1. To establish connection with the computer using an acoustic coupler:
 - a. Turn the coupler's power switch to the on position.
 - b. Pick up the telephone receiver and listen for a dial tone.
 - c. Dial the telephone number of the computer.
 - d. When you hear a continuous tone, place the receiver in the rubber cups of the acoustic coupler, making sure that the telephone headset is placed in the coupler correctly. Most couplers are marked with the word cord to help you.

You are now successfully connected to the computer and ready to log in.

2. To establish connection with the computer using a data set:
 - a. Pick up the telephone receiver.
 - b. Press the TALK button down and listen for a dial tone. The light in the button should be on.
 - c. Dial the telephone number of the computer.
 - d. When you hear a continuous tone, press the DATA button. The light in this button comes on, and the light in the TALK button goes out.
 - e. Put down the telephone receiver.

You are now successfully connected to the computer and ready to log in.

Entering the System

If the terminal and modem are properly set, the computer prints header information (information identifying the computer system) on the screen and waits for you to log in. If nothing appears on the screen, press the RETURN key on the terminal. If the computer still has not begun communication, hang up the phone. Refer to the manufacturer's instructions on operating the modem and the terminal. Recheck both the modem and the terminal and repeat step 1 or 2 under Connecting Your Terminal. If you are again unsuccessful, a central computer malfunction probably exists, and you must try again later.

NOTE

If you are entering ICEM DDN through the Engineering Data Library (EDL), the information you are prompted for will differ from that described here. For more information about entering ICEM DDN through EDL, refer to the ICEM Engineering Data Library Reference Manual (listed in the preface).

LOGGING IN

When communication is established, header information is printed followed by the line:

FAMILY:

Type in the family name you were given and press the RETURN key. If you are using the default family for the system, just press the RETURN key.

On the next line, type in your user name following the prompt:

USER NAME:

and press the RETURN key.

Next, type your password following the prompt:

PASSWORD:

and press the RETURN key. If no password is required, just press the RETURN key.

You can also enter your family name, user name, and password all on the first line using a comma as the separator. For example, if you are using the default family and if your user name is FRED and your password is SESAME, you can enter them all following the first prompt as follows:

FAMILY: ,FRED,SESAME (RETURN)

By typing a comma as the first entry, you are telling ICEM DDN to accept the default family name.

If your family name, user name, and password are acceptable, the computer allows you to select an application by issuing the following message:

termnam - APPLICATION:

termnam is the terminal name that was provided in the last part of the first line of header information. Type IAF and press the RETURN key.

Depending on your installation, it is possible that after your family name, user name, and password are validated, the terminal is automatically placed under the control of the Interactive Facility (IAF). In this case, the application prompt does not appear.

When the terminal is placed under IAF control, IAF responds on the next line by identifying the terminal number. For example:

TERMINAL: 101, NAMIAF

indicates that this is terminal number 101 of this installation's IAF. This number is not fixed; that is, the next time you log in on this terminal, IAF may issue a different terminal number.

NOTE

Always observe the terminal number. You will need it if your terminal is unexpectedly disconnected from the computer.

The next line that IAF displays is either:

RECOVER/SYSTEM:

or

RECOVER/CHARGE:

IAF is asking you to enter an IAF command (following RECOVER/SYSTEM), your charge number (following RECOVER/CHARGE), or the special command RECOVER.

NOTE

At some sites, this information is entered automatically at the time of log in. If you see a / displayed instead of the prompts above, you can skip the rest of this section and proceed to the Attaching ICEMDDN section.

If your user name does not require a charge number, the first form is displayed. At this point you can enter any valid IAF command. The login procedure is complete and you may ignore the following discussion of the charge number entry and proceed directly to Attaching ICEMDDN.

If, however, your user name requires a charge number, IAF displays RECOVER/CHARGE. Enter CHARGE followed by a comma, enter your charge and project numbers separated by a comma, and press the RETURN key. If you were not furnished with a project number, omit the second comma and press the RETURN key after entering your charge number. For example, if your charge number is 5923 and your project number is PROJ1, your response is:

```
RECOVER/CHARGE: CHARGE,5923,PROJ1 (RETURN)
```

If you were previously logged in and unexpectedly disconnected, enter RECOVER followed by a comma and the number that IAF previously assigned to the terminal.

IAF then responds with either a

```
/
```

or a

```
READY.
```

prompt. If / appears, proceed to Attaching ICEMDDN. If READY appears, type

```
BATCH (RETURN)
```

IAF responds with

```
$RFL=0  
/
```

which means that you are now in IAF's batch subsystem.

Attaching ICEMDDN

All the information in a computer system is organized into files. A file is a collection of information that can be accessed with a single name. In NOS, file names can be from 1 to 7 alphanumeric characters (A through Z or 0 through 9). Some of the product sets (FORTRAN, COBOL, and so forth) require that the first character be a letter (A through Z).

NOS contains a number of application routines that are organized as a library and referred to as the application library files. Each file has a system-supplied name that can be used to attach it to a terminal. One of these files is called ICEMDDN. This file contains the programs you need to create, modify, and save parts drawings during your work session.

ICEMDDN is a *direct access permanent file* which means that when it is attached to a terminal, it can be accessed directly without any intermediate local copy.

ICEMDDN is also a system file which means that it cannot be altered. As part of a library, it requires that you specify a library user name before you can attach the file to your terminal. (NOS requires user names whenever users access files other than their own.) The user name allows NOS to check user validation and resource limits.

To attach and use the routines in the ICEMDDN library file, you must type:

```
ATTACH, ICEMDDN/UN=APPLLIB, M=E (RETURN)
```

ATTACH calls in the NOS permanent file routine that searches for direct access files. UN=APPLLIB specifies APPLLIB as the user number of the system application library and the entry ICEMDDN attaches the library file with that name. M=E indicates that you are in the execution mode.

When the file is attached, the system indicates that it is waiting for another entry by displaying:

/

If you have made an error, the system displays a diagnostic message describing the nature of the error. Only when the error is corrected by reentering the command, will it give the prompt for another entry.

Basic File Manipulation

The system file ICEMDDN not only provides procedures for creating and modifying parts drawings, but it also provides a local file called TAPE3 which serves as a data base on which to store and retrieve the parameters that define your drawings.

As a local file, TAPE3 exists only until you log out, at which time it is deleted. To save a particular data base beyond the session in which it was created, you must make that data base a direct access permanent file before you begin using it. You can do this with the NOS control statement called DEFINE. The format of this control statement is:

```
DEFINE,TAPE3=pfn (RETURN)
```

pfn is the permanent file name (1 to 7 alphanumeric characters) you give your data base. By associating this name with the TAPE3 data base, your local file becomes a direct access permanent file, accessible with the permanent file name you supplied.

As soon as you press the RETURN key, all of your activity is automatically recorded on your permanent file (pfn) and continues to be held on that file when you log off.

For example, if you wanted to name your new data base, DBASE1, you would enter the following:

```
DEFINE,TAPE3=DBASE1 (RETURN)
```

When you want to modify and enhance this data base file, you must attach it with the following NOS control statement:

```
ATTACH,TAPE3=pfn/M=W (RETURN)
```

M=W means that you are in write mode, which allows you to alter this file and/or add new information. You must have your data base file attached in write mode if you want to alter the file in any way since the default mode for the ATTACH statement is read. The reason for this default is that direct access files are permanently altered by any write operation the moment it is entered and NOS, accordingly, includes this additional caution to the user.

For example, to attach your previously created data base, DBASE1, enter:

```
ATTACH,TAPE3=DBASE1/M=W (RETURN)
```

Once you have defined or attached your permanent file (pfn), you can begin using ICEM DDN. To do this, type:

```
ICEMDDN (RETURN)
```

You are then asked for the baud rate you are using. The baud rate is the speed at which the data is transmitted from the computer to your terminal via phone lines.

```
ENTER BAUD RATE.
```

SPECIFYING YOUR TERMINAL TYPE

You must respond by entering the first digit of the baud rate. The available baud rates are as follows:

Baud Rate

1200

2400

300

4800

9600

For brevity, you can also enter the baud rate on the same line as the ICEMDDN command using the following format:

ICEMDDN,number (RETURN)

number is the first digit of the baud rate.

Specifying Your Terminal Type

After you identify the baud rate, you are asked to select the type of graphics terminal you are using:

GRAPHICS TERMINAL TYPE

1. TEKTRONIX 4014
2. TEKTRONIX 4105
3. TEKTRONIX 4107
4. TEKTRONIX 4109
5. TEKTRONIX 4113
6. TEKTRONIX 4114
7. TEKTRONIX 4115
8. TEKTRONIX 4125
9. CDC VIKING 721
10. CDC IEW 790 WITH TEKEM

Enter the number that corresponds to your terminal type. It is generally written on a plate somewhere on the front of the terminal.

Once you have identified your terminal type, you are asked some additional questions to help ICEM DDN identify the specific characteristics of your graphics terminal. These prompts vary depending upon the type of terminal you are using. The prompts shown here are those that apply to a Tektronix 4115 terminal.

The first prompt is used to direct the menu text (also known as dialog area text) to the appropriate area. In this manual, we assume that you have directed the menu text to your graphics terminal (1. GRAPHICS TERMINAL).

MENU AREA

1. GRAPHICS TERMINAL
2. CDC 722

The next prompt asks you if you want to use the graphics tablet:

TABLET

1. OFF
2. TEKTRONIX OPTION 13
3. TEKTRONIX OPTION 14
4. TEKTRONIX 4957

If you have a tablet and wish to use it, enter the number that corresponds to your tablet type. If not, enter 1 in response to this prompt.

The next prompt asks you if you want to use the local display file (LDF). The LDF is actually memory that is internal to your graphics terminal. The LDF is very useful for rapid repainting, blanking, and deleting of entities. In this manual, we assume that your LDF is on.

LOCAL DISPLAY FILE
1.ON
2.OFF

This prompt appears only on those terminal types that support an LDF.

Finally, you are asked to select whether or not you want to use the local character set. Local character sets are stored in the local memory of your terminal. If on, they can help speed up the generation of text in your part.

LOCAL CHARACTER SET
1.ON
2.OFF

Refer to the ICEM Design/Drafting Introduction and System Controls manual for a complete explanation of each of the selections above.

Entering a Part

To create a new part or retrieve a part you worked on previously, you need to enter the part name and sheet number of the part.

The part name can be from 1 to 70 alphanumeric characters (including blank spaces). The sheet number can be any number from 0 to 9999. This number represents up to 10,000 different drawings associated with a part name.

For now, enter PART1 as your part name and 1 as the sheet number:

```
ENTER PART NAME  PART1 (RETURN)
SHEET NUMBER =   1 (RETURN)
```

By selecting sheet number 1, you are telling ICEM DDN that you wish to work on a drawing identified as sheet 1 of PART1. You can also do sheet 2 of PART1, sheet 3, and so on. The different sheets for any part name are completely independent drawings.

If you enter the part name and number of an old part, ICEM DDN takes you directly to that part. If not, you are asked to select the unit of measure that you are using. In either case, enter:

```
M (RETURN)
```

to turn the display of all menus on.

From the Units of Measure menu, enter 2 and press the RETURN key to work in US customary units.

```
--- UNITS OF MEASURE
1.METRIC (MM)
2.ENGLISH (INCH)
3.ENGLISH (FOOT/INCH)  2 (RETURN)
```

You are then asked to select the drafting standard that you will be using. Enter 1 and press the RETURN key to use 1982 ANSI standard dimensioning.

```
--- DRAFTING STANDARD
1.ANSI 1982
2.ANSI 1973
3.
4.
5.
6.
7.
8.
9.          1 (RETURN)
```

NOTE

Within the Drafting Standard menu, selections 3 through 9 are reserved for DIN, French, British, Swedish, Japanese, and Chinese drafting standards. If your site is using one of these drafting standards, it will appear in the menu.

As soon as you press the RETURN key, the screen clears and the letters ICEMDDN appear on your screen. You are now ready to begin drawing. ICEMDDN is the main menu heading:

```
ICEMDDN  L=0  V=1  D=0.00  P=0  C=0
```

As you become familiar with the use of ICEM DDN, you may wish to use a short cut method to enter information that you are prompted for at the beginning of every session. For example, you can select your terminal type, activate your tablet, and select the unit of measure that you wish to use—all in a single step. To do this, you add parameters to the ICEM DDN call statement using the following format. The call statement is used to make ICEM DDN available to you at your terminal.

ICEMDDN,[baud rate],p1,p2 ... pn (RETURN)

The most frequently used parameters appear in appendix B of this manual. Refer to the ICEM Design/Drafting Introduction and System Controls manual for more information on parameters.

Selecting Menus

You direct ICEM DDN by making selections from a list of options, called a menu. This menu, with the heading of ICEMDDN, is called the top-level menu or the main menu. ICEM DDN's menus are organized in a tree-like structure. Any selection you make from the main menu results in the presentation of a new menu or option at the next lower level. Most choices from this menu result in the presentation of a new menu at the next lower level, and so on. You proceed from the most general choices to the most specific, but eventually each branch of the tree comes to an end beyond which there are no lower level menus. Once you have made your selection, you must specify the details of its development. For example, enter:

9 (RETURN)

to get to the Point menu. A new menu and menu heading appear. Now press:

1 (RETURN)

to select the Screen Position function. You are at the end of this branch. ICEM DDN seeks specifics, asking where you want the point.

You will return to this menu later to study points, but for now, press:

]

to signal operation complete to ICEM DDN. In this case, it means you are done creating points by screen position; ICEM DDN returns you to the menu at the next higher level to make a new selection. Press:

] (RETURN)

to go back to the main menu. Each time you make a selection from a menu you move down a level until you reach the end of a branch. Each time you signal operation complete, you move up a level until you reach the main menu.

Enter:

1 (RETURN)

to get to the Modals and Fonts menu. This is an important menu to which you may return often during ICEM DDN sessions.

Modals and fonts control how ICEM DDN operates. For example, should ICEM DDN use three or six decimal places in displaying numerical quantities? Are lines in your drawings to be drawn with solid or dashed lines? Should entire menus or only the menu headings be displayed? Each of these questions is answered by correctly setting an ICEM DDN modal or font appearing in the Modals and Fonts menu.

For now, enter:

] (RETURN)

to go back to the main menu.

SELECTING MENUS

You have already altered one modal from its default value. When you entered the letter M at the first menu earlier in this terminal session, you were actually setting the menu display modal to on. This causes entire menus to be displayed instead of just the menu headings. If you were to enter that sequence again, you would toggle the menu modal back to off and would again see only menu *headings* on the screen. You did not have to return to the Modals and Fonts menu to do this. This is one of several more direct menu routes ICEM DDN allows. For another, press:

9 (RETURN)

Then press:

1 (RETURN)

to go to SCREEN POSITION on the Point menu.

Now press:

F

and you are back to the main menu, without having to indicate operation complete twice. The F key gets you back to the main menu no matter where you are in the menu tree. Similarly, holding down (CTRL) while pressing the P, L, or A key gets you to the Point, Line, or Arc/Circle/Fillet menu from any other location within ICEM DDN:

Enter:

(CTRL) P + (RETURN) to go directly to the Point menu.

(CTRL) L + (RETURN) to go directly to the Line menu.

(CTRL) A + (RETURN) to go directly to the Arc/Circle/Fillet menu.

If you are using the keyboard, there is one more direct route, called the type-ahead facility, you should be aware of. Rather than waiting for each new menu level to appear before entering the selection for that level, you can type several selections ahead of the display, provided you know the numbers of the menu items you want. For example, suppose you are at the main menu and want to get to SCREEN POSITION of the Point menu. Rather than typing a 9, waiting for the Point menu to be displayed, and then typing a 1 to get SCREEN POSITION, you can type:

9 (TAB) 1 (RETURN)

from the main menu. ICEM DDN stops at SCREEN POSITION of the Point menu without further guidance. This type-ahead facility is particularly helpful when the number of menu levels increases. Simply separate the successive menu choices by pressing the TAB key after each one.

NOTE

With the type-ahead facility, intermediate menu headings are not displayed.

There are several other function control keys used within ICEM DDN. These keys are an important part of your daily work activities. They are located on the terminal keyboard and may be active or inactive, depending upon the context in which they are used. You may have to press the RETURN key after entering a function control key. Whether this is necessary depends on both the type of terminal you are using and the type of entry required.

Table 1-1 lists and describes function control keys.

Table 1-1. Function Control Keys

Key	Title	Description
]	Operation complete	This key tells the system that you have completed the current operation. If you entered all required entries before you pressed the] key or if you wish to accept default values, the current operation is performed and the next logical prompt is displayed. If you did not enter all of the necessary input, you will return to the next higher level menu or prompt, thus cancelling the current operation. The] key is always valid.
[Operation reject	This key tells the system to cancel the current operation or prompt, if possible, and returns to a preceding message. This key is always valid. As with the operation complete key, special uses of the reject key are described in the ICEM DDN manuals.
Y	Yes	This key means yes when entered in response to a prompt. This key is treated as normal data input when you are in the text input mode.
N	No	This key means no when entered in response to a prompt. This key is treated as normal data input when you are in the text input mode.
M	Change menu display	This key changes the menu display status. If the menus are off and you select M, all selected menus are displayed until M is selected again. This key is treated as normal data input when you are in the text input mode.
R	Repaint display	This key repaints the display. After the screen has repainted, you will return to the menu or prompt that was active before you selected R. This key is treated as normal data input when you are in the text input mode.

(Continued)

Table 1-1. Function Control Keys (Continued)

Key	Title	Description
G	Change grid status	This key changes the grid activation status. The grid, which appears as an array of dots on your screen, is used for screen position selection. If the grid is active and you select G, the system deactivates the grid. If you select G again, the system reactivates the grid. This key is treated as normal data input when you are in the text input mode.
Z	Window (zoom)	This key takes you directly to the Zoom menu, which allows you to modify the position and size of the part on the screen. After you select from this menu and give the new zoom constraints, the display is redrawn and you return to the menu or prompt that was active before you selected Z. This key is treated as normal data input when you are in the text input mode.
D	Change depth	This key causes the prompt 1.DEPTH=xxxx to display with the current zt depth (the z-axis value of the work plane in front of or behind the plane of the screen). You may modify it as desired. After you enter a value or press the [or] key, you return to the menu or prompt that was active before you selected D. This key is treated as normal data input when you are in the text input mode.
(CTRL) F + (RETURN) or F	Return to the main menu	These keys take you to the main menu. They are treated as normal data input when you are in the text input mode.
(CTRL) P + (RETURN)	Go to the Point menu	This key takes you directly to the Point menu. It is a valid key at any time.
(CTRL) L + (RETURN)	Go to the Line menu	This key takes you directly to the Line menu. It is a valid key at any time.

(Continued)

Table 1-1. Function Control Keys (Continued)

Key	Title	Description
CTRL V + RETURN or V	Other options	This key allows you to set modals associated with the current function, display a menu that was bypassed because of a default modal setting, or display additional menu options.
CTRL E or E	Display the Entity Selection menu	This key is used to display the Entity Selection menu from which you can choose any entity selection method.
CTRL A + RETURN	Go to the Arc/ Circle/Fillet menu	This key takes you directly to the Arc/Circle/Fillet menu. It is a valid key at any time.
CTRL D + RETURN	Delete last entity	This key is used to delete the most recently created entity. It is a valid key at any time.
C or space bar	Read graphics cursor	These keys tell the system to read the position of the graphics cursor. They are valid only when the cursor is active. During entity selection, the C key is the standard key used to capture the closest entity whose type is legal for the current operation.
!	Capture and accept data	This key captures and saves the value of a parameter that has been displayed by ICEM DDN and associates that value with a variable name. Subsequently, the variable name can be typed in response to a parameter prompt. This has the same effect as typing in the value that was captured.
?	Help	This key is used to explain the current prompt. The explanation may consist of several paragraphs. You must enter either] or [after each paragraph. Enter] to show the next paragraph, if any, and enter [to end the help message display. If you use this key during entity selection, a list of selectable entities is displayed.
P	Select point	This key is used to capture the closest point to the graphics cursor. All other selectable entity types are temporarily locked out.
L	Select line	This key is used to capture the closest line to the graphics cursor. All other selectable entity types are temporarily locked out.

(Continued)

Table 1-1. Function Control Keys (Continued)

Key	Title	Description
P	Select point	This key is used to capture the closest point to the graphics cursor. All other selectable entity types are temporarily locked out.
L	Select line	This key is used to capture the closest line to the graphics cursor. All other selectable entity types are temporarily locked out.
A	Select arc, circle, or fillet	This key is used to capture the arc, circle, or fillet closest to the graphics cursor. All other selectable entity types are temporarily locked out.
O	Select other curves	This key is used to capture conics, composite curves, vectors, point-set curves, three-dimensional splines, and machining curves.
S	Select spline	This key is used to capture the closest two-dimensional spline. All other selectable entity types are temporarily locked out.
T	Select text	This key is used to capture dimensioning entities with text only. These are linear, circular, diameter, and angular dimensions, labels, and notes.
H	Select section lining	This key is used to capture the closest section lining. All other selectable entity types are temporarily locked out.
N	Select N/C toolpath	This key is used to capture the closest N/C toolpath. All other entity types are temporarily locked out. The selectable positions on a toolpath are at any corner or GOTO point along its path.

Using the Tablet

Control Data supports a graphics tablet with which you can enter menu selections, text, and graphic input into ICEM DDN. Designing a part using the tablet is often faster than designing it using the keyboard. Refer to the ICEM Design/Drafting Introduction and System Controls manual for information on how to set up the tablet and use the tablet overlays.

If you didn't activate your tablet at the beginning of the session, you can activate it by making the following menu selections at the keyboard:

- Select 1.MODALS AND FONTS from the main menu.
- Select 15.DATA ENTRY MODE from the Modals and Fonts menu.
- Select 1.MENU/CURSOR MODE from the Data Entry Mode menu.
- Select 4.TABLET/TABLET from the Menu/Cursor Mode menu.

Next, you are asked to select the type of tablet you are using:

```
411X TABLET OPTION
1.OPTION 13
2.OPTION 14
3.4957 TABLET
```

Enter the number that corresponds to your tablet type. You can now enter menu, text, and graphic data from the tablet. You can still, however, enter data from the keyboard but the RETURN key is necessary after each entry.

The tablet accepts data when the READY light on the tablet power module is on for the Tektronix model 4014 and 4016 and when the graphics cursor appears for the 4113, 4114, 4115, and 4125. To send data from the tablet, press the pen against the tablet. This lights the PEN light on the power module for the 4014 and 4016. It is important to remember to pick up the pen soon after pressing it against the tablet to prevent sending extra, useless, often erroneous data to ICEM DDN.

The tablet is divided into three separate areas. The top third of the tablet has squares on it that represent the keys or functions used most frequently. This area on the tablet is fixed and always remains enabled.

The lower two-thirds of tablet page can be changed, depending upon the menus you wish to use. These areas on the tablet are called the upper and lower areas. The page of menus which is enabled in either the upper or lower area of the tablet is called an overlay.

When you use the tablet, you do not need to follow the tree-like structure of the menus as you do when using the keyboard. For example, to create a point, you simply select the square for the specific point operation directly on the tablet overlay without first selecting the point menu. The correct menu choices are automatically entered in response to your tablet selection.

To try it, locate the Point block of squares. Press the pen in the Screen Position square. The Screen Position square is located in the upper area of the tablet, on the overlay called Basic Functions Upper (BFU).

BFU	Point		Line		Arc	
	Screen Position	Transform XT,YT,ZT Model X,Y,Z	Screen Position	Transform XT,YT,ZT Model X,Y,Z	Screen Position and Radius	Transform XT,YT,ZT,R Model X,Y,Z,R
Delta	Curve End Point	Join 2 Points	Join 2 Curves	Center Point and Radius	Center Point Point on Edge	
Regenerate Spline Points	On an Arc at an Angle	Parallel to a Line at a Distance	Thru a Point & Parallel to a Line	Thru 3 Points	Tangent to 3 Curves	
Arc Center	Intersect Curve On 1st Curve On 2nd Curve Current Depth True		Thru Point Horiz Vert Other	Thru Point Perpto Line	Fillet Specify Radius Unspec Radius	
Along a Curve at a Distance	On a Curve at a Parameter	Thru Point Tangent to Curve	Parallel to Line Tangent to Curve	Center Point Tangent Line	Auto Ctr No Trim	Auto Ctr Trim

Next, you are asked to indicate the location of a point:

INDICATE POSITION

Move the pen across the tablet surface, keeping it within approximately 3 mm (0.125 in) of the surface or barely on the surface. The graphics cursor on the terminal screen follows the path of the pen. ICEM DDN accepts a pen position as input only when you press the pen against the surface.

Try defining a few points by pressing the pen on the tablet in several places. When you have finished, press the pen in the] square located on the fixed area of the tablet.

You can also create points with the tablet by entering their coordinates. Press the pen in the Transform XT, YT, ZT square in the Point block. This square is also located on the BFU overlay.

BFU	Point		Line		Arc	
	Screen Position	Transform XT,YT,ZT Model X,Y,Z	Screen Position	Transform XT,YT,ZT Model X,Y,Z	Screen Position and Radius	Transform XT,YT,ZT,R Model X,Y,Z,R
Delta	Curve End Point	Join 2 Points	Join 2 Curves	Center Point and Radius	Center Point Point on Edge	
Regenerate Spline Points	On an Arc at an Angle	Parallel to a Line at a Distance	Thru a Point & Parallel to a Line	Thru 3 Points	Tangent to 3 Curves	
Arc Center	Intersect Curve On 1st Curve On 2nd Curve Current Depth True		Thru Point Horiz Vert Other	Thru Point Perpto Line	Fillet Specify Radius Unspec Radius	
Along a Curve at a Distance	On a Curve at a Parameter	Thru Point Tangent to Curve	Parallel to Line Tangent to Curve	Center Point Tangent Line	Auto Ctr No Trim	Auto Ctr Trim

ICEM DDN then asks you to enter the xt- and yt-coordinate values for the point. Press the pen in the numbered squares on the fixed area of the tablet to enter these values.

Press the RETURN square after entering the xt-coordinate and the] square after entering the yt-coordinate to tell ICEM DDN you have finished with that entry. A second RETURN is not required after the] because ICEM DDN inserts this automatically with tablet input only. When you have finished, press the pen in the] square.

To define a line using the tablet, locate the Line block and press the pen in the Join 2 Points square.

BFU	Point		Line		Arc	
	Transform XT.YT.ZT Screen Position	Model X.Y.Z	Transform XT.YT.ZT Screen Position	Model X.Y.Z	Transform XT.YT.ZT.R Screen Position and Radius	Model X.Y.Z.R
Basic Functions Upper	Delta	Curve End Point	Join 2 Points	Join 2 Curves	Center Point and Radius	Center Point Point on Edge
	Regenerate Spline Points	On an Arc at an Angle	Parallel to a Line at a Distance	Thru a Point & Parallel to a Line	Thru 3 Points	Tangent to 3 Curves
	Arc Center	Intersect Curve On 1st Curve On 2nd Curve Current Depth True	Thru Point Horiz Vert Other	Thru Point Perpto Line	Fillet Specify Radius Auto Ctr No Trim	Unspec Radius Auto Ctr Trim
	Along a Curve at a Distance	On a Curve at a Parameter	Thru Point Tangent to Curve	Parallel to Line Tangent to Curve	Center Point Tangent Line	Center Point Tangent Circle

ICEM DDN then asks you to specify an existing point with the prompt:

INDICATE POINT

Use the pen to locate and select two points. ICEM DDN uses the lower left corner of the graphics cursor on the screen to determine the location of graphic input positions.

When you press the pen near a point, the ICEM DDN tablet interface uses C as the entity select character. (Table 1-1 contains a complete list of entity select characters). To use P or any other entity select character, press the pen in the P square or in the appropriate square of another entity select character *before* you indicate an entity using the tablet pen. For example, if you touch the P square, only points are selectable. You can change the entity select character anytime during graphic input by simply touching another entity selection square. The entity select character is automatically reset to C when you exit from a menu selection.

To practice using the tablet, try creating several more lines between points. Press the pen in the] square to tell ICEM DDN that you have finished creating lines.

USING THE TABLET

To delete entities using the tablet, locate the Delete block on the BFU overlay and press the pen in the Select square.

Arc		Display Control			Display Views					BFU
Screen Position and Radius	Transform X.T.Y.T.Z.T.R	Change Work Space	Chg Wk View	Top-6 Front-1	Front-1 Right-3	ISO1-8	One	Last Layout		
Center Point and Radius	Model X.Y.Z.R	Define Work Space	Define Aux View	Align Views	Two Views		Std Four	Std Eight	Last Multi View	
Thru 3 Points	Center Point Point on Edge	Delete			Run	List	Recover			
Tangent to 3 Curves		Region In	Region Out	All Points	Level	Continue	Change Library	Manage Files		
Fillet		Blank			Selection Methods					
Specify Radius	Unspec Radius	Select	Level	All	All of a Type	Screen	Chan	Last Entity	Name	
Auto Ctr No Trim	Auto Ctr Trim	Unblank			Region In	Region Out	All Dsplyd	Modals		
Center Point Tangent Line	Center Point Tangent Circle	Constraint Sets			Trim/Extend Curves					
		Use	Clear	List	Include Level	Include Font	Include Type	Include Color	Include Pen	
		Define	Name	Delete	Exclude Level	Exclude Font	Exclude Type	Exclude Color	Exclude Pen	
		One End Screen			Two Ends Screen		Middle Screen		Two Curves at Intersection	
		One End Bound			Two Ends Bound		Middle Bound			

Use the pen to select one or two points and a line to delete. Press the] square to complete the deletions.

NOTE

If you accidentally select the wrong entity to be deleted, just enter [(operation reject). In doing this, you are telling the system not to delete that entity. Once you reject an entity using [, you won't be able to select it again for deletion until you select another entity or you exit and reenter the delete function.

All of the ICEM DDN menus and operations cannot fit on the main tablet page. For this reason, five other overlays are provided to supply all the menus and operations. Before you can use another overlay, you must tell ICEM DDN which one you want to use and in which area you will place it.

Using the drafting overlay as an example, locate the DFT Upper square in the top row of the fixed area. Press the pen in this square to tell ICEM DDN that you want to use this overlay.

BFU	BFL	AD Upper	AD Lower	DFT Upper	DFT Lower	N C Upper	N C Lower	DBM Upper	DBM Lower	DC Upper	DC Lower	SC1 Upper	SC2 Lower	USER Upper	USER Lower	Change Level Pen	Change Color	Delete Last Entity	
[Y]			N	Repaint		Return		Display Current Menu	File	Continue	Susprd	Upper Case	Do Not File	Pan			
Operation Reject	Yes	Operation Complete			No	Local	Most		Note	Quit	New Part	Lower Case	Quit	Continue	Susprd	New Part	↑	↓	
A Arc	B	C Curves	D Depth	E	F	G Grid	H Section	I Space	? Help	! Emb Txt	7	8	9	/	Zoom				
J	K	L Line	M Menus	N Tool Pth	O	P Point	Q	R Repaint	Back Space	Man Menu	4	5	6	*	Auto Max-Min	Diag Position	1/2 X	2X	
S 2D Sph	T Text	U	V	W	X	Y	Z Zoom	(o) Degree	±	:	Entity Select Menu	1	2	3	-	Save as Base	New Center	New Center Scale	Enter Scale
!	"	#	\$	%	&	'	()	'	\	Other Options	0	.	=	+	Return to Base	Retrieve Zoom	Name Zoom	Zoom from Hdcopy	
11	12	13	14	15	16	17	18	19	'	\	0	.	=	+	Change Zoom Mode				
															Entire Layout	Work View	Select Window	All Windows	

The following message is then displayed:

DFT UPPER ACTIVATED

After activating the DFT overlay, place it on the upper portion of the tablet (over BFU). ICEM DDN now accepts drafting operations rather than operations from the Basic Functions Upper overlay.

NOTE

Many users find it helpful to attach overlays to the tablet using some type of nonpermanent tape.

On the drafting overlay, press the pen in either the Horizontal Dimension square or the Vertical Dimension square.

DFT	Create Drafting Entities				Modify Drafting Entities			
	Horizontal Dimension	Vertical Dimension	Parallel Dimension	Parallel Dimension	New Text Origin	New Display Values		
Angular Dimension	Radius Dimension	Diameter Dimension	Rect Dim		Add Change Tolerance	Balloon Modification		
			Define Origin	Horiz	Vert	Add Change Limits	Datum Target Modification	
Thickness Dimension	Taper Dim Conical Flat		Chamfer Dimension	Arc Length Dimension	Delete Tolerance Limits	Feature Frame Modification		
Note	Datum Target	Datum Feature	Geometric Tol		Modify Level Pen	Basic		
			Φ	005	A	B	Modify Decimal Places	(Ref)
Label	Balloon	Arrowhead		Surf Texture		Character Set		Dimension Origin
		Curve End	On Curve	Arrow	Entity	Std	User	
Detail Magnify	Section Lining	Section Arrow	Center Line		Modify Text		Extension Line Status	
			Circle	Point	Bolt Circle	Delete Line	Insert Line	Dimension Line Status

Use the pen to select two curves and to indicate the text origin. ICEM DDN displays a horizontal or vertical dimension. Press the] square to exit from that tablet selection.

You must tell ICEM DDN when you want to finish using the drafting overlay and return to the previous overlay. To do this, press the pen in the BFU square in the top row of the fixed area if the drafting overlay is currently over the upper portion of the tablet, or press the pen in the BFL square if your drafting overlay is currently over the lower portion of the tablet.

To practice using the tablet, try performing the exercises previously described in this section.

Saving Your Drawing

Much of your daily work will involve creating new parts and revising old parts. For this reason, it is important that you understand the use of the file/exit sequence.

If you are using the tablet, you can file your part in single step by touching one of the FILE tablet squares with your tablet pen.

If you are using the keyboard, select 4.FILE/EXIT from the main menu to display the following menu:

- FILE CURRENT PART/EXIT ICEM DDN
- 1.FILE - CONTINUE CURRENT PART
- 2. - GET DIFFERENT PART
- 3. - QUIT SESSION
- 4. - SUSPEND SESSION
- 5.DO NOT FILE - CONTINUE FROM LAST FILE
- 6. - GET DIFFERENT PART
- 7. - QUIT SESSION
- 8. - SUSPEND SESSION

If the current part is a new one, you will probably want to save it for future use using one of the File options.

If the current part is an old one, you may or may not want to save the work that you did since the last time the part was saved. When you retrieve an old part, you are really working on a temporary or scratch copy of the part. It isn't until you file the part that the changes you make become permanent. If you select one of the File options, ICEM DDN replaces the old version of the part with the newer, revised version. If you select one of the Do Not File options, the old version in storage remains unchanged.

NOTE

In this section, only the first three choices in the File/Exit menu are discussed. For a detailed explanation of all of the File/Exit options, refer to the ICEM Design/Drafting Introduction and System Controls manual.

If you want to save the work you have done since the last time you filed your part, enter the following:

Keyboard Selection

Select 1.FILE - CONTINUE CURRENT PART from the File Current Part/Exit ICEM DDN menu.

Tablet Selection

File	
Continue	Suspnd
Quit	New Part

As soon as ICEM DDN has successfully filed your part, the following message is displayed:

FILE COMPLETE

NOTE

It is important that you save your drawing every 15 to 20 minutes so that it can be easily recovered if your computer system goes down or if you make a mistake such as accidentally deleting your previous work.

If you want to file the current part and begin work on a different part, enter the following:

Keyboard Selection

Select 2.FILE - GET DIFFERENT PART from the File Current Part/Exit ICEM DDN menu.

Tablet Selection

File	
Contnue	Suspnd
Quit	New Part

Next, you are asked to identify the part you want to work on:

ENTER PART NAME
SHEET NUMBER =

You can enter LIST in response to the ENTER PART NAME to obtain a list of the drawings you have saved.

At this time, you can either enter a new part name, if you want to start a new drawing, or you can enter the name of an old part that you want to continue working on. If you enter a new name (or a new sheet number for an old part), you again see the Units of Measure and Drafting Standard menus. If you enter an old part name, you are not asked to reenter this information since the part already exists and has this information stored with it.

If you want to file the current part and exit from ICEM DDN, enter the following:

Keyboard Selection

Select 3.FILE - QUIT SESSION from the File Current Part/Exit ICEM DDN menu.

Tablet Selection

File	
Contnue	Suspnd
Quit	New Part

As soon as you exit from ICEM DDN, you will enter the Interactive Facility (IAF) of the Network Operating System (NOS). ICEM DDN has replaced your part in the parts file, which is a permanent file under your user name. The next time you log in to the system, you can attach the parts file before your ICEM DDN session and continue working on your filed part.

When you have finished, log out of the computer by typing BYE after the next /:

BYE (RETURN)

After the message LOGGED OUT appears, hang up the phone, press the DIALOG key to remove text from your screen, and put the terminal in local mode (if your terminal is hardwired) by pressing the LOCAL key.

Your terminal session is now complete.

Reader's Guide

The ICEM DDN Instant provides a reproduction of the ICEM DDN menus.

The ICEM DDN manuals contain explanations of the prompts that the system issues. They also explain the input needed for each procedure in which you are engaged. These manuals should be available to you during your sessions at the terminal.

The ICEM DDN Tablet Overlays are plastic sheets to be used with the tablet.

Control Data manuals applicable to the ICEM DDN system are listed in the preface.

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In this chapter, you will learn how to create points, lines, and arcs using ICEM DDN. Then, using this geometry, you are shown how to use the Trim/Extend Curves and Delete functions.

Defining Points

There are several ways to define points within the Point menu. In this section, four of these ways are discussed:

- 1.SCREEN POSITION
- 2.KEY-IN
- 3.POLAR
- 4.DELTA

If you are using the keyboard, you can enter the Point menu either by selecting number 9, if you are in the ICEM DDN main menu, or by typing:

(CTRL) P + (RETURN)

If you are using the tablet, you can go directly to each of these menu selections by touching that square with your tablet pen.

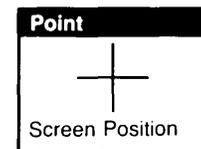
Screen Position

With the Screen Position menu selection, you can define a point on the screen at the position you indicate using the graphics cursor.

Keyboard Selection

Select 1.SCREEN POSITION from the Point menu.

Tablet Selection



You are first asked to select the position of the point on the screen:

INDICATE POSITION

If you are using the keyboard, the graphics cursor comes up on the screen (if it does not appear, twirl the thumbwheels, located at the right of the keyboard, until it does). Use the thumbwheels to move the cursor to the upper left corner of the screen; press C. C is the entity select character that tells ICEM DDN to accept the current position of the graphics cursor (refer to table 1-1).

If you are using the tablet, use the pen to move the graphics cursor to the upper left corner of the screen and press down with the tablet pen.

A small x appears at the position of the graphics cursor. This tells you that there is now a point located at that position on the screen. Move the graphics cursor and create another point. Repeat this several times to create points near each of the four corners of the screen. When you are finished, enter] to return to the Point menu, so you can learn how to use a different operation to define a point.

Keying in Points

Instead of indicating the position of the point by moving the graphics cursor to that position on the screen, you can enter the exact (x,y,z) coordinates of a point.

Keyboard Selection

Select 2.KEY-IN from the Point menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

Tablet Selection

Transform XT.YT.ZT
Model X.Y.Z

NOTE

The transform and model coordinate systems are discussed in detail in the ICEM Design/Drafting Introduction and System Controls manual.

You are first asked to enter numerical values in response to the following display:

```

1.XT      =
2.YT      =
3.ZT      = 0.0000

```

The prompt displays once in its entirety and then redisplayes each line in the prompt individually. This is to show you what information ICEM DDN expects you to enter.

Because you are working in US customary units, the screen is laid out in inches, actual scale, with (0,0) in the extreme lower left corner of the screen.

Enter 1.5 as the xt value:

1.5 (RETURN)

Enter 1.5 as the yt value:

1.5 (RETURN)

The cursor (the blinking square that indicates ICEM DDN can accept input) pauses after the yt; it does not go on to zt nor does it plot your point. At this time, you have completed the coordinates of a point at (1.5, 1.5, 0). The zt value defaults to the current depth, which is initially 0.

You must now decide whether or not you want to accept this point. Any time ICEM DDN requests input through a series of entries, as here, you are given the opportunity to change your mind before ICEM DDN uses the information.

If all of the values are acceptable, enter] (and RETURN if you are using the keyboard).

If you want to change a coordinate value, enter the number of the coordinate choice (for example, 3 for the zt-coordinate) followed by RETURN. Then, enter the new value and again, either accept the point with a] or change another coordinate value by entering the number of that coordinate choice.

To practice using this function, try performing the following exercise:

1. Define points at the following locations:

Location	Description
(4,5)	xt=4, yt=5, and zt=0
(6,1)	xt=6, yt=1, and zt=0
(1,4,2)	xt=1, yt=4, and zt=2

2. Enter] to return to the Definition Space menu.
3. Enter] again to return to the Point menu.

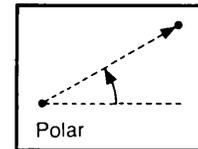
Polar Points

With the Polar menu selection, you can create a new point at an angle and distance from an existing point.

Keyboard Selection

Select 3.POLAR from the Point menu.

Tablet Selection



You are first asked to select an existing point:

INDICATE POINT

Position the graphics cursor near a point and select it using the C key or the tablet pen. ICEM DDN displays an attention indicator on the point to indicate that you have selected that particular point. If the attention indicator does not appear on the correct point, reject the selected point by entering [and try selecting it again. The correctly selected point becomes the base point.

Next, you are asked to enter the angle and straight line distance from the base point:

1. ANGLE =
2. RADIUS =

Enter the angle in degrees followed by RETURN, enter the distance in the units of measure that you selected, and enter] to indicate that the values you entered are correct.

DEFINING POINTS

A small x appears at the location where the point has been created. The angle that you entered is measured on the display from the direction of the positive x -axis, that is, from an imaginary horizontal line that starts at the base point and extends to the right of that base point. A positive angle is measured in a counterclockwise direction from the positive x -axis; a negative angle is measured in a clockwise direction from the positive x -axis.

If you enter $]$ instead of the numeric information, you return to the INDICATE POINT prompt.

To practice using this function, try performing the following exercise:

1. Create four points (north, south, east, and west) of a base point, each 1 inch from that base point.
2. Use the Polar function to define two points which, with the base point, serves as the vertices of an equilateral triangle with 1.5-inch sides. Later in this chapter you will create the lines that connect these points.

Delta Points

If a point already exists on your screen, you can create a new point at an (x,y,z) distance from that existing point using the Delta menu selection.

This selection is very similar to the Key-In selection *except* that you select the base point from which the (x,y,z) distance is measured. In contrast, the Key-In selection always uses the origin [the point $(0,0,0)$] as the base position from which to measure.

Drawing Lines

The Line menu provides several ways to create lines. In this section, we will show you three ways to create a line:

1. SCREEN POSITION
2. KEY-IN
3. JOIN 2 POINTS

If you are using the keyboard, you can get to the Line menu either by returning to the main menu and selecting 10, or by going directly to the Line menu by typing:

(CTRL) L + (RETURN)

If you are using the tablet, you can go directly to each of these menu selections by touching that square on the tablet with your tablet pen.

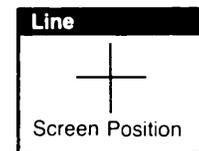
Drawing Lines By Screen Position

You can draw lines by indicating the positions of their endpoints on the screen.

Keyboard Selection

Select 1.SCREEN POSITION from the Line menu.

Tablet Selection



You are first asked to select the endpoints of the line:

INDICATE POSITION

Indicate the position of one endpoint of the line using the graphics cursor and the C key, or the tablet pen. A very small dot appears at the position you select as the first endpoint. Move the graphics cursor and indicate the position of the other endpoint. As soon as you have indicated the second endpoint, a line is drawn connecting these two end positions. At that time, you can enter two more positions for the endpoints of another line to be created or enter] to return to the Line menu.

To practice using this menu selection, try performing the following exercises:

1. Create two lines that connect diagonal corners of the display and that cross in the (approximate) center of the screen.
2. Create a box or a rectangle in the same way.
3. Enter] to return to the Line menu.

Drawing Lines by Keying in Coordinates

You can create a line by specifying the coordinates of the endpoints in the same way that you previously specified the coordinates of a point.

Keyboard Selection

Select 2.KEY-IN from the Line menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

Tablet Selection

Transform X,T,Y,T,ZT
Model X,Y,Z

ICEM DDN then displays the following prompt:

```

1.XT1      =
2.YT1      =
3.ZT1      = 0.0000
4.XT2      =
5.YT2      =
6.ZT2      = 0.0000

```

At this time, ICEM DDN is waiting for you to enter the coordinates of the two endpoints of the line you want to create. Just as when you entered the coordinates of a point earlier in this session, the values that you now enter are interpreted as distances from the origin [the point (0,0,0)] using the units of measure that you selected for the part (in this case, inches).

The origin is located in the extreme lower left corner of the display in the plane of the screen (that is, neither behind nor in front of the screen).

The prompt first displays in its entirety and then redisplay each of the prompt lines individually, *except* that it skips the display of lines 3 and 6 (ZT1 and ZT2):

```

1.XT1      = (the x value of the first endpoint)
2.YT1      = (the y value of the first endpoint)
4.XT2      = (the x value of the second endpoint)
5.YT2      = (the y value of the second endpoint)

```

The system then pauses, just as it did when you were entering the (x,y,z) coordinate values for a point earlier in this session. At this time, you can:

- Accept the default z value of 0.00 by entering] following the YT2 coordinate value.
- Enter a value other than 0 for ZT1 or ZT2 by entering:

3 (RETURN)

or

6 (RETURN)

followed by the new ZT1 or ZT2 value and].

To practice using this menu selection, try performing the following exercises:

1. Create lines between the following pairs of points.
 - (2,2,2) and (3,4,0)
 - (1,1,1) and (3,1,0)
2. Create an X formed by two lines that are perpendicular to each other and have a length of 2 inches.

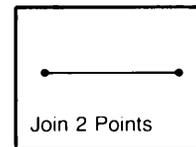
Drawing Lines By Joining Two Points

You can also draw a line between two existing points, the advantage being that you can define a line that starts precisely at the location of one existing point and ends precisely at the location of another. Before you can use this menu selection, however, you must have at least two points on the screen; if you have none in the display, use one of the operations in the Point menu to create some (refer to Defining Points earlier in this chapter).

Keyboard Selection

Select 3.JOIN 2 POINTS from the Line menu.

Tablet Selection



You are first asked to select the endpoints of the line:

INDICATE POINT

Select the point that is to be one endpoint of the line by moving the graphics cursor near that point and selecting it using the P key or the tablet pen. P is the entity select character that selects points (refer to table 1-1).

Because the display contains two different entities (points and lines), it is helpful to use the P key, which selects only points, rather than the C key, which selects the entity closest to the graphics cursor.

If a point and a line are so close together that positioning the graphics cursor precisely on the point is difficult, use the P key (rather than the C key) to register your selection, and the line is ignored. Or, if you want to select the line, use the L key. There are also entity select characters for selecting only arcs or circles (A key), curves (other than splines) constructed from the Other Curves menu (O key), splines (S key), and text (T key). These entity select characters help in selecting items from a crowded display.

The entity select characters are also valid when using the tablet. To activate an entity select character, touch the tablet square with that letter on it *before* you make your selection. The entity select character remains in effect until you exit from the menu function, at which time, the entity select character is automatically set back to the default character, C.

If an attention indicator appears on the point you selected, ICEM DDN captured the correct point. If not, enter [(operation reject) to erase that selection and try again. Select a second existing point for the other endpoint. A line between the two points that you indicated then displays. At this time, you can define additional lines in the same way or go back to the Line menu by entering].

To practice using this menu selection, try the following exercise:

1. Use the Join 2 Points menu selection to create the triangle formed by the three points defined as vertices (of an imaginary equilateral triangle).

Drawing Circles

You can draw circles and arcs using the Arc/Circle/Fillet menu. For each menu selection, you are asked to provide information that ICEM DDN needs to determine:

- The location of the circle or arc.
- The radius.
- The beginning and ending angles for the arc. For a full circle, these angles are 0° and 360° . An arc is a portion of a full circle.

In this section, you will learn three ways in which to create arcs or circles:

- 1.SCREEN POS AND RADIUS
- 2.KEY-IN CENTER AND RADIUS
- 3.CENTER AND RADIUS

If you are using the keyboard, you can get to the Arc/Circle/Fillet menu by returning to the main menu and selecting 11, or by typing:

(CTRL) A + (RETURN)

If you are using the tablet, you can go directly to each of these menu selections by touching that square on the tablet with your tablet pen.

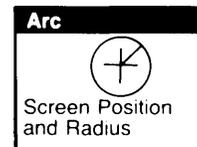
Drawing Circles by Screen Position and Radius

You can draw a circle by using the graphics cursor to select the location of the center of the circle.

Keyboard Selection

Select 1.SCREEN POS AND RADIUS from the Arc/Circle/Fillet menu.

Tablet Selection



The following prompt displays:

INDICATE POSITION

Select the position that you want to use as the center of your circle using the graphics cursor and the C key, or the tablet pen.

After you have indicated the center, the next prompt appears:

1.RADIUS =0.2500
 2.STARTING ANGLE =0.0000
 3.ENDING ANGLE =360.0000

Enter the radius, the starting angle, and the ending angle for the circle or arc that you want to create. Starting and ending angles are measured from the positive x-axis (a horizontal line from the center of the circle to the right) and in a counterclockwise direction. For example, 90° is straight up and 180° is straight to the left.

If the currently displayed value is the same as the value that you wish to enter, it is not necessary to reenter that value. Instead, enter RETURN to move to the next prompt in the sequence. If the rest of the values in the list are correct, you can enter] to accept them. For example, to create a full circle with a radius of 0.5 inch, you would simply enter .5 as the radius value followed by]:

```
1.RADIUS           =0.2500           .5 ]
2.STARTING ANGLE  =0.0000
3.ENDING ANGLE    =360.0000
```

To practice using this menu selection, try the following exercises:

1. Create circles with the following radii: 1/4, 1/2, 1, and 2 inches.
2. Create four semicircles: one right-side up, one upside down, and two standing on their sides facing opposite directions.

Drawing Circles by Keying in Center

You can create a circle or an arc by entering the (x,y,z) coordinates of the center.

Keyboard Selection

Select 2.KEY-IN CENTER AND RADIUS from the Arc/Circle/Fillet menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

Tablet Selection

Transform XT.YT.ZT.R

Model X.Y.Z.R

Enter the (xt,yt,zt) coordinates of the center of the circle followed by]:

```
1.XT =
2.YT =
3.ZT = 0.0000
```

After you have entered these values, you are asked to enter the radius and the starting and ending arc angles.

```
1.RADIUS           =0.5000
2.STARTING ANGLE  =0.0000
3.ENDING ANGLE    =360.0000
```

The arc or circle appears on your screen as soon as you enter]. You are again prompted to enter the coordinates for the center of the circle so that you can create another circle.

After drawing several circles, enter] to return to the Arc/Circle/Fillet menu.

To practice creating circles, try the following exercises:

1. Define circles with a 1-inch diameter at each of the following locations:

(1,1,0), (2,2,0), (1,4,0)

2. Draw five concentric circles with the center at (4,5) and with radii of 1.1, 1.2, 1.3, 1.4, and 1.5.

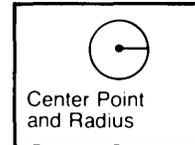
Drawing Circles by Center and Radius

You can draw a circle using an existing point as the center of your circle.

Keyboard Selection

Select 3.CENTER AND RADIUS from the Arc/Circle/Fillet menu.

Tablet Selection



You are first asked to select a point in the display that serves as the circle center:

INDICATE POINT

Use the graphics cursor to select a point. As you have already done in this session, enter the radius and the starting and ending angles. Since you can create many circles with the same center point, this is the best method for constructing concentric circles.

To help you learn how to use this menu selection, try the following exercises:

1. Pick an existing point on the screen. Define arcs that have radii of 0.25, 0.5, 1, and 1.5 and that are in the northeastern, northwestern, southwestern, and southeastern quadrants, respectively. (Each arc should extend through 90°.)
2. Create three concentric circles around the same point.

Trimming Curves

An important part of the drafting process is to clean up the drawings you have created. The Trim/Extend Curves menu selections give you the tools to do this automatically. With these selections, you can trim away a segment of an entity or extend an entity to reach a selected boundary.

Keyboard Selection

Select 12.OTHER CURVES from the main menu.

Select 8.TRIM/EXTEND CURVES from the Other Curves menu.

Tablet Selection

Trim/Extend Curves			
One End Screen 	Two Ends Screen 	Middle Screen 	Two Curves at Intersection 
One End Bound 	Two Ends Bound 	Middle Bound 	

The Trim/Extend Curves menu is as follows:

- ```

TRIM/EXTEND CURVES
1.ONE END - SCREEN POSITION
2. - CURVE
3.TWO ENDS - SCREEN POSITION
4. - CURVE
5.MIDDLE - SCREEN POSITION
6. - CURVE
7.TWO CURVES AT INTERSECTION

```

### NOTE

---

Since all of the Trim/Extend Curves operation described in this user's guide are those that use curves as boundaries, we have limited our discussion here to the Curve options. The Screen Position options are described in detail in the ICEM Design/Drafting Basic Construction manual.

---

If you are trimming one end of a curve, you are asked to indicate a boundary from which you would like to trim:

**INDICATE BOUNDARY**

Whenever you see this prompt, ICEM DDN is asking you to select a curve that will act as a boundary during the trim operation. A curve can be a point, line, or arc.

If you are trimming two ends or the middle of a curve, you are asked to indicate the boundaries from which to trim:

**INDICATE FIRST BOUNDARY**  
**INDICATE SECOND BOUNDARY**

## TRIMMING CURVES

As soon as you have selected your boundary or boundaries, you are asked to select the curve or curves you wish to trim using the boundary or boundaries you just indicated:

### INDICATE CURVE

You may select one or several curves (up to 20 curves maximum). When you have selected all of the curves you wish to trim, enter ].

If you are trimming two ends of a curve or the middle of a curve, they are then trimmed automatically.

If you are trimming one end of a curve, you are asked to indicate on which side of the boundary you would like the curves to be trimmed:

### INDICATE SIDE

The side you choose is *the side that is trimmed away*.

If you are trimming two curves that intersect, the prompts vary from those above. Instead of indicating bounds, you are asked to indicate the two entities that are to be trimmed, near the endpoints of each of the lines:

### INDICATE TRIM END 1

This prompt is repeated for the second curve:

### INDICATE TRIM END 2

When trimming two intersecting curves, remember to select them near the ends that are to be trimmed.

## Deleting Entities

Deleting an entity removes it from your display.

If you are using the keyboard, you must first go back to the main menu (use the F function control key) and select the Delete menu.

If you are using the tablet, you can go directly to the Delete menu selections by touching the tablet square associated with that selection.

There are several ways to delete unwanted entities from your display. In this section, four of these ways will be discussed:

DELETE  
1.SELECT

Enter **(CTRL)** E to display additional entity selection methods:

CONSTRAINT SET=NONE  
ENTITY SELECTION  
1.SCREEN SELECT  
2.CHAIN  
3.REGION IN

·  
·  
·

2.LEVEL RANGE  
3.ALL POINTS

## Deleting Single Entities

The Select option provides several different ways in which to delete entities, the most common of which is to select the entities that are to be deleted, one at a time, using the graphics cursor.

### Keyboard Selection

Select 1.SELECT from the Delete menu.

### Tablet Selection

| Delete    |            |            |       |
|-----------|------------|------------|-------|
| Select    |            | Screen     | Chain |
| Region In | Region Out | All Points | Level |

Using the graphics cursor, select a few of the points in the display using the P key or the tablet pen. Attention indicators appear on the points you select. When you have finished selecting points, enter ] to delete the entities you have selected and return to the Delete menu.

## Deleting Entities in a Chain

If you have a series of connected entities that form either an open or a closed set of entities, you can select these entities using chain selection. The equilateral triangle you created earlier in this session is a good example of when chain selection would be particularly helpful.

### Keyboard Selection

Select 1.SELECT from the Delete menu.

Enter **(CTRL)** E to display the Entity Selection menu.

Select 2.CHAIN from the Entity Selection menu.

### Tablet Selection

| Delete    |            |            |       |
|-----------|------------|------------|-------|
| Select    |            | Screen     | Chain |
| Region In | Region Out | All Points | Level |

Using the graphics cursor and the C key or the tablet pen, select the first entity in a series of connected entities.

Next, you are asked to indicate the direction that is to be followed in selecting the entities:

#### INDICATE DIRECTION

Move the graphics cursor a short distance along the outline of connected entities and enter C or press down with the tablet pen to indicate the direction for chaining.

ICEM DDN then places attention indicators on each of the contiguous curves that make up the chain. At this time, check to make sure that ICEM DDN correctly selected all of the curves in the chain.

If the selection is correct, enter ] to accept the selected entities.

If the selection is incorrect, enter [ and repeat the steps outlined above.

#### NOTE

Another way that you can cancel incorrect entity selections is to set the Selection Modification modal to the on position as follows:

Select 7.MODALS from the Entity Selection menu.

Select 3.SELECTION MODIFICATION from the Modals menu.

Select 1.ON from the Selection Modification menu.

As soon as you change the setting of this modal, the following message is displayed:

#### SELECTION MODIFICATION IS ON

From now on, each time you select entities using an entity selection method other than single selection, the Modify Selection menu is displayed. At that time, you can either accept the currently selected entities by entering ] or cancel the selection by entering 17.CANCEL LAST MODIFICATION. If you cancel the selection, you return to the Entity Selection menu where you can reselect the entities.

## Deleting Entities in a Region

If you want to delete all of the entities within a particular region on the screen, you can select the entities inside that region using region selection.

### Keyboard Selection

Select 1.SELECT from the Delete menu.

Enter **CTRL** E to display the Entity Selection menu.

Select 3.REGION IN from the Entity Selection menu.

### Tablet Selection

| Delete    |            |            |       |
|-----------|------------|------------|-------|
| Select    |            | Screen     | Chain |
| Region In | Region Out | All Points | Level |

The graphics cursor should be visible on the screen along with the following prompt:

INDICATE A CORNER OF THE REGION

At this time, indicate the first of two positions on the display that are the diagonal corners of an imaginary rectangle. ICEM DDN will delete all the entities that lie completely or partially within this rectangular region.

Next, you are asked to indicate the opposite diagonal corner of the rectangle:

INDICATE THE OPPOSITE CORNER

As with chain selection, ICEM DDN places attention indicators on each of the entities to be deleted.

If the selection is correct, enter ] to accept the selected entities. As soon as you enter ], the entities are deleted and the Delete menu is redisplayed.

If the selection is incorrect, enter [ and repeat the steps outlined above. If the Selection Modification modal is on, select 17.CANCEL LAST MODIFICATION from the Modify Selection menu and try selecting the entities again.

## Deleting All Points

You can delete all of the displayed points in your part in a single step using the Delete All Points menu selection.

### Keyboard Selection

Select 3.ALL POINTS from the Delete menu.

### Tablet Selection

| Delete    |            |            |       |
|-----------|------------|------------|-------|
| Select    |            | Screen     | Chain |
| Region In | Region Out | All Points | Level |

You are then asked if you are sure that you want to delete all of the points in your part:

DELETE ALL POINTS?

Enter Y if you want to delete the points.

Enter N if you decide that you don't want to delete the points.

0

0

0

00

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|                                                            |      |
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In this chapter, you will learn how the drafting functions of the ICEM DDN system can help you with tasks such as dimensioning and section lining.

## Dimensioning

The following discussion explains how to put dimension information into a drawing. Under the Dimension menu, there are separate functions that allow you to insert the dimension of each of the following:

- A horizontal or vertical length.
- A length perpendicular to any line.
- The distance between two points or curve ends.
- The size of an angle.
- The radius of a circle or arc.
- The diameter of a circle or arc.

In using each of these types of dimensioning, you select an item from the Dimension menu, select the object in the display to be dimensioned, and provide some additional information, such as the location of the dimension information and, in some cases, the positioning of the arrows. These steps allow you to add the dimensions of almost any object in your display. Additional control of the dimension operations is provided by some of the drafting modals in 1.DRAFTING MODALS in the Drafting menu.

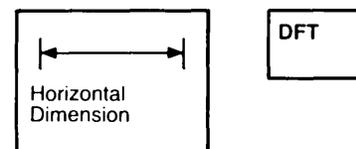
## Horizontal Dimensioning

Horizontal dimensioning provides the horizontal distance between two entities (the distance along the x-axis) even if the line along which you want dimensions is not horizontal. Dimensions that are neither horizontal nor vertical are described under Parallel Dimensioning.

### Keyboard Selection

- Select 16.DRAFTING from the main menu.
- Select 2.DIMENSION from the Drafting menu.
- Select 1.HORIZONTAL from the Dimension menu.

### Tablet Selection



The following prompt then displays:

**INDICATE ENTITY**

Use the graphics cursor to select the two entities between which you want a dimension (or pick a single entity's endpoints). This menu selection calculates the distance between these and writes that distance (in the units of measure that you selected earlier in this session) at a location you specify in response to the prompt:

**INDICATE TEXT ORIGIN**

Use the graphics cursor to indicate where the *lower left corner* of the dimension text is to be written. ICEM DDN then displays the extension lines, the arrows, and the dimension information. To return to the Dimension menu, enter ].

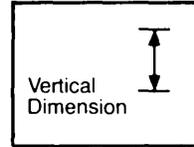
## Vertical Dimensioning

Vertical dimensioning is similar to horizontal dimensioning *except* that you are measuring the vertical distance between two entities.

### Keyboard Selection

Select 2.VERTICAL from the Dimension menu.

### Tablet Selection



Use the graphics cursor to pick two entities from which to measure. The vertical distance between these two entities is written between two witness lines extending from the objects and connected by arrows.

As with horizontal dimensioning, you again see the prompt:

INDICATE TEXT ORIGIN

Use the graphics cursor to select a location for the dimension information and enter ] to return to the Dimension menu.

## Parallel Dimensioning

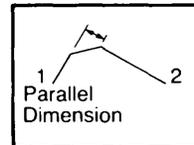
Parallel dimensioning is used to measure the distance between two entities. With this menu selection, you can:

- Measure and write the distance from one endpoint of an entity to the other.
- Measure and write the distance between a line and another entity. The distance measured is *perpendicular to the line*. (You may wish to create a few parallel lines on your display so that you can try this operation. 10.PARALLEL TO LINE in the Line menu produces parallel lines.)

### Keyboard Selection

Select 3.PARALLEL from the Dimension menu.

### Tablet Selection



If you are using the keyboard, you are asked to indicate whether the measurement is the normal distance from one line to the end of another entity. This prompt is answered automatically if you are using the tablet.

NORMAL TO PARALLEL LINES?

For now, enter Y in response to this prompt.

Next, you are asked to select a line or centerline from which a perpendicular dimension is to be made:

INDICATE ENTITY

Use the graphics cursor to select the two entities between which you want to measure the distance.

As with both horizontal and vertical dimensions, you are then asked to indicate the origin position of the dimension:

**INDICATE TEXT ORIGIN**

Use the graphics cursor to specify where ICEM DDN is to write the lower left corner of the dimension text. ICEM DDN calculates the distance between the two objects, draws witness lines if they are needed, draws the arrows, and writes the dimension with the first numeral at the location that you indicated for the origin.

To practice creating parallel dimensions, try the following exercises:

1. Use menu selections in the Line menu to create boxes and rectangles. Add horizontal and vertical dimensions to these figures. Place some dimensions inside and others outside the rectangles.
2. Use 1.SCREEN POSITION and 10.PARLEL TO LINE from the Line menu to produce several pairs of parallel lines. Add the dimensions that specify the distance between these pairs of lines.

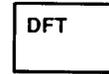
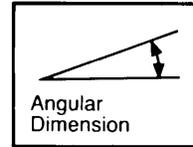
## Angular Dimensioning

Angular dimensioning is used to write the size of the angle between two lines that cross or extend from a common location or point.

### Keyboard Selection

Select 5.ANGULAR from the Dimension menu.

### Tablet Selection



First, you are asked to indicate the lines or center lines that form the angle you wish to measure:

#### INDICATE ENTITY

It is important to remember that the order in which you select these two lines must be counterclockwise around the vertex of the angle you are dimensioning. For example, in figure 3-1 (the upper portion) the line AB was selected before the line BC to obtain the dimension shown. Selecting these lines in the order BC then AB produces the dimension shown in figure 3-1 (the lower portion).

The way in which you select the ends of the lines is also very important. The lines in figure 3-1 (the upper and lower portion) were selected at the A and C ends rather than at the B end. Refer to the ICEM Design/Drafting Drafting Functions manual for the eight possible angular dimensions.

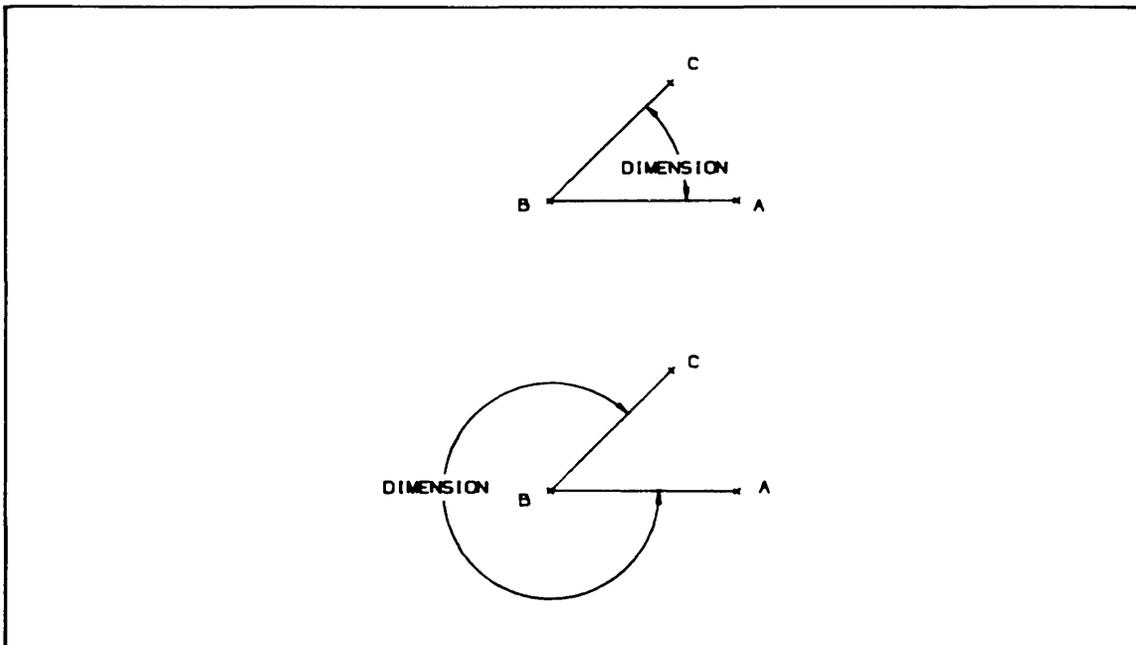


Figure 3-1. Angular Dimensions

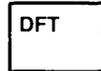
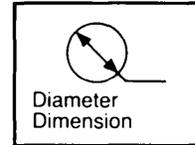
## Diameter Dimensioning

Diameter dimensioning is used to measure the diameter of a circle.

### Keyboard Selection

Select 7.DIAMETER from the Dimension menu.

### Tablet Selection



You are first asked to select a circle and to indicate the position of the first numeral of the dimension by displaying the prompts:

**INDICATE ENTITY**

and

**INDICATE TEXT ORIGIN**

In response to the first prompt, use the graphics cursor to select an arc or a circle. In response to the second prompt, indicate where you want the dimension text to start.

This operation produces an arrow that crosses the circle and extends to the text that gives the dimension.

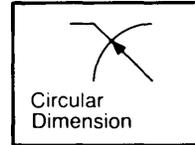
## Radius Dimensioning

Radius dimensioning is used to measure the radius of a circle.

### Keyboard Selection

Select 6.RADIUS from the Dimension menu.

### Tablet Selection



You are first asked to select an arc or a circle by displaying:

**INDICATE ARC OR CIRCLE**

You are then asked to indicate the origin position of the note:

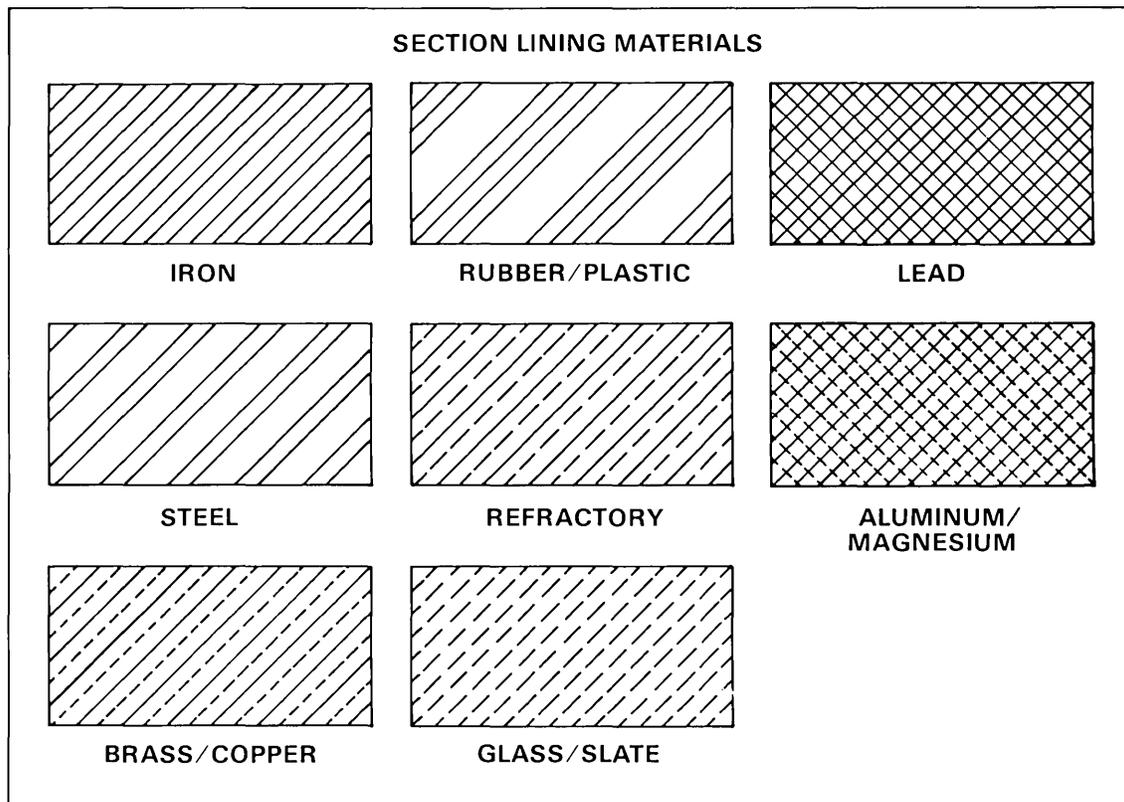
**INDICATE TEXT ORIGIN**

Use the graphics cursor and the C key or the tablet pen to indicate where the arrow is to join the text.

## Section Lining

Another important drafting function that you can do automatically with the ICEM DDN system is section lining (crosshatching). Section lining is part of the Drafting menu.

Before you begin, you must first decide which type of section lining you want to use (refer to figure 3-2). These choices are part of the section lining modals, 16.3.1 MODALS. The default section lining material is 1.IRON.



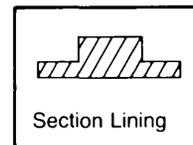
**Figure 3-2. Types of Section Lining**

To begin section lining, enter the following keyboard or tablet selection:

### Keyboard Selection

- Select 16.DRAFTING from the main menu.
- Select 3.SECTION LINING from the Drafting menu.
- Select 2.DEFINITION from the Section Lining menu.

### Tablet Selection



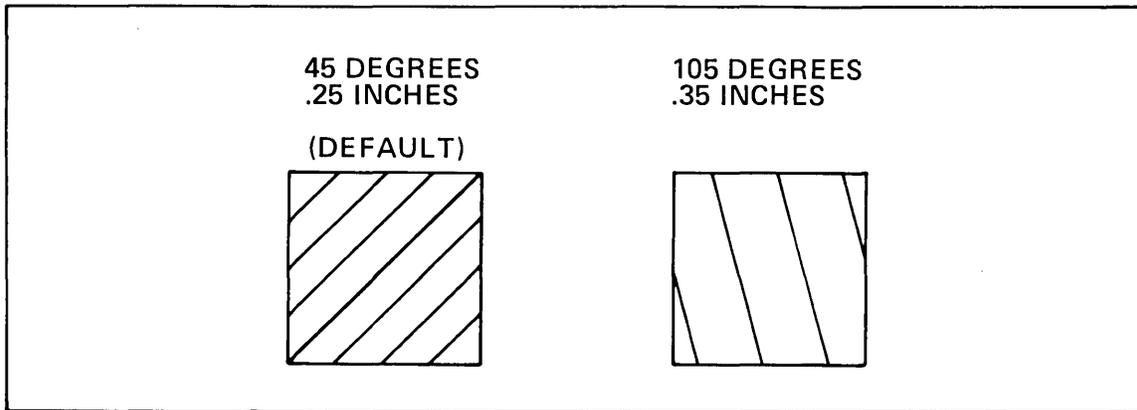
## SECTION LINING

You are first asked to enter the angle, distance, and tolerance values of the section lining:

**SECTIONING DEFINITION**  
1.ANGLE            =45.0000  
2.DISTANCE        =0.2500  
3.TOLERANCE       =0.0100

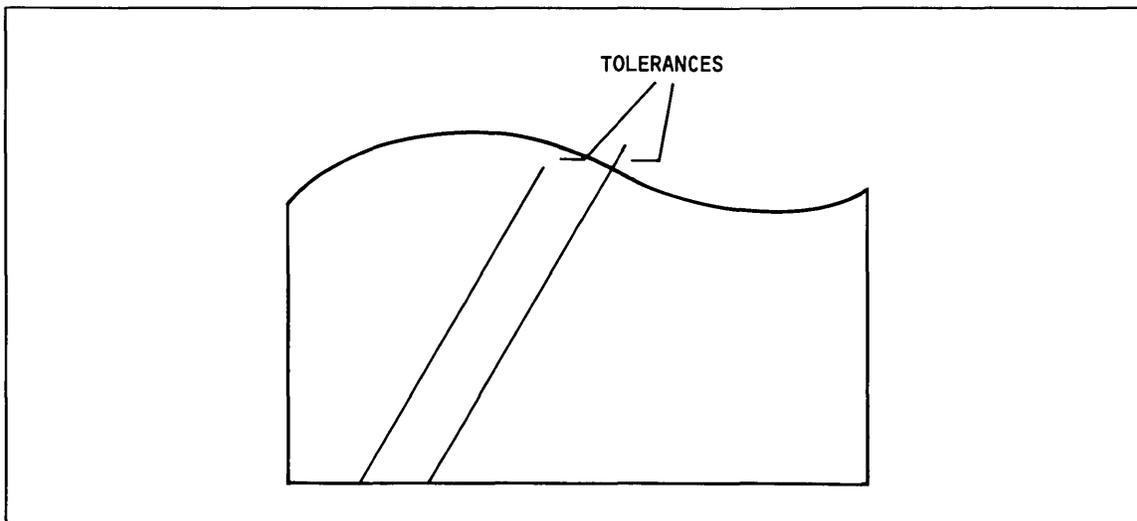
You can accept the currently displayed values by entering ] or you can change them to whatever values you wish. When you enter ], you are telling the system that you want to accept all of the remaining values in the list.

Figure 3-3 shows you two section lining samples in which different angle and distance values were used.



**Figure 3-3. Section Lining Samples**

The tolerance value is the distance a curve is allowed to fall short of or overshoot the boundary. This value affects the graphic display of the intersection of two curves as shown in figure 3-4.



**Figure 3-4. Tolerance Values for Section Lining**

The default tolerance value is 0.0100 and should suffice for most of the work that you do. If the section lining does not match up very well with an irregular curve, you may want to change the tolerance values. As with all modals, if you change this value it stays that way until you change it again.

The next step is to indicate the boundaries for the section lining as shown in figure 3-5.

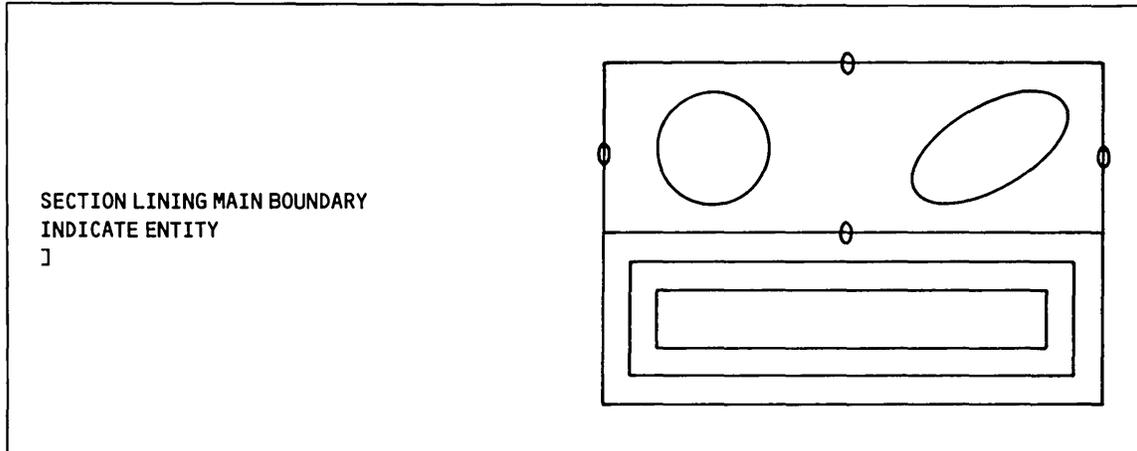


Figure 3-5. Selecting the Main Boundary

### Section Lining Islands

Next, you are asked if there are any islands within the boundaries you just selected. We selected both the circle and the ellipse as shown in figure 3-6.

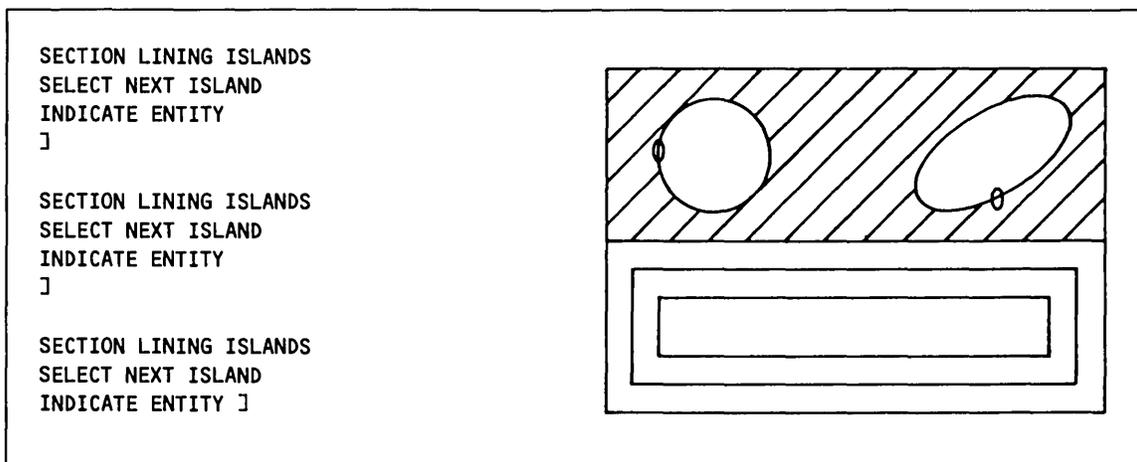


Figure 3-6. Section Lining Islands

## Section Lining Islands Within Islands

With the Section Lining function, you can also perform section lining of islands within islands by starting with the outermost island and working inward (figure 3-7). To help illustrate how the angle and distance values can affect the display, we changed these values to 135 and 0.125, respectively.

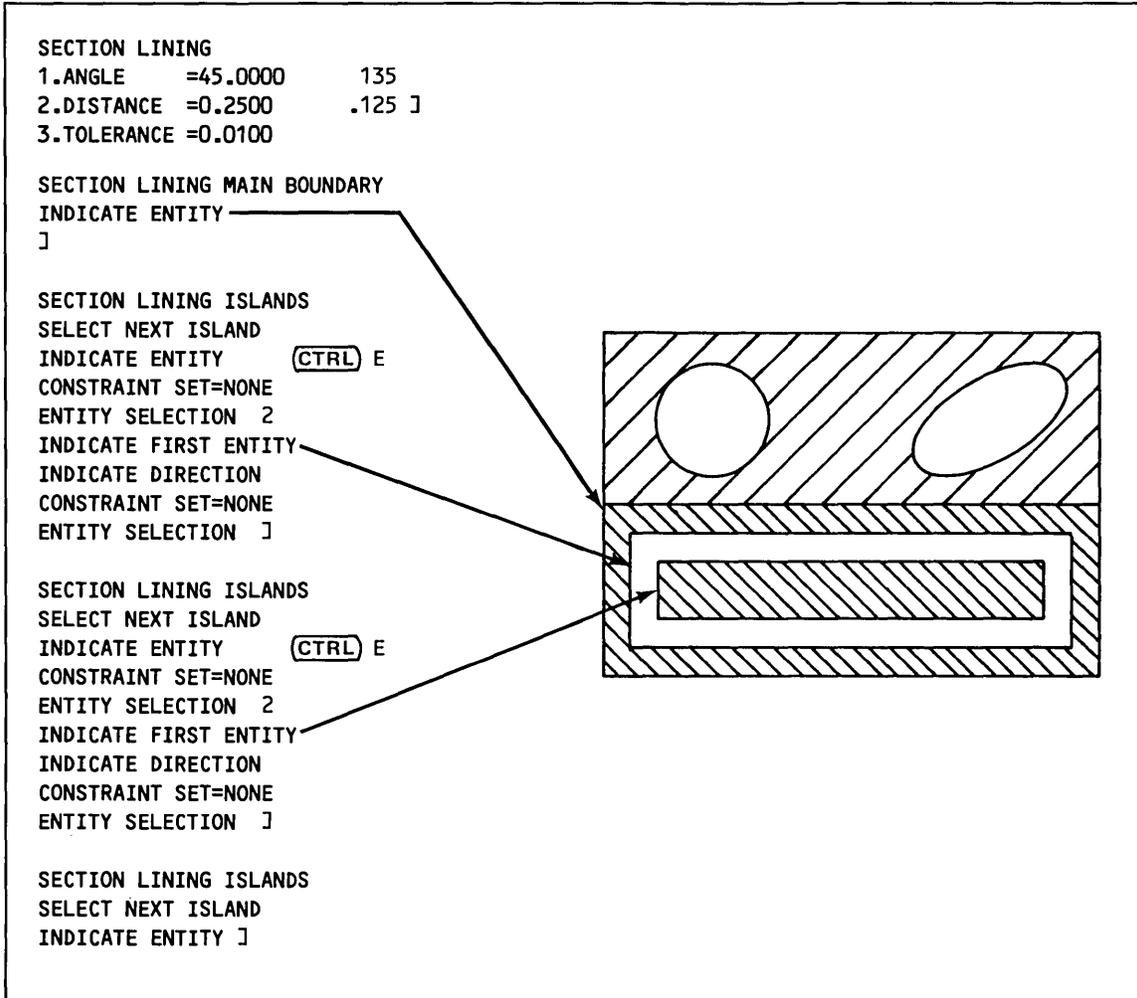


Figure 3-7. Section Lining Islands Within Islands

## Section Lining Islands Within Islands Within Islands

Just as you can perform automatic section lining of islands within islands, you can take this function one step further to perform automatic section lining of islands within islands within islands. In the example shown in figure 3-8, the section lining material modal (16.3.1 MODALS) was set to 7.LEAD.

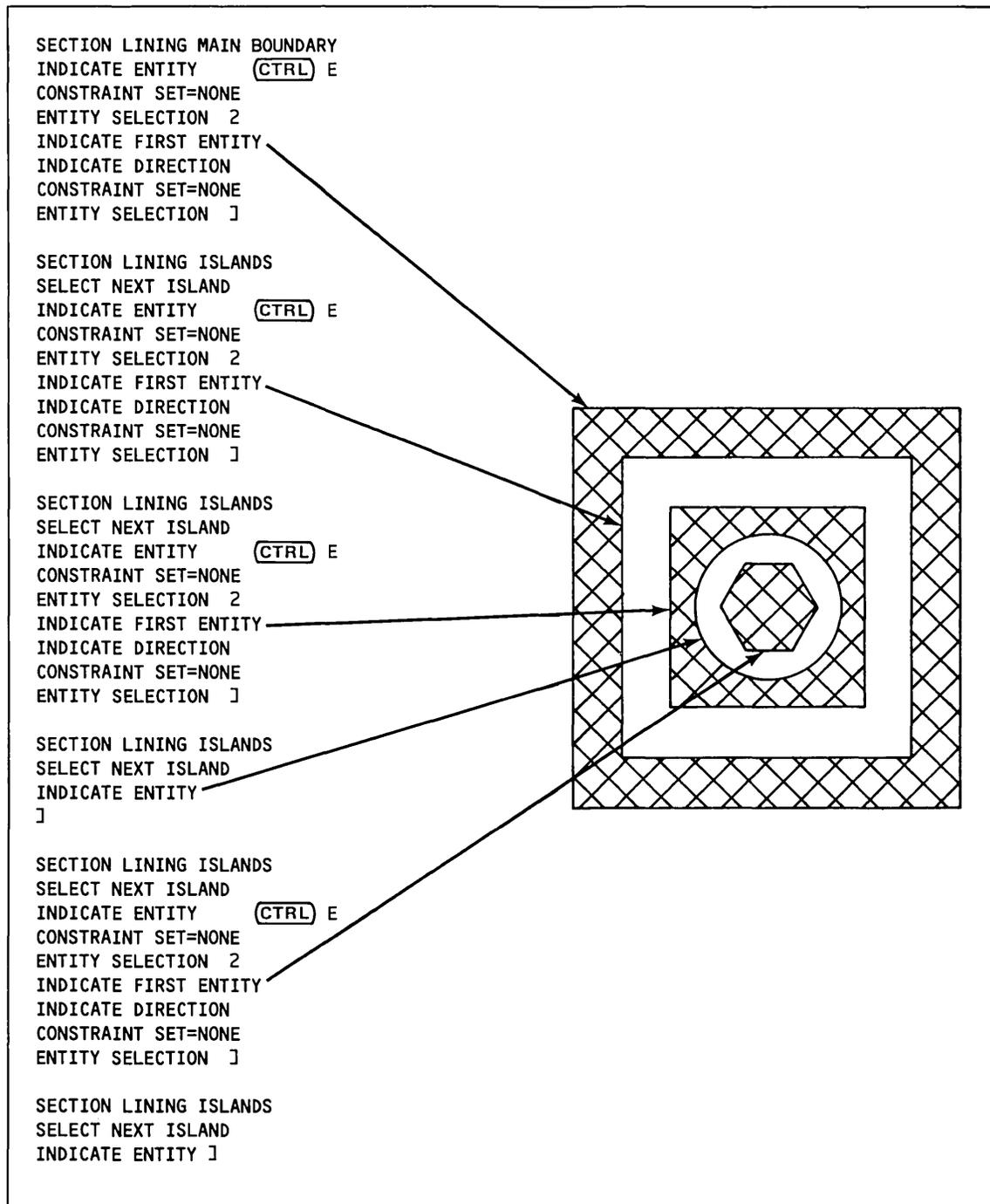


Figure 3-8. Section Lining Islands Within Islands Within Islands

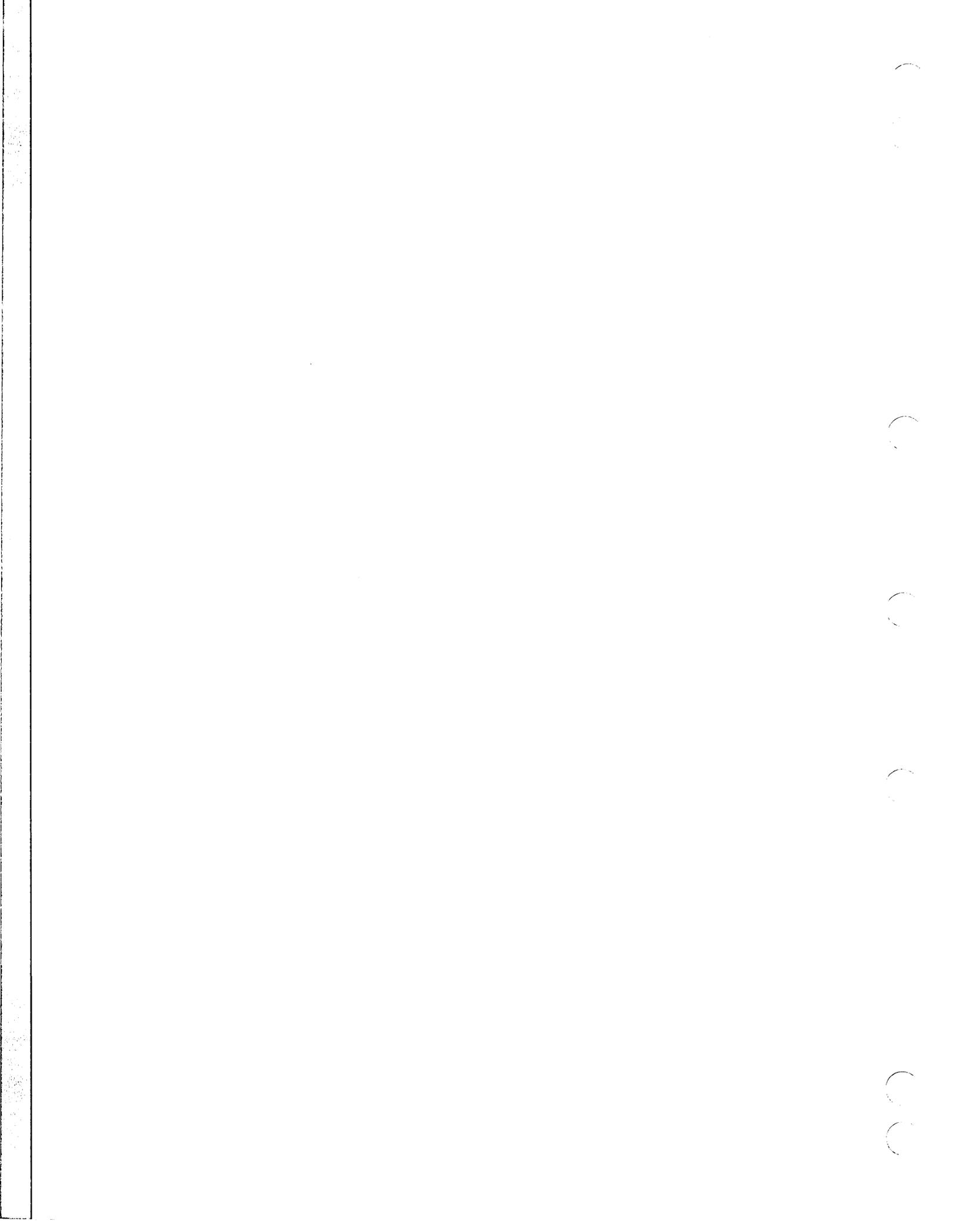


# Constructing a Bolt Plate

---

4

|                                               |      |
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| Getting Started .....                         | 4-1  |
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In this practice session, you will learn how to construct a bolt plate using basic construction procedures: Point, Line, Arc/Circle/Fillet, and Rectangular Array.

## Getting Started

1. Log in using the procedures described in the introduction to this manual.
2. Create a new part. The sample terminal session in figure 4-1 helps to illustrate this procedure.

| Terminal Session                                                                           | Comments                                                                                                                         |
|--------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------------------------------------------|
| <pre>ENTER PART NAME  BOLTPLATE</pre>                                                      | <p>Enter the name and sheet number of your new part. In this example, we called the part BOLTPLATE with a sheet number of 1.</p> |
| <pre>SHEET NUMBER = 1</pre>                                                                |                                                                                                                                  |
| <pre>NEW PART ASSUMED</pre>                                                                |                                                                                                                                  |
| <pre>--- UNITS OF MEASURE  M 1.METRIC (MM) 2.ENGLISH (INCH) 3.ENGLISH (FOOT/INCH)  2</pre> | <p>First, enter M to turn the menu display on. Next, select menu choice 2 to work in English units.</p>                          |
| <pre>--- DRAFTING STANDARD 1.ANSI 1982 2.ANSI 1973 3. 4. 5. 6. 7. 8. 9.          1</pre>   | <p>Select menu choice 1 to use 1982 ANSI standard dimensions.</p>                                                                |
| <pre>ICEMDDN  L=0  V=1  D=0.00  P=0  C=0</pre>                                             |                                                                                                                                  |

Figure 4-1. Start Up Procedure for Constructing a Bolt Plate

## GETTING STARTED

Before you begin, we recommend that you read the hints below. They will help you as you create this part.

- If you enter the wrong key, enter [ to reject that entry.
- If you accidentally create an entity that you don't want:

Press the CTRL key and the D key simultaneously, followed by the RETURN key:

**(CTRL)** D + **(RETURN)**

or

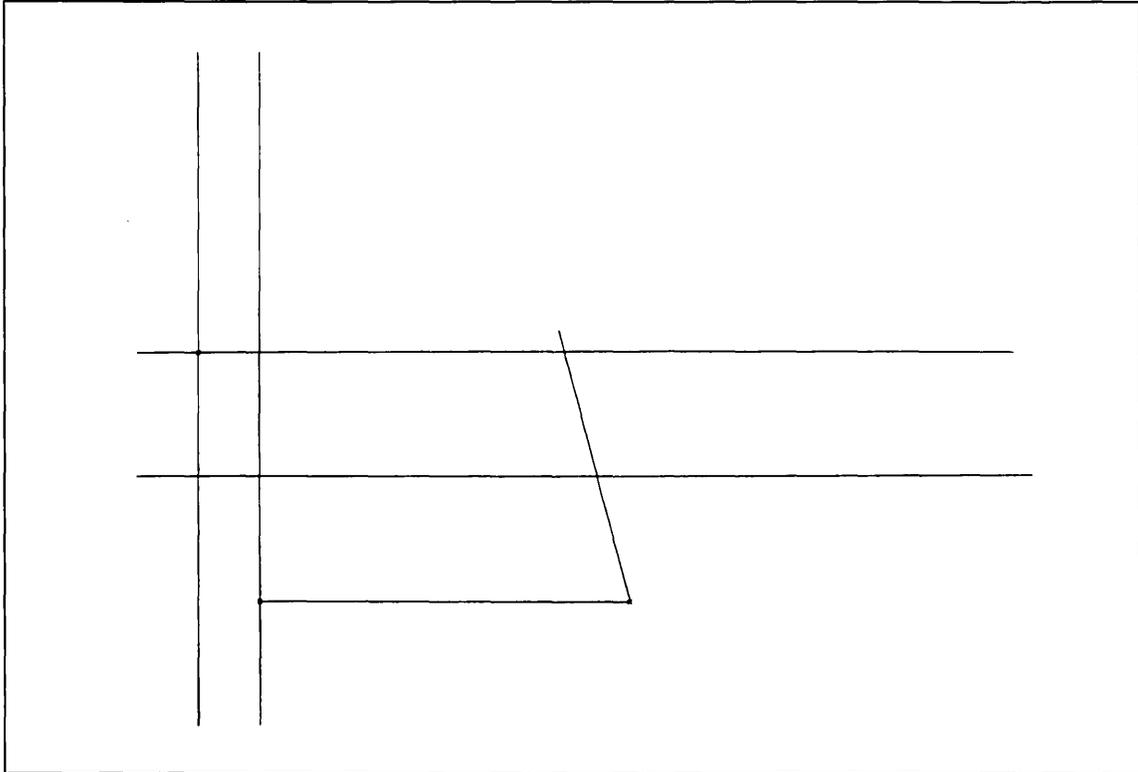
Select the Delete Last Entity square with your tablet pen:

|                          |
|--------------------------|
| Delete<br>Last<br>Entity |
|--------------------------|

- If your screen fades, press the shift key on your keyboard to brighten the display. Some terminals dim if you do not use them within a certain period of time.

## Creating Basic Lines

In the first part of this session, you will learn how to use the Point and Line menus to create the basic outline of the bolt plate. Figure 4-2 shows the points and lines that make up this outline.



**Figure 4-2. Basic Lines of a Bolt Plate**

### NOTE

If you are using the keyboard, every entry that you make must be followed by a RETURN. If you are using the tablet, the only time a RETURN is required is when you are entering numeric data entries, for example, x- and y-coordinate values.

Define and display the base point of the bolt plate as follows:

#### Keyboard Selection

Select 9.POINT from the main menu.

Select 2.KEY-IN from the Point menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

#### Tablet Selection

|                       |
|-----------------------|
| Transform<br>XT,YT,ZT |
| Model<br>X,Y,Z        |

Enter the following xt- and yt-coordinates (transform coordinates):

- 1.XT = 5
- 2.YT = 2 ]
- 3.ZT =0.0000

As soon as you enter ], the point appears on your screen as a small x.

Enter ] to return to the Definition Space menu.

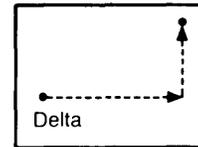
Enter ] to return to the Point menu.

Define a point 6 inches away from the base point in the positive x direction as follows:

**Keyboard Selection**

**Tablet Selection**

Select 4.DELTA from the Point menu.



- Position the graphics cursor near the base point you just created. If you are using the keyboard, select the point using the P key. If you are using a tablet, select it using the tablet pen. An attention indicator appears on the point.

Select 1.TRANSFORM COORDINATES from the Definition Space menu.

Enter the following xt- and yt-coordinates:

- 1.DELTA XT = 6
2. YT = 0 ]
3. ZT =0.0000

A point appears 6 inches to the right of the base point.

Enter ] to return to the INDICATE POINT prompt.

Enter ] again to return to the Point menu.

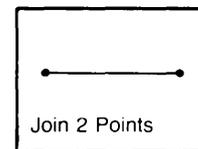
Join the two points you now have on your screen as follows:

**Keyboard Selection**

**Tablet Selection**

**(CTRL) L + (RETURN)**

Select 3.JOIN 2 POINTS from the Line menu.



- Position the graphics cursor on the left side of the screen and select the first point using the tablet pen or the P key.
- Position the graphics cursor on the right side of the screen and select the second point in the same way.

A line joining the two points appears on your screen.

Enter ] to return to the Line menu.

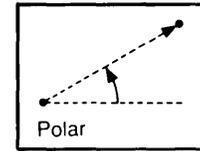
Create the sloped line on the right side of the bolt plate as follows:

---

**Keyboard Selection**

Select 7.POLAR from the Line menu.

---

**Tablet Selection**


- Position the graphics cursor on the right side of the screen and select the point on the right using the P key or the tablet pen. The polar line begins at this location.
- Position the graphics cursor on the horizontal line that you just created and select it using the L key or the tablet pen. This line serves as the reference line from which the angle is measured.

---

**NOTE**

Angles are always measured from 0° in a counterclockwise direction.

---

Enter the following angle and distance values for the new line:

```
1.ANGLE =45.0000 105
2.DISTANCE =0.2500 4.5]
```

A sloped line originating at the right end of the horizontal line, at an angle of 105° and a distance of 4.5 inches, then appears.

Enter ] to return to the INDICATE POINT prompt.

Enter ] to return to the Line menu.

Define a point near the upper left-hand side of your bolt plate as follows:

---

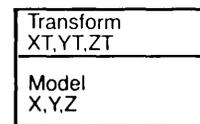
**Keyboard Selection**

**(CTRL) P + (RETURN)**

Select 2.KEY-IN from the Point menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

---

**Tablet Selection**


Enter the following xt- and yt-coordinates (transform coordinates):

```
1.XT = 4
2.YT = 6]
3.ZT =0.0000
```

A new point appears on your screen.

Enter ] to return to the Definition Space menu.

Enter ] again to return to the Point menu.

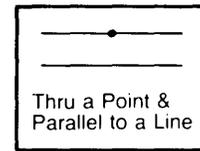
Create a line that runs parallel to the horizontal line through the point you just created as follows:

**Keyboard Selection**

**(CTRL) L + (RETURN)**

Select 8.POINT-PARLEL TO LINE from the Line menu.

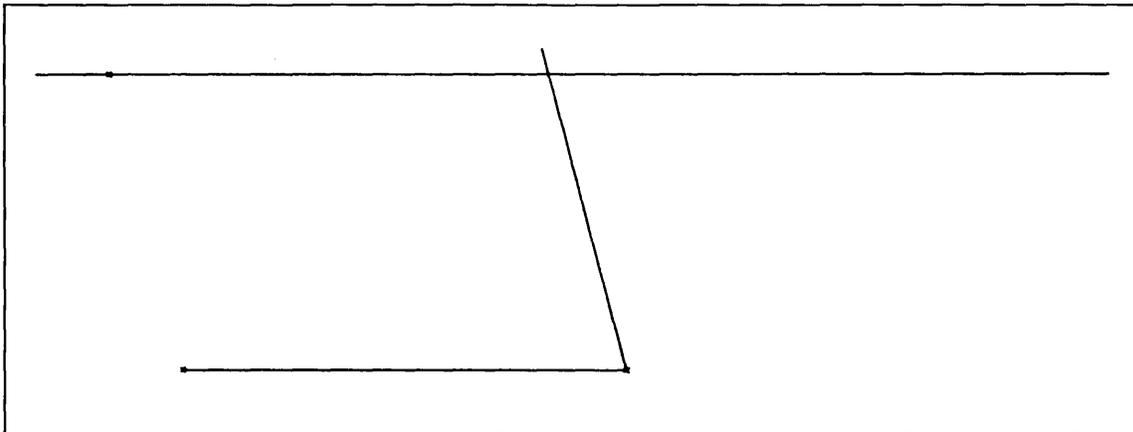
**Tablet Selection**



- Position the graphics cursor near the upper left point and select it using the P key or the tablet pen. This is the point through which the new line passes.
- Position the graphics cursor near the bottom horizontal line and select it using the L key or the tablet pen. The new line is parallel to this line.

Enter ] to return to the Line menu.

The display on your terminal should now match figure 4-3.



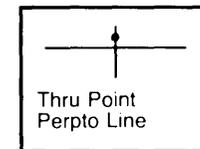
**Figure 4-3. Parallel Line**

Create a vertical line that runs through the upper left point and perpendicular to the bottom horizontal line as follows:

**Keyboard Selection**

Select 9.POINT-PERPTO LINE from the Line menu.

**Tablet Selection**



- Position the graphics cursor near the upper left point and select it using the P key or the tablet pen. This is the point through which the new line passes.
- Position the graphics cursor near the top horizontal line and select it using the L key or the tablet pen. The new line is perpendicular to this line.

A new vertical line appears on your screen.

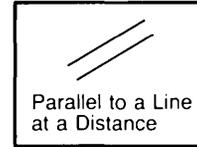
Enter ] to return to the Line menu.

Create a line that runs parallel to the top horizontal line as follows:

#### Keyboard Selection

Select 10.PARLEL TO LINE from the Line menu.

#### Tablet Selection



- Position the graphics cursor near the top horizontal line and select it using the L key or the tablet pen. This line serves as the reference line.

Next, you are asked to specify how far away you want your new parallel line to be from the reference line.

Enter 2 as the distance:

```
DISTANCE =0.2500 2]
```

- Position the graphics cursor *below* the top horizontal line and select that position using the C key or the tablet pen. In doing this, you are telling the system that you want the line to appear on the bottom side (with respect to the screen) of the reference line at a distance of 2 inches.

A new parallel line appears on your screen.

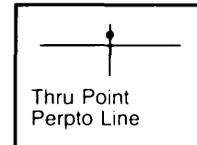
Enter ] to return to the Line menu.

Create a vertical line that runs through the lower left point and is perpendicular to the middle horizontal line as follows:

#### Keyboard Selection

Select 9.POINT-PERPTO LINE from the Line menu.

#### Tablet Selection



- Position the graphics cursor near the lower left point on the screen and select it using the P key or the tablet pen. This is the point through which the new line passes.
- Position the graphics cursor near the middle horizontal line and select it using the L key or the tablet pen. The new line is perpendicular to this line.

A new vertical line appears on your screen.

Enter ] to return to the Line menu.

The display on your terminal screen should now match figure 4-2.

## Trimming Lines

If you have not yet read Trimming Curves in chapter 2 of this manual, we recommend that you take the time to go back and read it before you try to trim the lines of your bolt plate. The Trimming Curves discussion gives you some hints to help you use the trim functions.

Trim the unnecessary, infinite ends off the bolt plate as follows:

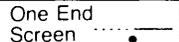
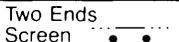
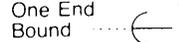
### Keyboard Selection

Select 12.OTHER CURVES from the main menu.

Select 8.TRIM/EXTEND CURVES from the Other Curves menu.

Select 2.ONE END-CURVE from the Trim/Extend Curves menu.

### Tablet Selection

| Trim/Extend Curves                                                                                |                                                                                                     |
|---------------------------------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------|
| One End Screen  | Two Ends Screen  |
| One End Bound   | Two Ends Bound   |

- Position the graphics cursor on the top horizontal line and select it as the boundary using the L key or the tablet pen.
- Position the graphics cursor on the right (sloped) line and select it as the line to be trimmed using the L key or the tablet pen. Since you are only trimming one line, enter ] to tell ICEM DDN that you have finished selecting curves to be trimmed.
- Position the graphics cursor above the top end of the right line and enter C or press down on the tablet pen to indicate that you want the line trimmed on that side of the boundary.

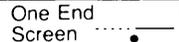
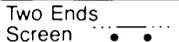
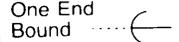
Enter ] to return to the Trim/Extend Curves menu.

Trim both ends of the top horizontal line as follows:

### Keyboard Selection

Select 4.TWO ENDS-CURVE from the Trim/Extend Curves menu.

### Tablet Selection

| Trim/Extend Curves                                                                                  |                                                                                                       |
|-----------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------|
| One End Screen  | Two Ends Screen  |
| One End Bound   | Two Ends Bound   |

- Position the graphics cursor on the right (sloped) line and select it as the first boundary using the L key or the tablet pen.
- Position the graphics cursor on the left vertical line and select it as the second boundary using the L key or the tablet pen.
- Position the graphics cursor on the top horizontal line and select it as the curve to be trimmed using the L key or the tablet pen, followed by ].

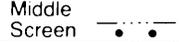
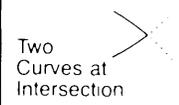
Enter ] to return to the Trim/Extend Curves menu.

Trim the ends of the left vertical line and the middle horizontal line at their intersection as follows:

### Keyboard Selection

Select 7.TWO CURVES AT INTERSECTION from the Trim/Extend Curves menu.

### Tablet Selection

|                                                                                                    |                                                                                                                  |
|----------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------|
| Middle Screen  | Two Curves at Intersection  |
| Middle Bound   |                                                                                                                  |

- Position the graphics cursor on the left vertical line *near the bottom of the screen* and select it as the first curve to be trimmed using the L key or the tablet pen.
- Position the graphics cursor on the middle horizontal line *near the left side of the screen* and select it as the second curve to be trimmed using the L key or the tablet pen.

Enter ] to return to the Trim Curves menu.

Trim the ends of the middle horizontal line and the middle vertical line at their intersection as follows:

**Keyboard Selection**

Select 7.TWO CURVES AT INTERSECTION from the Trim/Extend Curves menu.

**Tablet Selection**

|               |  |
|---------------|--|
| Middle Screen |  |
| Middle Bound  |  |

- Position the graphics cursor on the middle *horizontal* line near the right side of the screen and select it as the first line to be trimmed using the L key or the tablet pen.
- Position the graphics cursor on the middle *vertical* line near the top of the screen and select it as the second curve to be trimmed using the L key or the tablet pen.

Enter ] to return to the Trim/Extend Curves menu.

Trim one end of the left vertical line as follows:

**Keyboard Selection**

Select 2.ONE END-CURVE from the Trim/Extend Curves menu.

**Tablet Selection**

| Trim/Extend Curves |                 |
|--------------------|-----------------|
| One End Screen     | Two Ends Screen |
| One End Bound      | Two Ends Bound  |

- Position the graphics cursor on the top horizontal line and select it as the boundary using the L key or the tablet pen.
- Position the graphics cursor on the left vertical line and select it as the line to be trimmed using the L key or the tablet pen, followed by a ].
- Position the graphics cursor near the top end of the left vertical line and enter C, or press down on the tablet pen to indicate the side to be trimmed.

Enter ] to return to the Trim/Extend Curves menu.

Complete the outline of the bolt plate as follows:

**Keyboard Selection**

Select 2.ONE END-CURVE from the Trim/Extend Curves menu.

**Tablet Selection**

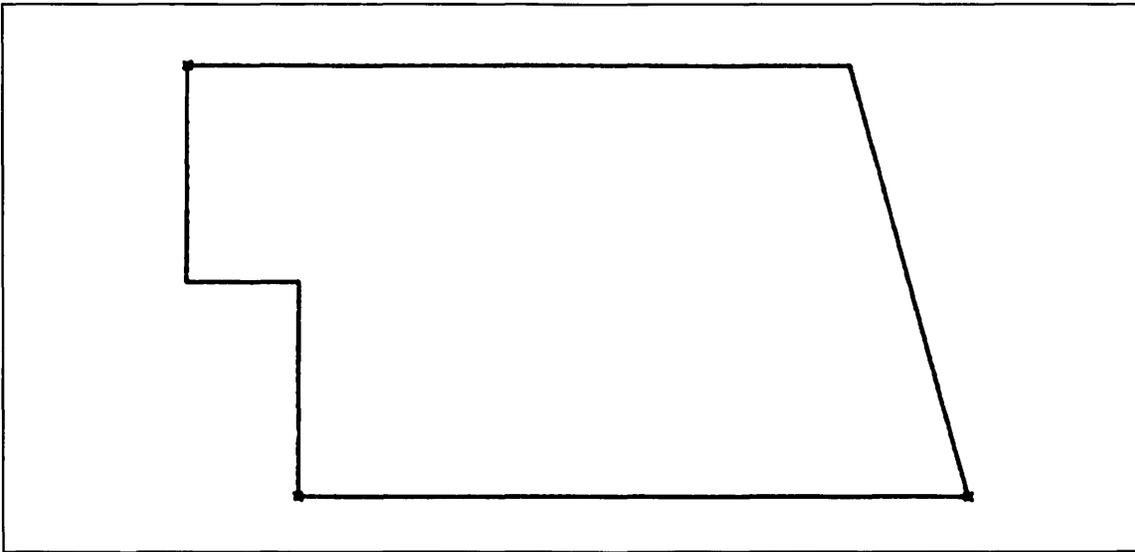
| Trim/Extend Curves |                 |
|--------------------|-----------------|
| One End Screen     | Two Ends Screen |
| One End Bound      | Two Ends Bound  |

## TRIMMING LINES

- Position the graphics cursor on the bottom horizontal line and select it as the boundary using the L key or the tablet pen.
- Position the graphics cursor on the middle vertical line and select it as the line to be trimmed using the L key or the tablet pen, followed by ].
- Position the graphics cursor near the bottom end of the middle vertical line and enter C, or press down with the tablet pen to indicate the side to be trimmed.

Enter ] to return to the Trim/Extend Curves menu.

The outline of your bolt plate should match the result shown in figure 4-4.



**Figure 4-4. Line Ends Trimmed**

## Creating the Fillets

Create all of the fillets of the bolt plate in a single step using multiple fillet construction as follows:

### Keyboard Selection

**(CTRL)** A + **(RETURN)**

Select 9.FILLET from the Arc/Circle/Fillet menu.

Select 1.SPECIFIED RADIUS as the fillet radius mode.

Select 1.TRIM/EXTEND as the fillet trim mode.

Select 1.AUTOMATIC CENTER as the fillet center mode.

Enter a radius value of 0.4 inch:

RADIUS = 0.2500 .4

### NOTE

The 0.2500 that appears is the default radius value. By entering a value of .4, you are changing the default from 0.25 to 0.4. The next time you see this prompt displayed, a radius of 0.4000 is displayed.

Enter **(CTRL)** E to display the Entity Selection menu.

Select 2.CHAIN from the Entity Selection menu.

- Position the graphics cursor on the left vertical line and select it as the first entity using the L key or the tablet pen.
- Move the graphics cursor a short distance along the outline of the bolt plate and enter C or press down with the tablet pen to indicate the direction for chaining the contiguous curves.

If there are attention indicators on each of the line segments that make up the bolt plate, enter ] to accept the selected entities.

If not, enter [ and repeat the steps outlined above. If the Selection Modification modal is on, select 17.CANCEL LAST MODIFICATION from the Modify Selection menu and try selecting the entities again.

The system then automatically calculates the center of each fillet, creates each fillet, and trims the ends of each line segment.

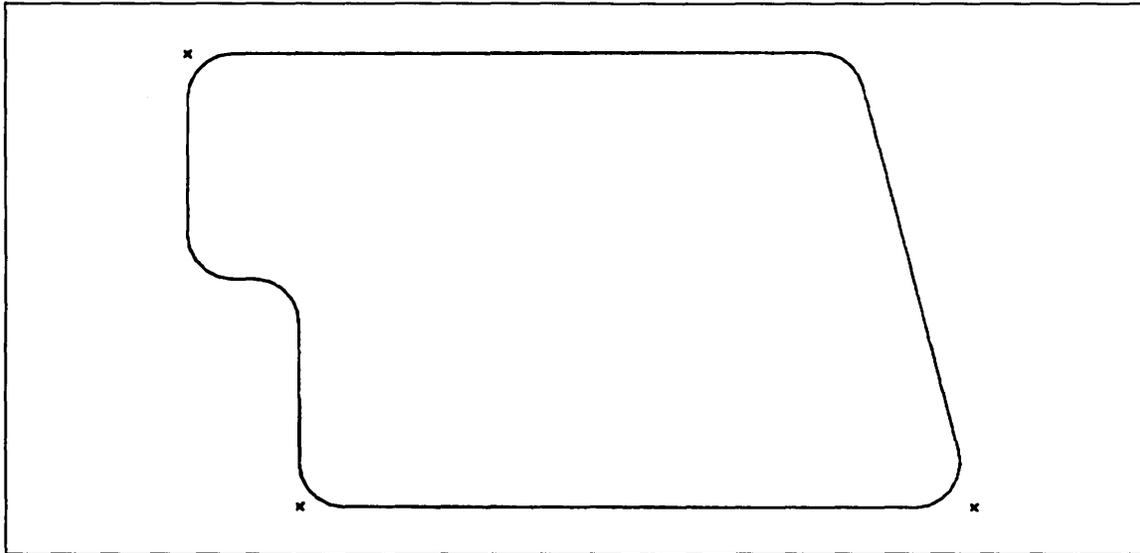
Enter ] to return to the Arc/Circle/Fillet menu.

### Tablet Selection

| Fillet                                                                                                  |                                                                                                      |
|---------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------|
| Specify Radius                                                                                          | Unspec Radius     |
| Auto Ctr<br>No Trim  | Auto Ctr<br>Trim  |

CREATING THE FILLETS

The display on your terminal should now match figure 4-5.



**Figure 4-5. Plate with Fillets**

## Placing an Array of Circles on the Plate

Create one circle to use as the base element from which to create an array of circles on the bolt plate as follows:

### Keyboard Selection

**(CTRL) A + (RETURN)**

Select 2.KEY-IN CENTER AND RADIUS from the Arc/Circle/Fillet menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

### Tablet Selection

|                         |
|-------------------------|
| Transform<br>XT,YT,ZT,R |
| Model<br>X,Y,Z,R        |

Enter the following x and y transform coordinates:

- 1.XT = 5
- 2.YT = 5 ]
- 3.ZT =0.0000

Accept 0.4 as the radius and 0° and 360° as the starting and ending angles of the circle by entering ]:

- 1.RADIUS =0.4000 ]
- 2.STARTING ANGLE=0.0000
- 3.ENDING ANGLE =360.0000

A full circle with a radius of 0.4 inch appears on your screen.

Enter [ to return to the Definition Space menu.

Enter ] to return to the Arc/Circle/Fillet menu.

Enter F to return to the main menu.

Create a rectangular array using the base circle you just created as follows:

### Keyboard Selection

Select 13.ENTITY MANIPULATION from the main menu.

Select 1.RECT ARRAY from the Entity Manipulation menu.

### Tablet Selection

| Entity Manipulation |            |             |                |
|---------------------|------------|-------------|----------------|
| Mirror              | Trans      | Rotate      | Rect Array     |
| Dupl Mirror         | Dupl Trans | Dupl Rotate | Circular Array |

- Position the graphics cursor on the circle and select it as the entity to be copied using the A key or the tablet pen. An attention indicator appears on the circle.

Select 2.KEY-IN as the origin mode.

Enter the following values for the origin:

- 1.ORIGIN XT = 7
- 2. YT = 3 ]

Next, you are asked to enter the number of columns (horizontal number) and rows (vertical number) in the array, the displacement between these columns (horizontal increment) and rows (vertical increment), and the angle of rotation (tilt) for the array.

## PLACING AN ARRAY

Accept 2 as the horizontal number and the vertical number by entering RETURN after each line:

- 1.HORIZONTAL NUMBER = 2 **RETURN**
- 2.VERTICAL NUMBER = 2 **RETURN**

Enter 2 as the horizontal and the vertical delta value and accept 0 as the tilt value by entering ]:

- 3.HORIZONTAL DELTA = 1.0000 2
- 4.VERTICAL DELTA = 1.0000 2 ]
- 5.TILT = 0.0000

Up to this point, you have defined a rectangular array of circles made up of two rows and two columns, with the origin of each circle located 2 inches from the origin of each of the other circles in the array (figure 4-6).

The next step is to exclude the circle in the upper right-hand corner from the array (figure 4-6). To do this, select 2.EXCLUDE SOME as the inclusion/exclusion condition.

Enter 2 as the x position and the y position:

- 1.POSITION X = 2
2. Y = 2 ]

Enter ] again to complete this operation.

The display on your terminal should now match figure 4-7.

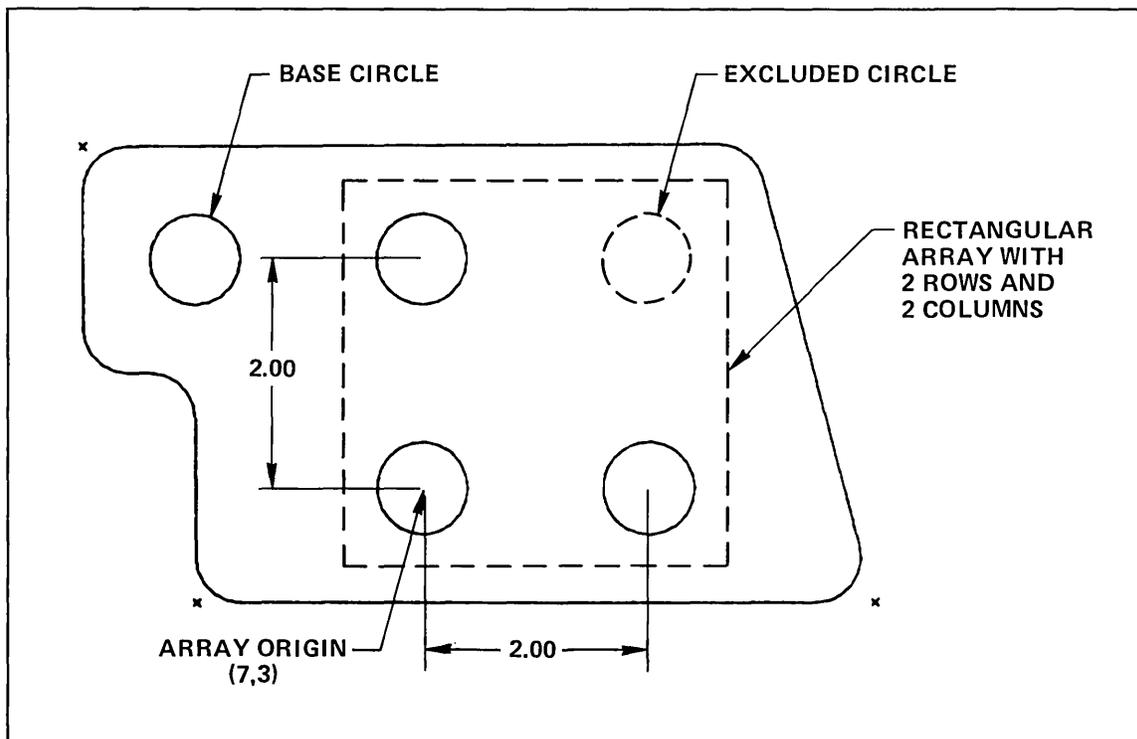
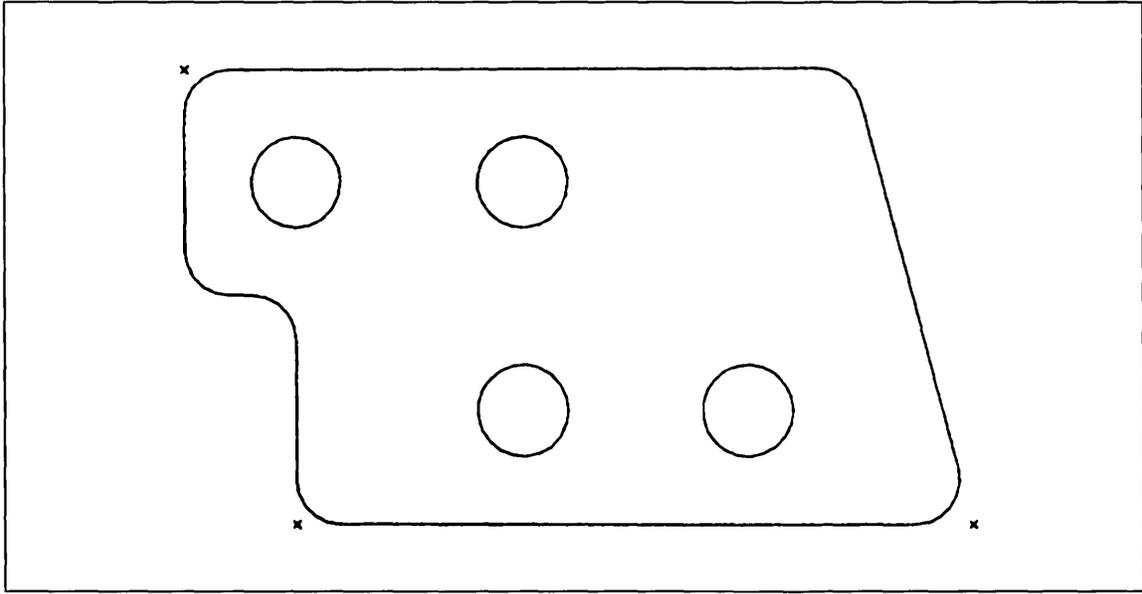


Figure 4-6. Constructing the Rectangular Array



**Figure 4-7. Completed Bolt Plate**

## Filing and Exiting

Figure 4-8 outlines the steps to end this terminal session and save your part.

| Terminal Session                                                                                                                                                                                                                                                                                                                              | Comments                                                                                                                                                                                                                                                                                                                                   |
|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <pre> ICEMDDN L=0 V=1 D=0.00 P=0 C=1 4  FILE CURRENT PART/EXIT ICEM DDN 1.FILE - CONTINUE CURRENT PART 2.   - GET DIFFERENT PART 3.   - QUIT SESSION 4.   - SUSPEND SESSION 5.DO NOT FILE - CONTINUE FROM LAST FILE 6.   - GET DIFFERENT PART 7.   - QUIT SESSION 8.   - SUSPEND SESSION 3  ICEMDDN TERMINATED.  /                     </pre> | <p>If you are using the keyboard, enter 4 to select the File Current Part/Exit ICEM DDN menu.</p> <p>If you are using the tablet, you can file your part and exit from ICEM DDN in a single step by selecting the FILE/QUIT square with your tablet pen.</p> <p>Select 3.FILE - QUIT SESSION to save your part and exit from ICEM DDN.</p> |

Figure 4-8. Sample File/Exit Sequence

# Constructing a Three-Dimensional Cam

---

5

|                                                |      |
|------------------------------------------------|------|
| Getting Started .....                          | 5-1  |
| Defining Base Points .....                     | 5-2  |
| Generating Cam Shaft Curves .....              | 5-3  |
| Generating the Cam Shaft Keyway .....          | 5-8  |
| Deleting Unnecessary Points .....              | 5-10 |
| Projecting the Cam into Three Dimensions ..... | 5-11 |
| Displaying the Cam .....                       | 5-13 |
| Filing and Exiting .....                       | 5-15 |

CCCCCCCC

In this practice session, you will learn how to construct a three-dimensional cam using fillets, strings, and varying depths.

## Getting Started

1. Log in using the procedures described in the introduction to this manual.
2. Create a new part. The sample terminal session in figure 5-1 helps to illustrate this procedure.

| Terminal Session                                                                         | Comments                                                                                                                   |
|------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------------------------------------|
| <pre>ENTER PART NAME CAM</pre>                                                           | <p>Enter the name and sheet number of your new part. In this example, we called the part CAM with a sheet number of 1.</p> |
| <pre>SHEET NUMBER = 1</pre>                                                              |                                                                                                                            |
| <pre>NEW PART ASSUMED</pre>                                                              |                                                                                                                            |
| <pre>--- UNITS OF MEASURE M 1.METRIC (MM) 2.ENGLISH (INCH) 3.ENGLISH (FOOT/INCH) 2</pre> | <p>First, enter M to turn the menu display on. Next, select menu choice 2 to work in English units.</p>                    |
| <pre>--- DRAFTING STANDARD 1.ANSI 1982 2.ANSI 1973 3. 4. 5. 6. 7. 8. 9.          1</pre> | <p>Select menu choice 1 to use 1982 ANSI standard dimensions.</p>                                                          |
| <pre>ICEMDDN L=0 V=1 D=0.00 P=0 C=0</pre>                                                |                                                                                                                            |

Figure 5-1. Start Up Procedure for Constructing a Three-Dimensional Cam

## Defining Base Points

The first step is to define the base points of the cam:

### Keyboard Selection

Select 9.POINT from the main menu.

Select 2.KEY-IN from the Point menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

### Tablet Selection

|                       |
|-----------------------|
| Transform<br>XT,YT,ZT |
| Model<br>X,Y,Z        |

Enter the following values to define the first base point:

```
1.XT = 7
2.YT = 4]
3.ZT =0.0000
```

A small x appears on your screen.

### NOTE

You can also enter data using the automatic type-ahead facility. For example, you could enter 7 and 4 as your xt and yt values by typing:

```
7 (TAB) 4] (RETURN)
```

Enter the following xt and yt values to define the next two base points:

```
1.XT = 4.35 or enter 4.35 (TAB) 4] (RETURN)
2.YT = 4]
3.ZT =0.0000
```

```
1.XT = 9.65 or enter 9.65 (TAB) 4] (RETURN)
2.YT = 4]
3.ZT =0.0000
```

There should now be three small x's on your screen. These points will serve as the base points for the rest of the cam.

Enter ] to return to the Definition Space menu.

## Generating Cam Shaft Curves

In the next part of this session, you will learn how to use points, arcs, and depth variation to create the basic shape of the cam.

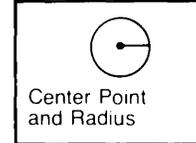
Create the top arc of the cam as follows:

### Keyboard Selection

**(CTRL) A + (RETURN)**

Select 3.CENTER AND RADIUS from the Arc/Circle/Fillet menu.

### Tablet Selection



- Position the graphics cursor near the center point and select it as the arc center using the P key or the tablet pen. An attention indicator appears on the point.

Enter the following radius, starting angle, and ending angle values:

```
1.RADIUS =0.2500 3.5 or enter 3.5 (TAB) 45 (TAB) 135] (RETURN)
2.STARTING ANGLE=0.0000 45
3.ENDING ANGLE =360.0000 135]
```

### NOTE

The default or current values display first. You can accept these values by entering ] or change them by entering new values. In the example above, all three of the values are changed.

Using the same center point you just used to create the top arc, define the bottom arc of the cam as follows:

- Position the graphics cursor near the center point and select it again as the center of the arc using the P key or the tablet pen. An attention indicator appears on the point.

Enter the following radius, starting angle, and ending angle values:

```
1.RADIUS =3.5000 2.25 or enter 2.25 (TAB) 225 (TAB) 315] (RETURN)
2.STARTING ANGLE=0.0000 225
3.ENDING ANGLE =360.0000 315]
```

Enter ] to return to the Arc/Circle/Fillet menu.

The next step is to create the top of the cam at a depth of 0.5 inch. To do this, you must first modify the current depth (0.0000) by entering the D key followed by RETURN or touching the D tablet square with your tablet pen.

Enter 0.5 as the new depth (0.5 inch):

```
DEPTH =0.0000 .5
```

The Arc/Circle/Fillet menu then redisplay.

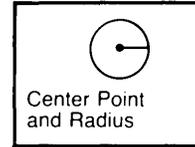
All of the entities that you create from now on are at a depth of 0.5 inch until you modify the depth again.

Next, define the cam shaft opening without the keyway as follows:

**Keyboard Selection**

**Tablet Selection**

Select 3.CENTER AND RADIUS from the Arc/Circle/Fillet menu.



Just as you did before, select the center point using the P key or the tablet pen.

Enter the following radius, starting angle, and ending angle values:

- 1.RADIUS =2.2500 .5 or enter .5 (TAB) 104.5 (TAB) 75.5 ] (RETURN)
- 2.STARTING ANGLE=0.0000 104.5
- 3.ENDING ANGLE =360.0000 75.5 ]

In the same way that you just created the cam shaft opening, define the full circle that forms the outside of the shaft as follows:

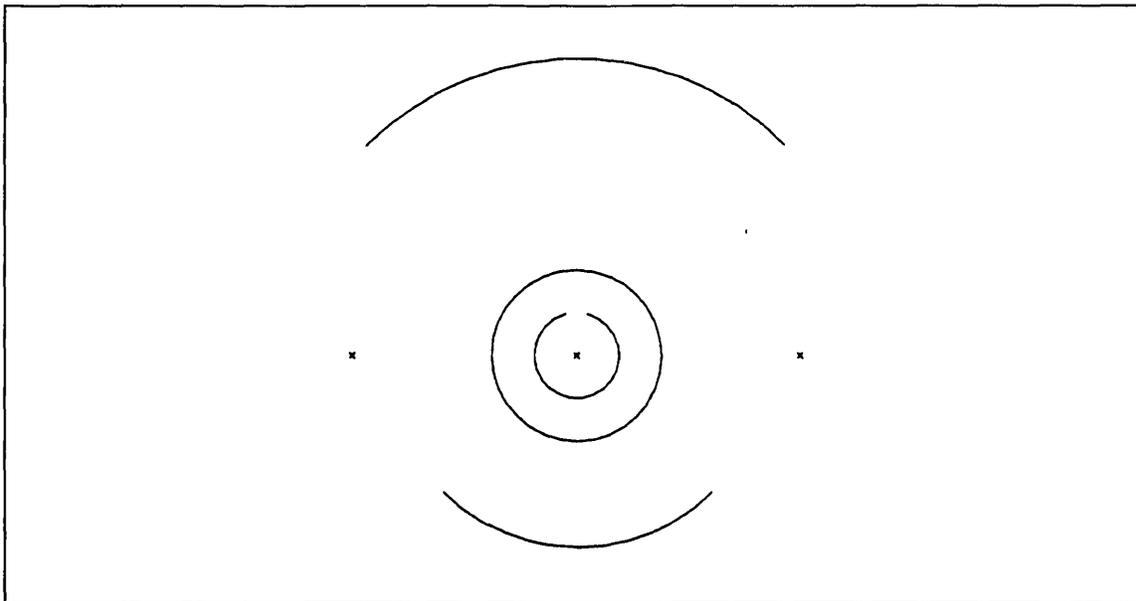
Select the center point again using the P key or the tablet pen.

Enter 1 as the radius and accept 0° and 360° as the starting and ending angles, respectively, by entering ]:

- 1.RADIUS =0.5000 1 ]
- 2.STARTING ANGLE=0.0000
- 3.ENDING ANGLE =360.0000

Enter ] to return to the Arc/Circle/Fillet menu.

The display on your screen should now match the display in figure 5-2.



**Figure 5-2. Initial Arcs for Cam**

Change the depth back to a value of 0 by entering the D key and RETURN, followed by 0 and ].

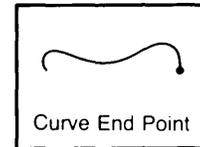
The next step toward finishing the cam outline is to define points at both ends of the top and bottom arcs:

#### Keyboard Selection

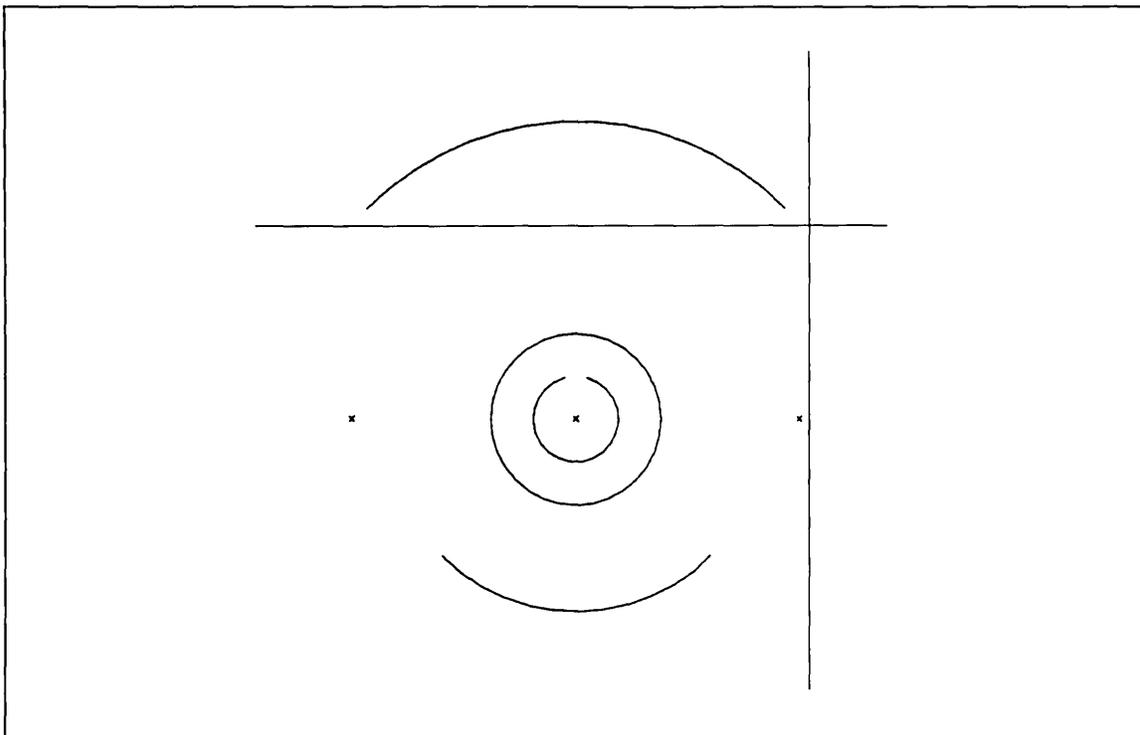
**(CTRL)** P + **(RETURN)**

Select 8.CURVE END from the Point menu.

#### Tablet Selection



- Position the graphics cursor near the right end of the top arc as shown in figure 5-3 and select the curve using the C key or the tablet pen. A point appears at the right end of the arc.



**Figure 5-3. Graphics Cursor Position**

- Position the graphics cursor near the left end of the top arc and select the curve using the C key or the tablet pen. A point appears at the left end of the arc.
- Position the graphics cursor near the left end of the bottom arc and select the curve using the C key or the tablet pen. A point appears at the left end of the arc.
- Position the graphics cursor near the right end of the bottom arc and select the curve using the C key or the tablet pen. A point appears at the right end of the arc.

Enter ] to return to the Point menu.



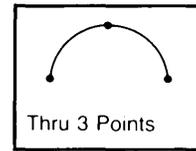
The last step is to draw the arcs that will complete the outline of the cam:

**Keyboard Selection**

**(CTRL) A + (RETURN)**

Select 7.THROUGH 3 POINTS from the Arc/Circle/Fillet menu.

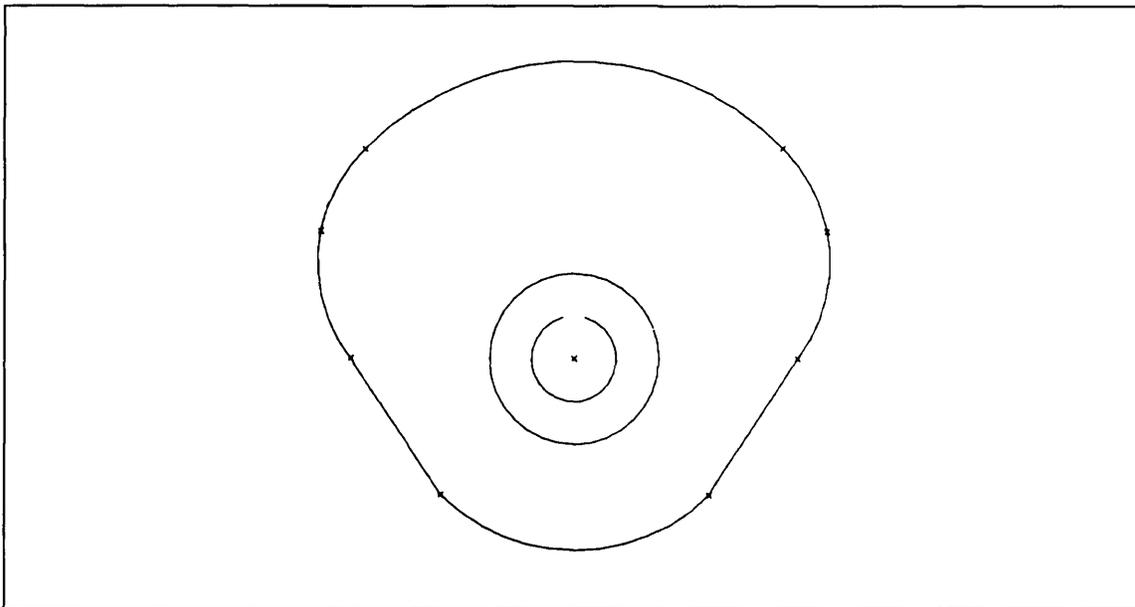
**Tablet Selection**



- Position the graphics cursor near the point at the left end of the top arc and select it using the P key or the tablet pen.
- Select the next two left points in a counterclockwise direction, ending with the point at the top end of the left slanted line.
- Position the graphics cursor near the point at the top end of the right slanted line and select it using the P key or the tablet pen.
- Select the next two right points in a counterclockwise direction, ending with the point at the right end of the top arc.

After the system displays the arc, enter F to return to the main menu.

The display on your terminal should now match figure 5-4.



**Figure 5-4. Cam and Cam Shaft Curves**

## Generating the Cam Shaft Keyway

Next, you will learn how to use the String function to create the keyway for the shaft hole.

Because you created the shaft at a depth of 0.5 inch, you must again modify the depth before you can create the keyway. To do this, enter D and RETURN followed by the new depth of .5:

```
DEPTH =0.0000 .5]
```

The main menu redisplay.

### NOTE

If you ever forget what depth you are working at, display the main menu heading and look for the depth value (abbreviated as D=x.xx). Now, your main menu heading should look like this:

```
ICEMDDN L=0 V=1 D=0.50 P=0 C=0
```

Before you can create the keyway using the String function, you must first tell the system where you want your string to begin:

### Keyboard Selection

Select 12.OTHER CURVES from the main menu.

Select 4.STRING from the Other Curves menu.

Select 4.CONNECT TO CURVE as the starting point.

### Tablet Selection

| String          |       |
|-----------------|-------|
| Screen Position | Coord |
| Point           | Curve |

- Position the graphics cursor near the upper left end of the open center circle and select it as the string starting point using the C key or the tablet pen. Note that *two attention indicators appear on the curve*, one in the middle of the curve and one at the left end. The attention indicator in the middle tells you that you have successfully selected that curve. The attention indicator at the end of the curve tells you which end serves as the starting point for the string.

Enter ] to return to the String menu.

Define the horizontal and vertical string segments that make up the keyway as follows:

Select 4.DELTA from the String menu.

Enter the following delta xt and yt values:

1.DELTA-XT = 0                      A vertical segment appears.  
2.        YT = .125 ]

1.DELTA-XT = .25                    A horizontal segment appears.  
2.        YT = 0 ]

Enter ] to return to the String menu.

Close the string with a vertical line from the horizontal line to the right side of the arc as follows:

Select 9.CONNECT TO CURVE from the String menu.

- Position the graphics cursor near the upper right end of the open circle in the center and select it using the C key or the tablet pen.

Enter ] to return to the String menu.

Enter ] again to indicate that you have finished the string.

Finally, you are asked if you want to define your string as individual lines and arcs:

DEFINE AS LINES AND ARCS?

Enter Y in response to this prompt.

Enter F to return to the main menu.

## Deleting Unnecessary Points

Next, you will learn how to use the Delete menu to remove all of the unnecessary points on your cam.

### Keyboard Selection

Select 3.DELETE from the main menu.

Select 3.ALL POINTS from the Delete menu.

### Tablet Selection

| Delete    |            |            |       |
|-----------|------------|------------|-------|
| Select    |            | Screen     | Chain |
| Region In | Region Out | All Points | Level |

You are asked if you want to delete all of the points on your drawing. This prompt gives you the opportunity to change your mind before all of the points on your part are deleted.

DELETE ALL POINTS?

Enter Y in response to this prompt.

The display on your terminal should now match figure 5-5.

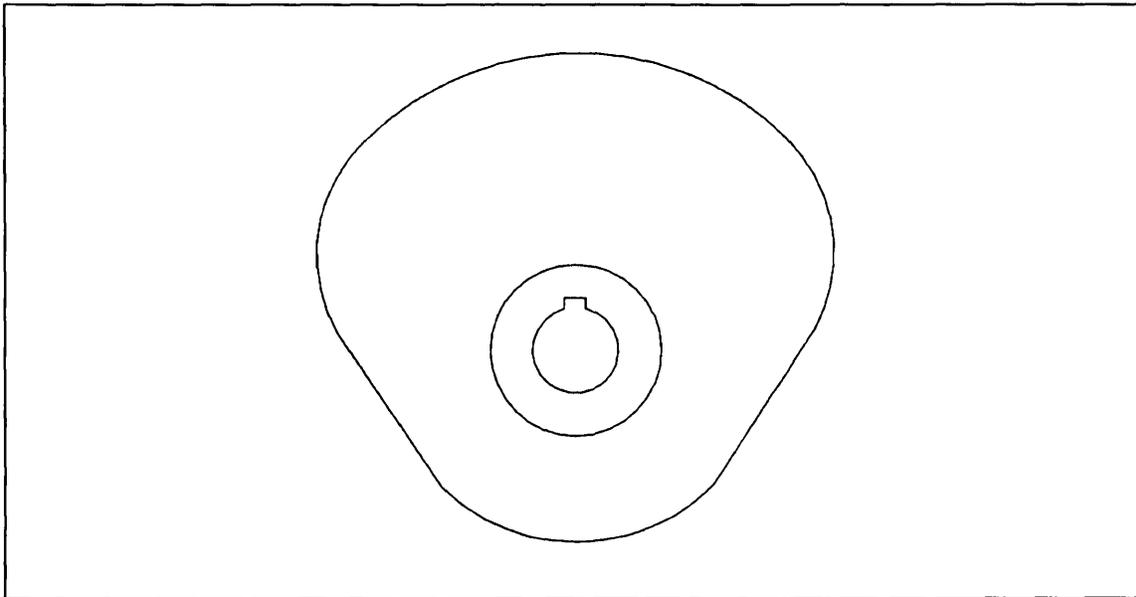


Figure 5-5. Completed Cam Shaft Keyway

## Projecting the Cam into Three Dimensions

In this part of the session, you will learn how to use the projected entities operation to create a copy of the set of entities that make up the cam. The copy is created at a distance from the original cam in a perpendicular direction and is connected to the original cam with lines.

### Keyboard Selection

Select 13.ENTITY MANIPULATION from the main menu.

Select 16.PROJECTED ENTITIES from the Entity Manipulation menu.

### Tablet Selection

| Entity Manipulation |                   |                   |                  |
|---------------------|-------------------|-------------------|------------------|
| Mirror              | Trans             | Rotate            | Rect Array       |
| Dupl Mirror         | Dupl Trans        | Dupl Rotate       | Circular Array   |
| Stretch             | Trans Rotate      | Rotate Trans      | Array Explode    |
| Group               | Dupl Trans Rotate | Dupl Rotate Trans | Project Entities |

In this example, you will project the cam into the screen (negative zt direction) at a distance of 1.25 inches:

DELTA ZT = 0.0000 -1.25 (RETURN)

Enter (CTRL) E to display the Entity Selection menu.

Select 2.CHAIN from the Entity Selection menu.

- Position the graphics cursor on the circle at the center of the part (the one with the keyway) and select it as the first entity using the A key or the tablet pen.
- Move the graphics cursor a short distance along the outline of the cam and enter C or press down with the tablet pen to indicate the direction for chaining the contiguous curves.

If there are attention indicators on each of the curves that make up the cam, enter ] to accept the selected entities.

If not, enter [ and repeat the steps outlined above. If the Selection Modification modal is on, select 17.CANCEL LAST MODIFICATION from the Modify Selection menu and try selecting the curves again.

Now, you are again prompted to enter a delta zt value. Enter RETURN to accept -1.25 as the delta zt value.

DELTA ZT = -1.25 (RETURN)

## PROJECTING THE CAM INTO THREE DIMENSIONS

- Position the graphics cursor to the left of the full circle and select it using the A key or the tablet pen.

Enter ] to define the projected entity.

In the same way you just projected the cam shaft opening, project the rest of the cam as follows:

Enter -.75 as the new delta zt value. Negative zt values are projected into the screen.

DELTA ZT = -1.25    -.75 (RETURN)

Enter (CTRL) E to display the Entity Selection menu.

Select 2.CHAIN from the Entity Selection menu.

- Position the graphics cursor above the top arc and select it as the first entity using the A key or the tablet pen.
- Move the graphics cursor a short distance along the outline of the cam and enter C or press down with the tablet pen to indicate the direction for chaining the contiguous curves.

If there are six attention indicators on the cam outline, enter ] to accept the selected entities.

If not, enter [ and repeat the steps outlined above. If the Selection Modification modal is on, select 17.CANCEL LAST MODIFICATION from the Modify Selection menu and try selecting the curves again.

Enter [ to return to the Entity Manipulation menu.

Enter F to return to the main menu.

## Displaying the Cam

In this part of the session, you will learn how to display the finished three-dimensional cam in the bottom and auxiliary views.

### Keyboard Selection

Select 8.DISPLAY CONTROL from the main menu.

Select 3.DISPLAY A SINGLE VIEW from the Display Control menu.

### Tablet Selection

| Display Views    |                    |             |              |                       |
|------------------|--------------------|-------------|--------------|-----------------------|
| Top-6<br>Front-1 | Front-1<br>Right-3 | ISO1-8      | One          | Last<br>Layout        |
| Two Views        |                    | Std<br>Four | Std<br>Eight | Last<br>Multi<br>View |
| Above<br>Below   | Side by<br>Side    |             |              |                       |

Display the bottom view of the cam at a scale that fills the entire screen as follows:

**VIEW NUMBER = 2**

You are then asked to select a zoom scale for the display. Select 5.AUTOMATICALLY MAXIMIZED from the menu or touch the tablet square marked AUTO MAX/MIN with your tablet pen. With an automatically maximized zoom scale, the system calculates the largest possible scale that the part can be displayed at and still fit on your screen.

The bottom view of your cam should now be displayed on the screen.

Display the auxiliary view of the cam (view 4) in the same way that you just displayed the bottom view of the cam:

### Keyboard Selection

Select 3.DISPLAY A SINGLE VIEW from the Display Control menu.

### Tablet Selection

| Display Views    |                    |             |              |                       |
|------------------|--------------------|-------------|--------------|-----------------------|
| Top-6<br>Front-1 | Front-1<br>Right-3 | ISO1-8      | One          | Last<br>Layout        |
| Two Views        |                    | Std<br>Four | Std<br>Eight | Last<br>Multi<br>View |
| Above<br>Below   | Side by<br>Side    |             |              |                       |

Enter 4 as the new view number:

**VIEW NUMBER = 4**

Select 5.AUTOMATICALLY MAXIMIZED as the zoom scale. Just as before, the display of the view fills the entire screen.

## DISPLAYING THE CAM

Display the bottom and the auxiliary view of your cam together on the screen as follows:

### Keyboard Selection

Select 4.DISPLAY MULTIPLE VIEWS from the Display Control menu.

Select 4.TWO VIEWS - TOP AND BOTTOM to display two views of the cam.

### Tablet Selection

| Display Views |              |          |           |                 |
|---------------|--------------|----------|-----------|-----------------|
| Top-6         | Front-1      | ISO1-8   | One       | Last Layout     |
| Front-1       | Right-3      |          |           |                 |
| Two Views     |              | Std Four | Std Eight | Last Multi View |
| Above Below   | Side by Side |          |           |                 |

To display the auxiliary view in the top half of the screen, enter 4 as the top view:

SELECT TOP VIEW  
VIEW NUMBER = 4

To display the bottom view in the bottom half of the screen, enter 2 as the bottom view:

SELECT BOTTOM VIEW  
VIEW NUMBER = 2

Select 5.AUTOMATICALLY MAXIMIZED as the zoom scale or touch the tablet square marked AUTO MAX/MIN with your tablet pen. The display fills the entire screen. The result is shown in figure 5-6.

Enter F to return to the main menu.

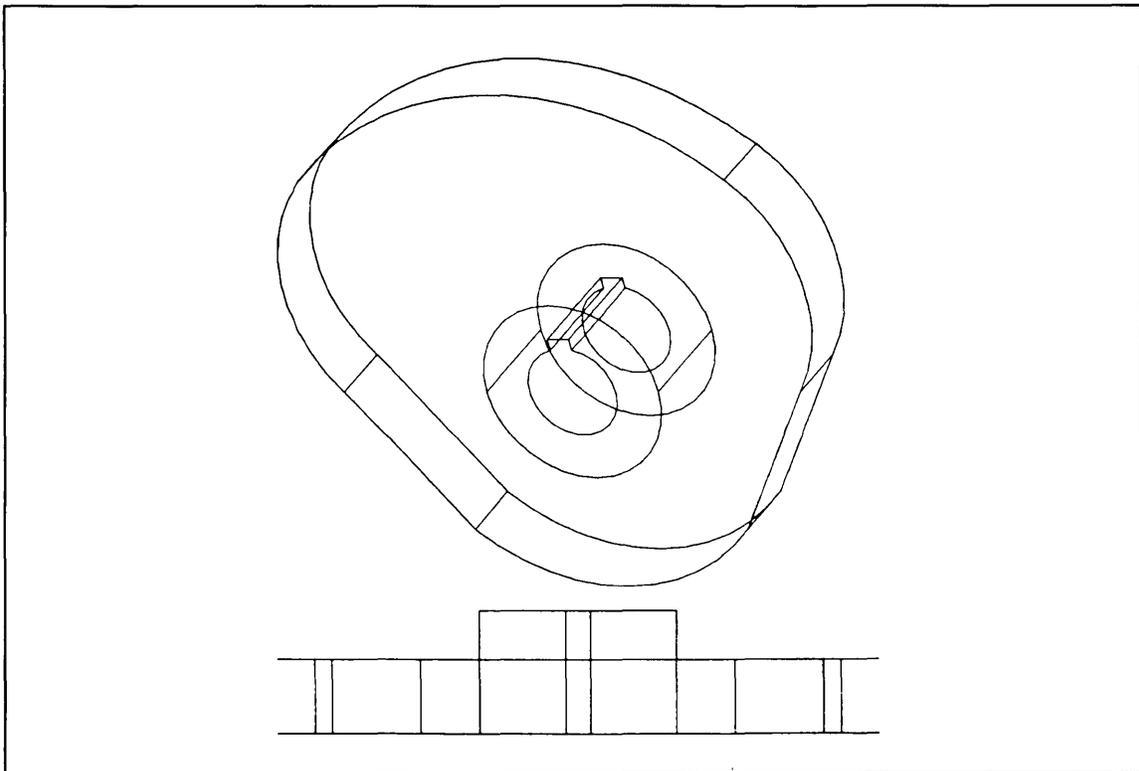


Figure 5-6. Auxiliary and Bottom Views of Cam

## Filing and Exiting

Figure 5-7 outlines the steps to end this terminal session and save your part.

| Terminal Session                                                                                                                                                                                                                                               | Comments                                                                                                                                                                                                                          |
|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ICEMDDN L=0 V=1 D=0.00 P=0 C=1 4                                                                                                                                                                                                                               | If you are using the keyboard, enter 4 to select the File Current Part/Exit ICEM DDN menu.                                                                                                                                        |
| FILE CURRENT PART/EXIT ICEM DDN<br>1.FILE - CONTINUE CURRENT PART<br>2. - GET DIFFERENT PART<br>3. - QUIT SESSION<br>4. - SUSPEND SESSION<br>5.DO NOT FILE - CONTINUE FROM LAST FILE<br>6. - GET DIFFERENT PART<br>7. - QUIT SESSION<br>8. - SUSPEND SESSION 3 | If you are using the tablet, you can file your part and exit from ICEM DDN in a single step by selecting the FILE/QUIT square with your tablet pen.<br><br>Select 3.FILE - QUIT SESSION to save your part and exit from ICEM DDN. |
| ICEMDDN TERMINATED.                                                                                                                                                                                                                                            |                                                                                                                                                                                                                                   |
| /                                                                                                                                                                                                                                                              |                                                                                                                                                                                                                                   |

Figure 5-7. Sample File/Exit Sequence

(

)

(

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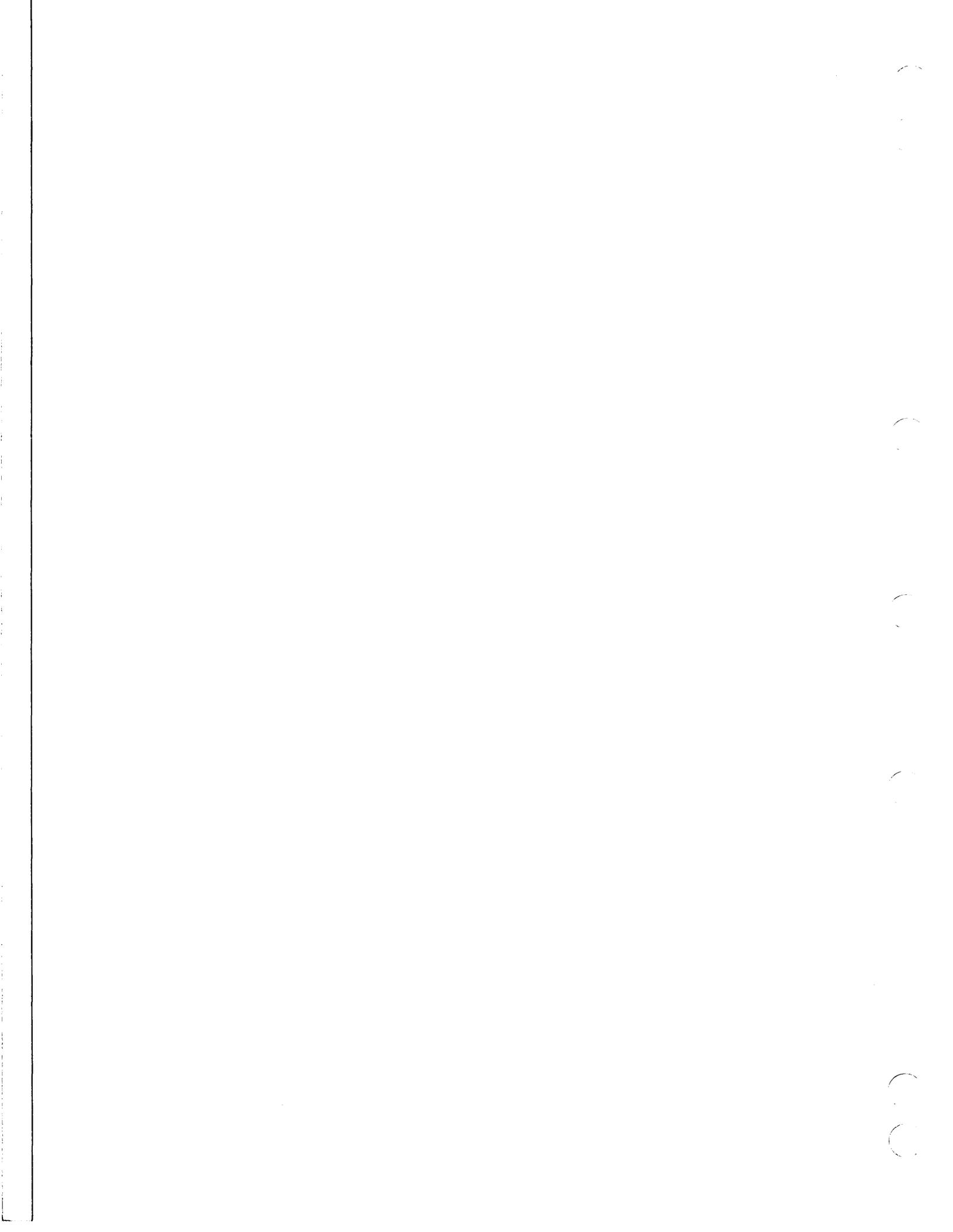
)

# Constructing a Latch

---

6

|                                    |      |
|------------------------------------|------|
| Getting Started .....              | 6-1  |
| Generating the Keyway .....        | 6-2  |
| Generating the Latch .....         | 6-4  |
| Closing the Latch.....             | 6-9  |
| Magnifying the Keyway Region ..... | 6-11 |
| Renaming the Part .....            | 6-15 |
| Filing and Exiting .....           | 6-16 |
| Practice Session .....             | 6-17 |



In this practice session, you will learn how to construct a latch using basic geometry and drafting. You will also learn how to use strings and fillets to construct a simple, two-dimensional part and how to use detail magnification to view an enlarged portion of the part.

## Getting Started

Create a new user part in which to create the latch as follows:

1. Log in using the procedures described in the introduction to this manual.
2. Create a new part. The sample terminal session in figure 6-1 helps to illustrate this procedure.

| Terminal Session                                                                                                             | Comments                                                                                                                     |
|------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------|
| <pre> ENTER PART NAME  LATCH  SHEET NUMBER = 1  NEW PART ASSUMED                     </pre>                                  | <p>Enter the name and sheet number of your new part. In this example, we called the part LATCH with a sheet number of 1.</p> |
| <pre> --- UNITS OF MEASURE  M   1.METRIC (MM)   2.ENGLISH (INCH)   3.ENGLISH (FOOT/INCH)  2                     </pre>       | <p>First, enter M to turn the menu display on. Next, select menu choice 2 to work in English units.</p>                      |
| <pre> --- DRAFTING STANDARD   1.ANSI 1982   2.ANSI 1973   3.   4.   5.   6.   7.   8.   9.      1                     </pre> | <p>Select menu choice 1 to use 1982 ANSI standard dimensions.</p>                                                            |

Figure 6-1. Start Up Procedure for Constructing a Latch

## Generating the Keyway

In the first part of this session, you will create the small keyway shown in figure 6-2 using the String function.

### NOTE

As you begin entering values, don't be concerned if you can't see each segment of the string as you create it. Because you are working in actual scale, some of the smaller string segments will be difficult to see. For example, a string segment with a length of 0.02 appears on your screen at exactly that length. Later in this session, you will learn how to magnify this area using the Detail Magnification function.

### Keyboard Selection

Select 12.OTHER CURVES from the main menu.

Select 4.STRING from the Other Curves menu.

Select 2.KEY-IN as the starting point.

### Tablet Selection

| String          |       |
|-----------------|-------|
| Screen Position | Coord |
| Point           | Curve |

Enter the following values to define the starting point of the string:

```
1.XT = 8
2.YT = 8]
```

An attention indicator appears on your screen at the coordinates (8,8).

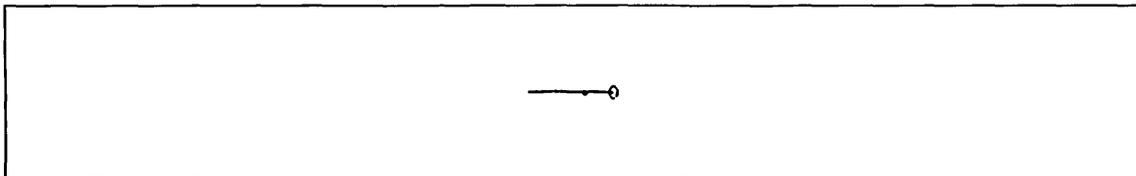
Enter ] to return to the String menu.

Select 4.DELTA from the String menu and enter the following delta xt and yt values:

- |                   |                                    |
|-------------------|------------------------------------|
| 1.DELTA-XT = -.24 | A horizontal line segment appears. |
| 2. YT = 0 ]       |                                    |
| 1.DELTA-XT = 0    | A vertical line segment appears.   |
| 2. YT = -.02 ]    |                                    |
| 1.DELTA-XT = -.02 | A horizontal line segment appears. |
| 2. YT = 0 ]       |                                    |
| 1.DELTA-XT = 0    | A vertical line segment appears.   |
| 2. YT = .01 ]     |                                    |
| 1.DELTA-XT = .01  | A horizontal line segment appears. |
| 2. YT = 0 ]       |                                    |
| 1.DELTA-XT = 0    | A vertical line segment appears.   |
| 2. YT = .01 ]     |                                    |
| 1.DELTA-XT = -.5  | A horizontal line segment appears. |
| 2. YT = 0 ]       |                                    |

Enter ] to return to the String menu.

The display on your screen should now match figure 6-2.



**Figure 6-2. Keyway**

## Generating the Latch

Create the rest of the latch outline as follows:

Select 7.CW ARC (clockwise arc) from the String menu and enter the following radius, arc length, and delta angle values:

```

1.RADIUS = 0.2500 .1875 A clockwise arc appears.
2.ARC LENGTH = 0.0000 (RETURN)
3.DELTA ANGLE = 0.0000 180]

```

---

### NOTE

A RETURN is used to tell the system to accept the current or default value.

---

Select 4.DELTA from the String menu and enter the following delta xt and yt values:

```

1.DELTA-XT = .75 A horizontal line segment appears.
2. YT = 0]

```

```

1.DELTA-XT = 0 A vertical line segment appears.
2. YT = .375]

```

```

1.DELTA-XT = -1.25 A horizontal line segment appears.
2. YT = 0]

```

Enter ] to return to the String menu.

Select 8.CCW ARC from the String menu and enter the following radius, arc length, and delta angle values:

```

1.RADIUS = 0.1875 .75 A counterclockwise arc appears.
2.ARC LENGTH = 0.0000 (RETURN)
3.DELTA ANGLE = 0.0000 90]

```

The display on your terminal should now match figure 6-3.

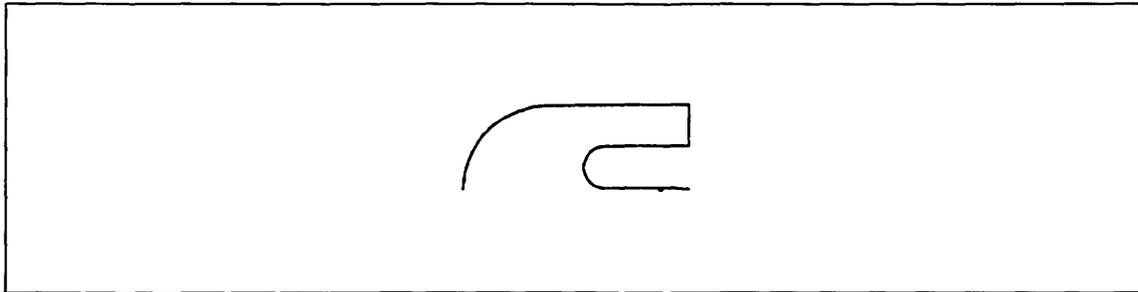


Figure 6-3. Starting the Latch



GENERATING THE LATCH

Select 7.CW ARC from the String menu and enter the following radius, arc length, and delta angle values:

- 1.RADIUS = 0.5000 (RETURN) A clockwise arc appears.
- 2.ARC LENGTH = 0.0000 (RETURN)
- 3.DELTA ANGLE = 0.0000 90 ]

Select 4.DELTA from the String menu and enter the following delta xt and yt values:

- 1.DELTA-XT = 0 A vertical line segment appears.
- 2. YT = -1 ]

Enter ] to return to the String menu.

Select 8.CCW ARC from the String menu and enter the following radius, arc length, and delta angle values:

- 1.RADIUS = 0.5000 .75 A counterclockwise arc appears.
- 2.ARC LENGTH = 0.0000 (RETURN)
- 3.DELTA ANGLE = 0.0000 180 ]

Select 4.DELTA from the String menu and enter the following delta xt and yt values:

- 1.DELTA-XT = 0 A vertical line segment appears.
- 2. YT = 2 ]

Enter ] to return to the String menu.

Select 8.CCW ARC from the String menu and enter the following radius, arc length, and delta angle values:

- 1.RADIUS = 0.7500 (RETURN) A counterclockwise arc appears.
- 2.ARC LENGTH = 0.0000 (RETURN)
- 3.DELTA ANGLE = 0.0000 90 ]

The display on your terminal should now match figure 6-4.

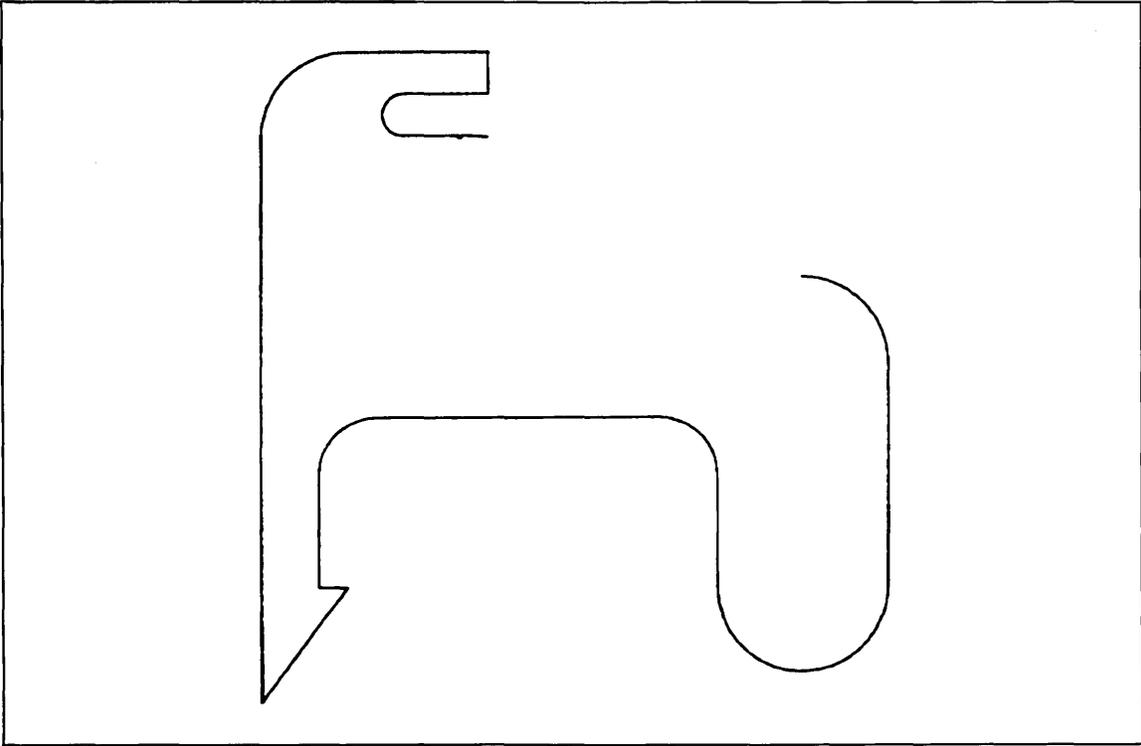


Figure 6-4. Creating the Latch



## Closing the Latch

Connect the two ends of the latch outline using fillet construction as follows:

### Keyboard Selection

**(CTRL) A + (RETURN)**

Select 9.FILLET from the Arc/Circle/Fillet menu.

Select 1.SPECIFIED RADIUS as the fillet radius mode.

Select 1.TRIM/EXTEND as the fillet trim mode.

Select 2.INDICATE CENTER as the fillet center mode.

Enter .25 as the new radius value:

RADIUS = 0.7500 .25 ]

- Position the graphics cursor at the end of the last line created and select it using the C key or the tablet pen.
- Position the graphics cursor at the right end of the first line segment created and select it using the C key or the tablet pen.

If attention indicators appear on each of the lines you just selected, enter ] to accept the selected entities. If not, enter [ for each incorrectly selected line. Entering ] tells ICEM DDN to cancel the selection you just made.

Next, you are asked to indicate the approximate location of the fillet center.

- Position the graphics cursor between the two attention indicators that appear on the two previously selected lines and select that position as the center of the fillet using the C key or the tablet pen.

As soon as you indicate the center, the fillet is drawn and the lines are trimmed automatically.

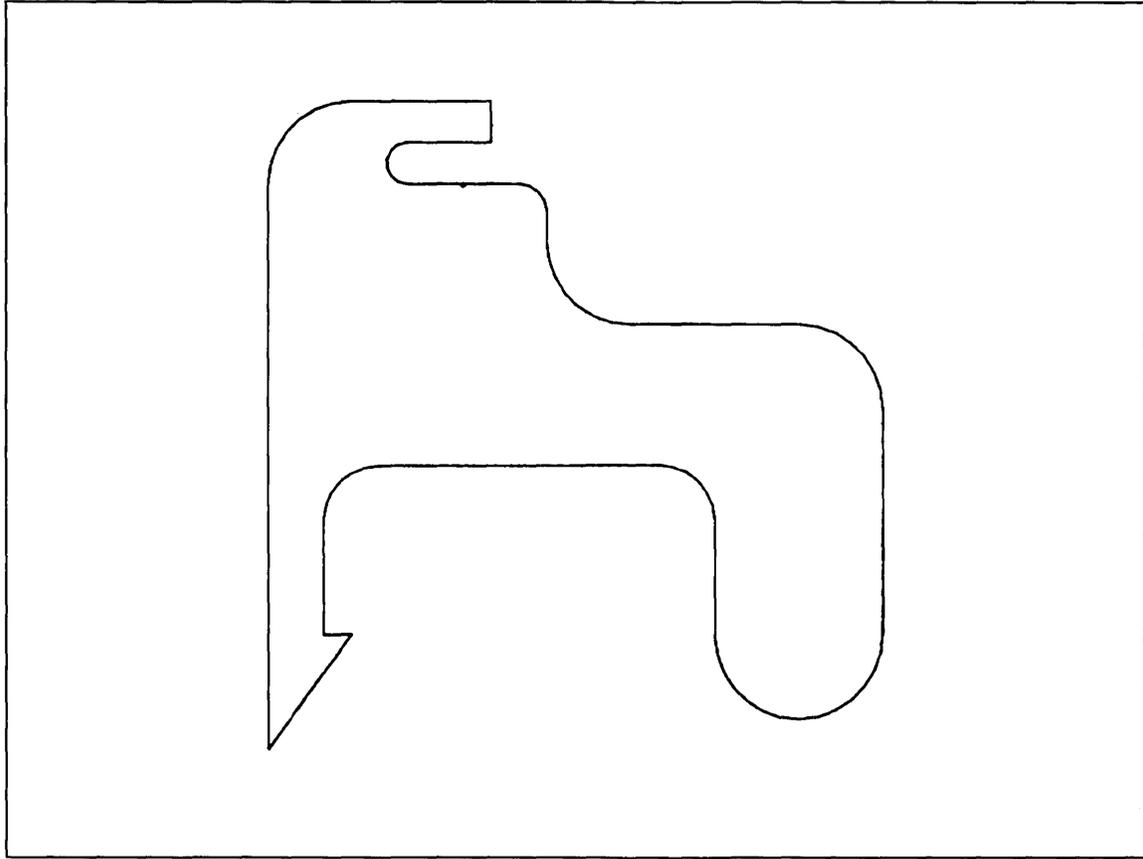
Enter F to return to the main menu.

### Tablet Selection

| Fillet                                                                                               |                                                                                                   |
|------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------|
| Specify Radius                                                                                       | Unspec Radius  |
| Auto Ctr No Trim  | Auto Ctr Trim  |

CLOSING THE LATCH

The display on your terminal should now match figure 6-5.



**Figure 6-5. Completed Latch**

## Magnifying The Keyway Region

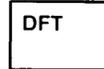
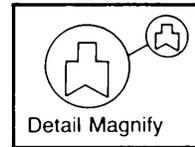
Create a detail magnification circle of the keyway as follows:

### Keyboard Selection

Select 16.DRAFTING from the main menu.

Select 14.DETAIL MAGNIFY from the Drafting menu.

### Tablet Selection



You are first asked if you want to use an existing circle for the detail magnification. Because there are no existing circles on the screen, enter N in response to the prompt:

EXISTING CIRCLE FOR REGION TO MAGNIFY?

- Position the graphics cursor on the keyway and select it as the center of the circle for the area to be magnified using the C key or the tablet pen.
- Position the graphics cursor as shown in figure 6-6 (slightly above and to the right of the keyway) and indicate the border of the circle for the area to be magnified using the C key or the tablet pen. The system draws a phantom circle around the keyway.

Select 2.INDICATE CENTER AND EDGE POSITIONS to specify the type of detail magnification circle.

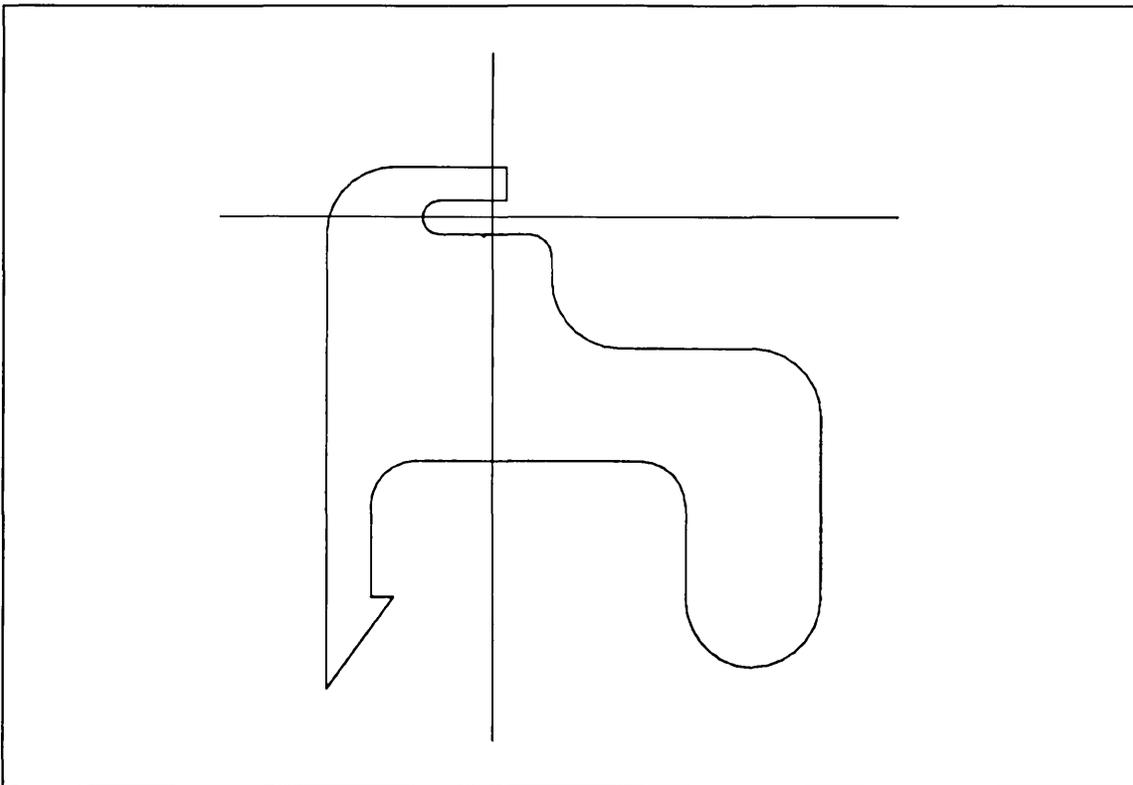


Figure 6-6. Select Detail Area

## MAGNIFYING THE KEYWAY REGION

To create a border around the detail magnification area, enter Y in response to the prompt:

DO YOU WANT THE REGION BORDERED?

- Position the graphics cursor as shown in figure 6-7 (above and to the right of the latch) and indicate the center of the area of magnification using the C key or the tablet pen.
- Position the graphics cursor as shown in figure 6-8 (below and to the right of the last position indicated) and indicate the outer edge of the detail magnification circle using the C key or the tablet pen. The system displays a circle using the phantom line font.

If the circle looks the way that you want it to, enter Y in response to the prompt:

MAGNIFICATION REGIONS OK?

The completed detail magnification appears in figure 6-9.

When you see the EXISTING CIRCLE FOR REGION TO MAGNIFY? prompt, enter F to return to the main menu.

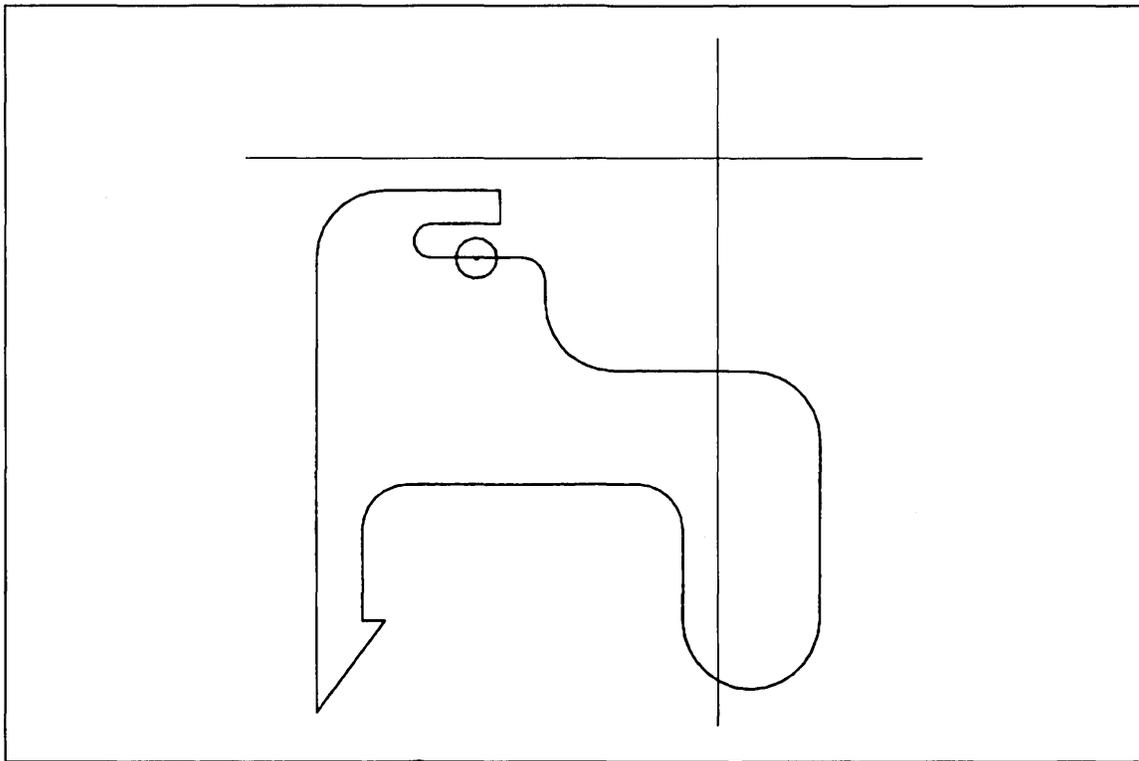


Figure 6-7. Select Center of a Detail Circle

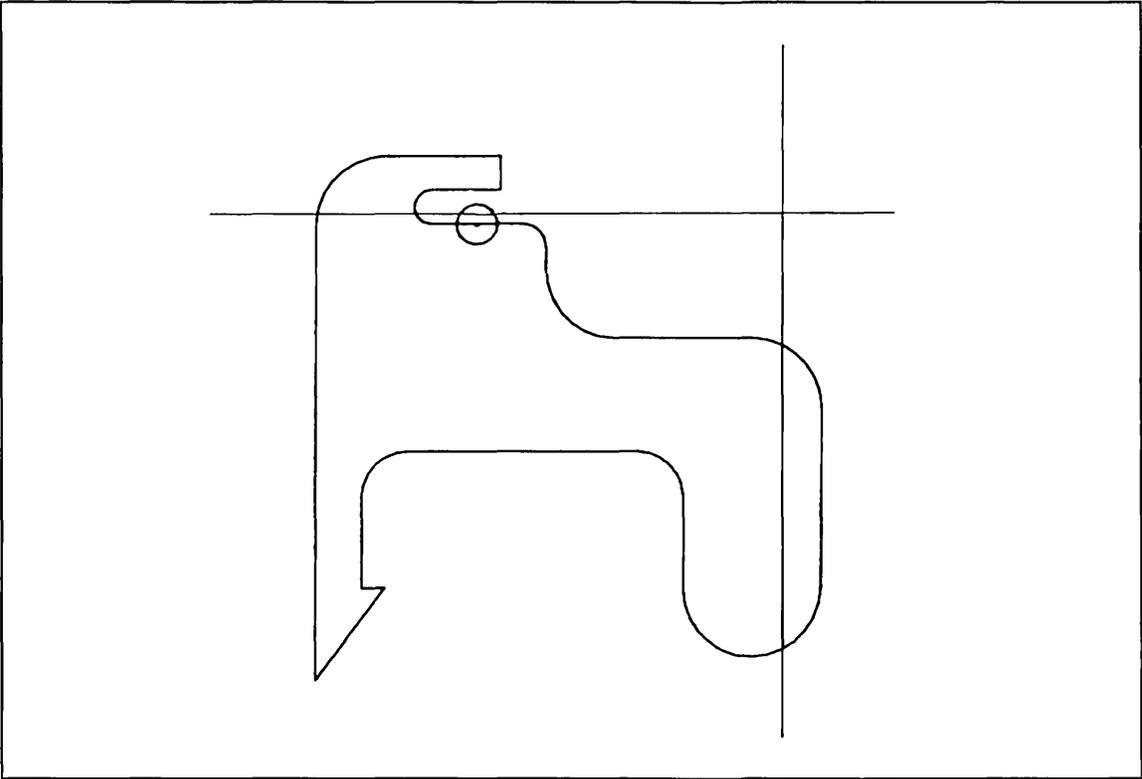


Figure 6-8. Select Outer Edge of a Detail Circle

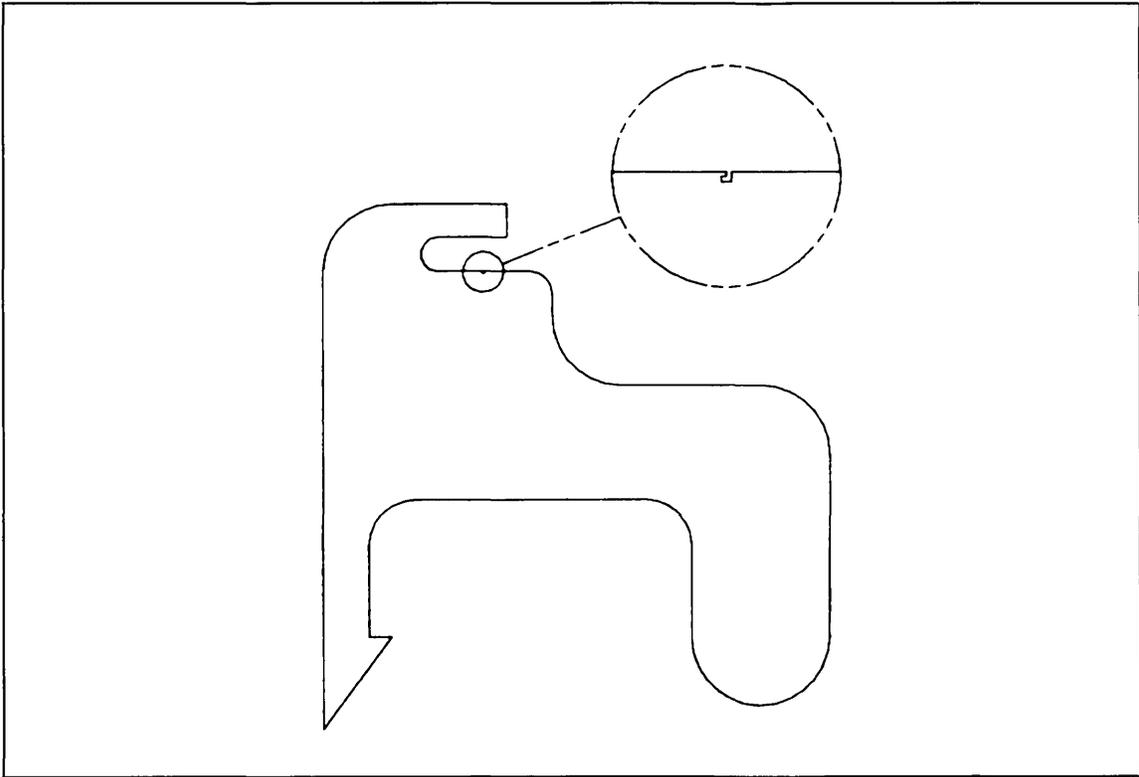


Figure 6-9. Completed Detail Magnification Circle

## Renaming the Part

Rename the current part name (LATCH, sheet 1) to a new part name (LATCH, sheet 2). The part named LATCH, sheet 2 is used in the next chapter of this manual.

### Keyboard Selection

Select 6.DATA BASE MANAGE-  
MENT from the main menu.

Select 1.PART MANAGEMENT  
from the Data Base Management  
menu.

Select 4.RENAME CURRENT  
PART from the Part Management  
menu.

Enter LATCH in response to the prompt:

ENTER NEW PART NAME LATCH

Enter 2 in response to the prompt:

SHEET NUMBER = 2

You now have a new current part named LATCH, sheet 2. The part named LATCH, sheet 1, no longer exists because it was not saved before it was renamed.

Enter F to return to the main menu.

### Tablet Selection

| Part Management     |                |           |                     |
|---------------------|----------------|-----------|---------------------|
| Independent Save    | Global Save    | List File | Rename Current Part |
| Independent Restore | Global Restore | Delete    | Change Status       |

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## Filing and Exiting

Figure 6-10 outlines the steps necessary to file the current part named LATCH, sheet 2.

| Terminal Session                                                                                                                                                                                                                                                                    | Comments                                                                                                                                                                                                                                 |
|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ICEMDDN L=0 V=1 D=0.00 P=0 C=1 4                                                                                                                                                                                                                                                    | If you are using the keyboard, enter 4 to select the File Current Part/Exit ICEM DDN menu.                                                                                                                                               |
| <pre> FILE CURRENT PART/EXIT ICEM DDN 1.FILE - CONTINUE CURRENT PART 2.   - GET DIFFERENT PART 3.   - QUIT SESSION 4.   - SUSPEND SESSION 5.DO NOT FILE - CONTINUE FROM LAST FILE 6.   - GET DIFFERENT PART 7.   - QUIT SESSION 8.   - SUSPEND SESSION 3                     </pre> | <p>If you are using the tablet, you can file your part and exit from ICEM DDN in a single step by selecting the FILE/QUIT square with your tablet pen.</p> <p>Select 3.FILE - QUIT SESSION to save your part and exit from ICEM DDN.</p> |
| ICEMDDN TERMINATED.                                                                                                                                                                                                                                                                 |                                                                                                                                                                                                                                          |
| /                                                                                                                                                                                                                                                                                   |                                                                                                                                                                                                                                          |

**Figure 6-10. Sample File/Exit Sequence**

## Practice Session

If you would like more practice using the String function, try creating the latch again, but this time, use only the dimensioned latch shown in figure 6-11 as your guide. If you need help, simply go back to the step-by-step exercise you just completed.

If you would like to try this exercise, create a new part named DIM LATCH, sheet 1, on which to construct your latch.

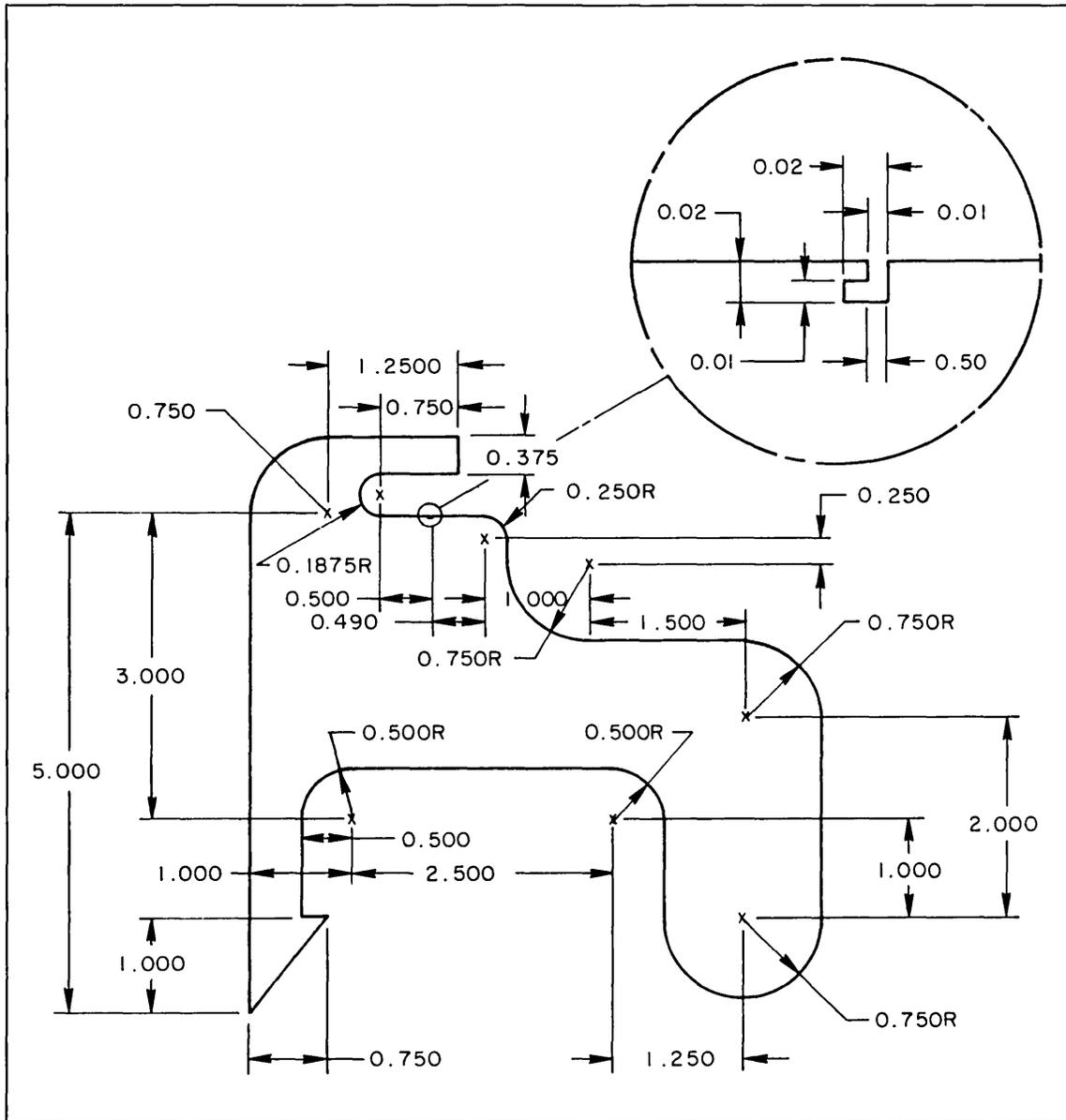


Figure 6-11. Dimensioned Latch



## Viewing the Latch

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|                                                  |      |
|--------------------------------------------------|------|
| Getting Started .....                            | 7-1  |
| Blanking the Magnified Area.....                 | 7-2  |
| Changing the Curve Font to Dashed .....          | 7-5  |
| Projecting the Latch into Three Dimensions ..... | 7-5  |
| Displaying the Latch in Four Views .....         | 7-6  |
| Displaying a Single View .....                   | 7-8  |
| Defining an Auxiliary View .....                 | 7-9  |
| Displaying the Latch in Two Views .....          | 7-11 |
| Zooming In on the Keyway .....                   | 7-13 |
| Filing and Exiting .....                         | 7-17 |

100

100

100

100

100

In this practice session, you will learn how to construct a three-dimensional entity from a two-dimensional entity and how to create new views to see details of the three-dimensional entity.

## Getting Started

Retrieve the latch that you created in the previous chapter as follows:

1. Log in using the procedures described in the introduction to this manual.
2. Retrieve the part named LATCH, sheet 2. The sample terminal session in figure 7-1 helps to illustrate this procedure.

| Terminal Session      | Comments                                                                                |
|-----------------------|-----------------------------------------------------------------------------------------|
| ENTER PART NAME LATCH | Enter the name and sheet number of the part that you want to retrieve (LATCH, sheet 2). |
| SHEET NUMBER = 2      |                                                                                         |

**Figure 7-1. Start Up Procedure for Viewing the Latch**

## Blanking the Magnified Area

It is often helpful to assign specific entities to a level other than 0 (the default level) so that you can blank all of the entities on that level. Blanking is the process of suppressing the entity *display*. A blanked entity still exists in the data base, but simply does not display on your screen until you unblank it.

First, assign the detail magnification entities to level 1 and then blank all entities on level 1 as follows:

### Keyboard Selection

Select 1.MODALS AND FONTS from the main menu.

Select 7.MODIFY ENTITY LEVEL/PEN NO. from the Modals and Fonts menu.

### Tablet Selection

| Modify        |
|---------------|
| Level Pen     |
| Surface Paths |

- Position the graphics cursor on the small circle and select it using the A key or the tablet pen.
- Position the graphics cursor on the line connecting the small circle and the detail magnification area and select it using the C key or the tablet pen.

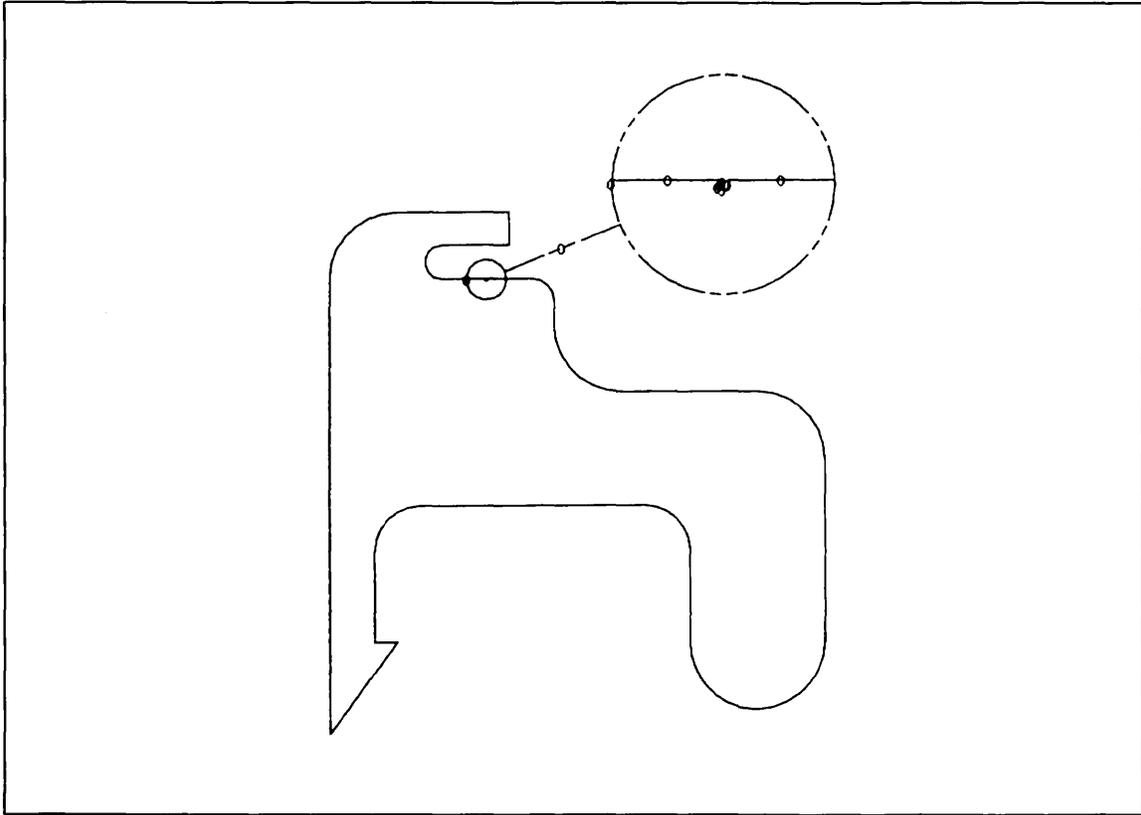
Enter **(CTRL)** E to display the Entity Selection menu.

Select 3.REGION IN from the Entity Selection menu.

- Position the graphics cursor to the lower left of the detail magnification area and select it using the C key or the tablet pen. The graphics cursor position should define one corner of a rectangle which encloses all of the detail magnification area.
- Position the graphics cursor to the upper right of the detail magnification area and select it using the C key or the tablet pen. The graphics cursor position should define the other corner of a rectangle which encloses all of the detail magnification area.

If the display on your screen matches figure 7-2, enter ] to accept the selected entities.

If not, enter [ and repeat the steps outlined above. If the Selection Modification modal is on, select 17.CANCEL LAST MODIFICATION from the Modify Selection menu and try selecting the entities again.



**Figure 7-2. Blanking the Magnified Area**

You are then asked if you want to retain the original levels for the entities you have just selected. Enter N in response to the prompt:

RETAIN ORIGINAL LEVEL NUMBERS?

By entering N, you are telling the system that you wish to modify the level of an entity or group of entities.

Enter 1 as the new level number:

LEVEL =0 1

You are asked if you want to retain the original pen numbers for the selected entities:

RETAIN ORIGINAL PEN NUMBERS?

Enter Y to return to the Entity Selection menu.

Since you have nothing more to modify, enter ] to return to the main menu.

## BLANKING THE MAGNIFIED AREA

Blank all of the entities on level 1 as follows:

### Keyboard Selection

Select 2.BLANK/UNBLANK from the main menu.

Select 2.BLANK-LEVEL RANGE from the Blank/Unblank menu.

### Tablet Selection

| Blank   |       |     |               |
|---------|-------|-----|---------------|
| Select  | Level | All | All of a Type |
| Unblank |       |     |               |
| Select  | Level | All | All of a Type |

Because you are blanking only one level, enter the range as level 1 to level 1 and accept the default increment value of 1:

1. FROM LEVEL = 0 1
  2. TO LEVEL = 0 1 ]
  3. BY INCREMENT = 1
- Default values are displayed first.

Enter F to return to the main menu.

Your screen should now match figure 7-3.

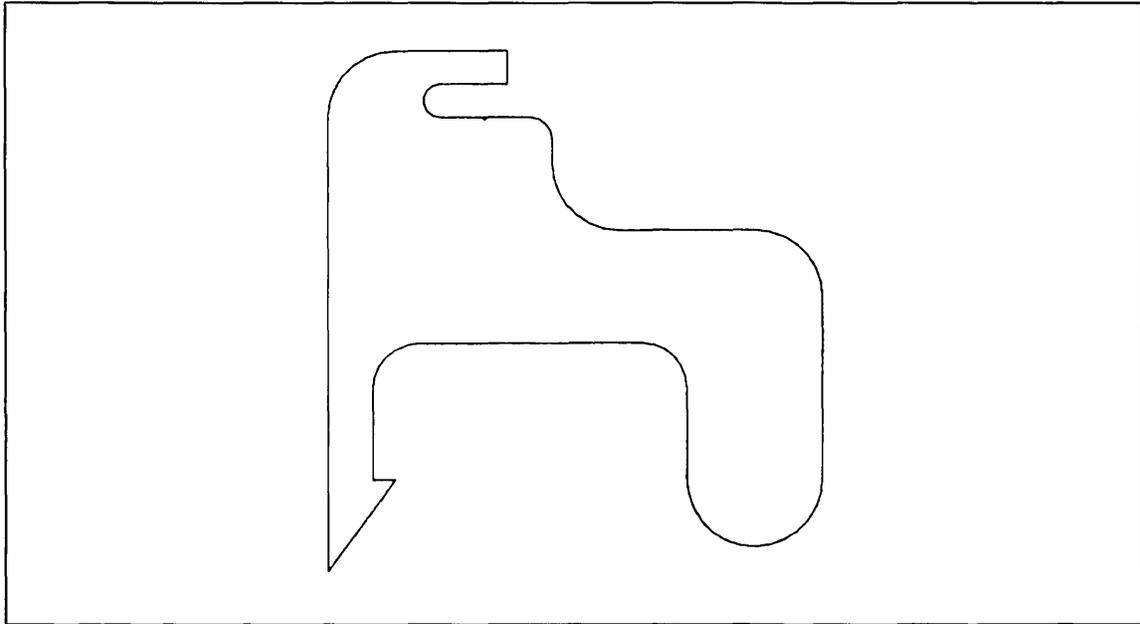


Figure 7-3. Latch Without Magnified Area

## Changing the Curve Font to Dashed

Before you project your latch in three dimensions, change the line font to dashed. This makes the lines that connect the latch with its projection appear as dashed lines.

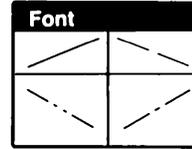
### Keyboard Selection

Select 1.MODALS AND FONTS from the main menu.

Select 5.CURVE FONT from the Modals and Fonts menu.

Select 2.DASHED as the curve font.

### Tablet Selection



Enter ] to return to the main menu.

All of the lines or arcs that you create from now on are dashed-until you change the curve font back to solid.

## Projecting the Latch into Three Dimensions

Create a projection of the latch as follows:

### Keyboard Selection

Select 13.ENTITY MANIPULATION from the main menu.

Select 16.PROJECTED ENTITIES from the Entity Manipulation menu.

### Tablet Selection

| Entity Manipulation |                   |                   |                  |
|---------------------|-------------------|-------------------|------------------|
| Mirror              | Trans             | Rotate            | Rect Array       |
| Dupl Mirror         | Dupl Trans        | Dupl Rotate       | Circular Array   |
| Stretch             | Trans Rotate      | Rotate Trans      | Array Explode    |
| Group               | Dupl Trans Rotate | Dupl Rotate Trans | Project Entities |

In this example, we will project the latch into the screen (negative zt direction) at a distance of 1 inch. Enter -1 to define a region projected 1 inch *into the screen*:

DELTA ZT =0.0000 -1 (RETURN)

Enter (CTRL) E to display the Entity Selection menu.

Select 2.CHAIN from the Entity Selection menu.

- Position the graphics cursor on a line segment of the figure and select it using the C key or the tablet pen. It does not matter which segment you choose because all line and arc segments of the latch are contiguous.
- Move the graphics cursor a short distance in the counterclockwise direction and enter C or press down with the tablet pen again to indicate the chaining direction.
- If all of the line and arc segments have been marked with attention indicators, enter ] to accept the selected entities.

If not, enter [ and repeat the steps outlined above. If the Selection Modification modal is on, select 17.CANCEL LAST MODIFICATION from the Modify Selection menu and try selecting the entities again.

The selected entities are then projected 1 inch in the negative zt direction and you are again prompted to enter a delta zt value. Now, enter [ (*operation reject*) to exit. If you enter ] (*operation complete*), you are telling the system that you wish to continue projecting entities using the currently displayed delta zt value.

Enter ] to return to the main menu.

## Displaying the Latch in Four Views

Display four views of the latch simultaneously as follows:

Select 8.DISPLAY CONTROL from the main menu.

Select 4.DISPLAY MULTIPLE VIEWS from the Display Control menu.

Select 6.FOUR QUADRANTS to display the latch in four views.

You are first asked to enter the number of the view you wish to display in each of the four quadrants on your screen. Enter 1 as the upper left view, 2 as the lower left view, 3 as the upper right view, and 4 as the lower right view.

### NOTE

---

You can also enter all of the view numbers in a single step by typing:

1 (TAB) 2 (TAB) 3 (TAB) 4 (RETURN)

---

SELECT UPPER LEFT VIEW  
VIEW NUMBER = 1

SELECT LOWER LEFT VIEW  
VIEW NUMBER = 2

SELECT UPPER RIGHT VIEW  
VIEW NUMBER = 3

SELECT LOWER RIGHT VIEW  
VIEW NUMBER = 4

You are then asked to select the zoom scale for the display. For this example, select 5.AUTOMATICALLY MAXIMIZED as the zoom scale. The display on your screen should now match figure 7-4.

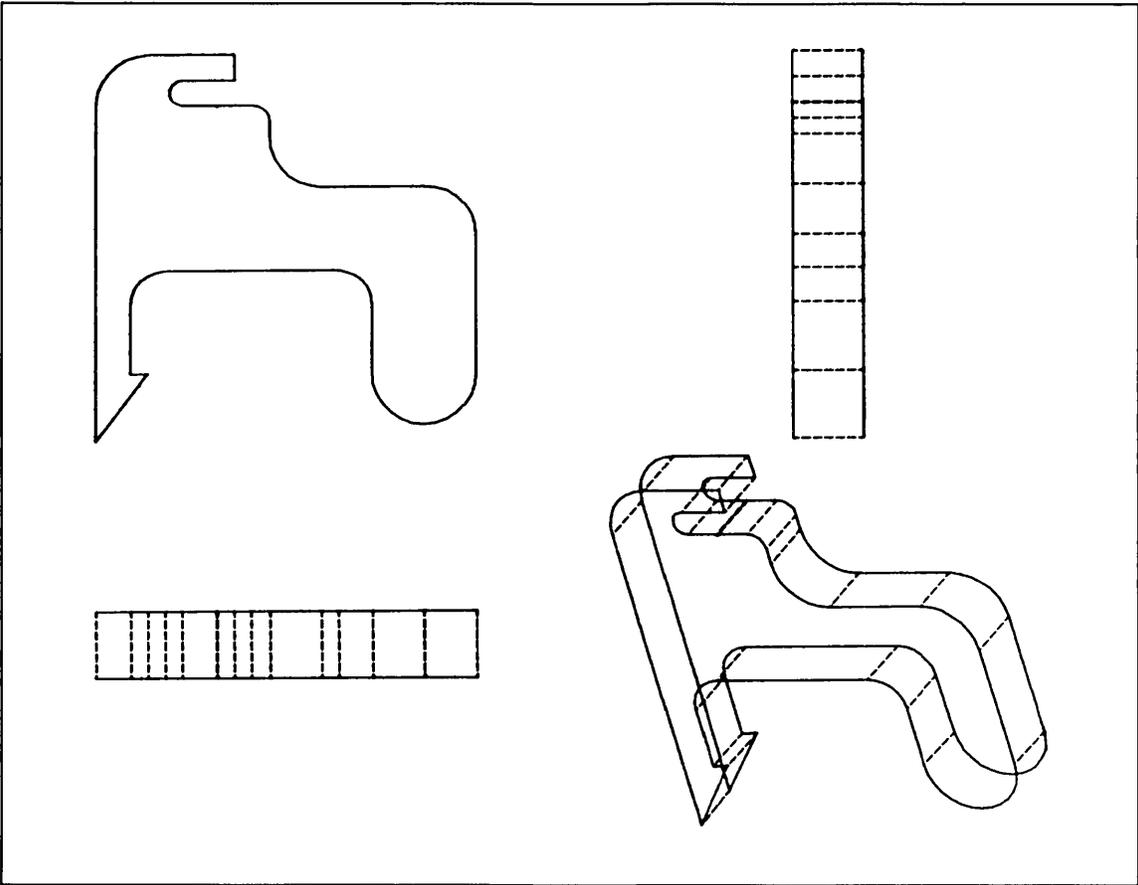


Figure 7-4. Four Views of Latch

## Displaying a Single View

In this part of the session, you will learn how to bring up a standard view and then, using that view, define a new auxiliary view.

### Keyboard Selection

Select 3.DISPLAY A SINGLE VIEW from the Display Control menu.

### Tablet Selection

| Display Views |              |          |           |                 |
|---------------|--------------|----------|-----------|-----------------|
| Top-6         | Front-1      | ISO1-8   | One       | Last Layout     |
| Front-1       | Right-3      |          |           |                 |
| Two Views     |              | Std Four | Std Eight | Last Multi View |
| Above         | Side by Side |          |           |                 |
| Below         |              |          |           |                 |

Enter the number of the view you wish to display. For this example, use view 1:

VIEW NUMBER = 1

Attention indicators appear around view 1.

Just as before, you are again asked to select the zoom scale you wish to use for the display. Select 5.AUTOMATICALLY MAXIMIZED as the zoom scale.

## Defining an Auxiliary View

Create a new view by rotating view 1 about the horizontal, vertical, and normal axes as follows:

### Keyboard Selection

Select 9.DEFINE AUXILIARY VIEW from the Display Control menu.

### Tablet Selection

| Display Control   |                 |             |
|-------------------|-----------------|-------------|
| Change Work Space | Chg Wk View     |             |
| Define Work Space | Define Aux View | Align Views |

To rotate the top of the latch about the horizontal axis, select 3.HORIZONTAL AXIS, TOP OUT from the Auxiliary View menu and enter 30° as the angle of rotation:

ROTATION =45.0000 30

To rotate the right side of the latch about the vertical axis, select 5.VERTICAL AXIS, RIGHT OUT from the Auxiliary View menu and enter 15° as the angle of rotation:

ROTATION =30.0000 15

To display this auxiliary view, enter ] to return to the ZOOM SCALE prompt and 5.AUTOMATICALLY MAXIMIZED as the zoom scale. Figure 7-5 shows the display.

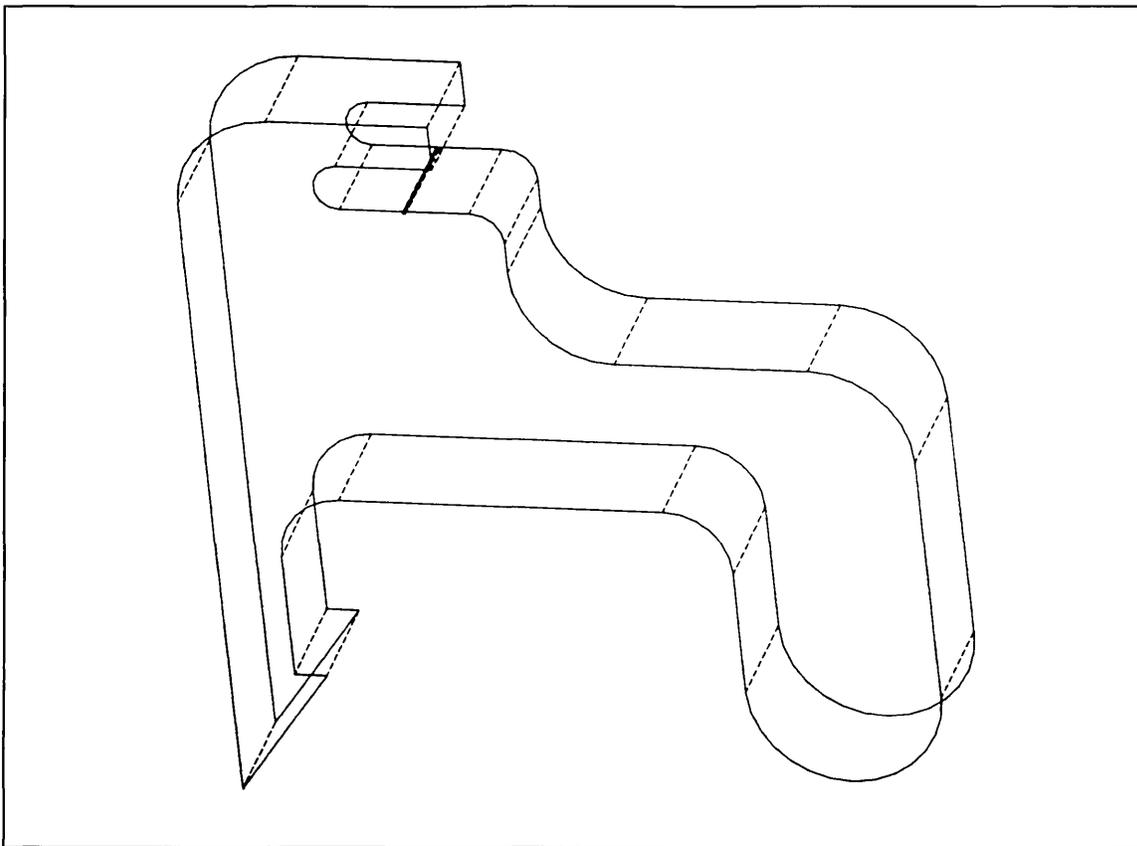


Figure 7-5. Defining an Auxiliary View

DEFINING AN AUXILIARY VIEW

Now you are asked if you want to continue defining your auxiliary view:

CONTINUE DEFINITION OF VIEW 09?

Enter Y in response to this prompt.

To rotate the latch in a clockwise direction about the normal axis, select 1.NORMAL AXIS, CW from the Auxiliary View menu and enter 45° as the angle of rotation:

ROTATION =15.0000 45

Enter ] to return to the Zoom Scale prompt and 5.AUTOMATICALLY MAXIMIZED as the zoom scale. Figure 7-6 shows the display.

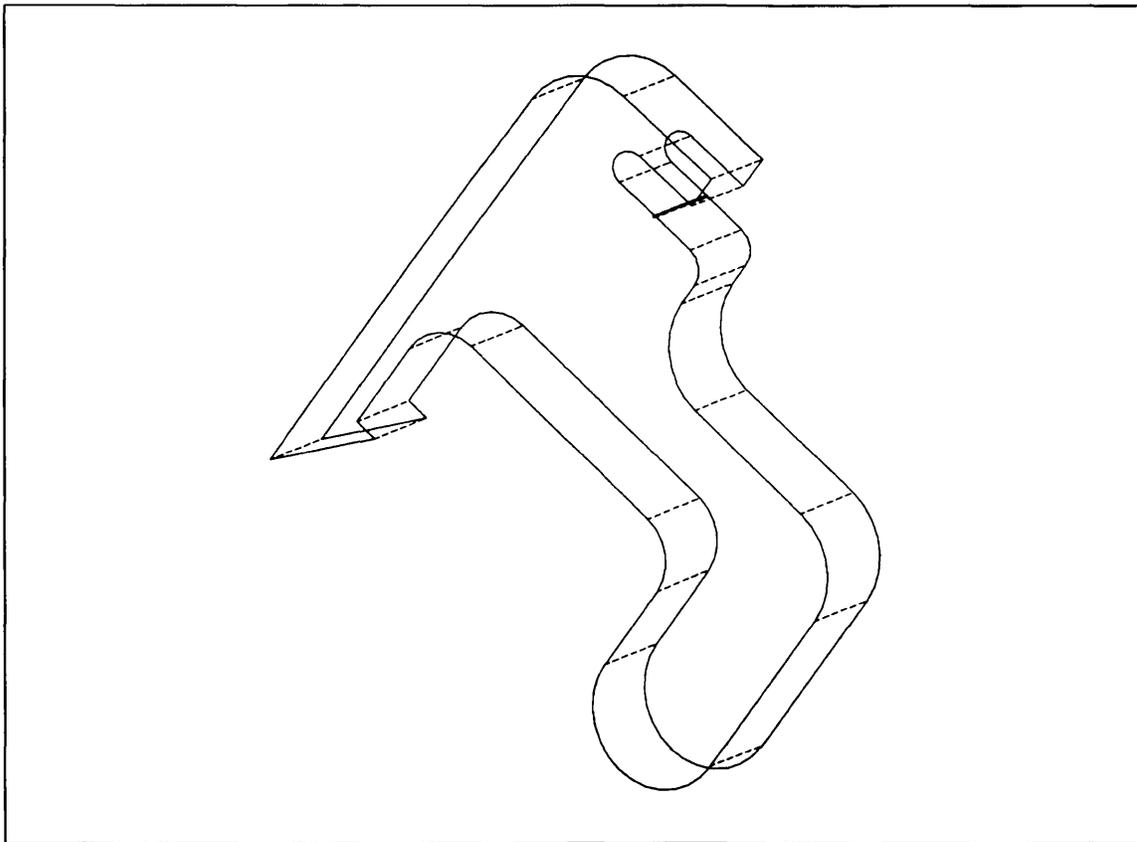


Figure 7-6. View 9

You are again asked if you want to continue defining your auxiliary view. This time enter N in response to the prompt. By entering N, you are telling the system that you have finished defining the view and wish to save it as a view.

## Displaying the Latch in Two Views

Display two views of the latch simultaneously as follows:

### Keyboard Selection

Select 4.DISPLAY MULTIPLE VIEWS from the Display Control menu.

Select 4.TWO VIEWS - TOP AND BOTTOM to display two views of the cam.

### Tablet Selection

| Display Views |              |          |           |                 |
|---------------|--------------|----------|-----------|-----------------|
| Top-6         | Front-1      | ISO1-8   | One       | Last Layout     |
| Front-1       | Right-3      |          |           |                 |
| Two Views     |              | Std Four | Std Eight | Last Multi View |
| Above Below   | Side by Side |          |           |                 |

To display the standard auxiliary view in the top half of the screen, enter 4 as the top view:

SELECT TOP VIEW  
VIEW NUMBER = 4

To display the auxiliary view you just created in the bottom half of the screen, enter 9 as the bottom view:

SELECT BOTTOM VIEW  
VIEW NUMBER = 9

DISPLAYING THE LATCH IN TWO VIEWS

Select 5.AUTOMATICALLY MAXIMIZED as the zoom scale. The display fills the entire screen. Figure 7-7 shows the result.

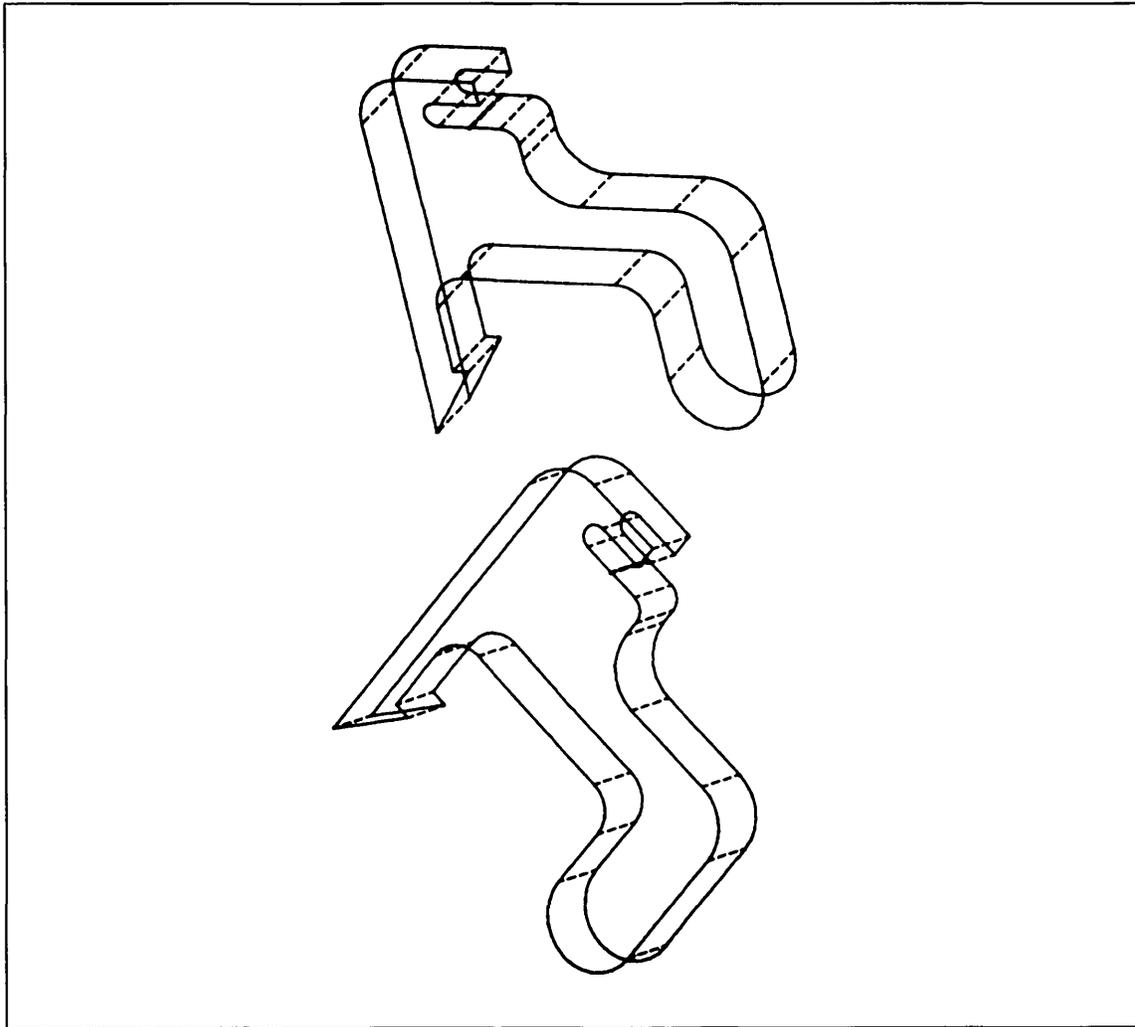


Figure 7-7. Two Views of Latch

## Zooming In on the Keyway

In this part of the session, you will learn how to zoom the keyway area of your latch to show more detail.

First, change the zoom mode to affect the current working view only:

### Keyboard Selection

Enter Z to go to the Zoom menu.

Select 17.CHANGE ZOOM MODE from the Zoom menu.

Select 2.WORK VIEW as the zoom mode.

### Tablet Selection

| Change Zoom Mode |           |               |             |
|------------------|-----------|---------------|-------------|
| Entire Layout    | Work View | Select Window | All Windows |

Next, change the work view to view 4 as follows:

### Keyboard Selection

Select 5.CHANGE WORK VIEW from the Display Control menu.

### Tablet Selection

| Display Control   |                 |             |
|-------------------|-----------------|-------------|
| Change Work Space | Work View       | Chg Wk View |
| Define Work Space | Define Aux View | Align Views |

Enter view 4 as the new work view:

```
WORK VIEW IS 9 SELECT ANOTHER
VIEW NUMBER = 4
```

Zoom the keyway area of the latch by indicating the opposite corners (diagonal screen positions) of that region as follows:

### Keyboard Selection

Select 6.ZOOM from the Display Control menu.

Select 9.DIAGONAL SCREEN POSITIONS from the Zoom menu.

### Tablet Selection

| Zoom         |               |                  |             |
|--------------|---------------|------------------|-------------|
| Auto Max-Min | Diag Position | 1/2 X            | 2X          |
| Save as Base | New Center    | New Center Scale | Enter Scale |

- Position the graphics cursor 0.5 inch above and to the left of the keyway in the top view and select it using the C key or the tablet pen.
- Position the graphics cursor 0.5 inch below and to the right of the keyway in the top view and select it using the C key or the tablet pen. An enlarged view of the selected area then displays in the upper half of the screen (figure 7-8). The bottom view remains unchanged.

ZOOMING IN ON THE KEYWAY

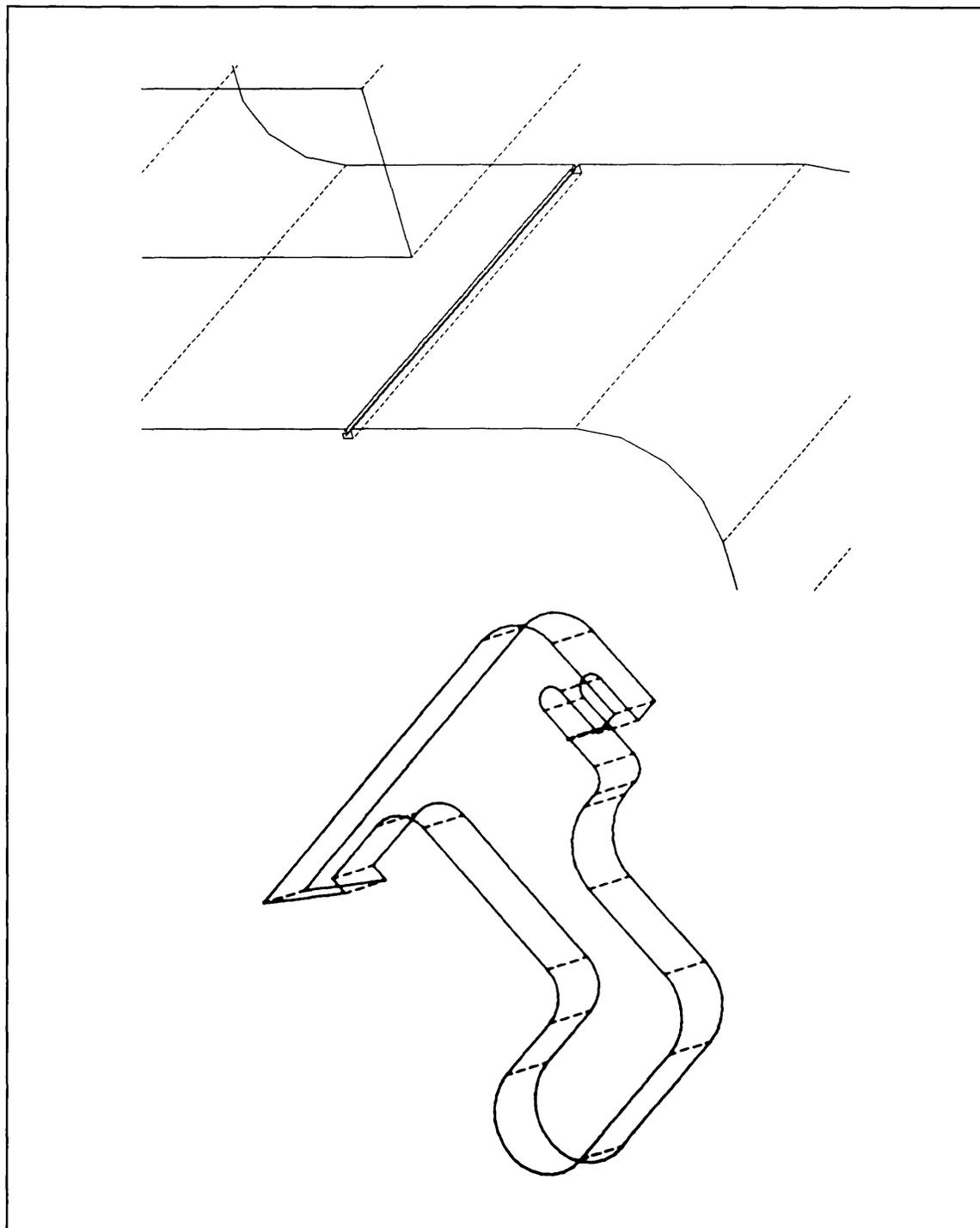


Figure 7-8. Zoomed View of the Keyway

Return to the scale at which the latch was originally constructed as follows:

**Keyboard Selection**

Select 6.ZOOM from the Display Control menu.

Select 1.RETURN TO BASE SCALE from the Zoom menu.

**Tablet Selection**

| <b>Zoom</b>    |               |                  |                  |
|----------------|---------------|------------------|------------------|
| Auto Max-Min   | Diag Position | 1/2 X            | 2X               |
| Save as Base   | New Center    | New Center Scale | Enter Scale      |
| Return to Base | Retrieve Zoom | Name Zoom        | Zoom from Hdcopy |

The original part as shown in figure 7-7 redisplay.

If you want to zoom another view, you must first change that view to the working view. This is because the zoom mode has been limited to the work view only.

Change the work view to the bottom half of the screen as follows:

**Keyboard Selection**

Select 5.CHANGE WORK VIEW from the Display Control menu.

**Tablet Selection**

| <b>Display Control</b> |                 |             |
|------------------------|-----------------|-------------|
| Change Work Space      |                 | Chg Wk View |
| Define Work Space      | Define Aux View | Align Views |

Enter 9 as the new work view:

WORK VIEW IS 4 SELECT ANOTHER  
VIEW NUMBER = 9

Move the keyway area to the center of the screen as follows:

**Keyboard Selection**

Select 6.ZOOM from the Display Control menu.

Select 4.NEW CENTER, ENTER SCALE from the Zoom menu.

**Tablet Selection**

| <b>Zoom</b>  |               |                  |             |
|--------------|---------------|------------------|-------------|
| Auto Max-Min | Diag Position | 1/2 X            | 2X          |
| Save as Base | New Center    | New Center Scale | Enter Scale |

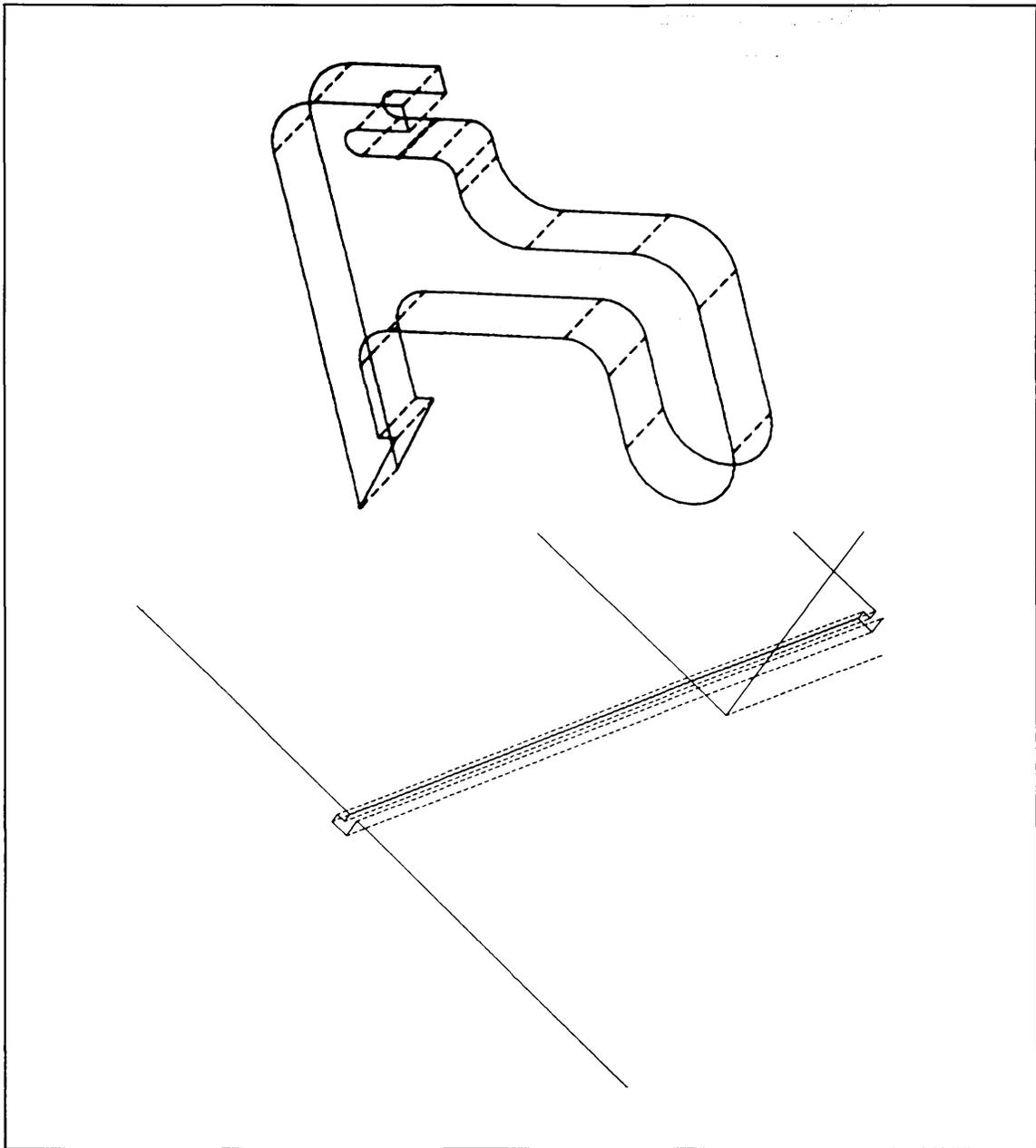
- Position the graphics cursor on the keyway in the bottom view and select it as the new center using the C key or the tablet pen.

Enter a new scale factor of 20 and accept the default ratio of 1:

1.SCALE =1.3591 20 ]  
2.RATIO =1.0000

ZOOMING IN ON THE KEYWAY

The zoomed keyway is displayed in the center of the bottom half of the screen (figure 7-9). The top view remains unchanged.



**Figure 7-9. Another Zoomed View of the Keyway**

## Filing and Exiting

Figure 7-10 outlines the steps to end this terminal session and save your part.

| Terminal Session                                                                                                                                                                                                                                                                     | Comments                                                                                                                                                                                                                                 |
|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ICEMDDN L=0 V=1 D=0.00 P=0 C=1 4                                                                                                                                                                                                                                                     | If you are using the keyboard, enter 4 to select the File Current Part/Exit ICEM DDN menu.                                                                                                                                               |
| <pre> FILE CURRENT PART/EXIT ICEM DDN 1.FILE - CONTINUE CURRENT PART 2.   - GET DIFFERENT PART 3.   - QUIT SESSION 4.   - SUSPEND SESSION 5.DO NOT FILE - CONTINUE FROM LAST FILE 6.   - GET DIFFERENT PART 7.   - QUIT SESSION 8.   - SUSPEND SESSION  3                     </pre> | <p>If you are using the tablet, you can file your part and exit from ICEM DDN in a single step by selecting the FILE/QUIT square with your tablet pen.</p> <p>Select 3.FILE - QUIT SESSION to save your part and exit from ICEM DDN.</p> |
| ICEMDDN TERMINATED.                                                                                                                                                                                                                                                                  |                                                                                                                                                                                                                                          |
| /                                                                                                                                                                                                                                                                                    |                                                                                                                                                                                                                                          |

Figure 7-10. Sample File/Exit Sequence



|                                       |      |
|---------------------------------------|------|
| Getting Started .....                 | 8-2  |
| Creating the Pattern Geometry .....   | 8-3  |
| Creating a Pentagon .....             | 8-3  |
| Creating a Hexagon .....              | 8-4  |
| Creating a Right Triangle .....       | 8-5  |
| Creating Patterns .....               | 8-7  |
| Listing Your Patterns .....           | 8-9  |
| Deleting the Geometry .....           | 8-10 |
| Retrieving Patterns .....             | 8-11 |
| Retrieving the Pentagon Pattern ..... | 8-11 |
| Retrieving the Triangle Pattern ..... | 8-14 |
| Retrieving the Hexagon Pattern .....  | 8-15 |
| Filing and Exiting .....              | 8-18 |

11

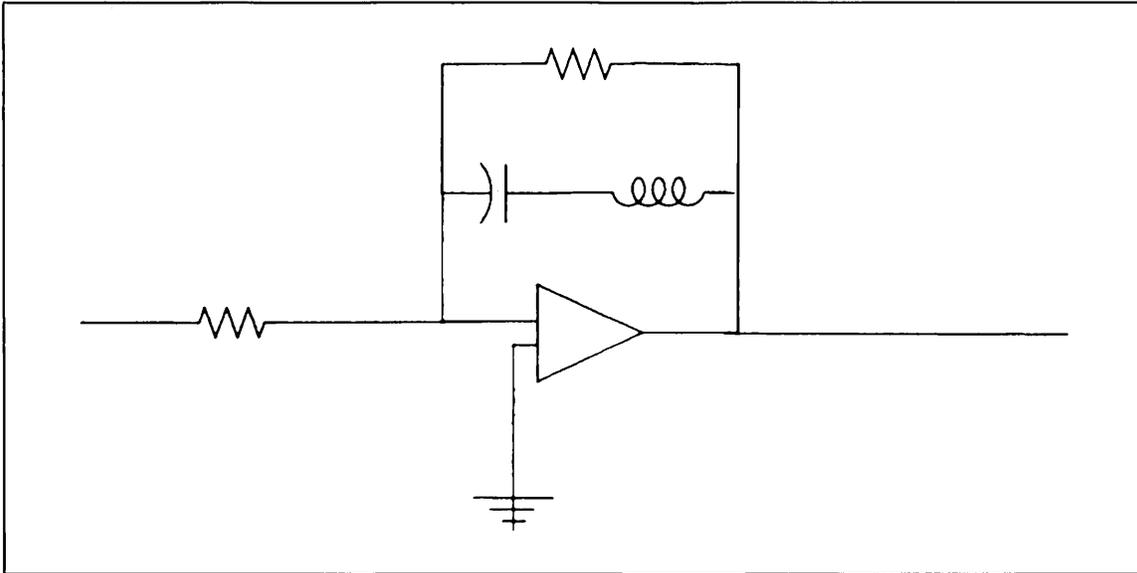
12

13

14

15

Patterns are useful for duplicating a master part many times. For example, many users find patterns helpful for drawing such things as gears or electrical circuits. To help illustrate the use of patterns, the electrical circuit shown in figure 8-1 was drawn using patterns to define each of the components.



**Figure 8-1. Patterns for Electrical Circuit Components**

In this practice session, you will learn how to create, retrieve, and delete patterns. For simplicity, a pentagon, hexagon, and triangle are used to create the sample patterns.

## Getting Started

1. Log in using the procedures described in the introduction to this manual.
2. Create a new part. The sample terminal session in figure 8-2 helps to illustrate this procedure.

| Terminal Session                                                                           | Comments                                                                                                                        |
|--------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------|
| <pre>ENTER PART NAME PATTERNS</pre>                                                        | <p>Enter the name and sheet number of your new part. In this example, we called the part PATTERNS with a sheet number of 1.</p> |
| <pre>SHEET NUMBER = 1</pre>                                                                |                                                                                                                                 |
| <pre>NEW PART ASSUMED</pre>                                                                |                                                                                                                                 |
| <pre>--- UNITS OF MEASURE  M 1.METRIC (MM) 2.ENGLISH (INCH) 3.ENGLISH (FOOT/INCH)  2</pre> | <p>First, enter M to turn the menu display on. Next, select menu choice 2 to work in English units.</p>                         |
| <pre>--- DRAFTING STANDARD 1.ANSI 1982 2.ANSI 1973 3. 4. 5. 6. 7. 8. 9.      1</pre>       | <p>Select menu choice 1 to use 1982 ANSI standard dimensions.</p>                                                               |
| <pre>ICEMDDN L=0 V=1 D=0.00 P=0 C=0</pre>                                                  |                                                                                                                                 |

Figure 8-2. Start Up Procedure for Creating Patterns

## Creating the Pattern Geometry

Before you can create a pattern, the entities that are to make up that pattern must be displayed on your screen. In the next three sections, you are guided through the procedures necessary to create a regular pentagon, hexagon, and right triangle. These geometric shapes are used later when you learn how to create patterns.

### Creating a Pentagon

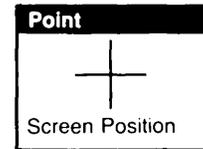
The first step toward creating a regular pentagon is to define the center point.

#### Keyboard Selection

Select 9.POINT from the main menu.

Select 1.SCREEN POSITION from the Point menu.

#### Tablet Selection



- Position the graphics cursor near the center of the upper right portion of the screen and select that position using the C key or the tablet pen. This point is labeled as point A in figure 8-3.

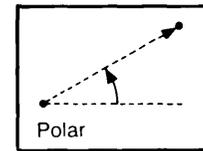
Enter ] to return to the Point menu.

Using the point you just created, define the endpoints of the pentagon sides as follows:

#### Keyboard Selection

Select 3.POLAR from the Point menu.

#### Tablet Selection



- Position the graphics cursor on the center point you just defined and select it as the base point using the P key or the tablet pen.

Define five points with a radius of 2 and angles of 0°, 72°, 144°, 216°, and 288°. These points will serve as the endpoints of the pentagon.

1.ANGLE = 0  
2.RADIUS = 2 ]

1.ANGLE = 72  
2.RADIUS = 2 ]

1.ANGLE = 144  
2.RADIUS = 2 ]

1.ANGLE = 216  
2.RADIUS = 2 ]

1.ANGLE = 288  
2.RADIUS = 2 ]

Enter ] to tell ICEM DDN that you have finished defining points.

Enter ] again to return to the Point menu.

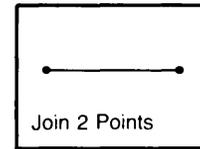
Connect the points you just created to form the sides of the pentagon as follows:

**Keyboard Selection**

**Tablet Selection**

**(CTRL)** L + **(RETURN)**

Select 3.JOIN 2 POINTS from the Line menu.



- Position the graphics cursor on one of the endpoints and select it using the P key or the tablet pen.
- Continue selecting points in this way until you have completed the pentagon.

Enter ] to return to the Line menu.

**Creating a Hexagon**

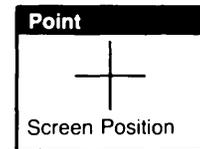
The next geometric shape that you will create is a regular hexagon. Just as with the pentagon, you first need to define a center point:

**Keyboard Selection**

**Tablet Selection**

**(CTRL)** P + **(RETURN)**

Select 1.SCREEN POSITION from the Point menu.



- Position the graphics cursor near the center of the upper left portion of the screen and select that position using the C key or the tablet pen. This point is labeled as point B in figure 8-3.

Enter ] to return to the Point menu.

Enter F to return to the main menu.

Using the center point you just created and an inscribed radius value of 2, create the hexagon as follows:

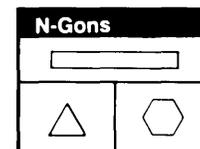
**Keyboard Selection**

**Tablet Selection**

Select 12.OTHER CURVES from the main menu.

Select 7.N-GON from the Other Curves menu.

Select 3.HEXAGON from the N-gon menu.



Select 1.INScribed RADIUS from the Center Plus menu.

Select 3.EXISTING POINT as the center position.

- Position the graphics cursor on the point you just created and select it as the center point of the hexagon using the P key or the tablet pen.

Enter 2 as the inscribed radius value:

IN RADIUS = 2

To make the flat sides of the hexagon horizontal (parallel with the xt-axis), select 1.XT-AXIS in response to the side of flat prompt.

You are then asked if you want to define your hexagon as individual lines in the data base:

DEFINE AS LINES?

Enter Y in response to this prompt.

Enter ] to return to the N-gon menu.

## Creating a Right Triangle

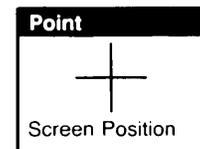
Create the base point of the triangle as follows:

### Keyboard Selection

(CTRL) P + (RETURN)

Select 1.SCREEN POSITION from the Point menu.

### Tablet Selection



- Position the graphics cursor near the bottom of the screen, approximately in the middle, and select that position using the C key or the tablet pen. This point is labeled as point C in figure 8-3.

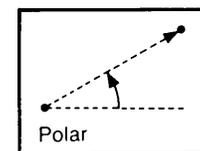
Enter ] to return to the Point menu.

Using the point you just created as the base point, define the other triangle endpoints as follows:

### Keyboard Selection

Select 3.POLAR from the Point menu.

### Tablet Selection



- Position the graphics cursor on the center point you just defined and select it as the base point using the P key or the tablet pen.

Place the first point at an angle of  $10^\circ$  from the base point and a distance of 3 inches:

1.ANGLE = 10  
2.RADIUS = 3 ]

Place the second point at an angle of  $100^\circ$  from the base point and a distance of 4 inches:

1.ANGLE = 100  
2.RADIUS = 4 ]

Enter ] to return to the INDICATE POINT prompt.

Enter ] to return to Point menu.

Enter F to return to the main menu.

Construct the triangle automatically using the N-gon function:

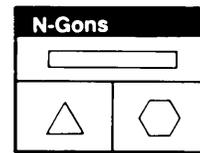
**Keyboard Selection**

Select 12.OTHER CURVES from the main menu.

Select 7.N-GON from the Other Curves menu.

Select 1.TRIANGLE from the N-gon menu.

**Tablet Selection**



Select 3.EXISTING POINTS as the base position.

- Position the graphics cursor on each of the points that make up the triangle and select them using the P key or the tablet pen.

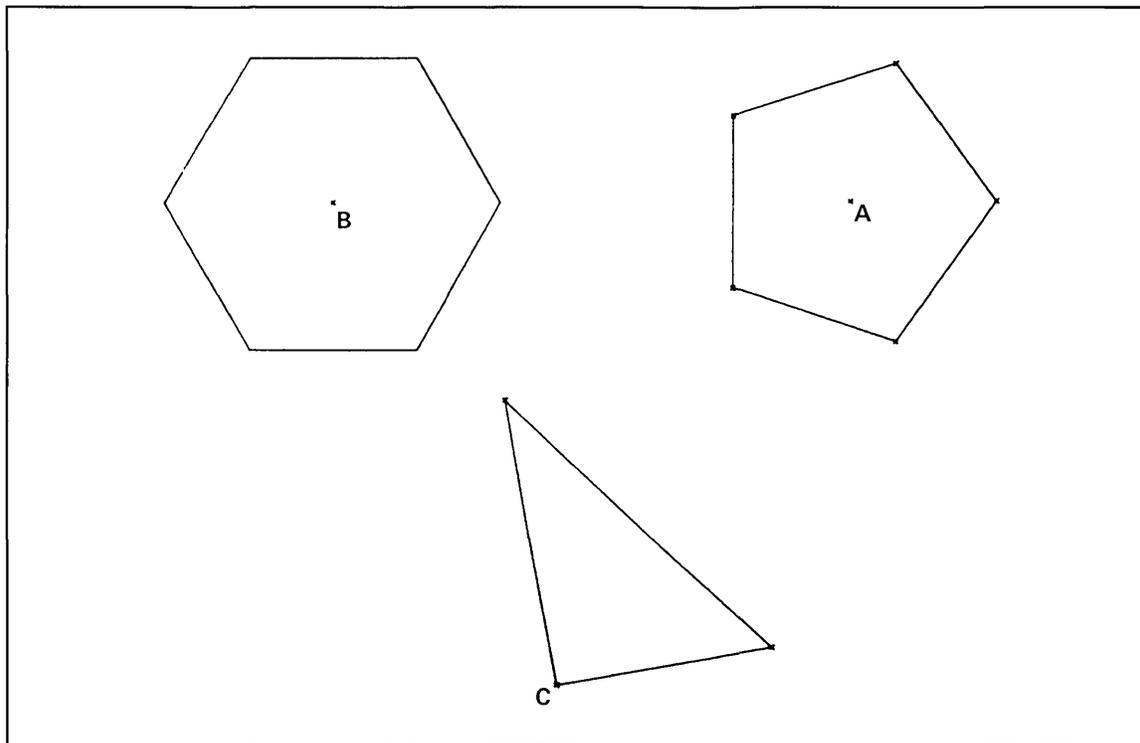
As soon as you have selected the third point, you are asked if you want to define the triangle as individual lines:

DEFINE AS LINES?

Enter Y in response to this prompt.

Enter F to return to the main menu.

The display on your terminal should now look similar to that shown in figure 8-3.



**Figure 8-3. Pentagon, Hexagon, and Triangle Geometry**

## Creating Patterns

Now that all of the geometry has been created, the next step is to create patterns of the geometric shapes. When you create a pattern, you are actually storing a master copy of the geometry in the primary pattern library. Then, each time you retrieve that pattern, a copy of the stored geometry is generated for you.

In this section, you will learn how to retrieve multiple copies of a pattern, change the scale factor of the pattern, and change the angle at which the pattern is rotated on the screen.

### Keyboard Selection

Select 6.DATA BASE MANAGEMENT from the main menu.

Select 2.PATTERN MANAGEMENT from the Data Base Management menu.

Select 2.CREATE from the Pattern Management menu.

You are first asked for the name of your pattern. For this example, call your pentagon pattern REGULAR PENTAGON:

```
CREATE
ENTER PATTERN NAME REGULAR PENTAGON
```

Next, you are prompted to select the entities that are to be included in the pattern.

Enter **(CTRL)** E to display the Entity Selection menu.

Select 3.REGION IN from the Entity Selection menu.

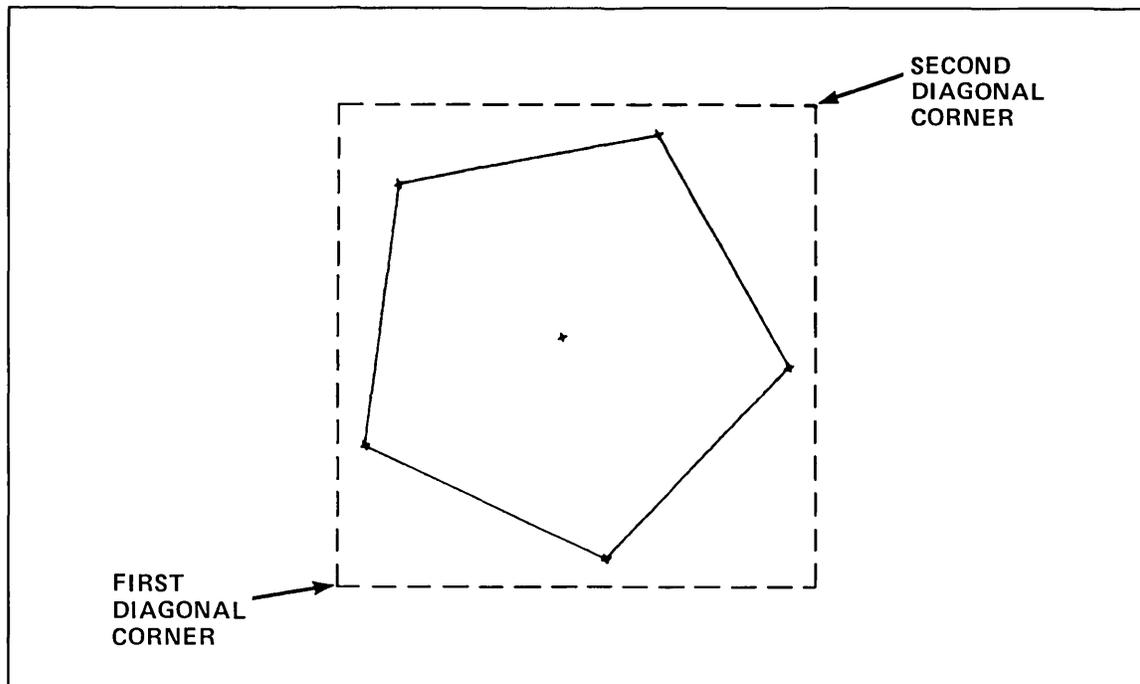
- Position the graphics cursor below and to the left of the pentagon and select that position using the C key or tablet pen (figure 8-4).
- Position the graphics cursor above and to the right of the pentagon to define the opposite diagonal corner of the region (figure 8-4).

### NOTE

In doing this, make sure that no entities cross the imaginary horizontal and vertical lines that define the region.

### Tablet Selection

| Pattern Management |             |          |
|--------------------|-------------|----------|
| Modals             |             | Create   |
| List Primary       | List Second | Retrieve |



**Figure 8-4. Region Selection for Pattern Creation**

Attention indicators should appear on each side of the pentagon as well as on all of the points. If they don't, enter [ and repeat the steps outlined above. If the Selection Modification modal is on, select 17.CANCEL LAST MODIFICATION from the Modify Selection menu and try selecting the entities again.

If the attention indicators are correct, enter ] to accept the selected entities.

After selecting entities, you are prompted for the pattern's origin:

- ORIGIN METHOD
- 1.SCREEN POSITION
  2. ENTER COORDINATES
  - 3.EXISTING POINT
  - 4.LOWER LEFT

Because the pentagon has a center point, it is convenient to use that point as the pattern origin. To do this, select 3.EXISTING POINT as the origin mode. Later, when you retrieve this pattern, the pattern origin (in this case, the center point of the pattern) will be used to place the pattern on the screen at the position you indicate.

- Position the graphics cursor on the center point of the pentagon and select it as the origin using the P key or the tablet pen.

As soon as your pattern has been created, the following message is displayed:

PATTERN CREATED

In the same way that you just created this pattern, create the hexagon and right triangle patterns. Name the hexagon pattern REGULAR HEXAGON and the triangle pattern RIGHT TRIANGLE.

#### NOTE

The hexagon, like the pentagon, has a center point which conveniently may be used as the origin of the pattern. The triangle, on the other hand, does not have a center point. In this case, the first point that you defined (the base point) would be a good choice for the origin of the pattern.

## Listing Your Patterns

List the patterns stored in your primary pattern file as follows:

### Keyboard Selection

Select 5.LIST from the Pattern Management menu.

Select 1.PRIMARY LIBRARY from the List Patterns menu.

### Tablet Selection

| Pattern Management |             |          |
|--------------------|-------------|----------|
| Modals             |             | Create   |
| List Primary       | List Second | Retrieve |

The list that you see on your terminal should look like the list shown in figure 8-5.

| PRIMARY LIBRARY LIST         |                 |                 |                 |                       |       |              |
|------------------------------|-----------------|-----------------|-----------------|-----------------------|-------|--------------|
| LIBRARY NAME = PATTERN       |                 | 8/14/85         | 13:41           | RELEASE REVISION 1.60 |       |              |
| TOTAL NUMBER OF PATTERNS = 3 |                 |                 |                 |                       |       |              |
| PATTERN NAME                 | DATE<br>CREATED | TIME<br>CREATED | ENTITY<br>COUNT | SIZE<br>(SECT.)       | UNITS | DRAFT<br>STD |
| REGULAR PENTAGON             | 8/14/85         | 13:38           | 11              | 2                     | ENG   | A 82         |
| RIGHT TRIANGLE               | 8/14/85         | 13:39           | 6               | 1                     | ENG   | A 82         |
| REGULAR HEXAGON              | 8/14/85         | 13:41           | 7               | 1                     | ENG   | A 82         |

Figure 8-5. Local Pattern List

Enter R to repaint your screen.

Enter F to return to the main menu.

## Deleting the Geometry

Because the pentagon, hexagon, and triangle patterns are now stored in your primary pattern file, you no longer need the geometry currently displayed on your screen.

Delete the geometry as follows:

### Keyboard Selection

Select 3.DELETE from the main menu.

Select 1.SELECT from the Delete menu.

### Tablet Selection

| Delete    |            |            |       |
|-----------|------------|------------|-------|
| Select    |            | Screen     | Chain |
| Region In | Region Out | All Points | Level |

Enter **(CTRL)** E to display the Entity Selection menu.

Select 4.ALL DISPLAYED from the Entity Selection menu.

As soon as you enter ], all of the entities on the screen are deleted.

Enter F to return to the main menu.

## Retrieving Patterns

Retrieving patterns allows you to copy the pattern anywhere on the same plane that the pattern was originally defined or in a parallel plane. For example, a pattern defined in the front view may be copied at any depth in or out of the screen. You cannot, however, copy it to a location on the side view or any other view. If you try to do this, the copy is drawn with respect to the plane in which the pattern was originally defined.

### Retrieving the Pentagon Pattern

Retrieve the pentagon pattern you created earlier as follows:

#### Keyboard Selection

Select 6.DATA BASE MANAGEMENT from the main menu.

Select 2.PATTERN MANAGEMENT from the Data Base Management menu.

Select 3.RETRIEVE from the Pattern Management menu.

#### Tablet Selection

| Pattern Management |             |          |
|--------------------|-------------|----------|
| Modals             |             | Create   |
| List Primary       | List Second | Retrieve |

Enter REGULAR PENTAGON in response to the prompt:

```
RETRIEVE
ENTER PATTERN NAME REGULAR PENTAGON
```

After you enter a pattern name, ICEM DDN begins looking for the pattern in the pattern libraries, beginning first with the primary library and then proceeding to the secondary library. Since your patterns are in the primary library, the following message is displayed followed immediately by the Modify Modals menu:

```
PATTERN FOUND ON PRIMARY LIBRARY
PATTERN

MODIFY MODALS
1.ORIGIN METHOD EXISTING POINT
2.SCALE FACTOR 1.00000
3.ROTATION ANGLE 0.00000
4.ROTATION AXIS Z/ZT-AXIS
5.ORIENTATION WORK SPACE
6.
7.ENTITY GROUPING OFF
8.LEVEL ORIGINAL
9.PEN ORIGINAL
10.COLOR ORIGINAL
11.ENTITY NAME DROPPED
```

Note that the current settings for each of these modals are also displayed. Each time you retrieve a pattern, it is important that you check these settings to make sure they are correct. For example, since you deleted all of your geometry, there is no existing point on the screen to use as the pattern origin location. Therefore, you need to change the pattern origin method from Existing Point to Screen Position.

Select 1.ORIGIN METHOD from the Modify Modals menu to tell ICEM DDN that you want to change this modal setting.

## RETRIEVING PATTERNS

The Origin Method menu is displayed:

```
ORIGIN METHOD
1.SCREEN POSITION
2. ENTER COORDINATES
3.EXISTING POINT
4.DELTA FROM CURVE END
5.NORMAL TO CURVE
```

Enter 1.SCREEN POSITION as the new origin method.

As soon as you change the origin method, the following message is displayed:

```
ORIGIN METHOD IS SCREEN POSITION
```

Next, the Modify Modals menu is redisplayed:

```
MODIFY MODALS
1.ORIGIN METHOD SCREEN POSITION
2.SCALE FACTOR 1.00000
3.ROTATION ANGLE 0.00000
4.ROTATION AXIS Z/ZT-AXIS
5.ORIENTATION WORK SPACE
6.
7.ENTITY GROUPING OFF
8.LEVEL ORIGINAL
9.PEN ORIGINAL
10.COLOR ORIGINAL
11.ENTITY NAME DROPPED
```

Note that the origin method has been changed to Screen Position. Each time you retrieve a pattern from now on, ICEM DDN will be expecting you to retrieve it using a screen position instead of an existing point.

Enter ] in response to the Modify Modals menu. This tells ICEM DDN that you are accepting the current modal settings.

The next prompt asks you to specify the pattern origin:

```
INDICATE ORIGIN POSITION
```

- Position the graphics cursor at the place on the screen that you would like the pattern to be placed and select that position using the C key or the tablet pen.

A pentagon appears on your screen with the origin at the screen position you selected.

Enter ] to return to the ENTER PATTERN NAME prompt.

You can change the other modal settings in the Modify Modals menu in the same way that you changed the origin method. Just enter the number that corresponds to the modal you wish to change and then enter the new setting.

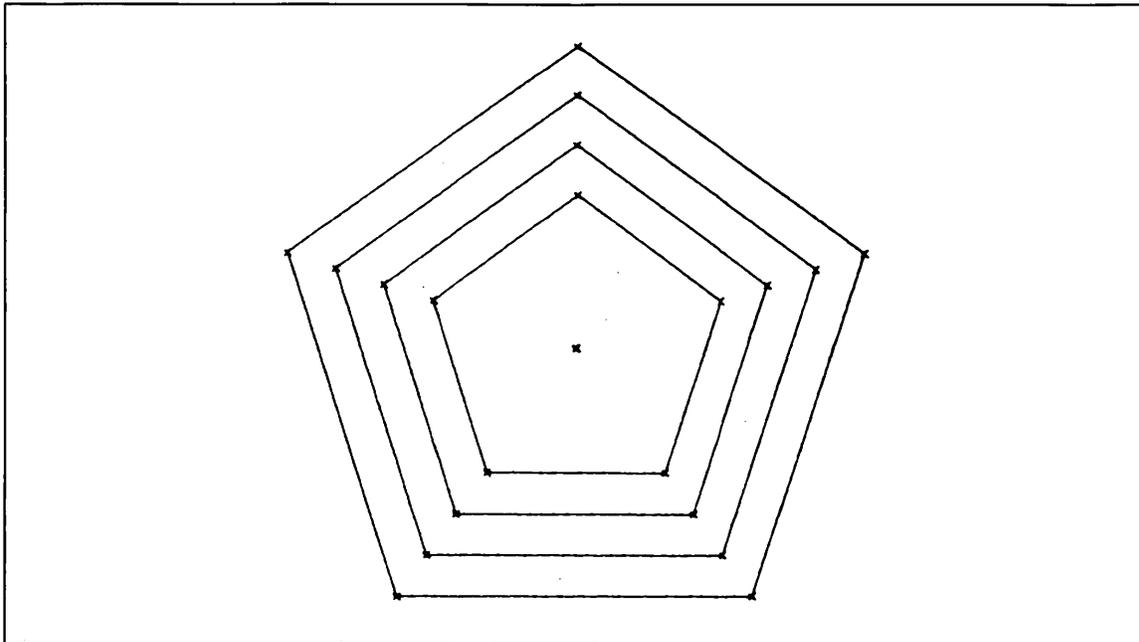
To practice retrieving the pentagon pattern using different modal settings, create four concentric pentagons (using the same center point each time), but change 2.SCALE FACTOR and 3.ROTATION ANGLE in the Modify Modals menu as follows:

**NOTE**

After you create the first pentagon, change 1.ORIGIN METHOD back to Existing Point to ensure that all four of the pentagons are concentric.

| Pentagon | Scale Factor | Rotation Angle |
|----------|--------------|----------------|
| 1        | 0.75         | 15             |
| 2        | 1.0          | 15             |
| 3        | 1.25         | 15             |
| 4        | 1.5          | 15             |

Figure 8-6 shows the result.



**Figure 8-6. Concentric Pentagons**

## Retrieving the Triangle Pattern

Retrieve the triangle pattern, called RIGHT TRIANGLE, in the same way that you just retrieved the pentagon pattern but change 1.ORIGIN METHOD in the Modify modals menu back to Screen Position.

Also, change 2.SCALE FACTOR and 3.ROTATION ANGLE as follows for each of the four triangles:

| Triangle | Scale Factor | Rotation Angle |
|----------|--------------|----------------|
| 1        | 1.00         | 0              |
| 2        | 0.75         | 180            |
| 3        | 0.25         | 90             |
| 4        | 0.50         | 270            |

Figure 8-7 shows the results.

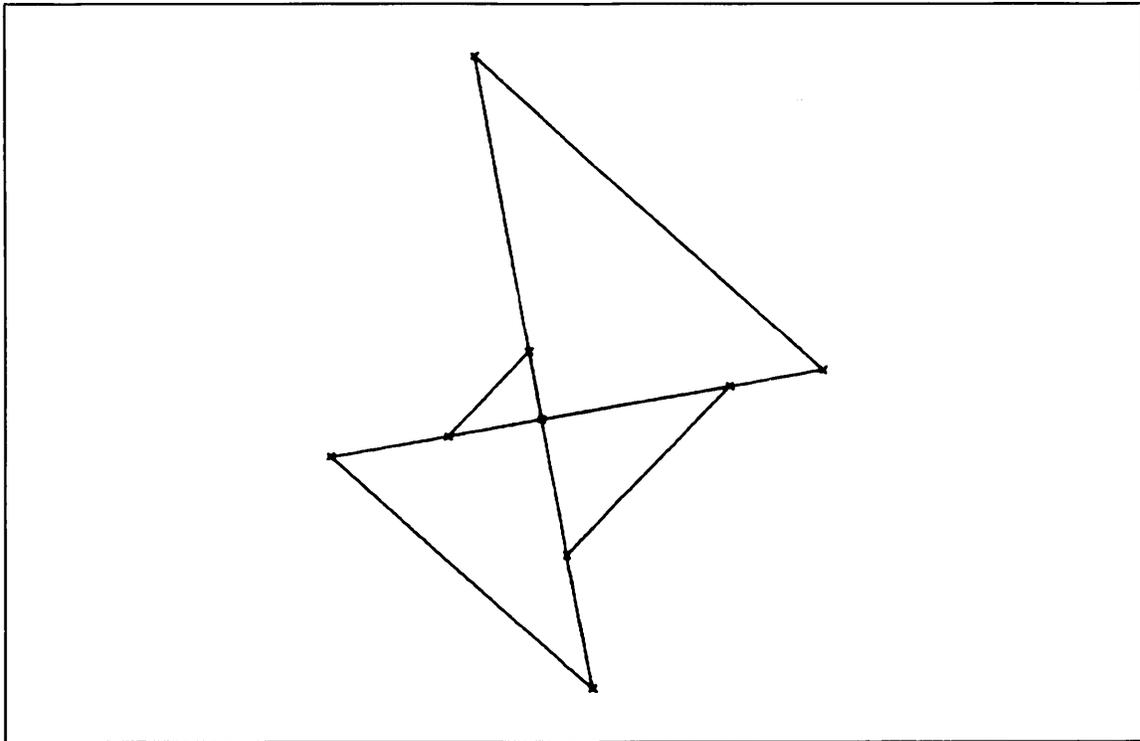


Figure 8-7. Triangle Patterns

## Retrieving the Hexagon Pattern

Retrieve the hexagon pattern, called REGULAR HEXAGON, in the same way that you just retrieved the pentagon and triangle patterns except this time, define the pattern as a *group* of entities by setting the Entity Grouping modal to the on position.

```

MODIFY MODALS
1.ORIGIN METHOD SCREEN POSITION
2.SCALE FACTOR 0.50000
3.ROTATION ANGLE 270
4.ROTATION AXIS Z/ZT-AXIS
5.ORIENTATION WORK SPACE
6.
7.ENTITY GROUPING ON
8.LEVEL ORIGINAL
9.PEN ORIGINAL
10.COLOR ORIGINAL
11.ENTITY NAME DROPPED

```

This defines the entire pattern as a *single entity* in the data base.

Before you retrieve this pattern, also change the scale factor and rotation angle to 2 and 45, respectively.

The Modify Modals menu should now match that shown below:

```

MODIFY MODALS
1.ORIGIN METHOD SCREEN POSITION
2.SCALE FACTOR 2.00000
3.ROTATION ANGLE 45
4.ROTATION AXIS Z/ZT-AXIS
5.ORIENTATION WORK SPACE
6.
7.ENTITY GROUPING ON
8.LEVEL ORIGINAL
9.PEN ORIGINAL
10.COLOR ORIGINAL
11.ENTITY NAME DROPPED

```

Enter ] to accept the current modal settings.

- Position the graphics cursor near the center of the screen and select that position as the pattern origin using the C key or the tablet pen.

## RETRIEVING PATTERNS

The hexagon pattern you just retrieved probably overlaps with at least one of the other patterns on the screen. To delete the hexagon, enter:

**(CTRL) D + (RETURN)**

or touch the DELETE LAST ENTITY square with your tablet pen.

Because you defined your hexagon pattern as a *group* of entities, all of the points and lines that make up the hexagon are marked with attention indicators.

### NOTE

---

If the Entity Grouping modal had been set to off, it would have been necessary for you to delete each of the entities in the pattern individually.

---

Next, you are asked if you are sure that you want to delete the group of entities:

**DELETE ENTITIES IN GROUP?**

Enter Y in response to this prompt.

Retrieve the pattern again at the center of the screen, but this time use the 5 as the scale factor.

Because the scale factor is quite large, some of the entities in the hexagon have been defined off screen.

### Keyboard Selection

Enter Z to go directly to the Zoom menu.

Enter 11.AUTO MAX-MINS from the Zoom menu.

Enter ] to return to the ENTER PATTERN NAME prompt.

Enter ] to return to the Pattern Management prompt.

Enter F to return to the main menu.

The display on your screen should look something like the display shown in figure 8-8.

### Tablet Selection

| Zoom           |               |                  |                  |
|----------------|---------------|------------------|------------------|
| Auto Max-Min   | Drag Position | 1/2 X            | 2X               |
| Save as Base   | New Center    | New Center Scale | Enter Scale      |
| Return to Base | Retrieve Zoom | Name Zoom        | Zoom from Hdcopy |

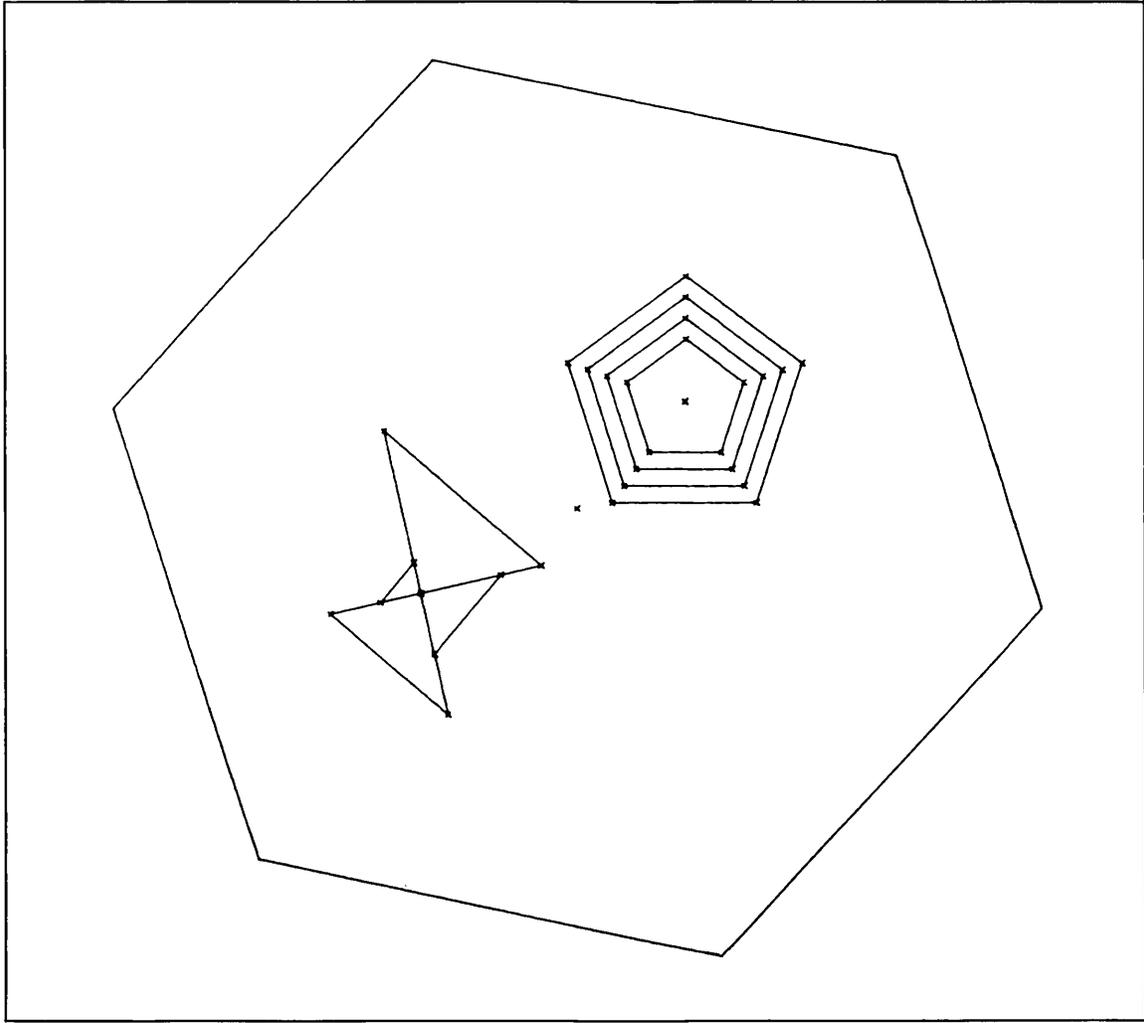


Figure 8-8. Rescaled Hexagon Pattern

## Filing and Exiting

Figure 8-9 outlines the steps to end this terminal session and save the work you just did.

| Terminal Session                                                                                                                                                                                                                                              | Comments                                                                                                                                                                                                                                 |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <pre>ICEMDDN L=0 V=1 D=0.00 P=0 C=1 4</pre>                                                                                                                                                                                                                   | <p>If you are using the keyboard, enter 4 to select the File Current Part/Exit ICEM DDN menu.</p>                                                                                                                                        |
| <pre>FILE CURRENT PART/EXIT ICEM DDN 1.FILE - CONTINUE CURRENT PART 2.   - GET DIFFERENT PART 3.   - QUIT SESSION 4.   - SUSPEND SESSION 5.DO NOT FILE - CONTINUE FROM LAST FILE 6.   - GET DIFFERENT PART 7.   - QUIT SESSION 8.   - SUSPEND SESSION 3</pre> | <p>If you are using the tablet, you can file your part and exit from ICEM DDN in a single step by selecting the FILE/QUIT square with your tablet pen.</p> <p>Select 3.FILE - QUIT SESSION to save your part and exit from ICEM DDN.</p> |
| <pre>ICEMDDN TERMINATED.</pre>                                                                                                                                                                                                                                |                                                                                                                                                                                                                                          |
| <pre>/</pre>                                                                                                                                                                                                                                                  |                                                                                                                                                                                                                                          |

Figure 8-9. Sample File/Exit Sequence

# Generating Attributes

---

9

|                                                                                 |      |
|---------------------------------------------------------------------------------|------|
| Getting Started .....                                                           | 9-1  |
| Generating Points and General Notes Which Represent Electronic Components ..... | 9-2  |
| Generating Common Attributes .....                                              | 9-4  |
| Generating Unique Attributes .....                                              | 9-5  |
| Modifying Subattribute Values Unique to Each Component .....                    | 9-8  |
| Generating Attribute Reports .....                                              | 9-10 |
| Generating a GOES INTO Attribute Report .....                                   | 9-10 |
| Retrieving the Attributes and Subattributes .....                               | 9-11 |
| Specifying the Ordering of the Report .....                                     | 9-12 |
| Formatting the Report.....                                                      | 9-13 |
| Generating an Attribute Report of the Total Data Base.....                      | 9-14 |
| Retrieving the Attributes and Subattributes .....                               | 9-15 |
| Specifying the Ordering of the Report .....                                     | 9-16 |
| Formatting the Report.....                                                      | 9-17 |
| Generating a Parts List with Costs Greater than \$2.49 .....                    | 9-21 |
| Retrieving the Attributes and Subattributes .....                               | 9-22 |
| Specifying the Ordering of the Report .....                                     | 9-23 |
| Formatting the Report.....                                                      | 9-24 |
| Filing and Exiting .....                                                        | 9-27 |



In this practice session, you will learn how to generate a management information data base for an electronics manufacturer. This manufacturer produces a product made up of 14 components. Once the components are constructed and formed into a part, attributes are appended to each component. These attributes make up the management information data base which is used to generate reports for invoicing, inventory control, and internal control. For simplicity, points are used to represent the 14 electronic components.

## Getting Started

Create the new electronic part which will have attributes appended to it as follows:

1. Log in using the procedures described in the introduction to this manual.
2. Create a new part. The sample terminal session in figure 9-1 helps to illustrate this procedure.

| Terminal Session                                                                           | Comments                                                                                                                           |
|--------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------|
| <pre>ENTER PART NAME  ELECTRONICS</pre>                                                    | <p>Enter the name and sheet number of your new part. In this example, we called the part ELECTRONICS with a sheet number of 1.</p> |
| <pre>SHEET NUMBER = 1</pre>                                                                |                                                                                                                                    |
| <pre>NEW PART ASSUMED</pre>                                                                |                                                                                                                                    |
| <pre>--- UNITS OF MEASURE  M 1.METRIC (MM) 2.ENGLISH (INCH) 3.ENGLISH (FOOT/INCH)  2</pre> | <p>First, enter M to turn the menu display on. Next, select menu choice 2 to work in English units.</p>                            |
| <pre>--- DRAFTING STANDARD 1.ANSI 1982 2.ANSI 1973 3. 4. 5. 6. 7. 8. 9.          1</pre>   | <p>Select menu choice 1 to use 1982 ANSI standard dimensions.</p>                                                                  |
| <pre>ICEMDDN  L=0  V=1  D=0.00  P=0  C=0</pre>                                             |                                                                                                                                    |

Figure 9-1. Start Up Procedure for Generating Attributes

## Generating Points and General Notes Which Represent Electronic Components

Create a list of electronic components with points and general notes as follows. This display is rescaled as necessary.

### Keyboard Selection

Select 9.POINT from the main menu.

Select 2.KEY-IN from the Point menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

### Tablet Selection

|                       |
|-----------------------|
| Transform<br>XT,YT,ZT |
| Model<br>X,Y,Z        |

Enter the xt- and yt- coordinates in table 9-1 to define the points that represent electronic components:

**Table 9-1. Electronic Component Point Coordinates**

| XT  | YT   | General Note |
|-----|------|--------------|
| 3.5 | 8.3  | PART A       |
| 4.5 | 7.5  | PART B       |
| 5.3 | 7.0  | PART D       |
| 5.3 | 6.1  | PART E       |
| 5.3 | 5.5  | PART F       |
| 6.0 | 4.7  | PART G       |
| 6.0 | 4.0  | PART H       |
| 6.0 | 3.4  | PART C       |
| 6.8 | 2.8  | PART I       |
| 6.8 | 2.1  | PART J       |
| 6.8 | 1.5  | PART F       |
| 7.5 | 0.9  | PART G       |
| 7.5 | 0.2  | PART H       |
| 8.5 | -0.6 | PART K       |

To identify each of the electronic components you just created, add the general notes listed in table 9-1 as follows:

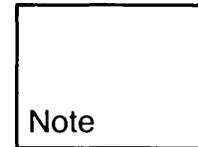
**Keyboard Selection**

**Tablet Selection**

Enter F to return to the main menu.

Select 16.DRAFTING from the main menu.

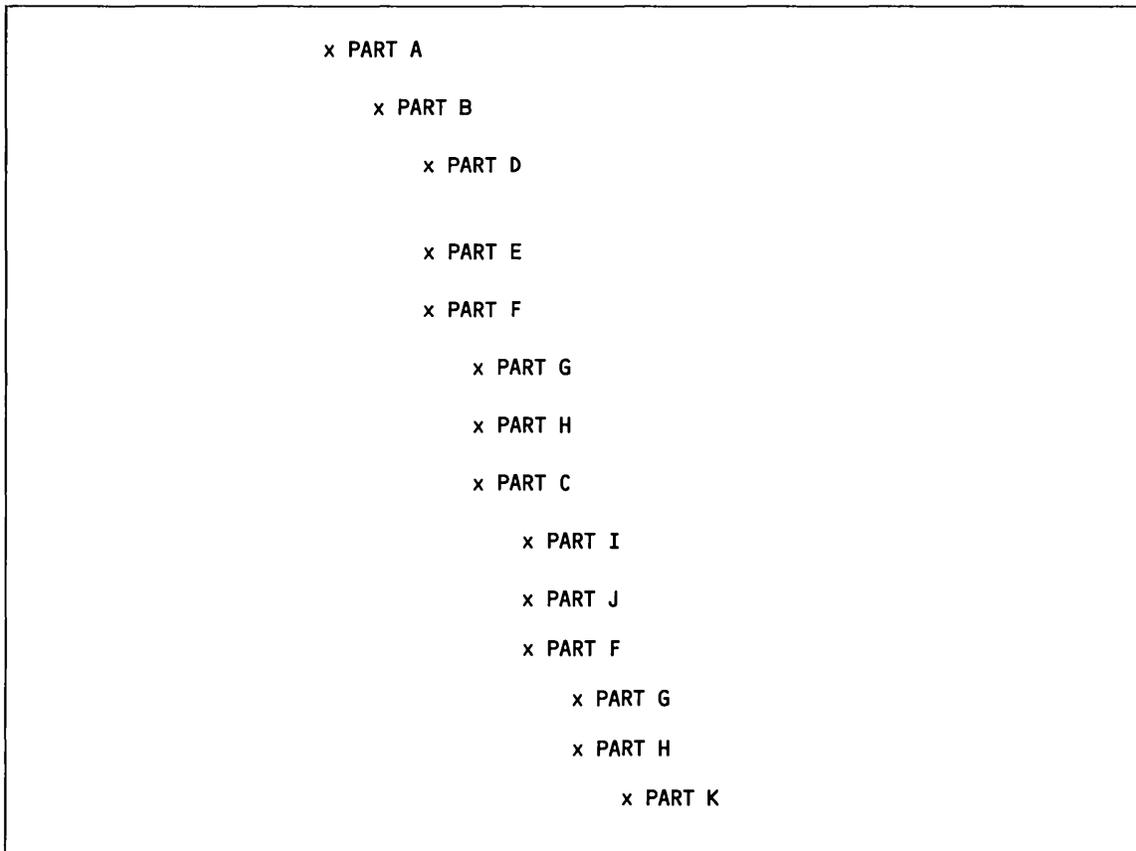
Select 10.NOTE from the Drafting menu.



- Position the graphics cursor *to the right* of each point to select that position as the text origin.

Type in the general notes associated with each point followed by a RETURN. Enter RETURN again to indicate that you have finished the note.

After the display is rescaled as necessary, the result should appear as shown in figure 9-2.



**Figure 9-2. Rescaled Display of Components**

## Generating Common Attributes

Define the common attributes associated with these components (points) as follows:

### Keyboard Selection

Select 5.6.1 CREATE from the Attribute Management menu.

### Tablet Selection

| Attribute Management |         |                   |          |        |
|----------------------|---------|-------------------|----------|--------|
| Create               | Copy    | Search            | Identify |        |
| List On File         | Display | Report Generation | Delete   | Modify |

|     |
|-----|
| DBM |
|-----|

- Position the graphics cursor near each of the points and individually select them using the P key or the tablet pen.

Enter ] to end the selection process.

You are then asked to enter the attribute name. Enter ELECTRONICS in response to the prompt:

```
ENTER ATTRIBUTE NAME
ELECTRONICS
```

You are then asked to select the type of subattribute you want to enter. Because there are no subattributes for this attribute, enter ] to return to the ENTER ATTRIBUTE NAME prompt.

```
SUBATTRIBUTE TYPE
1.NAME
2.VALUE
3.BOTH]
```

Enter SOURCE as the name of the next attribute:

```
ENTER ATTRIBUTE NAME
SOURCE
```

Next, select 3 (both names and values) in response to the SUBATTRIBUTE TYPE prompt and enter the following subattribute names and values:

```
ENTER SUBATTRIBUTE NAME 1
COST
VALUE 1 = 1

ENTER SUBATTRIBUTE NAME 2
DELIV
VALUE 2 = 2

ENTER SUBATTRIBUTE NAME 3
VENDOR
VALUE 3 = 3

ENTER SUBATTRIBUTE NAME 4
]
```

Enter ] three times to return to the Attribute Management menu.

## Generating Unique Attributes

In the next part of this session, you will learn how to generate attributes that are unique to each point.

There are three attributes (with subattributes) for each point. Table 9-2 contains the attribute information you will need to complete this part of the session.

**Table 9-2. Subattribute Values**

| Part Name | Manufacturer | Value | Goes Into |
|-----------|--------------|-------|-----------|
| PART B    | FAB          | 2     | PART A    |
| PART D    | WE MAKE      | 3     | PART B    |
| PART E    | WE MAKE      | 3     | PART B    |
| PART F    | FAB          | 3     | PART B    |
| PART G    | DEKO         | 4     | PART F    |
| PART H    | ACE MANUF    | 4     | PART F    |
| PART C    | FAB          | 2     | PART A    |
| PART I    | ABC CO       | 3     | PART C    |
| PART J    | WE MAKE      | 3     | PART C    |
| PART F    | FAB          | 3     | PART C    |
| PART G    | DEKO         | 4     | PART F    |
| PART H    | ACE MANUF    | 4     | PART F    |
| PART K    | WE MAKE      | 5     | PART H    |

To help guide you through this data entry, we have prepared the following sample terminal session (figure 9-3).

### NOTE

At any time during attribute creation, you can display the attributes you have created using 5.6.5 DISPLAY. If you find any typing and/or spelling errors, use 5.6.4 DELETE or 5.6.2 MODIFY to make the corrections.

| Terminal Session                                                                                                                                                                                                                                                        | Comments                                                                                                                                              |
|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------|
| <p>ATTRIBUTE MANAGEMENT<br/>           1.CREATE<br/>           2.MODIFY<br/>           3.COPY<br/>           4.DELETE<br/>           5.DISPLAY<br/>           6.IDENTIFY<br/>           7.SEARCH<br/>           8.LIST ON FILE<br/>           9.REPORT GENERATION 1</p> | <p>Enter 1 to create attributes.</p>                                                                                                                  |
| <p>INDICATE ENTITY</p>                                                                                                                                                                                                                                                  | <p>Select the point identified as PART A as the first entity to be assigned attributes.<br/><br/>           Enter ] to end the selection process.</p> |
| <p>ENTER ATTRIBUTE NAME<br/>           PART NAME</p>                                                                                                                                                                                                                    | <p>Enter PART NAME as the attribute name.</p>                                                                                                         |
| <p>SUBATTRIBUTE TYPE<br/>           1.NAME<br/>           2.VALUE<br/>           3.BOTH 1</p>                                                                                                                                                                           | <p>Enter 1 to assign a name to the attribute you just entered.</p>                                                                                    |
| <p>ENTER SUBATTRIBUTE NAME 1<br/>           PART A</p>                                                                                                                                                                                                                  | <p>Enter the subattribute name (refer to table 9-2 under the column heading Part Name).</p>                                                           |
| <p>ENTER SUBATTRIBUTE NAME 2<br/>           ]</p>                                                                                                                                                                                                                       | <p>Enter ] to return to the ENTER ATTRIBUTE NAME prompt.</p>                                                                                          |
| <p>ENTER ATTRIBUTE NAME<br/>           MANUF</p>                                                                                                                                                                                                                        | <p>Enter MANUF as the attribute name.</p>                                                                                                             |

Figure 9-3. Creating Attributes (Continued)

*(Continued)*

| Terminal Session                                   | Comments                                                                                               |
|----------------------------------------------------|--------------------------------------------------------------------------------------------------------|
| SUBATTRIBUTE TYPE<br>1.NAME<br>2.VALUE<br>3.BOTH 3 | Enter 3 to assign both a name and a value to the attribute.                                            |
| ENTER SUBATTRIBUTE NAME 1<br>FINAL                 | Enter the subattribute name (refer to table 9-2 under the column heading Manufacturer).                |
| VALUE 1 = 1                                        | Enter the value associated with that subattribute (refer to table 9-2 under the column heading Value). |
| ENTER SUBATTRIBUTE NAME 2<br>]                     | Enter ] to return to the ENTER ATTRIBUTE NAME prompt.                                                  |
| ENTER ATTRIBUTE NAME<br>GOES INTO                  | Enter GOES INTO as the attribute name.                                                                 |
| SUBATTRIBUTE TYPE<br>1.NAME<br>2.VALUE<br>3.BOTH 1 | Enter 1 to assign names only.                                                                          |
| ENTER SUBATTRIBUTE NAME 1<br>MAJOR                 | Enter the subattribute name (refer to table 9-2 under the column heading Goes Into).                   |
| ENTER SUBATTRIBUTE NAME 2<br>]                     | Enter ] to return to the ENTER SUBATTRIBUTE NAME prompt.                                               |
| ENTER ATTRIBUTE NAME<br>] ]                        | Enter ] twice to return to the Attribute Management menu.                                              |

**Figure 9-3. Creating Attributes**

Repeat the steps in figure 9-3 for each of the part names listed in table 9-2.

## Modifying Subattribute Values Unique to Each Component

Change the subattribute values for the common attributes of each component as follows:

### Keyboard Selection

Select 5.6.2 MODIFY from the Attribute Management menu.

### Tablet Selection

| Attribute Management |         |                   |          |        |
|----------------------|---------|-------------------|----------|--------|
| Create               | Copy    | Search            | Identify |        |
| List On File         | Display | Report Generation | Delete   | Modify |

DBM

Select 1.SCREEN SELECT ENTITY as the Selection mode.

- Position the graphics cursor on the point identified as PART A and select it as the entity to be modified using the P key or the tablet pen.

Enter Y in response to the prompt:

SELECTION OK? Y

Enter 2/1 in response to the prompt:

INDICATE MODIFICATION LINE BY:  
NO.ATTR/NO. SUB-ATTR. 2/1

### NOTE

All attributes and subattributes are displayed with numbers when selected. These are the numbers that should be used when selecting attributes and subattributes.

Accept the current subattribute name (COST) by entering RETURN in response to the prompt:

OLD SUB-ATTR.NAME=COST  
TYPE NEW SUB-ATTR. NAME (RETURN)

Enter 99.99 as the new cost value:

NEW VALUE = 1.0000 99.99 J

Enter 2/2 in response to the prompt:

INDICATE MODIFICATION LINE BY:  
NO.ATTR/NO. SUB-ATTR. 2/2

Accept the current subattribute name (DELIV) by entering RETURN in response to the prompt:

OLD SUB-ATTR.NAME=DELIV  
TYPE NEW SUB-ATTR. NAME (RETURN)

Enter 30 as the new value:

NEW VALUE = 2.0000 30 J

Enter 2/3 in response to the prompt:

INDICATE MODIFICATION LINE BY:  
NO.ATTR/NO. SUB-ATTR. 2/3

Accept the current subattribute name (VENDOR) by entering RETURN in response to the prompt:

OLD SUB-ATTR.NAME=VENDOR  
TYPE NEW SUB-ATTR. NAME (RETURN)

Enter 999 as the new value:

NEW VALUE = 3.0000 999 ]

Enter ] in response to the NO.ATTR/NO.SUB-ATTR prompt.

Use table 9-3 as your guide to modify the subattribute values for the rest of the points.

**NOTE**

To verify attribute modification, use 5.6.5 DISPLAY from the Attribute Management menu.

**Table 9-3. Subattribute Values for Modification**

| Part   | Cost | Delivery | Vendor |
|--------|------|----------|--------|
| PART B | 6.25 | 20       | 825    |
| PART D | 1.00 | 10       | 654    |
| PART E | 2.75 | 10       | 878    |
| PART F | 2.50 | 10       | 879    |
| PART G | 1.25 | 3        | 870    |
| PART H | 1.25 | 3        | 876    |
| PART C | 8.00 | 10       | 744    |
| PART I | 2.00 | 5        | 786    |
| PART J | 0.50 | 5        | 111    |
| PART F | 2.50 | 10       | 879    |
| PART G | 1.25 | 3        | 870    |
| PART H | 1.25 | 3        | 876    |
| PART K | 1.50 | 5        | 119    |

## Generating Attribute Reports

In the next part of this practice session, you will learn how attributes and subattributes are used together with the Attribute Reporting function to display reports containing some or all of the attribute data.

Each entity has attributes/subattributes associated with them in the format shown in figure 9-4.

| ATTRIBUTE   | SUBATTRIBUTE | VALUE        |
|-------------|--------------|--------------|
| ELECTRONICS |              |              |
| PART NAME   |              |              |
|             | PART NUMBER  |              |
| SOURCE      |              |              |
|             | COST         | VALUE(COST)  |
|             | DELIV        | VALUE(DAYS)  |
|             | VENDOR       | VALUE(CODE)  |
| MANUF       |              |              |
|             | VENDOR NAME  | VALUE(LEVEL) |
| GOES INTO   |              |              |
|             | PART NUMBER  |              |

Figure 9-4. Entity Attribute Listing Format

## Generating a GOES INTO Attribute Report

The first attribute report that you will generate is a GOES INTO list using the subattribute names PART NAME and GOES INTO. Figure 9-5 shows the report.

| ATTRIBUTE REPORT |           |
|------------------|-----------|
| GOES INTO LIST   |           |
| PART NAME        | GOES INTO |
| PART A           | MAJOR     |
| PART B           | PART A    |
| PART C           | PART A    |
| PART D           | PART B    |
| PART E           | PART B    |
| PART F           | PART C    |
| PART G           | PART F    |
| PART H           | PART F    |
| PART I           | PART C    |
| PART J           | PART C    |
| PART K           | PART H    |

Figure 9-5. GOES INTO List Attribute Report

**Retrieving the Attributes and Subattributes**

The first step is to tell the system which attributes and subattributes an entity must have to be included in your report. Figure 9-6 is a sample terminal session to help you with this task.

**Keyboard Selection**

**Tablet Selection**

Select 5.6.9 REPORT GENERATION from the Attribute Management menu.

| Attribute Management |         |                   |          |        |
|----------------------|---------|-------------------|----------|--------|
| Create               | Copy    | Search            | Identify |        |
| List On File         | Display | Report Generation | Delete   | Modify |

DBM

| Terminal Session                                                                                                                                                     | Comments                                                                                                                                                                                              |
|----------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ENTER ATTRIBUTE NAME 1<br>ELECTRONICS                                                                                                                                | Enter ELECTRONICS as the name of the attribute that all entities in the report have in common. Because ELECTRONICS is common to all entities, all of the attributes are extracted from the data base. |
| CONDITIONAL RELATIONS<br>1.UNCONDITIONAL<br>2.LESS THAN<br>3.LESS THAN OR EQUAL TO<br>4.EQUAL TO<br>5.NOT EQUAL TO<br>6.GREATER THAN OR EQUAL TO<br>7.GREATER THAN 1 | Enter 1 in response to this prompt. This places no restrictions on the subattribute values associated with the attribute name ELECTRONICS.                                                            |
| ENTER ATTRIBUTE NAME 2<br>]                                                                                                                                          | Since there are no other attributes to search for, enter ] in response to this prompt.                                                                                                                |
| ENTER SUBATTRIBUTE NAME 1<br>]                                                                                                                                       | Since there are also no more subattributes to search for, enter ] to end the retrieval stage of report generation.                                                                                    |

**Figure 9-6. Retrieving Entities with the ELECTRONICS Attribute**

### Specifying the Ordering of the Report

The next step is to tell the system how you want the data in your report to be ordered. Figure 9-7 is a sample terminal session to help you with this task.

| Terminal Session                                                | Comments                                                                                      |
|-----------------------------------------------------------------|-----------------------------------------------------------------------------------------------|
| ORDERED ITEM 1<br>1.SUBATTRIBUTE NAME<br>2.SUBATTRIBUTE VALUE 1 | Enter 1 to order alphabetically according to the subattribute names assigned to an attribute. |
| ENTER ATTRIBUTE NAME<br>PART NAME                               | Enter PART NAME as the attribute whose subattributes are reported alphabetically.             |
| ORDERED ITEM 2<br>1.SUBATTRIBUTE NAME<br>2.SUBATTRIBUTE VALUE 2 | Enter 2 to numerically order by the subattribute values assigned to a subattribute name.      |
| ENTER ATTRIBUTE NAME<br>MANUF                                   | Enter MANUF in response to this prompt.                                                       |
| ENTER SUBATTRIBUTE NAME<br>]                                    | Enter ] in response to this prompt.                                                           |
| NUMERIC ORDERING<br>1.ASCENDING<br>2.DECENDING 1                | Enter 1 again to report the values under MANUF in ascending order.                            |
| ORDERED ITEM 3<br>1.SUBATTRIBUTE NAME<br>2.SUBATTRIBUTE VALUE ] | Enter ] to end the ordering stage of report generation.                                       |

Figure 9-7. Ordering Report Data for the GOES INTO List

## Formatting the Report

After a short delay for the ordering, the system asks you for a report description (figure 9-8).

| Terminal Session                                                                                                                  | Comments                                                                                                                                     |
|-----------------------------------------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------|
| ENTER TITLE LINE 1<br>ATTRIBUTE REPORT                                                                                            | Enter ATTRIBUTE REPORT as the first line of the title.                                                                                       |
| ENTER TITLE LINE 2<br>GOES INTO LIST                                                                                              | Enter GOES INTO LIST as the second line of the title.                                                                                        |
| ENTER TITLE LINE 3<br>]                                                                                                           | Enter ] to end title line input.                                                                                                             |
| ENTER COLUMN HEADING 1<br>PART NAME                                                                                               | Enter PART NAME as the heading of the first column of the report.                                                                            |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 2 | Enter 2 to specify that the items in the column are subattribute names. The next prompt specifies the attribute name of these subattributes. |
| ENTER ATTRIBUTE NAME<br>PART NAME                                                                                                 | Enter PART NAME in response to this prompt.                                                                                                  |
| ENTER COLUMN HEADING 2<br>GOES INTO                                                                                               | Enter GOES INTO as the heading of the second column of the report.                                                                           |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 2 | Enter 2 to specify that the items in the column are subattribute names. The next prompt specifies the attribute name of these subattributes. |
| ENTER ATTRIBUTE NAME<br>GOES INTO                                                                                                 | Enter GOES INTO in response to this prompt.                                                                                                  |
| ENTER COLUMN HEADING 3<br>]                                                                                                       | Enter ] to end column heading input.                                                                                                         |

Figure 9-8. Formatting the Data for the GOES INTO List (Continued)

(Continued)

| Terminal Session                        | Comments                                                                                                                                                                                                                                        |
|-----------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 1.CHARACTER SIZE=0.1250<br>2.LEVEL =0 ] | Enter ] to accept the default character size and level.                                                                                                                                                                                         |
| INDICATE REPORT ORIGIN                  | Position the graphics cursor near the upper left hand corner of the screen and select that position using the C key or the tablet pen. This is done to avoid displaying the report on top of the electronic components currently on the screen. |
| DEFINE REPORT AS GROUP? N               | Enter N to define your report as individual entities.                                                                                                                                                                                           |
| RESCALE? Y                              | If your report is partially off of the screen, you are asked if you want to rescale. Enter Y in response to this prompt.                                                                                                                        |

Figure 9-8. Formatting the Data for the GOES INTO List

**NOTE**

You can keep your screen from becoming cluttered by assigning each of the reports you create to a different level using 1.7 MODIFY ENTITY LEVEL/PEN NO. Then, using 2.2 BLANK-LEVEL RANGE, you can blank one or more of the reports.

**Generating an Attribute Report of the Total Data Base**

The second report that you generate is an attribute report that displays the entire data base (figure 9-9).

| REPORT GENERATOR<br>TOTAL DATA BASE |           |       |           |       |       |            |           |             |
|-------------------------------------|-----------|-------|-----------|-------|-------|------------|-----------|-------------|
| QTY                                 | PART NAME | LEVEL | GOES INTO | COST  | FLOAT | VENDOR     | VEND CODE | DEPARTMENT  |
| 1                                   | PART A    | 1     | MAJOR     | 99.99 | 30    | FINAL      | 999       | ELECTRONICS |
| 1                                   | PART B    | 2     | PART A    | 6.25  | 20    | FAB        | 825       | ELECTRONICS |
| 1                                   | PART C    | 2     | PART A    | 8.00  | 10    | FAB        | 744       | ELECTRONICS |
| 1                                   | PART D    | 3     | PART B    | 1.00  | 10    | WE MAKE    | 654       | ELECTRONICS |
| 1                                   | PART E    | 3     | PART B    | 2.75  | 10    | WE MAKE    | 878       | ELECTRONICS |
| 2                                   | PART F    | 3     | PART C    | 2.50  | 10    | FAB        | 879       | ELECTRONICS |
| 2                                   | PART G    | 4     | PART F    | 1.25  | 3     | DEKO       | 870       | ELECTRONICS |
| 2                                   | PART H    | 4     | PART F    | 1.25  | 3     | ACE MANUF. | 876       | ELECTRONICS |
| 1                                   | PART I    | 3     | PART C    | 2.00  | 5     | ABC CO.    | 786       | ELECTRONICS |
| 1                                   | PART J    | 3     | PART C    | 0.50  | 5     | WE MAKE    | 111       | ELECTRONICS |
| 1                                   | PART K    | 5     | PART H    | 1.50  | 5     | WE MAKE    | 119       | ELECTRONICS |

Figure 9-9. Total Data Base Attribute Report

**Retrieving the Attributes and Subattributes**

The first step is to tell the system which attributes and subattributes you want included in your report. Figure 9-10 is a sample terminal session to help you with the task.

**Keyboard Selection**

**Tablet Selection**

Select 5.6.9 REPORT GENERATION from the Attribute Management menu.

| Attribute Management |         |                   |          |        |
|----------------------|---------|-------------------|----------|--------|
| Create               | Copy    | Search            | Identify |        |
| List On File         | Display | Report Generation | Delete   | Modify |

DBM

| Terminal Session                                                                                                                                                     | Comments                                                                                                                                                                                                                               |
|----------------------------------------------------------------------------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ENTER ATTRIBUTE NAME 1<br>ELECTRONICS                                                                                                                                | Enter ELECTRONICS in response to this prompt. This is the attribute name that an entity must have to be included in the report. Because ELECTRONICS is common to all entities, all of the attributes are extracted from the data base. |
| CONDITIONAL RELATIONS<br>1.UNCONDITIONAL<br>2.LESS THAN<br>3.LESS THAN OR EQUAL TO<br>4.EQUAL TO<br>5.NOT EQUAL TO<br>6.GREATER THAN OR EQUAL TO<br>7.GREATER THAN 1 | Enter 1 in response to this prompt. This places no restrictions on the subattribute values associated with the attribute name ELECTRONICS.                                                                                             |
| ENTER ATTRIBUTE NAME 2<br>]                                                                                                                                          | Since there are no other attributes to search for, enter ] in response to this prompt.                                                                                                                                                 |
| ENTER SUBATTRIBUTE NAME 1<br>]                                                                                                                                       | Since there are also no more subattributes to search for, enter ] to end the retrieval stage of report generation.                                                                                                                     |

**Figure 9-10. Retrieving Entities with the ELECTRONICS Attribute**

### Specifying the Ordering of the Report

The next step is to tell the system how you want the data in your report to be sorted. Figure 9-11 is a sample terminal session to help you with the data entry.

| Terminal Session                                                | Comments                                                                                      |
|-----------------------------------------------------------------|-----------------------------------------------------------------------------------------------|
| ORDERED ITEM 1<br>1.SUBATTRIBUTE NAME<br>2.SUBATTRIBUTE VALUE 1 | Enter 1 to order alphabetically according to the subattribute names assigned to an attribute. |
| ENTER ATTRIBUTE NAME<br>PART NAME                               | Enter PART NAME as the attribute whose subattributes are reported alphabetically.             |
| ORDERED ITEM 2<br>1.SUBATTRIBUTE NAME<br>2.SUBATTRIBUTE VALUE 2 | Enter 2 to numerically order by the subattribute values assigned to a subattribute name.      |
| ENTER ATTRIBUTE NAME<br>MANUF                                   | Enter MANUF in response to this prompt.                                                       |
| ENTER SUBATTRIBUTE NAME<br>]                                    | Enter ] in response to this prompt.                                                           |
| NUMERIC ORDERING<br>1.ASCENDING<br>2.DECENDING 1                | Enter 1 again to report the values under MANUF in ascending order.                            |
| ORDERED ITEM 3<br>1.SUBATTRIBUTE NAME<br>2.SUBATTRIBUTE VALUE ] | Enter ] to end the ordering stage of report generation.                                       |

Figure 9-11. Ordering Report Data for the Total Data Base Report

**Formatting the Report**

After a short delay for the ordering, the system asks you for a description of the report (figure 9-12).

| Terminal Session                                                                                                                  | Comments                                                                                                                                             |
|-----------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------|
| ENTER TITLE LINE 1<br>REPORT GENERATOR                                                                                            | Enter REPORT GENERATOR as the first line of the title.                                                                                               |
| ENTER TITLE LINE 2<br>TOTAL DATA BASE                                                                                             | Enter TOTAL DATA BASE as the second line of the title.                                                                                               |
| ENTER TITLE LINE 3<br>]                                                                                                           | Enter ] to end title line input.                                                                                                                     |
| ENTER COLUMN HEADING 1<br>QTY                                                                                                     | Enter QTY as the heading of the first column of the report.                                                                                          |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 5 | Enter 5 to compute the number of part names that have identical ordering rules. For example, there are two components named PART F in the data base. |
| ENTER COLUMN HEADING 2<br>PART NAME                                                                                               | Enter PART NAME as the heading of the second column of the report.                                                                                   |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 2 | Enter 2 to specify that the items in the next column are subattribute names. The next prompt specifies the attribute name of these subattributes.    |
| ENTER ATTRIBUTE NAME<br>PART NAME                                                                                                 | Enter PART NAME in response to this prompt.                                                                                                          |

**Figure 9-12. Formatting the Data for the Total Data Base Report (Continued)**

(Continued)

| Terminal Session                                                                                                                  | Comments                                                                                                                                          |
|-----------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------|
| ENTER COLUMN HEADING 3<br>LEVEL                                                                                                   | Enter LEVEL as the heading of the third column of the report.                                                                                     |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 3 | Enter 3 to produce an ascending list of MANUF subattribute values (refer to the next prompt).                                                     |
| ENTER ATTRIBUTE NAME<br>MANUF                                                                                                     | Enter MANUF in response to this prompt.                                                                                                           |
| ENTER SUBATTRIBUTE NAME<br>]                                                                                                      | Enter ] in response to this prompt.                                                                                                               |
| DECIMAL PLACES = 0                                                                                                                | Enter 0 to tell the system that you do not want decimal places used for the report entries in this column.                                        |
| ENTER COLUMN HEADING 4<br>GOES INTO                                                                                               | Enter GOES INTO as the heading of the fourth column of the report.                                                                                |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 2 | Enter 2 to specify that the items in the next column are subattribute names. The next prompt specifies the attribute name of these subattributes. |
| ENTER ATTRIBUTE NAME<br>GOES INTO                                                                                                 | Enter GOES INTO in response to this prompt.                                                                                                       |
| ENTER COLUMN HEADING 5<br>COST                                                                                                    | Enter COST as the heading of the fifth column of the report.                                                                                      |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 3 | Enter 3 to produce an ascending list of COST subattribute values (refer to the next prompt).                                                      |

Figure 9-12. Formatting the Data for the Total Data Base Report (Continued)

*(Continued)*

| Terminal Session                                                                                                                  | Comments                                                                                                                                          |
|-----------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------|
| ENTER ATTRIBUTE NAME<br>SOURCE                                                                                                    | Enter SOURCE in response to this prompt.                                                                                                          |
| ENTER SUBATTRIBUTE NAME<br>COST                                                                                                   | Enter COST in response to this prompt.                                                                                                            |
| DECIMAL PLACES = 2                                                                                                                | Enter 2 to tell the system that you want two decimal places used for the report entries in this column.                                           |
| ENTER COLUMN HEADING 6<br>FLOAT                                                                                                   | Enter FLOAT as the heading of the sixth column of the report.                                                                                     |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 3 | Enter 3 to produce an ascending list of DELIV subattribute values (refer to the next prompt).                                                     |
| ENTER ATTRIBUTE NAME<br>SOURCE                                                                                                    | Enter SOURCE in response to this prompt.                                                                                                          |
| ENTER SUBATTRIBUTE NAME<br>DELIV                                                                                                  | Enter DELIV in response to this prompt.                                                                                                           |
| DECIMAL PLACES = 0                                                                                                                | Enter 0 to tell the system that you do not want decimal places used for the report entries in this column.                                        |
| ENTER COLUMN HEADING 7<br>VENDOR                                                                                                  | Enter VENDOR as the heading of the seventh column of the report.                                                                                  |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 2 | Enter 2 to specify that the items in the next column are subattribute names. The next prompt specifies the attribute name of these subattributes. |
| ENTER ATTRIBUTE NAME<br>MANUF                                                                                                     | Enter MANUF in response to this prompt.                                                                                                           |
| ENTER COLUMN HEADING 8<br>VEND CODE                                                                                               | Enter VEND CODE as the heading of the eighth column of the report.                                                                                |

Figure 9-12. Formatting the Data for the Total Data Base Report *(Continued)*

(Continued)

| Terminal Session                                                                                                                  | Comments                                                                                                                                                                                                                                         |
|-----------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 3 | Enter 3 to produce an ascending list of VENDOR subattribute values (refer to the next prompt).                                                                                                                                                   |
| ENTER ATTRIBUTE NAME<br>SOURCE                                                                                                    | Enter SOURCE in response to this prompt.                                                                                                                                                                                                         |
| ENTER SUBATTRIBUTE NAME<br>VENDOR                                                                                                 | Enter VENDOR in response to this prompt.                                                                                                                                                                                                         |
| DECIMAL PLACES = 0                                                                                                                | Enter 0 to tell the system that you do not want decimal places used for the report entries in this column.                                                                                                                                       |
| ENTER COLUMN HEADING 9<br>DEPARTMENT                                                                                              | Enter DEPARTMENT as the heading of the ninth column of the report.                                                                                                                                                                               |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 1 | Enter 1 to display the attribute name (refer to the next prompt) associated with each subattribute in the report.                                                                                                                                |
| ENTER ATTRIBUTE NAME<br>ELECTRONICS                                                                                               | Enter ELECTRONICS in response to this prompt.                                                                                                                                                                                                    |
| ENTER COLUMN HEADING 10<br>]                                                                                                      | Enter ] to end column heading input.                                                                                                                                                                                                             |
| 1.CHARACTER SIZE=0.1250<br>2.LEVEL =0 ]                                                                                           | Enter ] to accept the default character size and level.                                                                                                                                                                                          |
| INDICATE REPORT ORIGIN                                                                                                            | Position the graphics cursor near the bottom left hand corner of the screen and select that position using the C key or the tablet pen. This is done to avoid displaying the report on top of the electronic components currently on the screen. |
| DEFINE REPORT AS GROUP? N                                                                                                         | Enter N to define your report as individual entities.                                                                                                                                                                                            |
| RESCALE? Y                                                                                                                        | Because your report is off the screen, enter Y in response to this prompt.                                                                                                                                                                       |

Figure 9-12. Formatting the Data for the Total Data Base Report

### Generating a Parts List with Costs Greater than \$2.49

The last report you will generate is an attribute report that displays a parts list with costs greater than \$2.49.

This report (shown in figure 9-13) is generated using PART NAME, COST, VENDOR CODE, and VENDOR NAME.

| ATTRIBUTE REPORT  |       |             |             |
|-------------------|-------|-------------|-------------|
| COST - GT. \$2.49 |       |             |             |
| PART NAME         | COST  | VENDOR CODE | VENDOR NAME |
| PART F            | 2.50  | 879         | FAB         |
| PART E            | 2.75  | 878         | WE MAKE     |
| PART B            | 6.25  | 825         | FAB         |
| PART C            | 8.00  | 744         | FAB         |
| PART A            | 99.99 | 999         | FINAL       |

Figure 9-13. Attribute Report

### Retrieving the Attributes and Subattributes

Select the attributes and subattributes that are to be included in the report using figure 9-14 as your guide.

#### Keyboard Selection

#### Tablet Selection

Select 5.6.9 REPORT GENERATION from the Attribute Management menu.

| Attribute Management |         |                   |          |        |
|----------------------|---------|-------------------|----------|--------|
| Create               | Copy    | Search            | Identify |        |
| List On File         | Display | Report Generation | Delete   | Modify |

|     |
|-----|
| DBM |
|-----|

| Terminal Session                                                                                                                                                     | Comments                                                                                                                            |
|----------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------|
| ENTER ATTRIBUTE NAME 1<br>]                                                                                                                                          | Enter ] to move to the next prompt.                                                                                                 |
| ENTER SUBATTRIBUTE NAME 1<br>COST                                                                                                                                    | Enter COST in response to this prompt.                                                                                              |
| CONDITIONAL RELATIONS<br>1.UNCONDITIONAL<br>2.LESS THAN<br>3.LESS THAN OR EQUAL TO<br>4.EQUAL TO<br>5.NOT EQUAL TO<br>6.GREATER THAN OR EQUAL TO<br>7.GREATER THAN 7 | Enter 7 to restrict the entities available for retrieval to those with a value greater than \$2.49 (refer to the next prompt).      |
| VALUE = 2.49                                                                                                                                                         | Enter 2.49 in response to this prompt. This extracts all entities with the subattribute value greater than 2.49 from the data base. |
| ENTER SUBATTRIBUTE NAME 2<br>]                                                                                                                                       | Enter ] in response to this prompt.                                                                                                 |

Figure 9-14. Retrieving Entities with the COST Subattribute

### Specifying the Ordering of the Report

Select the sorting criteria for your report using figure 9-15 as your guide.

| Terminal Session                                                | Comments                                                                                      |
|-----------------------------------------------------------------|-----------------------------------------------------------------------------------------------|
| ORDERED ITEM 1<br>1.SUBATTRIBUTE NAME<br>2.SUBATTRIBUTE VALUE 2 | Enter 2 to numerically order by the subattribute values assigned to a subattribute name.      |
| ENTER ATTRIBUTE NAME<br>SOURCE                                  | Enter SOURCE as the attribute whose subattributes are reported alphabetically.                |
| ENTER SUBATTRIBUTE NAME<br>COST                                 | Enter COST in response to this prompt.                                                        |
| ORDERED ITEM 2<br>1.SUBATTRIBUTE NAME<br>2.SUBATTRIBUTE VALUE 1 | Enter 1 to order alphabetically according to the subattribute names assigned to an attribute. |
| ENTER ATTRIBUTE NAME<br>PART NAME                               | Enter PART NAME in response to this prompt.                                                   |
| NUMERIC ORDERING<br>1.ASCENDING<br>2.DECENDING 1                | Enter 1 to report the values under PART NAME in ascending order.                              |
| ORDERED ITEM 3<br>1.SUBATTRIBUTE NAME<br>2.SUBATTRIBUTE VALUE ] | Enter ] to end the ordering stage of report generation.                                       |

Figure 9-15. Ordering Report Data for the Parts List Report

### Formatting the Report

After a short delay for the ordering, the system asks you for a report description (figure 9-16).

| Terminal Session                                                                                                                  | Comments                                                                                                                                     |
|-----------------------------------------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------|
| ENTER TITLE LINE 1<br>ATTRIBUTE REPORT                                                                                            | Enter ATTRIBUTE REPORT as the first line of the header.                                                                                      |
| ENTER TITLE LINE 2<br>COST - GT, \$2.49                                                                                           | Enter COST - GT, \$2.49 as the second line of the header.                                                                                    |
| ENTER TITLE LINE 3<br>]                                                                                                           | Enter ] to end title line input.                                                                                                             |
| ENTER COLUMN HEADING 1<br>PART NAME                                                                                               | Enter PART NAME as the heading of the first column of the report.                                                                            |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 2 | Enter 2 to specify that the items in the column are subattribute names. The next prompt specifies the attribute name of these subattributes. |
| ENTER ATTRIBUTE NAME<br>PART NAME                                                                                                 | Enter PART NAME in response to this prompt.                                                                                                  |
| ENTER COLUMN HEADING 2<br>COST                                                                                                    | Enter COST as the heading of the second column of the report.                                                                                |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 3 | Enter 3 to produce an ascending list of COST subattribute values (refer to the next prompt).                                                 |

Figure 9-16. Formatting the Data for the Parts List Report (Continued)

(Continued)

| Terminal Session                                                                                                                  | Comments                                                                                                                                     |
|-----------------------------------------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------------------------------------------------------|
| ENTER ATTRIBUTE NAME<br>SOURCE                                                                                                    | Enter SOURCE in response to this prompt.                                                                                                     |
| ENTER SUBATTRIBUTE NAME<br>COST                                                                                                   | Enter COST in response to this prompt.                                                                                                       |
| DECIMAL PLACES = 2                                                                                                                | Enter 2 to tell the system you want two decimal places used for the report entries in this column.                                           |
| ENTER COLUMN HEADING 3<br>VENDOR CODE                                                                                             | Enter VENDOR CODE as the heading of the third column of the report.                                                                          |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 3 | Enter 3 to produce an ascending list of VENDOR subattribute values (refer to the next prompt).                                               |
| ENTER ATTRIBUTE NAME<br>SOURCE                                                                                                    | Enter SOURCE in response to this prompt.                                                                                                     |
| ENTER SUBATTRIBUTE NAME<br>VENDOR                                                                                                 | Enter VENDOR in response to this prompt.                                                                                                     |
| DECIMAL PLACES = 0                                                                                                                | Enter 0 to tell the system that you do not want decimal places used for the report entries in this column.                                   |
| ENTER COLUMN HEADING 4<br>VENDOR NAME                                                                                             | Enter VENDOR NAME as the heading of the fourth column of the report.                                                                         |
| COLUMN CONTENTS<br>1.ATTRIBUTE NAME<br>2.SUBATTRIBUTE NAME<br>3.SUBATTRIBUTE VALUE<br>4.LITERAL STRING<br>5.QUANTITY<br>6.TOTAL 2 | Enter 2 to specify that the items in the column are subattribute names. The next prompt specifies the attribute name of these subattributes. |
| ENTER COLUMN HEADING 5<br>]                                                                                                       | Enter ] to end column heading input.                                                                                                         |
| ENTER ATTRIBUTE NAME<br>MANUF                                                                                                     | Enter MANUF in response to this prompt.                                                                                                      |
| 1.CHARACTER SIZE=0.1250<br>2.LEVEL =0 ]                                                                                           | Enter ] to accept the default character size and level.                                                                                      |

Figure 9-16. Formatting the Data for the Parts List Report (Continued)

*(Continued)*

| Terminal Session          | Comments                                                                                                                                                                       |
|---------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| INDICATE REPORT ORIGIN    | Position the graphics cursor near the left side of the screen (below the bottom of the first report you generated) and select that position using the C key or the tablet pen. |
| DEFINE REPORT AS GROUP? N | Enter N to define your report as individual entities.                                                                                                                          |
| RESCALE? Y                | If your report is partially off the screen, enter Y in response to this prompt.                                                                                                |

**Figure 9-16. Formatting the Data for the Parts List Report**

## Filing and Exiting

Figure 9-17 outlines the steps to end this terminal session and save the work you just did.

| Terminal Session                                                                                                                                                                                                                                                                                            | Comments                                                                                                                                                                                                                                 |
|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ICEMDDN L=0 V=1 D=0.00 P=0 C=1 4                                                                                                                                                                                                                                                                            | If you are using the keyboard, enter 4 to select the File Current Part/Exit ICEM DDN menu.                                                                                                                                               |
| <pre> FILE CURRENT PART/EXIT ICEM DDN 1.FILE - CONTINUE CURRENT PART 2.   - GET DIFFERENT PART 3.   - QUIT SESSION 4.   - SUSPEND SESSION 5.DO NOT FILE - CONTINUE FROM LAST FILE 6.           - GET DIFFERENT PART 7.           - QUIT SESSION 8.           - SUSPEND SESSION 3                     </pre> | <p>If you are using the tablet, you can file your part and exit from ICEM DDN in a single step by selecting the FILE/QUIT square with your tablet pen.</p> <p>Select 3.FILE - QUIT SESSION to save your part and exit from ICEM DDN.</p> |
| ICEMDDN TERMINATED.                                                                                                                                                                                                                                                                                         |                                                                                                                                                                                                                                          |
| /                                                                                                                                                                                                                                                                                                           |                                                                                                                                                                                                                                          |

Figure 9-17. Sample File/Exit Sequence



# Creating a Three-Dimensional Bracket

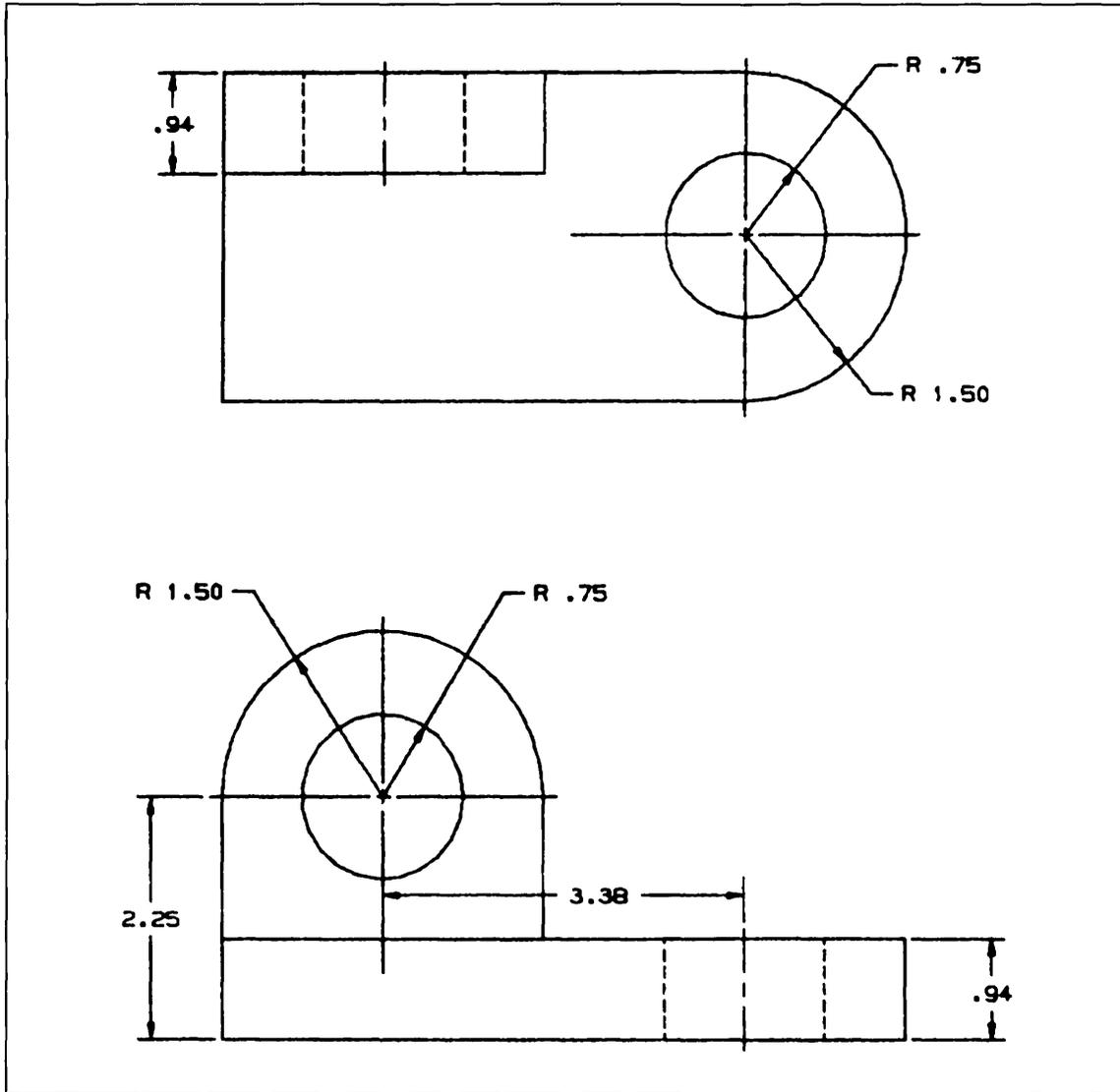
---

10

|                                                          |       |
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| Defining the Base Point .....                            | 10-3  |
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In this practice session, you will use many of the basic construction techniques you have already learned as well as some new techniques to create the three-dimensional bracket shown in figure 10-1.



**Figure 10-1. Three-Dimensional Bracket**

Because you will be referring to figure 10-1 frequently throughout this practice session, you may wish to remove this page and place it next to you as you work through the session.

## Getting Started

1. Log in using the procedures described in the introduction to this manual.
2. Create a new part. The sample terminal session in figure 10-2 helps to illustrate this procedure.

| Terminal Session                                                                                                                                                                                                                                                           | Comments                                                                                                                                                                                                                                                                                                        |
|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <pre> ENTER PART NAME BRACKET  SHEET NUMBER = 1  NEW PART ASSUMED ---UNITS OF MEASURE  M 1.METRIC (MM) 2.ENGLISH (INCH) 3.ENGLISH (FOOT/INCH)  2  ---DRAFTING STANDARD 1.ANSI 1982 2.ANSI 1973 3. 4. 5. 6. 7. 8. 9.          1  ICEMDDN  L=0  V=1  D=0.00  P=0  C=0 </pre> | <p>Enter the name and sheet number of your new part. In this example, we called the part BRACKET with a sheet number of 1.</p> <p>First, enter M to turn the menu display on. Next, select menu choice 2 to work in English units (inch).</p> <p>Select menu choice 1 to use 1982 ANSI standard dimensions.</p> |

Figure 10-2. Start Up Procedure for Creating a Bracket

## Defining the Base Point

Define the base point of the bracket at (0,0,0) as follows:

### Keyboard Selection

Select 9.POINT from the main menu.

Select 2.KEY-IN from the Point menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

### Tablet Selection

|                       |
|-----------------------|
| Transform<br>XT,YT,ZT |
| Model<br>X,Y,Z        |

Enter 0,0 to define the base point of the bracket:

- 1.XT = 0
- 2.YT = 0 ]
- 3.ZT =0.0000

Because the point you just created is located at the origin of the screen, part of it is actually off screen. For this reason, the system displays the following prompt:

**ENTITIES DEFINED OFF SCREEN**

Enter ] to return to the Definition Space menu.

Enter ] again to return to the Point menu.

## Changing the Screen Center

To make it easier to see your part on the screen as you create it, move the center of the screen to the approximate location of the point you just created as follows:

### Keyboard Selection

Enter Z to go directly to the Zoom menu.

Select 2.NEW CENTER from the Zoom menu.

### Tablet Selection

| Zoom            |                  |                        |                |
|-----------------|------------------|------------------------|----------------|
| Auto<br>Max-Min | Diag<br>Position | 1/2 X                  | 2X             |
| Save as<br>Base | New<br>Center    | New<br>Center<br>Scale | Enter<br>Scale |

When you define a new center of screen, you are asked to indicate the new screen center:

**INDICATE NEW SCREEN CENTER**

New users are sometimes confused by this prompt. ICEM DDN is simply asking you to indicate the position on the screen that you would like to be the new screen center.

- Position the graphics cursor on or near the point you just created and select that position as the new center of the screen using the C key or the tablet pen.

The point now appears at the center of the screen.

Because you entered the Zoom menu using the Z key, you automatically return to the menu you were in before you used the Zoom function which, in this case, is the Point menu.

## Defining the Front Drill Hole Center Point

Define the center point of the front drill hole as follows:

### **Keyboard Selection**

Select 2.KEY-IN from the Point menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

### **Tablet Selection**

|                       |
|-----------------------|
| Transform<br>XT,YT,ZT |
| Model<br>X,Y,Z        |

From figure 10-1, you can see that the xt- and yt-coordinate values of the front center point are 1.5 and 2.25, respectively. Enter these values in response to the following prompt:

- 1.XT = 1.5
- 2.YT = 2.25 ]
- 3.ZT =0.0000

Enter ] to return to the Definition Space menu.

Enter ] to return to the Point menu.

## Creating Two Arcs

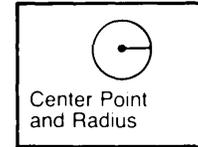
Create two arcs around the point you just created as follows:

### Keyboard Selection

**(CTRL)** A + **(RETURN)**

Select 3.CENTER AND RADIUS from the Arc/Circle/Fillet menu.

### Tablet Selection



- Position the graphics cursor on the point you just created and select it as the center of the circle using the P key or the tablet pen. An attention indicator appears on the point.

Enter .75 as the radius and accept 0 degrees and 360 degrees as the starting and ending angles, respectively, by entering ]:

```
1.RADIUS =0.2500 .75]
2.STARTING ANGLE =0.0000
3.ENDING ANGLE =360.0000
```

Select the same point again as the center point using the P key or the tablet pen.

Enter the following radius, starting angle, and ending angle values:

```
1.RADIUS =0.7500 1.5 or enter 1.5 (TAB) (TAB) 180] (RETURN)
2.STARTING ANGLE =0.0000 (RETURN)
3.ENDING ANGLE =360.0000 180]
```

Enter ] to return to the Arc/Circle/Fillet menu.

The display on your screen should now match the display in figure 10-3.

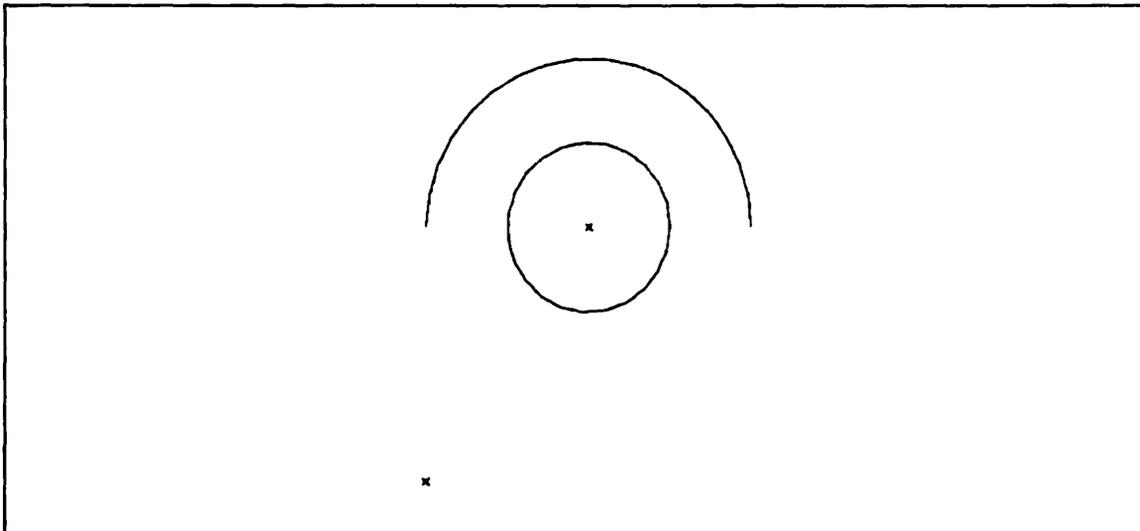


Figure 10-3. Creating Two Arcs

## Completing the Front Plate

The next step is to create the sides of the front plate. These lines are highlighted in blue in figure 10-4.

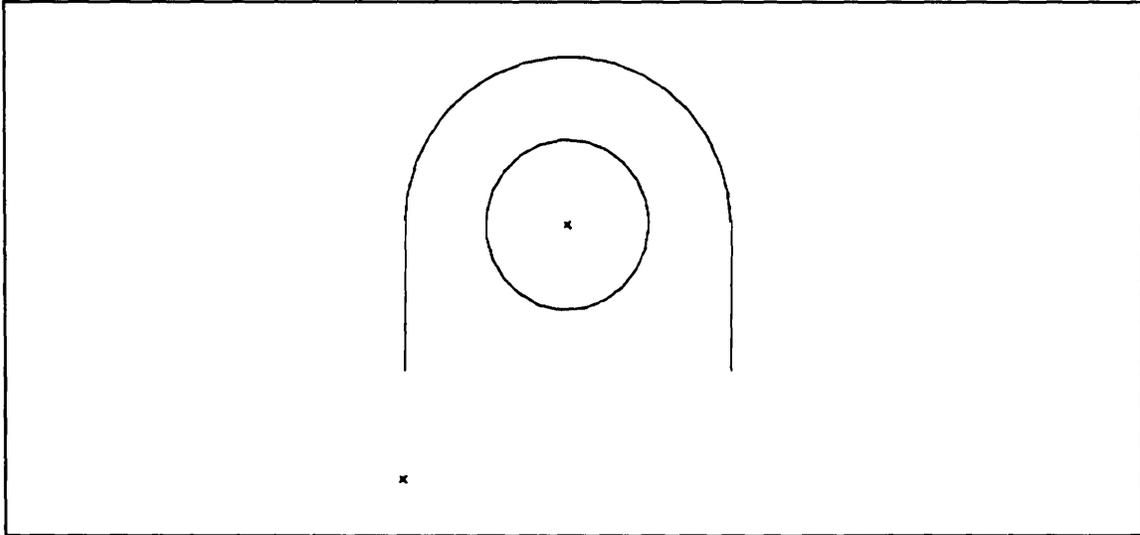


Figure 10-4. The Sides of the Front Plate

Create the sides of the front plate as follows:

### Keyboard Selection

**(CTRL) L + (RETURN)**

Select 2.KEY-IN from the Line menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

### Tablet Selection

|                       |
|-----------------------|
| Transform<br>XT,YT,ZT |
| Model<br>X,Y,Z        |

Enter the following values to create the sides of the plate:

1.XT1 = 0  
 2.YT1 = 2.25  
 3.ZT1 =0.0000  
 4.XT2 = 0  
 5.YT2 = .94 ]  
 6.ZT2 =0.0000

1.XT1 = 3  
 2.YT1 = .94  
 3.ZT1 =0.0000  
 4.XT2 = 3  
 5.YT2 = 2.25 ]  
 6.ZT2 =0.0000

Enter ] to return to the Definition Space menu.

Enter ] to return to the Line menu.

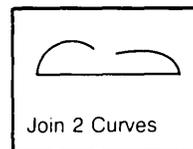
Join the two line segments you just created as follows:

---

**Keyboard Selection**

Select 14.JOIN 2 CURVES from the Line menu.

---

**Tablet Selection**


- Position the graphics cursor *near the bottom* of the left vertical line and select it using the L key or the tablet pen.
- Position the graphics cursor *near the bottom* of the right vertical line and select it using the L key or the tablet pen.

---

**NOTE**

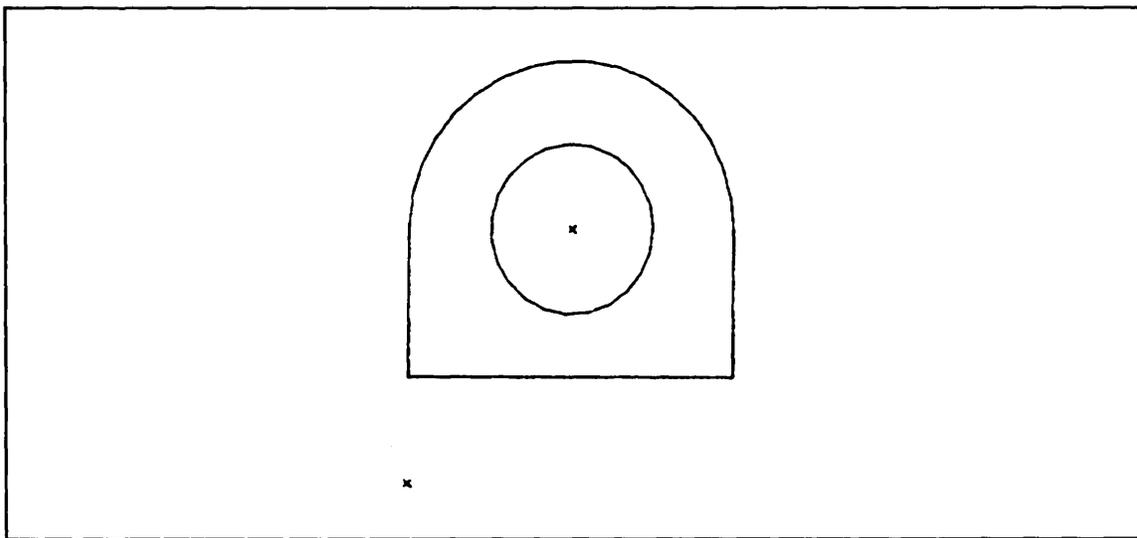
When joining two curves, it is very important that you select the entities you wish to join *near the ends* that you want joined.

---

Enter ] to return to the Line menu.

Enter F to return to the main menu.

The display on your screen should now match the display in figure 10-5.



**Figure 10-5. Completing the Plate**

## Projecting the Front Plate

Project the arcs and lines you just created as follows:

### Keyboard Selection

Select 13.ENTITY MANIPULATION from the main menu.

Select 16.PROJECTED ENTITIES from the Entity Manipulation menu.

### Tablet Selection

| Entity Manipulation |                   |                   |                  |
|---------------------|-------------------|-------------------|------------------|
| Mirror              | Trans             | Rotate            | Rect Array       |
| Dupl Mirror         | Dupl Trans        | Dupl Rotate       | Circular Array   |
| Stretch             | Trans Rotate      | Rotate Trans      | Array Explode    |
| Group               | Dupl Trans Rotate | Dupl Rotate Trans | Project Entities |

In this example, the arcs and lines are projected towards you (positive zt direction) at a distance of .94 inches:

DELTA ZT = 0.0000 .94 (RETURN)

- Position the graphics cursor on the full circle and select it using the A key or the tablet pen.
- Enter (CTRL) E to display the Entity Selection menu.

Select 2.CHAIN from the Entity Selection menu.

- Position the graphics cursor on the right vertical line segment and select it using the L key or the tablet pen.
- Move the graphics cursor a short distance along the outline of the plate and enter C or press down with the tablet pen to indicate the direction for chaining the contiguous curves.

Enter ] to tell ICEM DDN that you have finished selecting entities.

Enter [ to return to the Entity Manipulation menu.

Enter F to return to the main menu.

## Displaying Four Views of the Projected Plate

Display the front plate in four views as follows:

### Keyboard Selection

Select 8.DISPLAY CONTROL from the main menu.

Select 4.DISPLAY MULTIPLE VIEWS from the Display Control menu.

Select 9.VIEWS 6, 1, 3, 8 (TOP/FRONT/RIGHT/ISO 1) to display the plate in the four standard views.

Next, you are asked to select the zoom scale for the display:

#### ZOOM SCALE FOR MULTIPLE VIEWS

- 1.LAST SCALE USED
- 2.BASE SCALE
- 3.SAVED SCALE
- 4.ENTER SCALE
- 5.AUTOMATICALLY MAXIMIZED

Select 5.AUTOMATICALLY MAXIMIZED as the zoom scale.

### Tablet Selection

| Display Views |              |          |           |                 |
|---------------|--------------|----------|-----------|-----------------|
| Top-6         | Front-1      | ISO1-8   | One       | Last Layout     |
| Front-1       | Right-3      |          |           |                 |
| Two Views     |              | Std Four | Std Eight | Last Multi View |
| Above Below   | Side by Side |          |           |                 |

DISPLAYING FOUR VIEWS

The display on your screen should now match the display in figure 10-6.

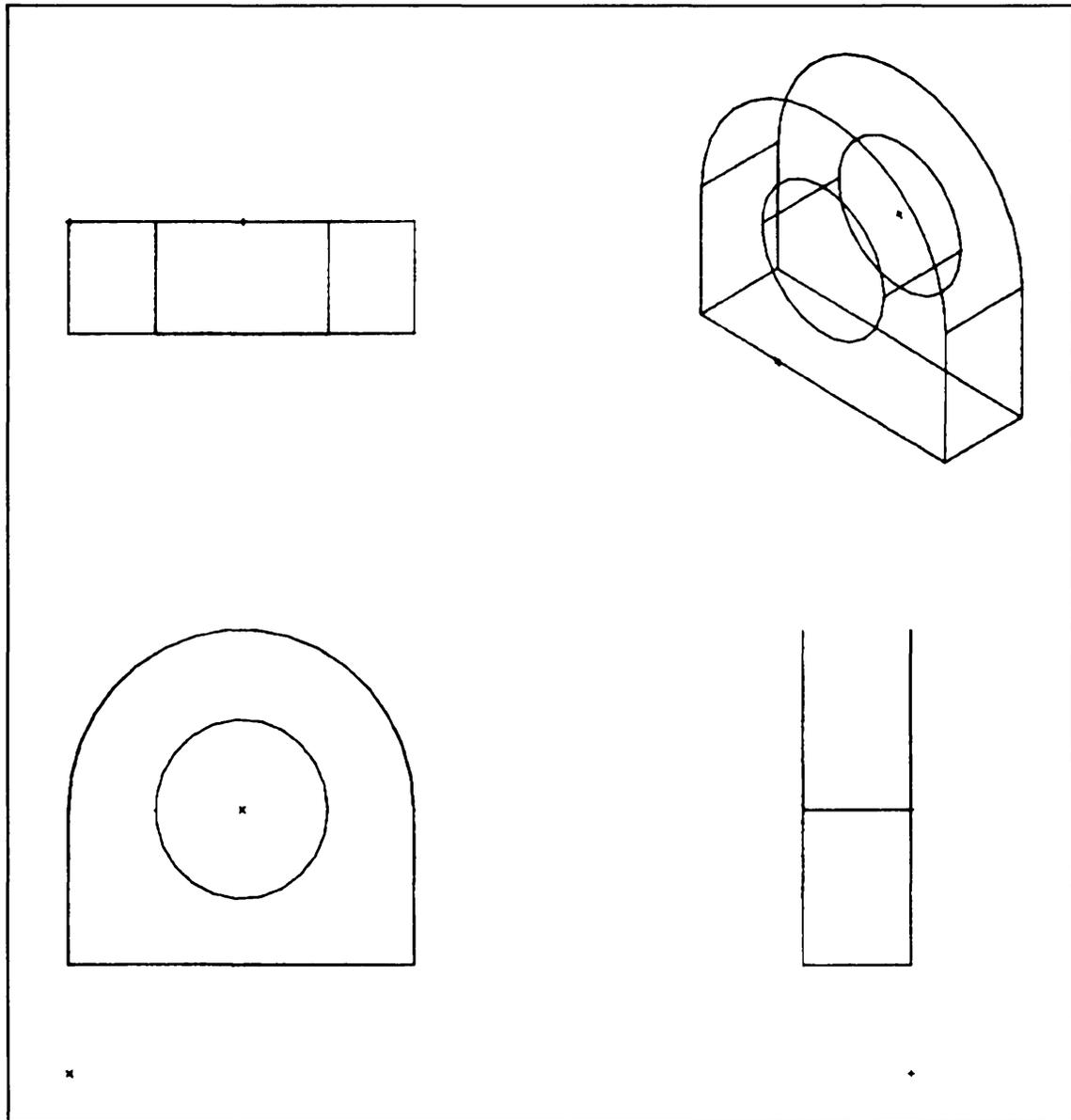


Figure 10-6. Four Views of the Projected Plate

## Changing the Zoom Mode to Entire Layout

The next step is to change the zoom mode to the entire layout. A layout is simply a set of views displayed on the screen in rectangular windows. In changing the zoom mode to the entire layout, you are telling ICEM DDN that you want all future zoom operations to affect the layout *windows* rather than the views themselves.

### Keyboard Selection

Select 6.ZOOM from the Display Control menu.

Select 17.CHANGE ZOOM MODE from the Zoom menu.

Select 1.ENTIRE LAYOUT from the Change Zoom Mode menu.

### Tablet Selection

#### Change Zoom Mode

|                  |              |                  |                |
|------------------|--------------|------------------|----------------|
| Entire<br>Layout | Work<br>View | Select<br>Window | All<br>Windows |
|------------------|--------------|------------------|----------------|

## Adding View Borders to the Display

When you have multiple views displayed on the screen, it is sometimes helpful to add view borders to the display. View borders are rectangular boxes that outline the view windows currently displayed on the screen.

Select 1.MODALS from the Display Control menu.

Select 5.VIEW BORDER DISPLAY MODE from the Modals menu.

Select 3.DISPLAY IN ALL VIEWS from the View Border Display Mode menu.

The view borders are displayed on the screen.

Enter ] to return to the Display Control menu.

## Zooming the Entire Layout

Scale down the entire layout to .75 as follows:

### Keyboard Selection

Select 6.ZOOM from the Display Control menu.

Select 8.ENTER SCALE from the Zoom menu.

### Tablet Selection

| Zoom         |               |                  |             |
|--------------|---------------|------------------|-------------|
| Auto Max-Min | Diag Position | 1/2 X            | 2X          |
| Save as Base | New Center    | New Center Scale | Enter Scale |

Next, you are asked to enter a new scale factor and ratio.

Accept the 1.0000 as the scale factor, but change the ratio to .75 as shown below. The ratio is used to establish a new scale relative to the current scale. In this example, the display scale will be .75 of the current scale factor.

- 1.SCALE = 1.0000 **(RETURN)**
- 2.RATIO = 1.0000 .75

The entire layout is scaled down to .75 of the current scale factor.

The display on your screen should now match the display in figure 10-7.

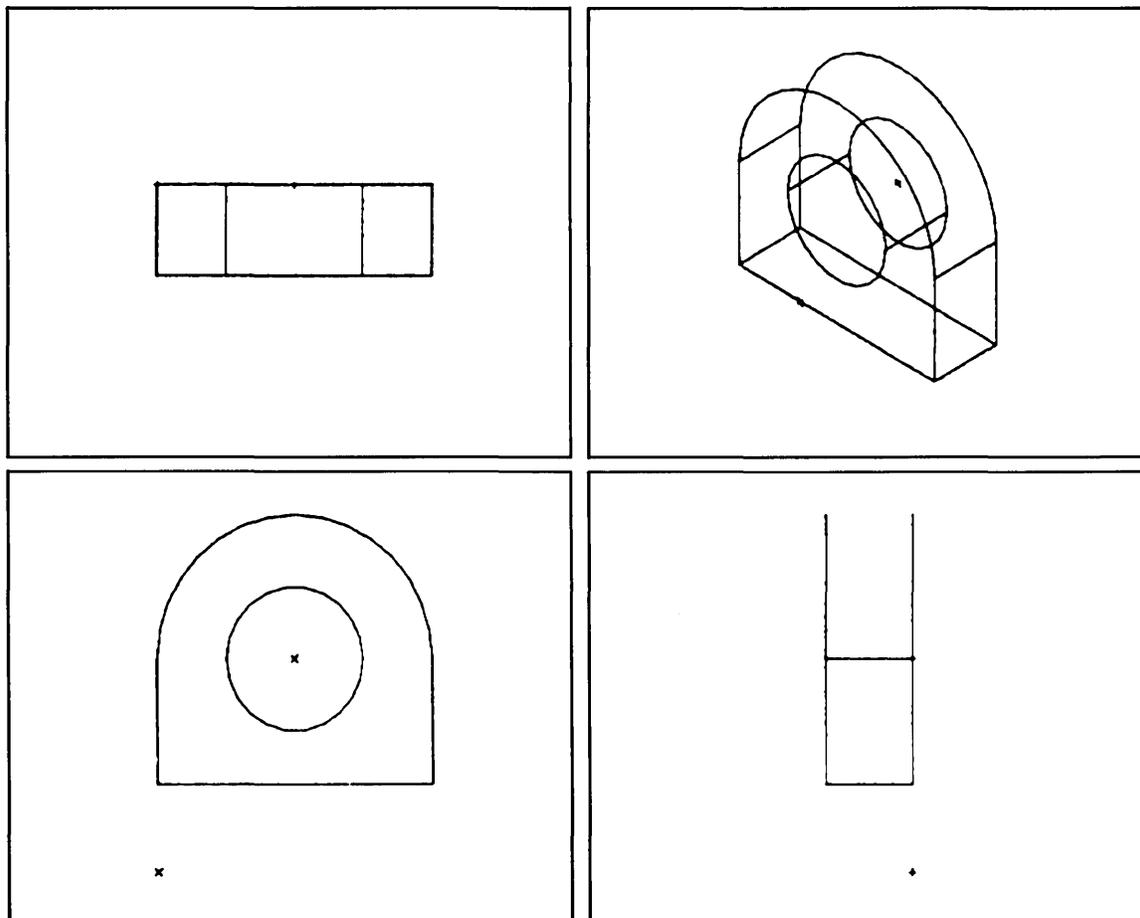


Figure 10-7. Multiple View Display at .75 Scale

## Defining the Center Point of the Bottom Drill Hole

The next step is to define the center point of the bottom drill hole. But before you do that, it is important that you change the work view to view 6, which is the *top view* of the part.

The work view tells you how you are looking at your part. For example, when you change the work view from the front view to the top view, it is as if you are now looking down at the top of the part—even though the display on your screen does not look any different than it did when the front view was the current work view.

If you have only one view of your part displayed on the screen, that view is always the work view. However, when there is more than one view displayed on the screen, as there is now, it is important that you know which of the displayed views is the current work view. If you're not sure, just go to the main menu heading and check to see what V is equal to. At this time, the work view is view 1, which is the front view:

```
ICEMDDN L=0 V=1 D=0.00 P=0 C=0
```

Change the current work view to the top view as follows:

### Keyboard Selection

Select 8.DISPLAY CONTROL from the main menu.

Select 5.CHANGE WORK VIEW from the Display Control menu.

### Tablet Selection

| Display Control   |                 |             |
|-------------------|-----------------|-------------|
| Change Work Space |                 | Chg Wk View |
| Define Work Space | Define Aux View | Align Views |

Enter 6 as the new work view:

```
WORK VIEW IS 1 SELECT ANOTHER
VIEW NUMBER = 6
```

Now define the center point of the bottom drill hole as follows:

### Keyboard Selection

**(CTRL)** P + **(RETURN)**

Select 2.KEY-IN from the Point menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

### Tablet Selection

|                       |
|-----------------------|
| Transform<br>XT,YT,ZT |
| Model<br>X,Y,Z        |

From figure 10-1, you can see that the xt- and yt-coordinate values of this point are 4.88 and -1.5, respectively. Enter these values in response to the following prompt:

```
1.XT = 4.88
2.YT = -1.5]
3.ZT =0.0000
```

Enter ] to return to the Definition Space menu.

Enter ] to return to the Point menu.

### NOTE

Note that the point you just created is out of the view window in some of the views on the screen. Next, you will learn how to change the zoom mode in order to display all currently defined entities.

Enter F to return to the main menu.

## Changing the Zoom Mode to Affect All Views

Change the zoom mode to display all of the currently defined entities as follows:

### Keyboard Selection

Select 8.DISPLAY CONTROL from the main menu.

Select 6.ZOOM from the Display Control menu.

Select 17.CHANGE ZOOM MODE from the Zoom menu.

Select 3.ALL VIEWS from the Change Zoom Mode menu.

### Tablet Selection

| Change Zoom Mode |           |               |             |
|------------------|-----------|---------------|-------------|
| Entire Layout    | Work View | Select Window | All Windows |

This tells ICEM DDN that you want all future zoom operations to affect all displayed *views* rather than the entire layout.

Now automatically maximize all of the views as follows:

### Keyboard Selection

Select 6.ZOOM from the Display Control menu.

Select 11.AUTO MAX-MINS from the Zoom menu.

### Tablet Selection

| Zoom         |               |                  |             |
|--------------|---------------|------------------|-------------|
| Auto Max-Min | Diag Position | 1/2 X            | 2X          |
| Save as Base | New Center    | New Center Scale | Enter Scale |

The display on your screen should now match the display in figure 10-8.

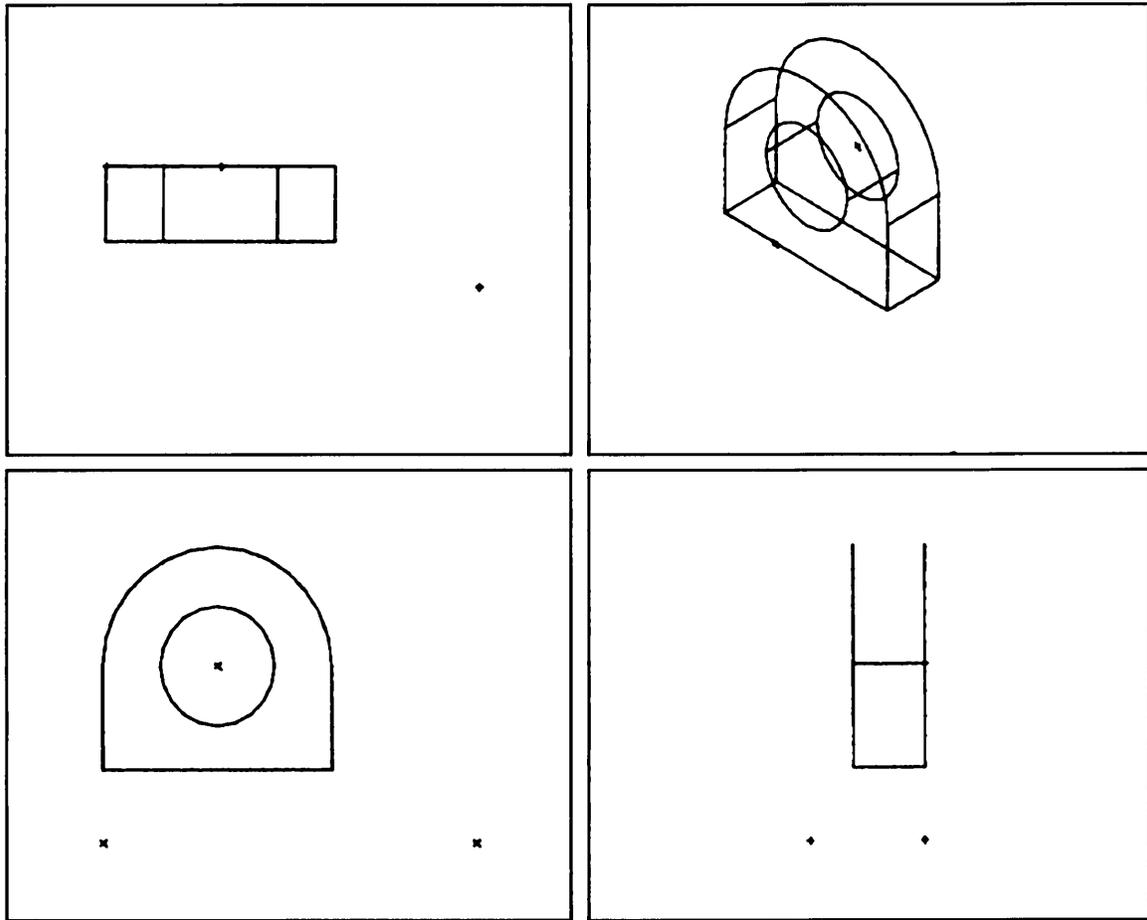


Figure 10-8. Automatically Maximized View Display

## Creating the Remaining Arcs

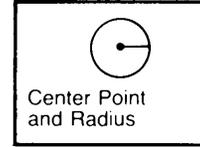
Create two arcs around the point you just created as follows:

### Keyboard Selection

**(CTRL) A + (RETURN)**

Select 3.CENTER AND RADIUS from the Arc/Circle/Fillet menu.

### Tablet Selection



- Position the graphics cursor on the point you just created and select it as the center of the circle using the P key or the tablet pen. An attention indicator appears on the point.

### NOTE

It may be helpful to select these points in the isometric view. With ICEM DDN, you can select entities in any view that you wish—even if the view is not the current work view. When selecting entities in a view other than the work view, attention indicators appear in both the view you are selecting entities in *and* in the work view.

Enter .75 as the radius and accept 0 degrees and 360 degrees as the starting and ending angles, respectively, by entering ]:

```
1.RADIUS =1.5000 .75]
2.STARTING ANGLE =0.0000
3.ENDING ANGLE =360.0000
```

Select the same point again as the center point using the P key or the tablet pen.

Enter the following radius, starting angle, and ending angle values:

```
1.RADIUS =0.7500 1.5 or enter 1.5 (TAB) 270 (TAB) 90] (RETURN)
2.STARTING ANGLE =0.0000 270
3.ENDING ANGLE =360.0000 90]
```

Enter ] to return to the Arc/Circle/Fillet menu.

Note that the arc you just created is out of the view window in some of the views on the screen. Automatically maximize all of the views to display all of the entities as follows:

### Keyboard Selection

Enter Z to go directly to the Zoom menu.

Select 11.AUTO MAX-MINS from the Zoom menu.

### Tablet Selection

| Zoom         |               |                  |             |
|--------------|---------------|------------------|-------------|
| Auto Max-Min | Diag Position | 1/2 X            | 2X          |
| Save as Base | New Center    | New Center Scale | Enter Scale |

The display on your screen should now match the display in figure 10-9.

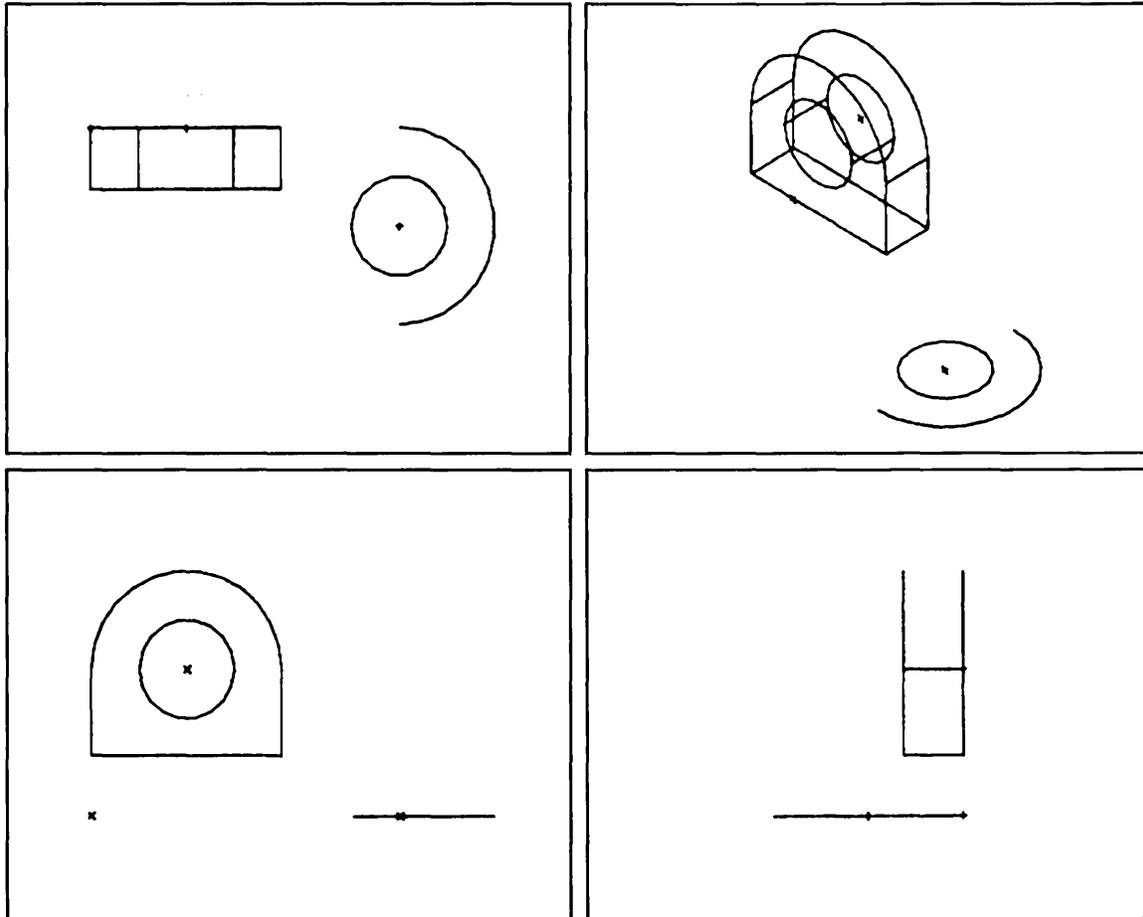


Figure 10-9. Creating the Remaining Two Arcs

## Creating the Front Corner Point

The next step is to create the front corner point of the bracket. This point is highlighted in blue in figure 10-10.

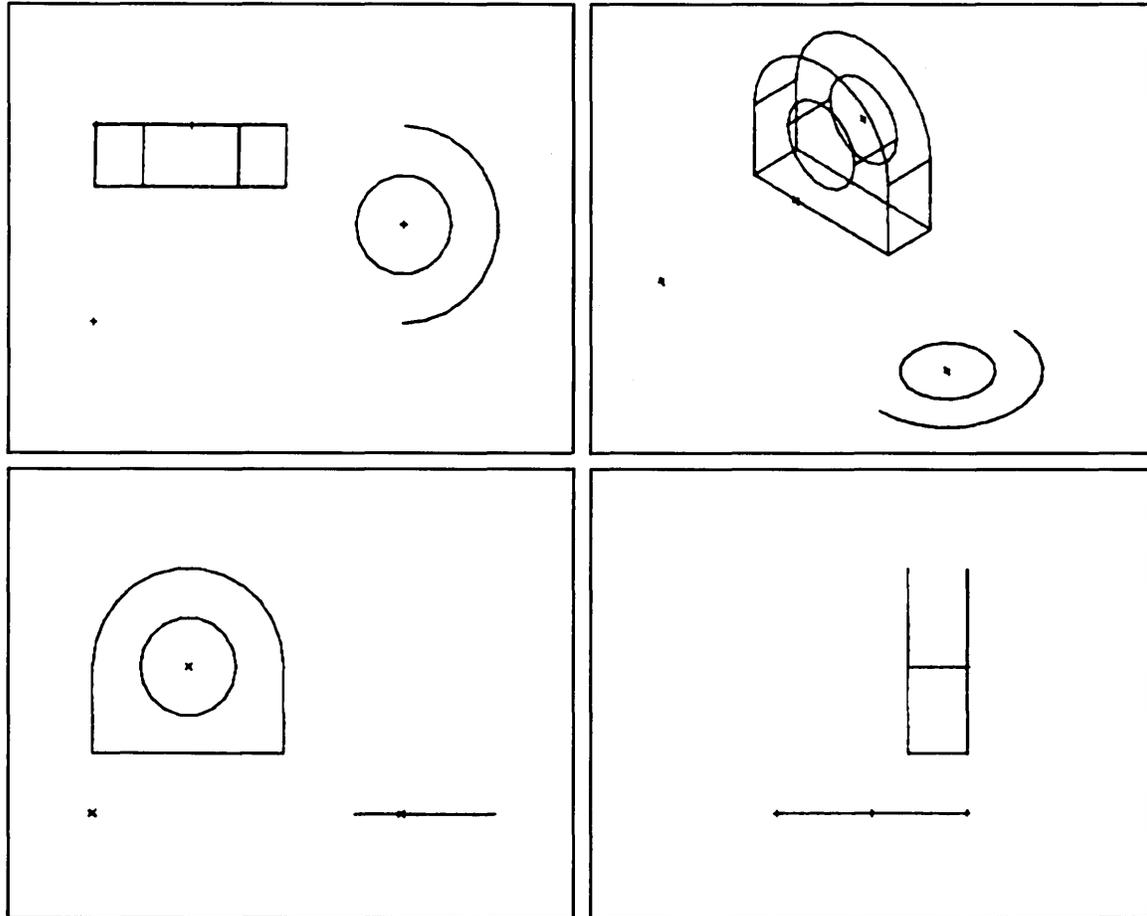


Figure 10-10. Creating the Right Front Corner Point

Create the front corner point as follows:

**Keyboard Selection**

**CTRL** P + **RETURN**

Select 2.KEY-IN from the Point menu.

Select 1.TRANSFORM COORDINATES as the definition space mode.

**Tablet Selection**

|                       |
|-----------------------|
| Transform<br>XT,YT,ZT |
| Model<br>X,Y,Z        |

From figure 10-1, you can see that the xt- and yt-coordinate values of this point are 0 and -3, respectively. Enter these values in response to the following prompt:

- 1.XT = 0
- 2.YT = -3 ]
- 3.ZT =0.0000

Enter ] to return to the Definition Space menu.

## Joining the Base Point and the Front Corner Point

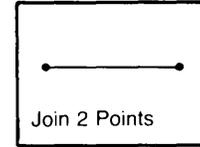
Join the base point and the front corner point as follows:

### Keyboard Selection

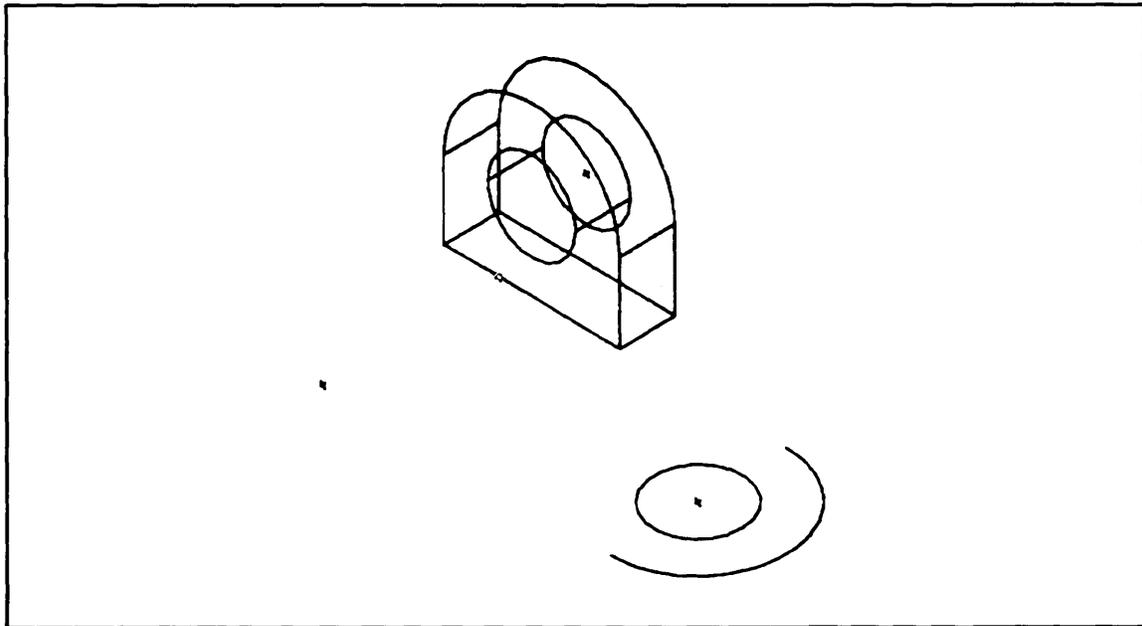
**(CTRL)** L + **(RETURN)**

Select 3.JOIN 2 POINTS from the Line menu.

### Tablet Selection



- Position the graphics cursor on the base point highlighted in figure 10-11 and select it using the P key or the tablet pen.



**Figure 10-11. Joining the Base Point and the Front Corner Point**

- Position the graphics cursor on the front corner point you just created and select it using the P key or the tablet pen.

A line joining these points appears on the screen.

Enter ] to return to the Line menu.

## Completing the Base Outline

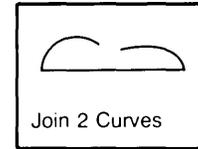
Complete the base outline of the bracket as follows:

### Keyboard Selection

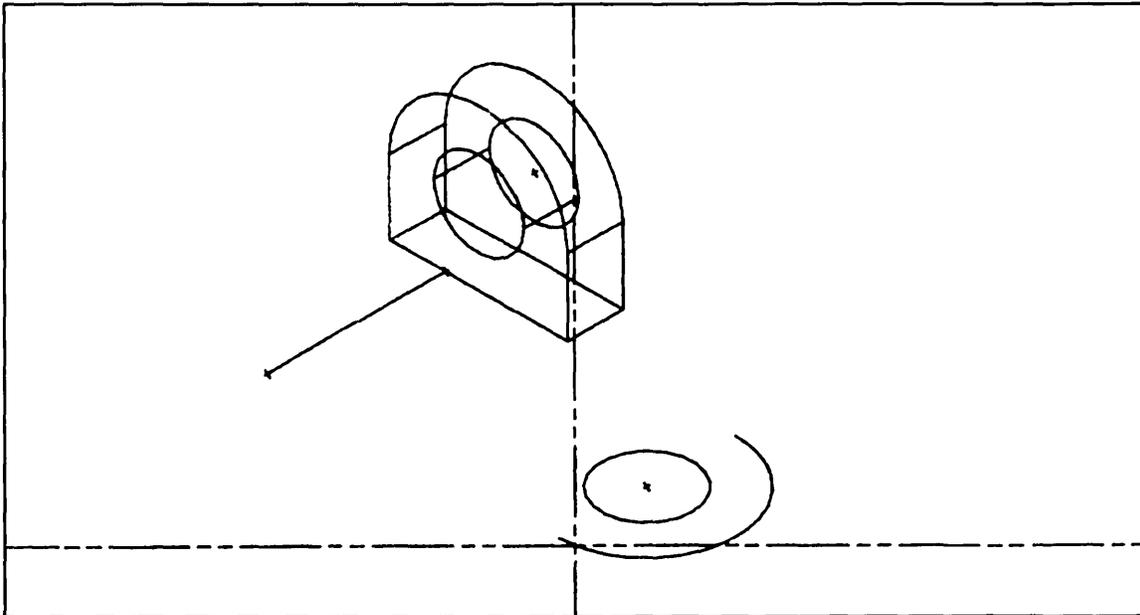
**(CTRL)** L + **(RETURN)**

Select 14.JOIN 2 CURVES from the Line menu.

### Tablet Selection



- Position the graphics cursor *near the bottom end* of the arc as shown in figure 10-12 and select it as the first curve using the A key or the tablet pen.



**Figure 10-12. Graphics Cursor Position for Joining Two Curves**

- Position the graphics cursor on the front corner point and select it as the second curve using the P key or the tablet pen. Remember that a curve can be a point, line, or arc.

A line joining the point and arc you selected appears on your screen.

Using what you just learned, complete the base outline by creating the line opposite the line you just created.

The display on your screen should now match the display in figure 10-13.

Enter F to return to the main menu.

COMPLETING THE BASE OUTLINE

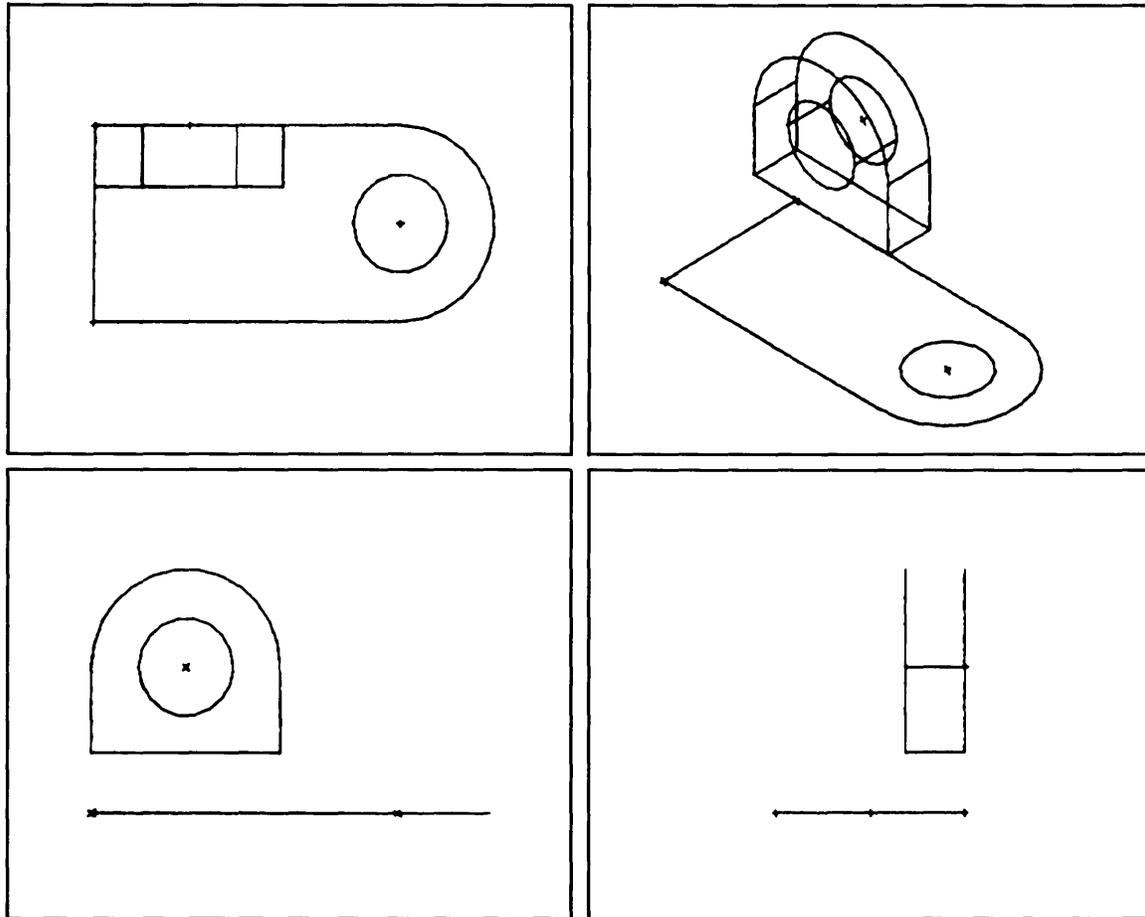


Figure 10-13. Completed Base Outline

## Projecting the Base of the Bracket

Project the base outline of the bracket as follows:

### Keyboard Selection

Select 13.ENTITY MANIPULATION from the main menu.

Select 16.PROJECTED ENTITIES from the Entity Manipulation menu.

### Tablet Selection

| Entity Manipulation |                   |                   |                  |
|---------------------|-------------------|-------------------|------------------|
| Mirror              | Trans             | Rotate            | Rect Array       |
| Dupl Mirror         | Dupl Trans        | Dupl Rotate       | Circular Array   |
| Stretch             | Trans Rotate      | Rotate Trans      | Array Explode    |
| Group               | Dupl Trans Rotate | Dupl Rotate Trans | Project Entities |

Enter RETURN to accept .94 as the delta zt value:

DELTA ZT = 0.9400 (RETURN)

- Position the graphics cursor on the full circle and select it using the A key or the tablet pen.
- Enter (CTRL) E to display the Entity Selection menu.

Select 2.CHAIN from the Entity Selection menu.

- Position the graphics cursor on the larger arc and select it using the A key or the tablet pen.
- Move the graphics cursor a short distance along the outline of the bracket and enter C or press down with the tablet pen to indicate the direction for chaining the contiguous curves.

### NOTE

It is important to remember that when you are using the chain selection method with multiple views displayed on the screen, you must select the first entity in the chain and indicate the direction of the chain selection *in the current work view*, which in this case, is the top view.

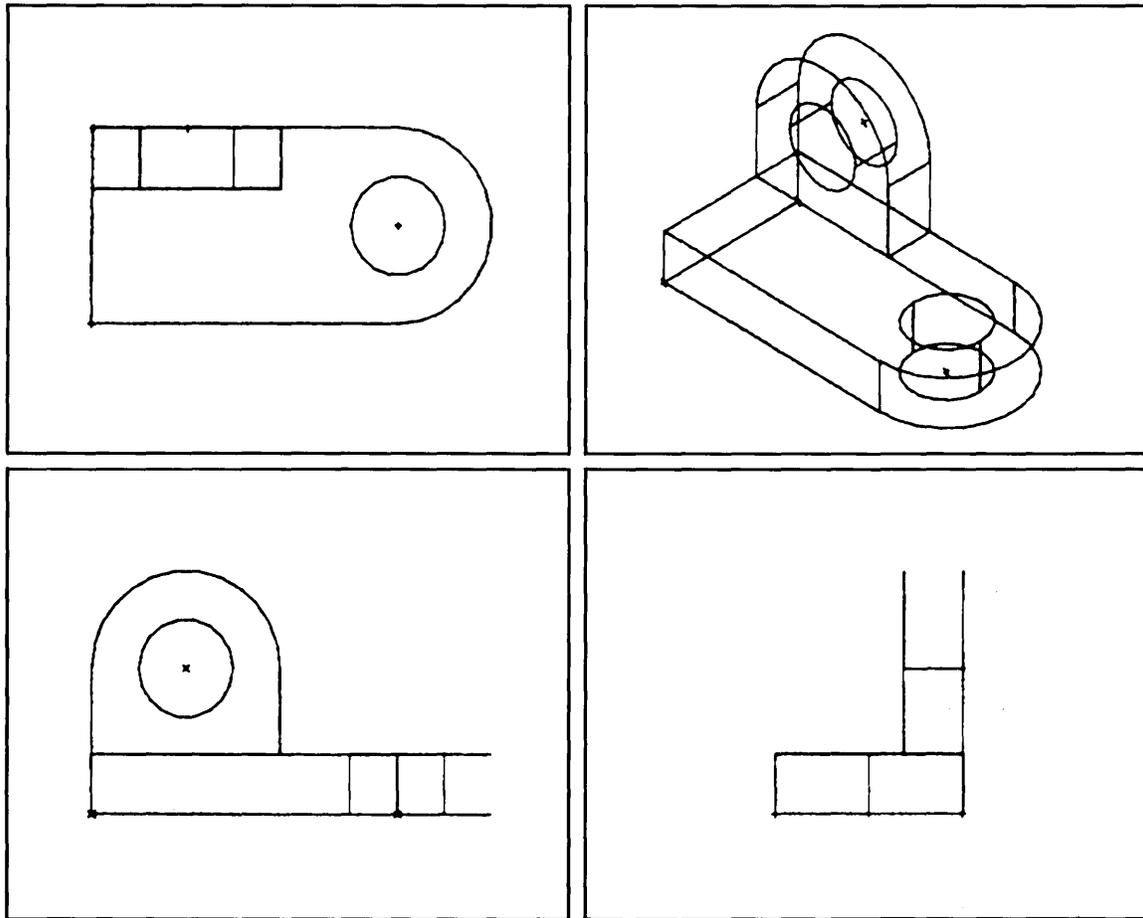
Enter ] to tell ICEM DDN that you have finished selecting entities.

Enter [ to return to the Entity Manipulation menu.

Enter F to return to the main menu.

PROJECTING THE BASE

The display on your screen should now match the display in figure 10-14.



**Figure 10-14. Completed Bracket**

## Filing and Exiting

Figure 10-15 outlines the steps to end this terminal session and save your part.

| Terminal Session                                                                                                                                                                                                                                                                      | Comments                                                                                                                                                                                                                                 |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <pre>ICEMDDN L=0 V=1 D=0.00 P=0 C=1 4</pre>                                                                                                                                                                                                                                           | <p>If you are using the keyboard, enter 4 to select the File Current Part/Exit ICEM DDN menu.</p>                                                                                                                                        |
| <pre>FILE CURRENT PART/EXIT ICEM DDN 1.FILE - CONTINUE CURRENT PART 2.   - GET DIFFERENT PART 3.   - QUIT SESSION 4.   - SUSPEND SESSION 5.DO NOT FILE - CONTINUE FROM LAST FILE 6.           - GET DIFFERENT PART 7.           - QUIT SESSION 8.           - SUSPEND SESSION 3</pre> | <p>If you are using the tablet, you can file your part and exit from ICEM DDN in a single step by selecting the FILE/QUIT square with your tablet pen.</p> <p>Select 3.FILE - QUIT SESSION to save your part and exit from ICEM DDN.</p> |
| <pre>ICEMDDN TERMINATED.  /</pre>                                                                                                                                                                                                                                                     |                                                                                                                                                                                                                                          |

Figure 10-15. Sample File/Exit Sequence

00

0

0

0

0

# Selective View Blanking and Hidden Line Removal

11

---

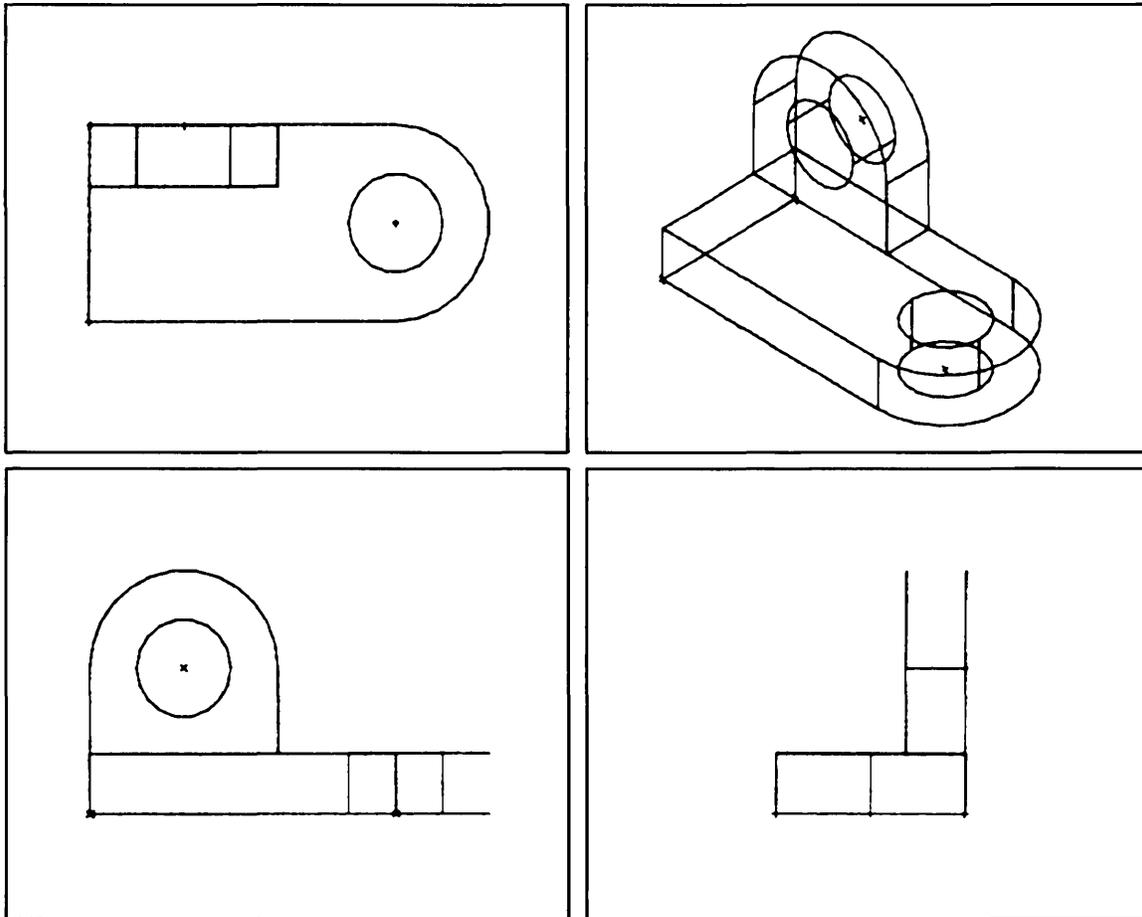
|                                                                                 |       |
|---------------------------------------------------------------------------------|-------|
| Getting Started .....                                                           | 11-2  |
| Creating Points .....                                                           | 11-3  |
| Creating the Curve Edges .....                                                  | 11-5  |
| Creating the Sides of the Drill Holes .....                                     | 11-6  |
| Selective View Blanking .....                                                   | 11-7  |
| Changing the Work View to the Isometric View.....                               | 11-8  |
| Blanking Entities in All Except One View .....                                  | 11-9  |
| Creating Tangent Lines .....                                                    | 11-13 |
| Blanking the Tangent Lines in All Views Except the Isometric View.....          | 11-15 |
| Modifying the Curve Font .....                                                  | 11-16 |
| Blanking Dashed Projection Lines in All Views Except the Top View .....         | 11-17 |
| Blanking Dashed Projection Lines in All Views Except the Front View .....       | 11-18 |
| Blanking the Remaining Projection Lines in All Views Except the Side View ..... | 11-19 |
| Deleting Unnecessary Projection Lines.....                                      | 11-20 |
| Deleting All Points.....                                                        | 11-21 |
| Hidden Line Removal .....                                                       | 11-22 |
| Displaying a Single View .....                                                  | 11-22 |
| Removing One End of a Line .....                                                | 11-23 |
| Removing Entire Entities .....                                                  | 11-26 |
| Removing One End of Two Arcs .....                                              | 11-28 |
| Removing Two Ends of a Circle.....                                              | 11-30 |
| Deleting All Points.....                                                        | 11-33 |
| Displaying the Four Standard Views of the Bracket.....                          | 11-34 |
| Filing and Exiting .....                                                        | 11-35 |



# Selective View Blanking and Hidden Line Removal

As a draftsman, an important part of your job is to make sure that your part drawings reflect acceptable drafting practice.

In looking at the part you created in chapter 10, you'll notice that there is still more work to be done before the part drawing is complete (refer to figure 11-1).



**Figure 11-1. Four Standard Views of the Bracket**

In this practice session, you will learn how to clean up this part using selective view blanking and hidden line removal.

## Getting Started

Retrieve the bracket that you created in chapter 11 as follows:

1. Log in using the procedures described in the introduction to this manual.
2. Retrieve the part named BRACKET, sheet 1. The sample terminal session in figure 11-2 helps to illustrate this procedure.

| Terminal Session                                | Comments                                                                                         |
|-------------------------------------------------|--------------------------------------------------------------------------------------------------|
| ENTER PART NAME BRACKET<br><br>SHEET NUMBER = 1 | Enter the name and sheet number of the part that you want to retrieve (BRACKET, sheet number 1). |

**Figure 11-2. Start Up Procedure for Selective View Blanking and Hidden Line Removal**

## Creating Points

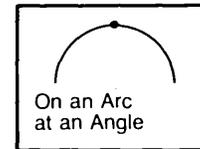
The first step is to add the lines that are missing from the curved edges of the part. To do this, it is first necessary to create the points from which to draw the lines.

### Keyboard Selection

Select 9.POINT from the main menu.

Select 7.ON A CIRCLE from the Point menu.

### Tablet Selection



Next, you are asked to indicate the arc or circle on which the point is to appear:

INDICATE ARC OR CIRCLE

Working in the isometric view, select the arc on the upper portion of the plate as shown in figure 11-3.

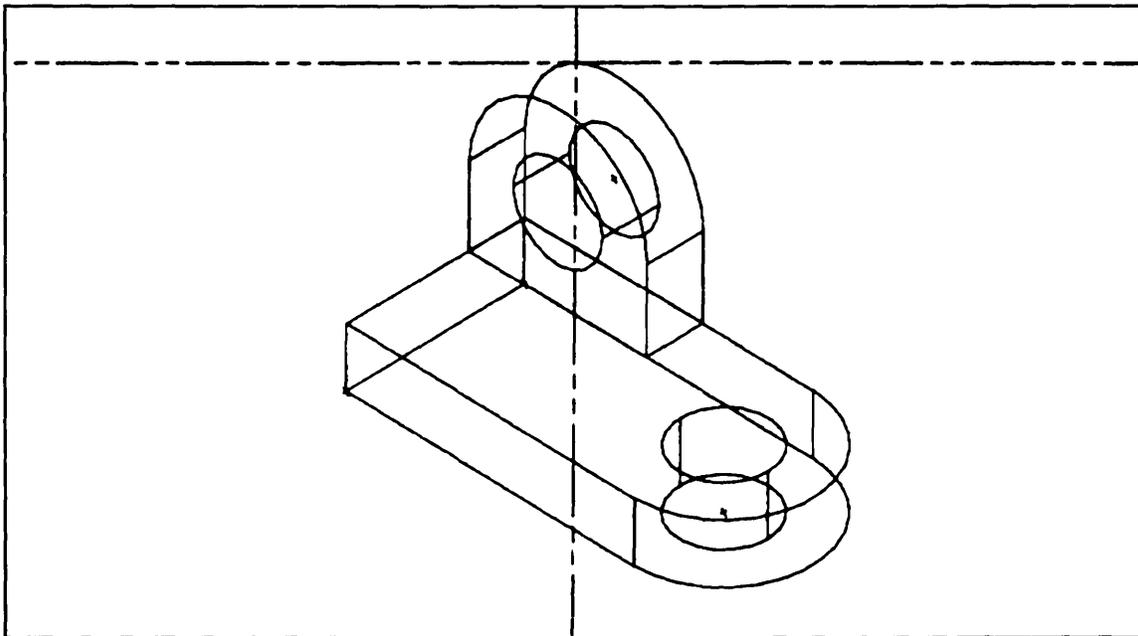


Figure 11-3. Graphics Cursor Position for Selecting the Curve Edges

### NOTE

Remember that when you are selecting entities in a view other than the current work view, attention indicators appear in both views.

## CREATING POINTS

The next prompt asks you where on the arc you would like the point. Since the line you want to draw is at the 90-degree position, enter 90 in response to this prompt:

ANGLE = 90

Enter ] to return to the Point menu.

Repeat this procedure until you have points at each of the positions shown in figure 11-4. Some notes have been added to figure 11-4 to help you decide what angle values to use as you create each of these points. Since all of the arcs you need to select are visible in the isometric view, it is easiest to select them in that view.

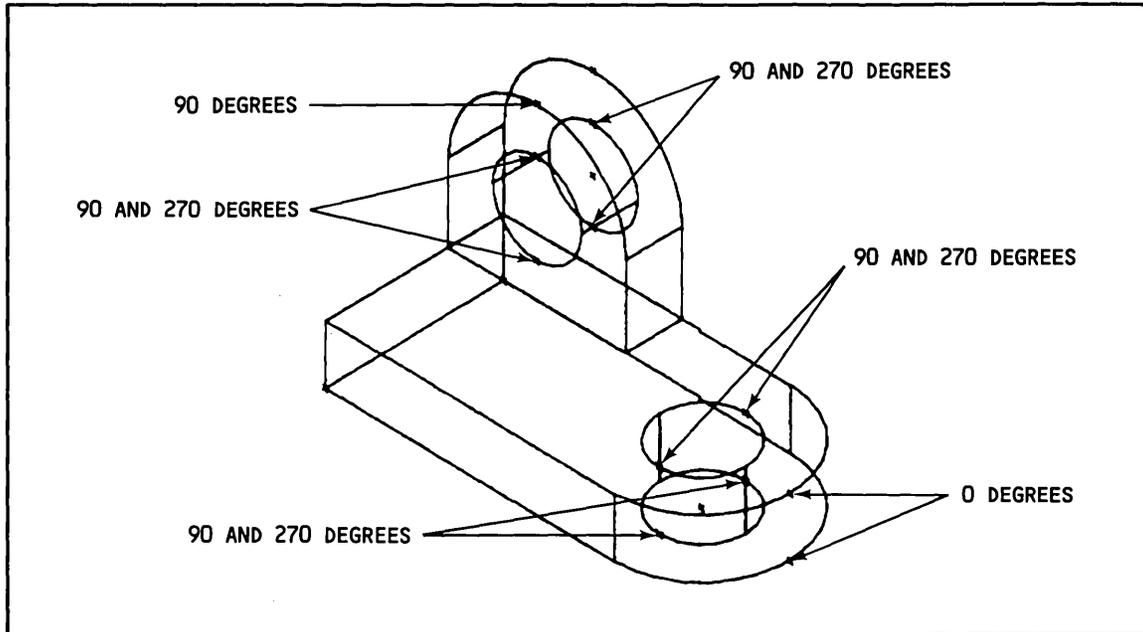


Figure 11-4. Creating Points

## Creating the Curve Edges

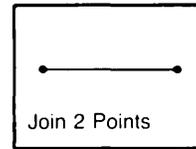
Now that you've created the necessary points, the next step is to connect the points to create the curve edges highlighted in figure 11-5.

### Keyboard Selection

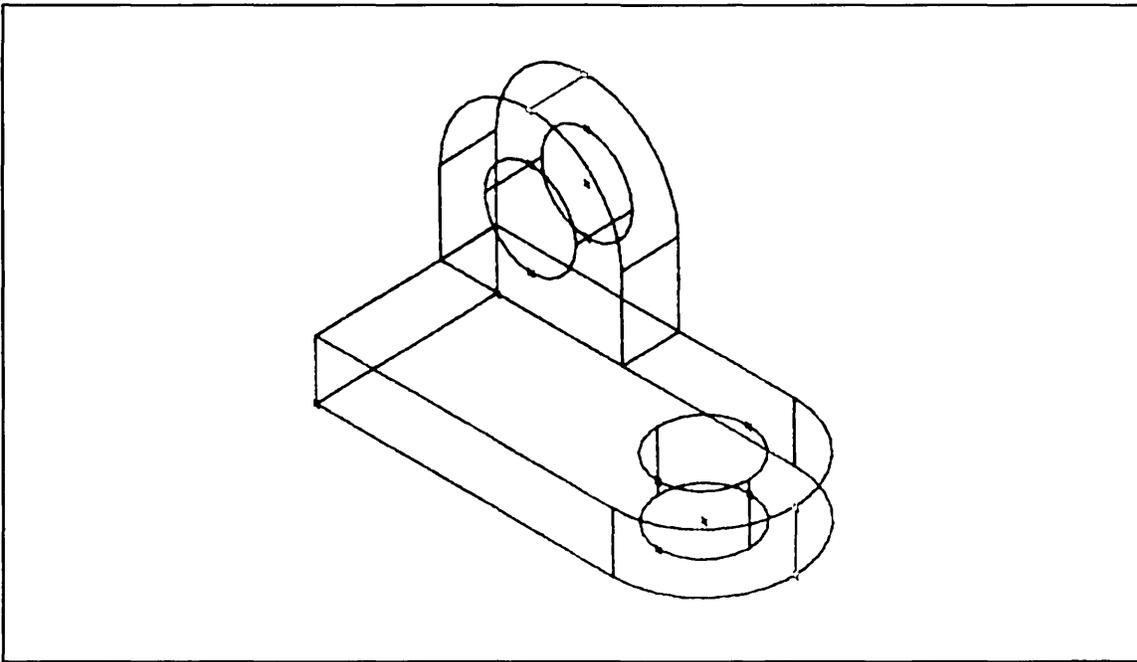
**(CTRL) L + (RETURN)**

Select 3.JOIN 2 POINTS from the Line menu.

### Tablet Selection



- Working in the isometric view, position the graphics cursor on the points needed to create the curve edges and select them using the C key or the tablet pen.



**Figure 11-5. Creating the Curve Edges**

Enter ] to return to the Line menu.

Enter F to return to the main menu.

## Creating the Sides of the Drill Holes

The next step is to create dashed lines at the sides of the drill holes by joining the points you just created.

Before you begin, change the line font to *dashed* to reflect standard drafting conventions.

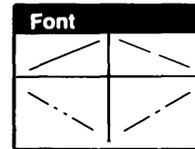
### Keyboard Selection

Select 1.MODALS AND FONTS from the main menu.

Select 5.CURVE FONT from the Modals and Fonts menu.

Select 2.DASHED from the Curve Font menu.

### Tablet Selection



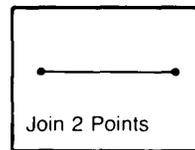
Create the dashed lines as follows:

### Keyboard Selection

**CTRL** L + **RETURN**

Select 3.JOIN 2 POINTS from the Line menu.

### Tablet Selection



- Working in the isometric view, position the graphics cursor on each of the remaining points to create the sides of the drill holes.

When you're finished, the isometric view should match that shown in figure 11-6.

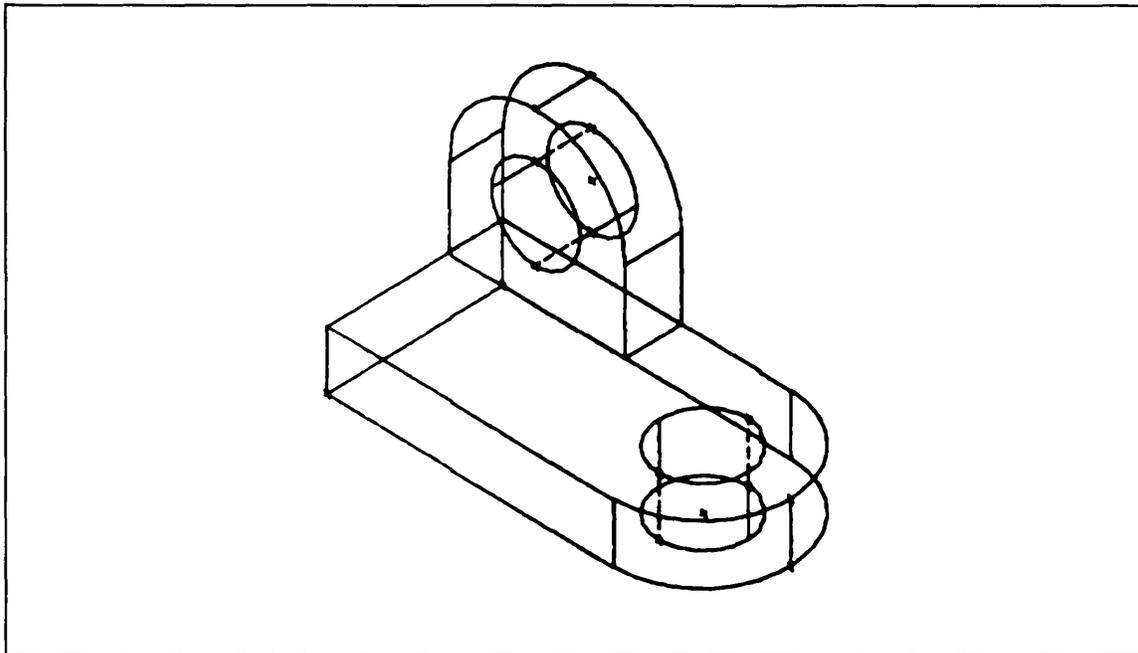


Figure 11-6. Creating the Sides of the Drill Holes

Enter ] to return to the Line menu.

Enter F to return to the main menu.

# Selective View Blanking

Look closely at the four views of the part shown in figure 11-7. Note that while you have created necessary lines in some views—you have also created some *unnecessary* lines in other views. Because these lines are needed in some views, but not in others, you need to use the function called Selective View Blanking.

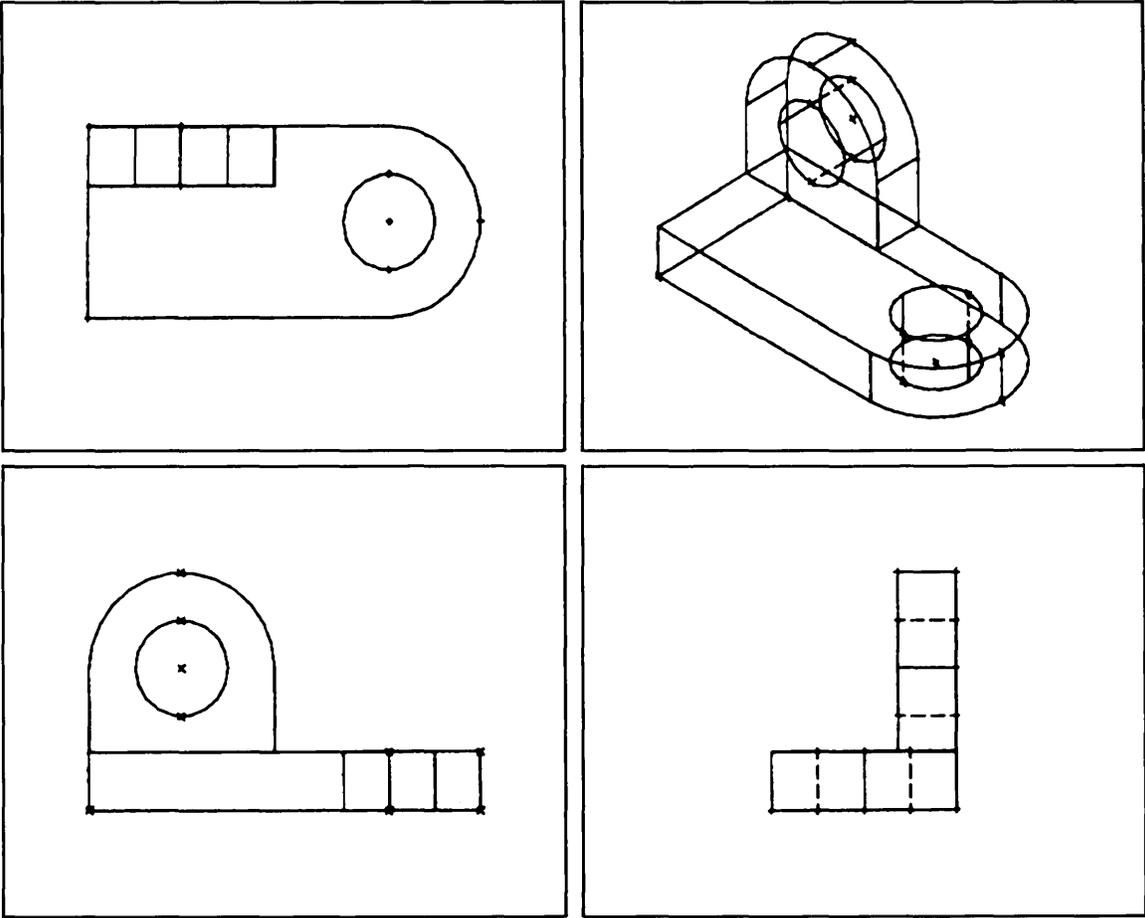


Figure 11-7. Blanking Entities in Selected Views

## SELECTIVE VIEW BLANKING

As the name implies, Selective View Blanking allows you to choose the view or views in which an entity is to be blanked.

To use Selective View Blanking, you must first set the 1.10 BLANK/UNBLANK VIEW SELECT modal to the on position as follows:

### Keyboard Selection

Select 1.MODALS AND FONTS from the main menu.

Select 10.BLANK/UNBLANK VIEW SELECT from the Modals and Fonts menu.

Select 1.ON from the View Selection menu.

### Tablet Selection

| Blank / Unblank |           |
|-----------------|-----------|
| Select Views    | All Views |
| Grid Parameters |           |

### NOTE

For simplicity, we have limited the scope of this practice session to the four standard views: front, top, side, and isometric.

## Changing the Work View to the Isometric View

Since most of the entities that you will selectively blank in this practice session are visible in the isometric view, change the work view from the top view to the isometric view as follows:

### Keyboard Selection

Select 8.DISPLAY CONTROL from the main menu.

Select 5.CHANGE WORK VIEW from the Display Control menu.

### Tablet Selection

| Display Control   |                 |             |
|-------------------|-----------------|-------------|
| Change Work Space | Chg Wk View     |             |
| Define Work Space | Define Aux View | Align Views |

Enter the isometric view as the new work view:

```
WORK VIEW IS 6 SELECT ANOTHER
VIEW NUMBER = 8
```

Enter F to return to the main menu.

## Blanking Entities in All Except One View

The best way to approach Selective View Blanking is to first decide which entities appear in only one view. For example, the line highlighted in figure 11-8 is needed only in the side view.

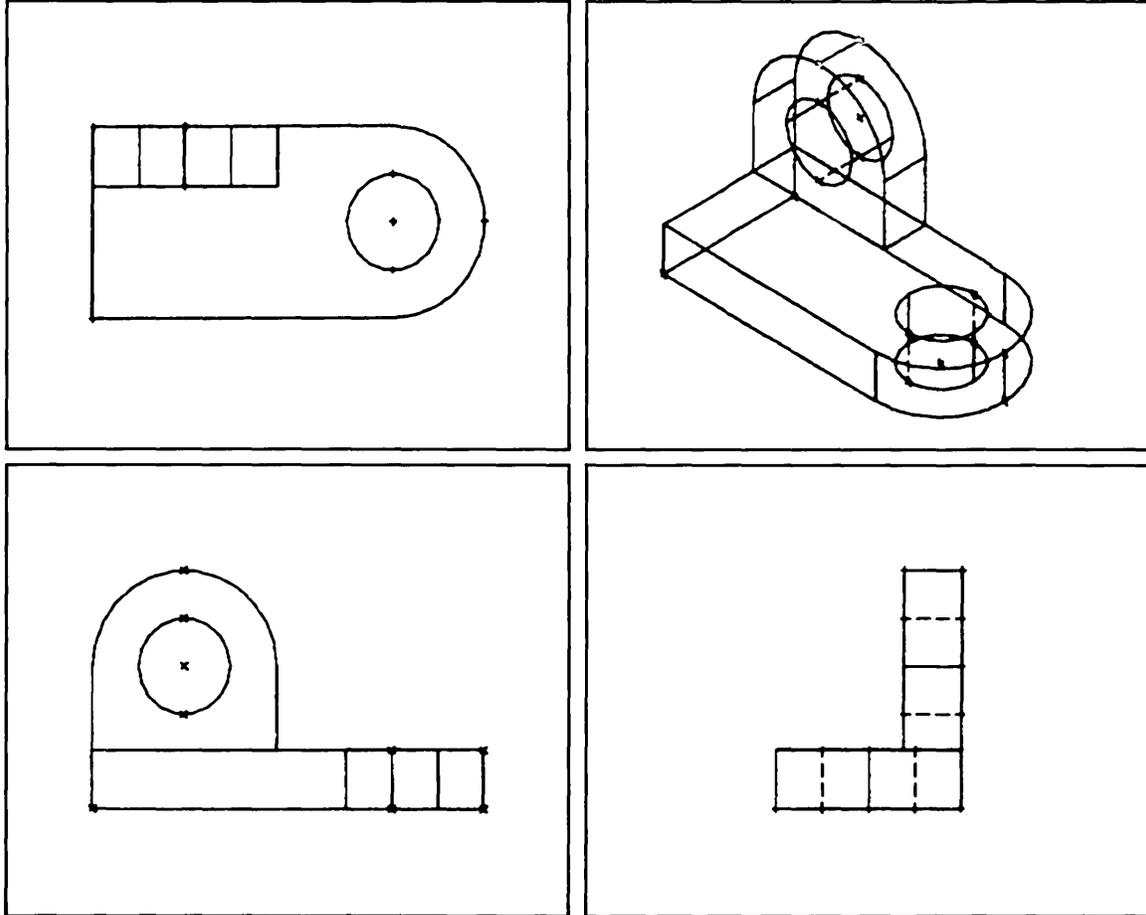


Figure 11-8. Blanking Entities in the Front, Top, and Isometric View

## BLANKING ENTITIES

Blank this line in views 1, 6, and 8 as follows:

### Keyboard Selection

Select 2.BLANK/UNBLANK from the main menu.

Select 1.BLANK - SELECT from the Blank/Unblank menu.

### Tablet Selection

| Blank   |       |     |               |
|---------|-------|-----|---------------|
| Select  | Level | All | All of a Type |
| Unblank |       |     |               |
| Select  | Level | All | All of a Type |

The first prompt is the view selection prompt:

```
VIEW SELECTION
1.RANGE
2.VIEW NUMBER
```

This prompt is asking you whether you want to blank a range of views (for example, views 1 through 6) or individual views (for example, views 1, 6, and 8). Since you are blanking individual views, select 2.

Enter the view numbers 1, 6, and 8 when prompted for them. When you have finished selecting views, enter ] on a line by itself in response to the view number prompt.

```
VIEW NUMBER = 1
VIEW NUMBER = 6
VIEW NUMBER = 8
VIEW NUMBER =]
```

The VIEW SELECTION prompt then redisplay. This gives you another opportunity to select additional views for blanking. Since you have finished selecting views, enter ] in response to this prompt.

- Position the graphics cursor on the line and select it using the L key or the tablet pen.

Enter ] to return to the Blank/Unblank menu.

The entity is blanked in views 1, 6, and 8, but still visible in view 3.

### NOTE

Some terminals may require a local or host repaint to remove blanked entities from the screen. If an entity still appears on your screen after you have blanked it, try repainting the screen using the PAGE key (local repaint) or the R key (host repaint) or touch the REPAINT tablet square in the appropriate place with your tablet pen.

The next step is to blank the line highlighted in figure 11-9 in all views except the front view, which is view 1.

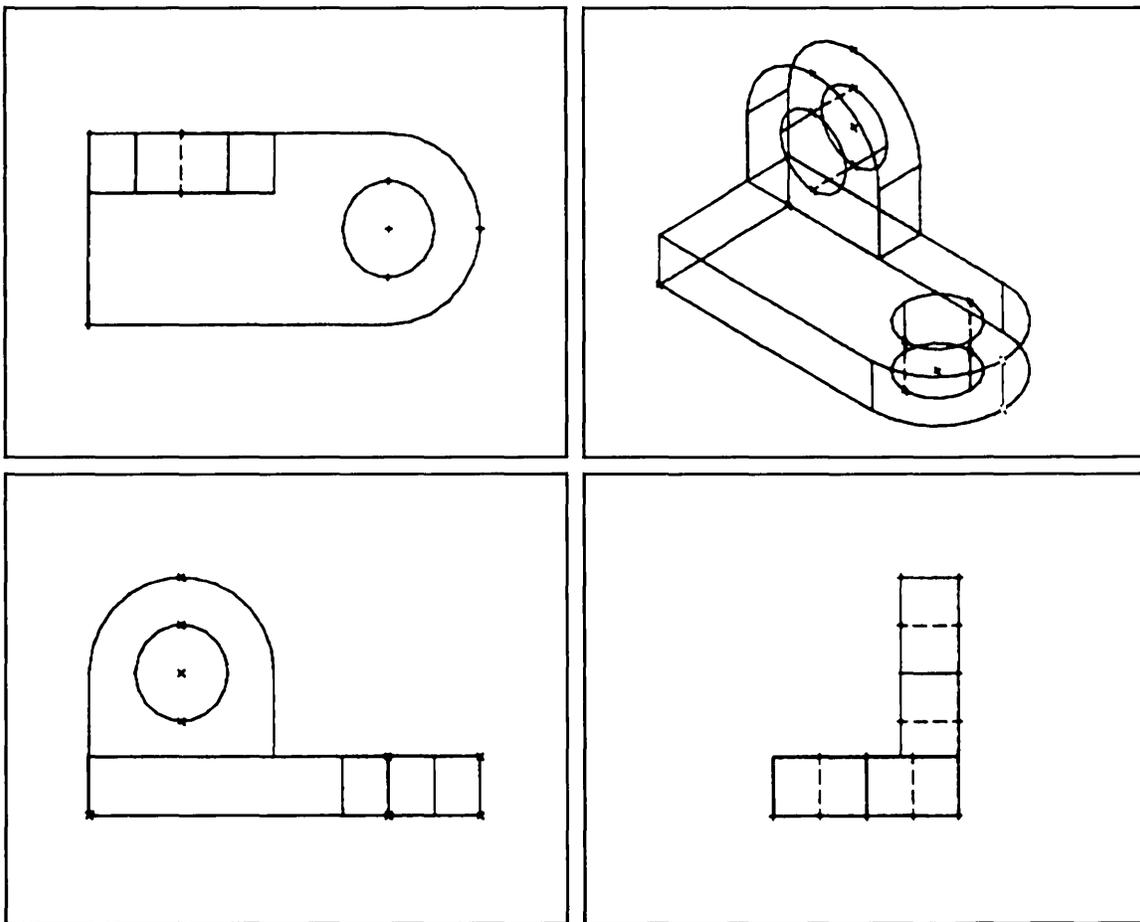


Figure 11-9. Blanking Entities in Views 2 Through 8

BLANKING ENTITIES

Blank this line in views 2 through 8 as follows:

**Keyboard Selection**

Select 1.BLANK - SELECT from the Blank/Unblank menu.

**Tablet Selection**

| <b>Blank</b>   |       |     |               |
|----------------|-------|-----|---------------|
| Select         | Level | All | All of a Type |
| <b>Unblank</b> |       |     |               |
| Select         | Level | All | All of a Type |

This time, select 1.RANGE from the View Selection menu.

Enter the view range followed by ] *on a line by itself* as follows:

- 1. FROM VIEW = 2
- 2. TO VIEW = 8 ]
- 1. FROM VIEW = ]
- 2. TO VIEW =

Enter ] again to tell ICEM DDN that you have finished selecting views.

- Position the graphics cursor on the line and select it using the L key or the tablet pen.

Enter ] to return to the Blank/Unblank menu.

Enter F to return to the main menu.

The entity is blanked in views 2 through 8, but still visible in view 1.

## Creating Tangent Lines

The next step is to create the tangent lines highlighted in figure 11-10.

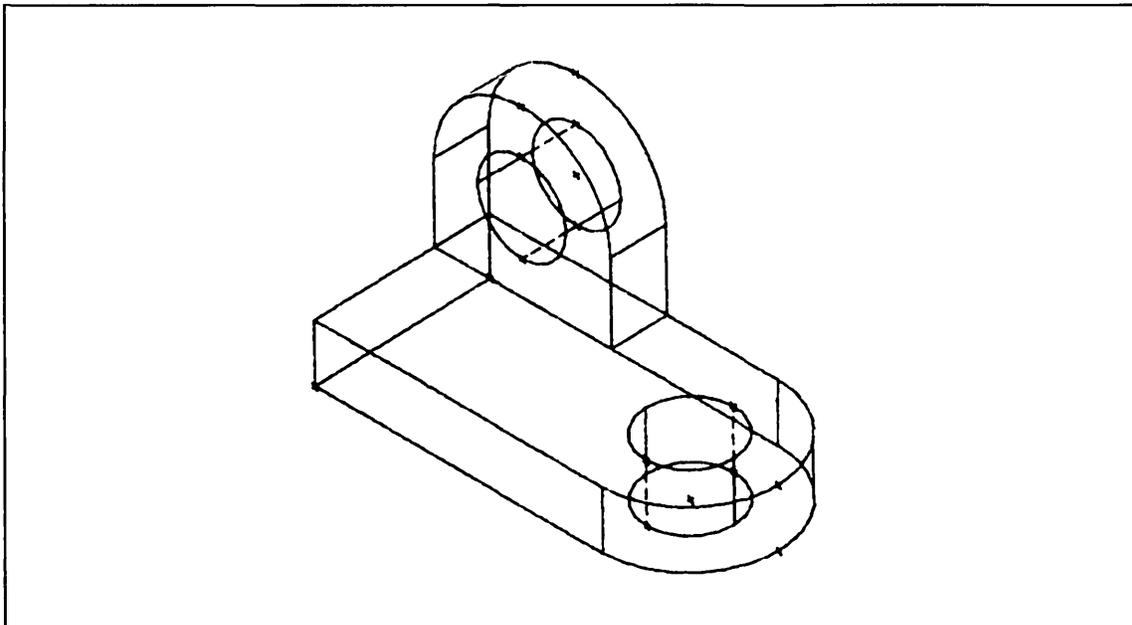


Figure 11-10. Creating Tangent Lines

Before you create the tangent lines, it is necessary to change the line font back to *solid*. Remember that you changed the line font to dashed earlier in this practice session so that you could create the sides of the drill holes.

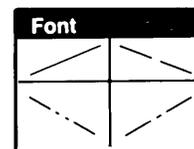
### Keyboard Selection

Select 1.MODALS AND FONTS from the main menu.

Select 5.CURVE FONT from the Modals and Fonts menu.

Select 1.SOLID from the Curve Font menu.

### Tablet Selection



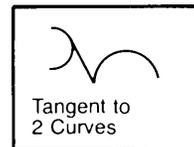
Create the tangent lines as follows:

### Keyboard Selection

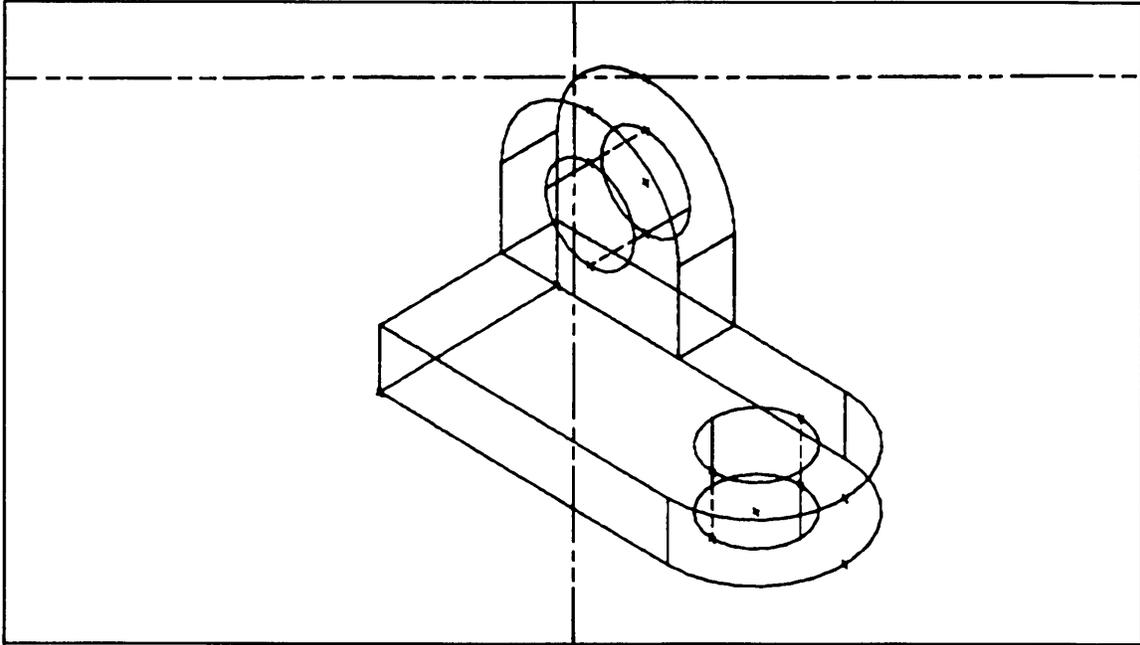
**(CTRL) L + (RETURN)**

Select 4.TANTO 2 CURVES from the Line menu.

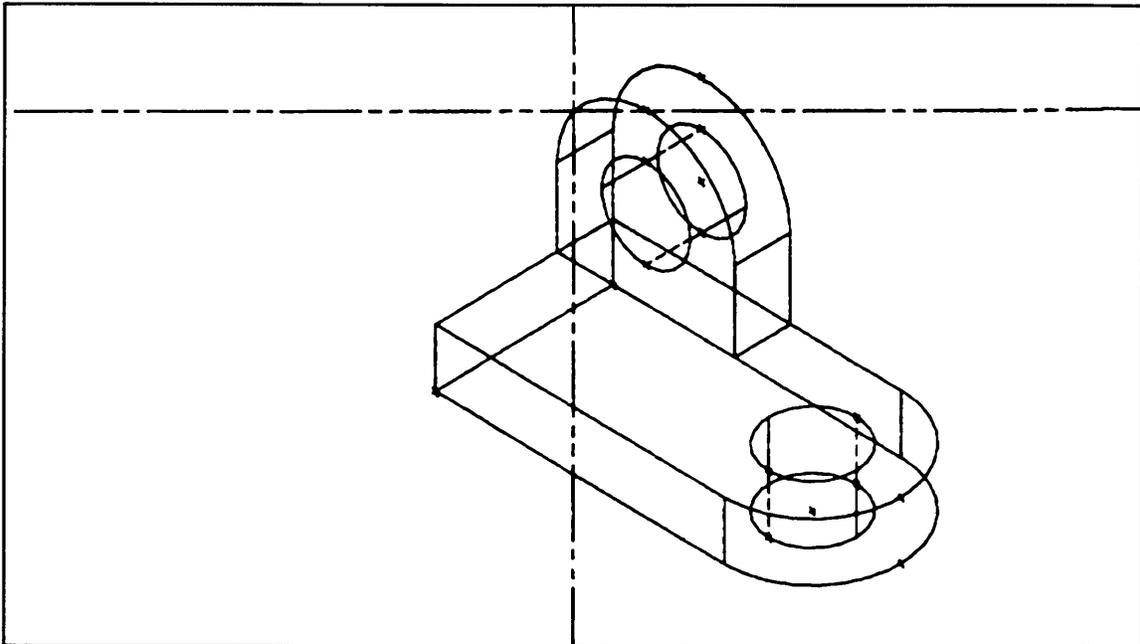
### Tablet Selection



- Working in the isometric view, position the graphics cursor on the arcs you wish to draw the tangent lines between and select them using the A key or the tablet pen. Figures 11-11 and 11-12 show you how to position the graphics cursor to create the top tangent line.



**Figure 11-11. First Graphics Cursor Position for Creating the Top Tangent Line**



**Figure 11-12. Second Graphics Cursor Position for Creating the Top Tangent Line**

Create the bottom tangent line in the same way that you just created the top tangent line.

After you've created both of the tangent lines, enter ] to return to the Line menu.

Enter F to return to the main menu.

## Blanking the Tangent Lines in All Views Except the Isometric View

Blank the tangent lines you just created in all views except the isometric view as follows:

### Keyboard Selection

Select 2.BLANK/UNBLANK from the main menu.

Select 1.BLANK - SELECT from the Blank/Unblank menu.

### Tablet Selection

| <b>Blank</b>   |       |     |               |
|----------------|-------|-----|---------------|
| Select         | Level | All | All of a Type |
| <b>Unblank</b> |       |     |               |
| Select         | Level | All | All of a Type |

Select 1.RANGE from the View Selection menu.

Enter the view range as follows:

- 1.FROM VIEW = 1
2. TO VIEW = 7 ]
- 1.FROM VIEW = ]
2. TO VIEW =

Enter ] again to tell ICEM DDN that you have finished selecting views.

- Working in the isometric view, position the graphics cursor on the tangent lines and select them using the L key or the tablet pen.

Enter ] to return to the Blank/Unblank menu.

Enter F to return to the main menu.

The lines are blanked in views 1 through 7, but still visible in view 8.

## Modifying the Curve Font

At this time, change the solid projection lines highlighted in figure 11-13 to the *dashed* font to reflect standard drafting conventions. These lines were created when you projected the circles that form the drill holes of your bracket.

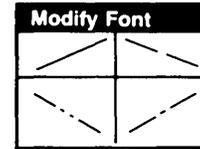
### Keyboard Selection

Select 1.MODALS AND FONTS from the main menu.

Select 6.MODIFY ENTITY FONT from the Modals and Fonts menu.

Select 2.DASHED from the Modify Entity Font menu.

### Tablet Selection



- Working in the isometric view, position the graphics cursor on each of the *solid* projection lines and select them using the L key or the tablet pen.

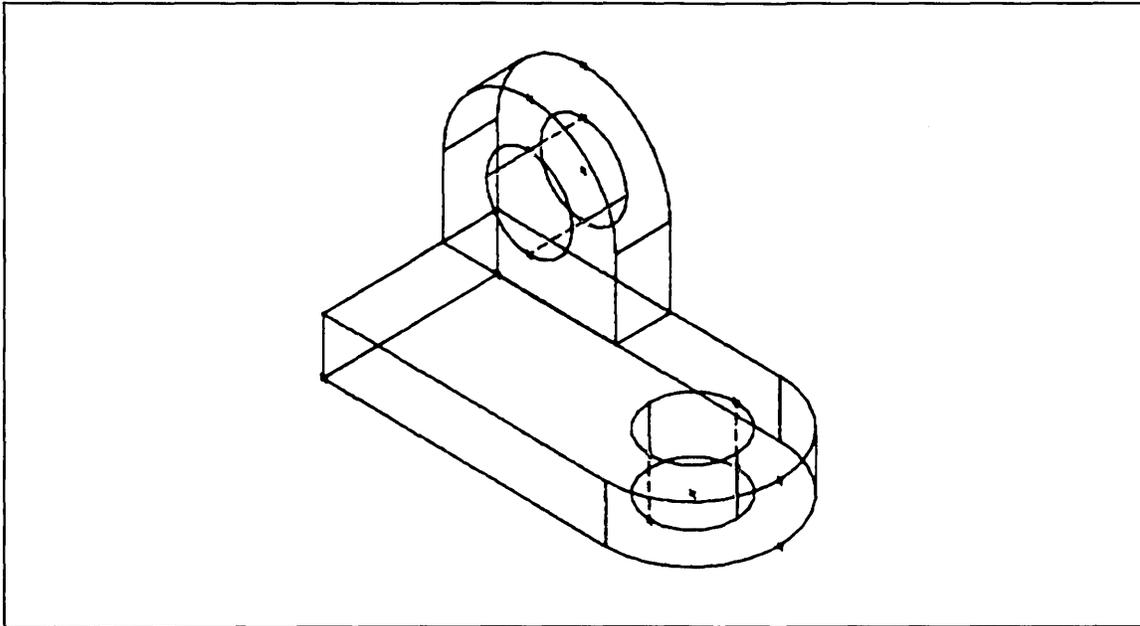


Figure 11-13. Modifying the Curve Font

## Blanking Dashed Projection Lines in All Views Except the Top View

Blank the dashed projection lines highlighted in figure 11-14 in all views except the top view as follows:

### Keyboard Selection

Select 2.BLANK/UNBLANK from the main menu.

Select 1.BLANK - SELECT from the Blank/Unblank menu.

### Tablet Selection

| Blank   |       |     |               |
|---------|-------|-----|---------------|
| Select  | Level | All | All of a Type |
| Unblank |       |     |               |
| Select  | Level | All | All of a Type |

Select 1.RANGE from the View Selection menu.

Enter the view range as follows:

1. FROM VIEW = 1
2. TO VIEW = 5 ]
1. FROM VIEW = 7
2. TO VIEW = 8 ]
1. FROM VIEW = ]
2. TO VIEW =

Enter ] again to tell ICEM DDN that you have finished selecting views.

- Working in the top view, position the graphics cursor on the dashed projection lines highlighted in figure 11-14 and select them using the L key or the tablet pen.

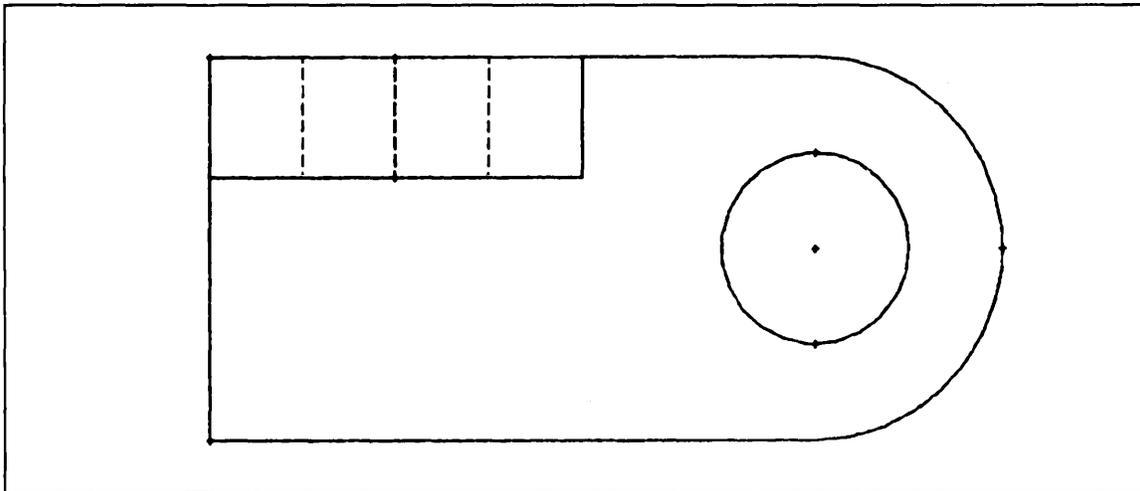


Figure 11-14. Blanking Dashed Projection Lines in All Views Except the Top View

Enter ] to return to the Blank/Unblank menu.

The projection lines are blanked in views 1 through 5, 7, and 8, but still visible in view 6.

## Blanking Dashed Projection Lines in All Views Except the Front View

Blank the dashed projection lines highlighted in figure 11-15 in all views except the front view as follows:

### Keyboard Selection

Select 1.BLANK - SELECT from the Blank/Unblank menu.

### Tablet Selection

| Blank   |       |     |               |
|---------|-------|-----|---------------|
| Select  | Level | All | All of a Type |
| Unblank |       |     |               |
| Select  | Level | All | All of a Type |

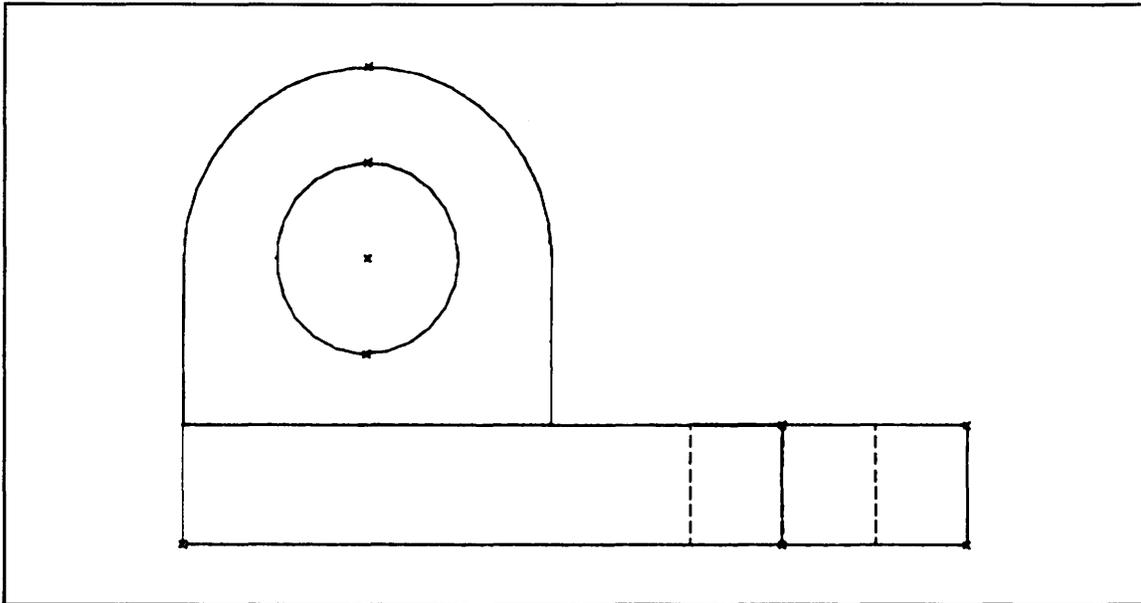
Select 1.RANGE from the View Selection menu.

Enter the view range as follows:

1. FROM VIEW = 2
2. TO VIEW = 8 ]
1. FROM VIEW = ]
2. TO VIEW =

Enter ] again to tell ICEM DDN that you have finished selecting views.

- Working in the front view, position the graphics cursor on the projection lines highlighted in figure 11-15 and select them using the L key or the tablet pen.



**Figure 11-15. Blanking Dashed Projection Lines in All Views Except the Front View**

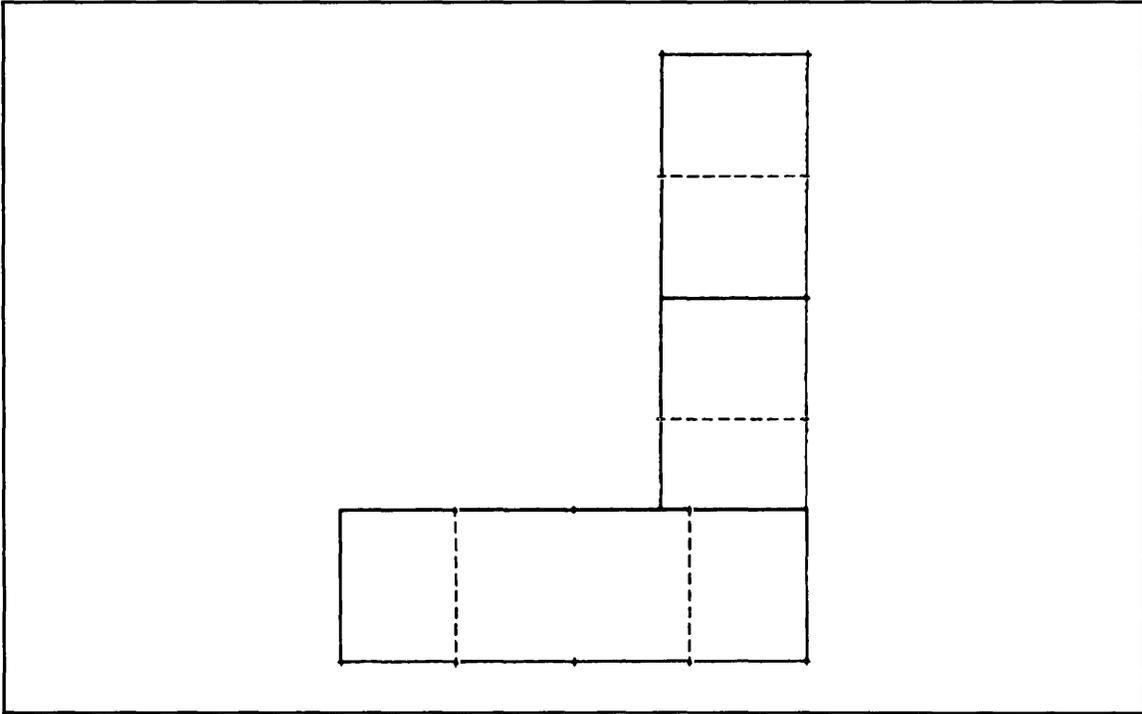
Enter ] to return to the Blank/Unblank menu.

Enter F to return to the main menu.

The projection lines are blanked in views 2 through 8, but still visible in view 1.

## Blanking the Remaining Projection Lines in All Views Except the Side View

Apply what you just learned to blank the projection lines highlighted in figure 11-16 in all views except the side view, which is view 3. If you need help, simply go back to the exercises you just completed.



**Figure 11-16. Blanking Dashed Projection Lines in All Views Except the Side View**

## Deleting Unnecessary Projection Lines

Delete the projection lines not needed in any view as follows:

### Keyboard Selection

Select 3.DELETE from the main menu.

Select 1.SELECT from the Delete menu.

### Tablet Selection

| Delete    |            |            |       |
|-----------|------------|------------|-------|
| Select    |            | Screen     | Chain |
| Region In | Region Out | All Points | Level |

- Working in the isometric view, position the graphics cursor on each of the projection lines highlighted in figure 11-17 and select them using the L key.

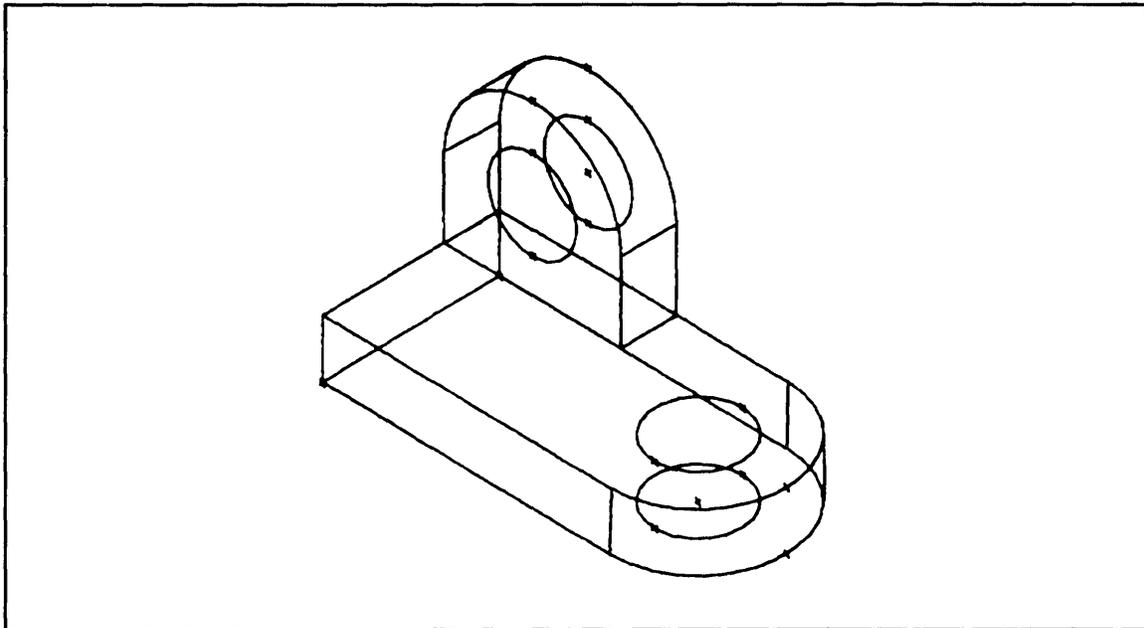


Figure 11-17. Deleting Unnecessary Projection Lines

Enter ] to return to the Delete menu.

## Deleting All Points

Delete all of the points in your part as follows:

### Keyboard Selection

Select 3.ALL POINTS from the Delete menu.

### Tablet Selection

| Delete    |            |            |       |
|-----------|------------|------------|-------|
| Select    |            | Screen     | Chain |
| Region In | Region Out | All Points | Level |

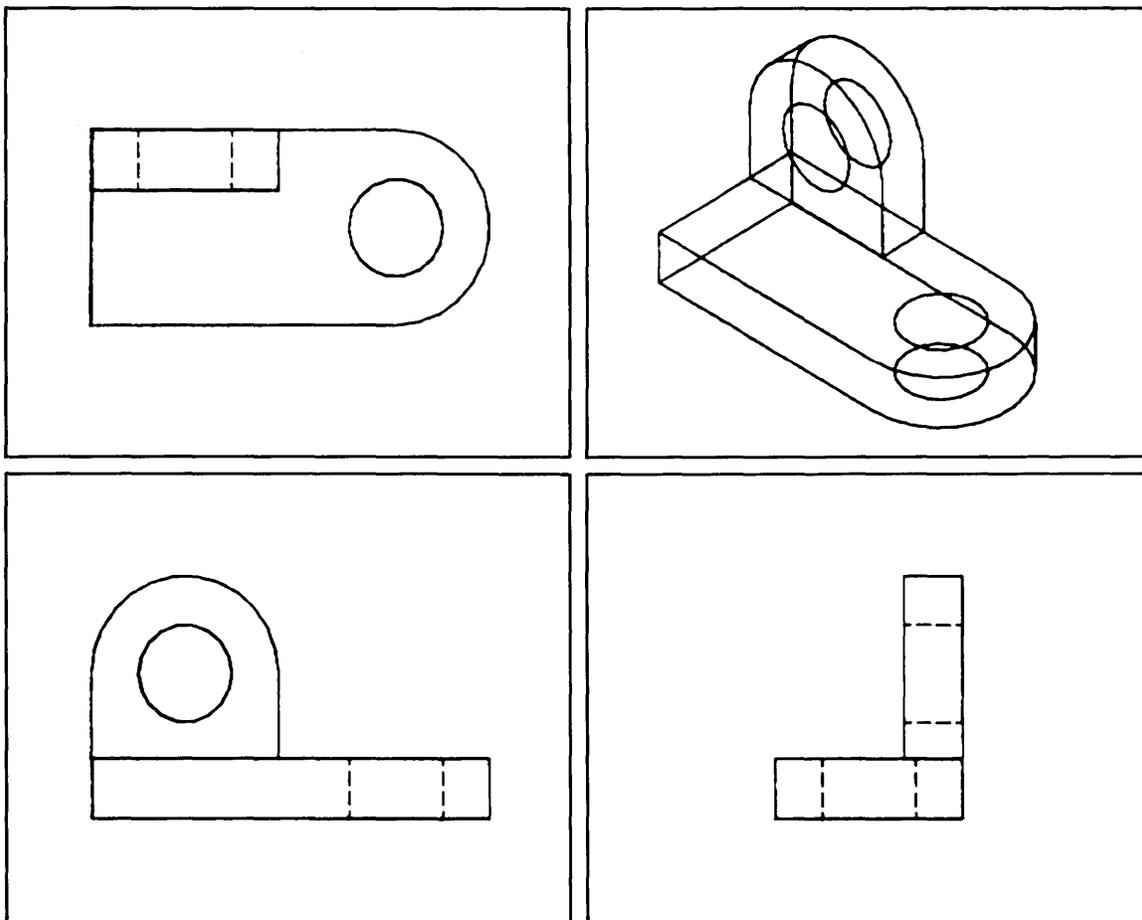
Next, you are asked if you are sure that you want to delete all of the points in your part:

**DELETE ALL POINTS?**

Enter Y in response to this prompt.

All of the points in your part are deleted.

The display on your screen should now match figure 11-18.



**Figure 11-18. Deleting All Points**

## Hidden Line Removal

In the next part of this practice session, you will learn how to use the Hidden Line Removal function. Hidden Line Removal is similar to Trim/Extend Curves in that you can remove one end, two ends, or the middle of an entity from the current work view.

With Hidden Line Removal, the entity that you select is selectively blanked in the current work view and a new entity or entities is created using one of the following line fonts:

- LINE FONT
- 1.SOLID
  - 2.DASHED
  - 3.PHANTOM
  - 4.CENTERLINE
  - 5.BLANKED

If you are using the tablet, the line font selections are part of the Hidden Line Removal block, which is located on the Display Control (DC) tablet overlay.

### Displaying a Single View

Since the rest of the work that you will be doing on the part is only in the isometric view, change the view display to a single view as follows:

#### Keyboard Selection

Select 8.DISPLAY CONTROL from the main menu.

Select 3.DISPLAY A SINGLE VIEW from the Display Control menu.

#### Tablet Selection

| Display Views |              |          |           |                 |
|---------------|--------------|----------|-----------|-----------------|
| Top-6         | Front-1      | ISO1-8   | One       | Last Layout     |
| Front-1       | Right-3      |          |           |                 |
| Two Views     |              | Std Four | Std Eight | Last Multi View |
| Above Below   | Side by Side |          |           |                 |

Enter the number of the view you wish to display. In this case, use the isometric view:

VIEW NUMBER = 8

Attention indicators appear around the isometric view.

Next, you are asked to select the zoom scale you wish to use for the display:

- ZOOM SCALE FOR SINGLE VIEW
- 1.LAST SCALE USED
  - 2.BASE SCALE
  - 3.SAVED SCALE
  - 4.ENTER SCALE
  - 5.AUTOMATICALLY MAXIMIZED

This time, select 4.ENTER SCALE as the zoom scale.

Enter .75 as the new scale factor as shown below:

- 1.SCALE = 1.5188     .75 (RETURN)
- 2.RATIO = 1.0000

The bracket appears in the full display at approximately the same size as it appeared in the view layout window.

Enter F to return to the main menu.

### Removing One End of a Line

The next step is to remove the end of the line highlighted in figure 11-19.

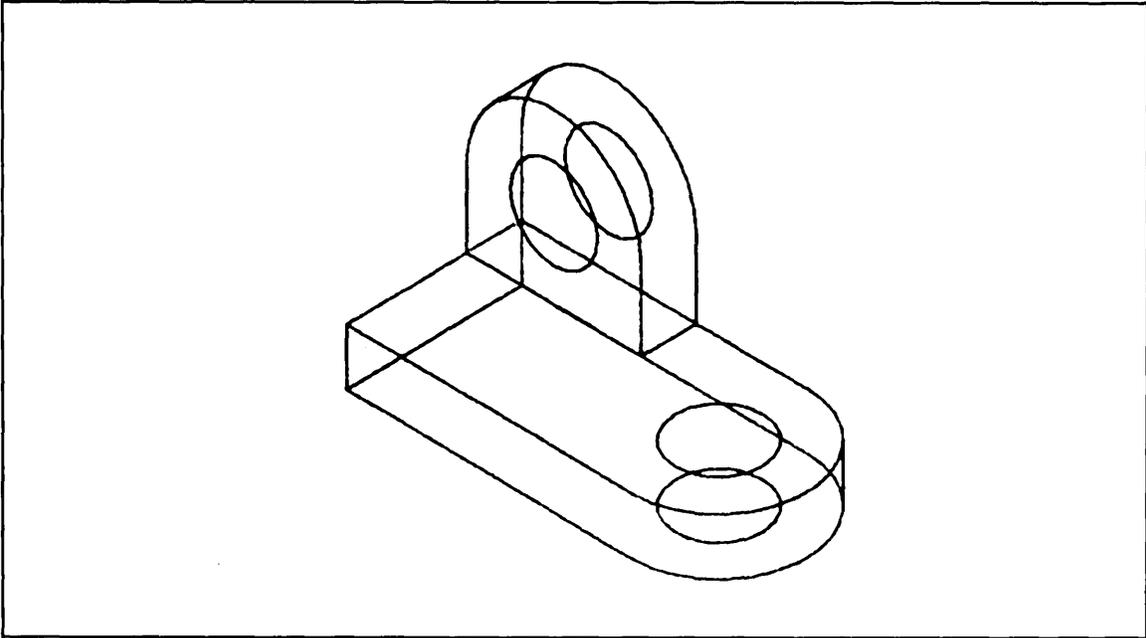


Figure 11-19. Hidden Line Removal of One End of an Entity

**Keyboard Selection**

Select 12.OTHER CURVES from the main menu.

Select 11.HIDDEN LINE REMOVAL from the Other Curves menu.

Select 5.BLANKED from the Line Font menu.

Select 1.ONE END from the Hidden Line Removal Mode menu.

- Position the graphics cursor on the left vertical line of the top plate and select it as the bound using the L key or the tablet pen.
- Position the graphics cursor as shown in figure 11-20 and select it using the L key or the tablet pen.

**Tablet Selection**

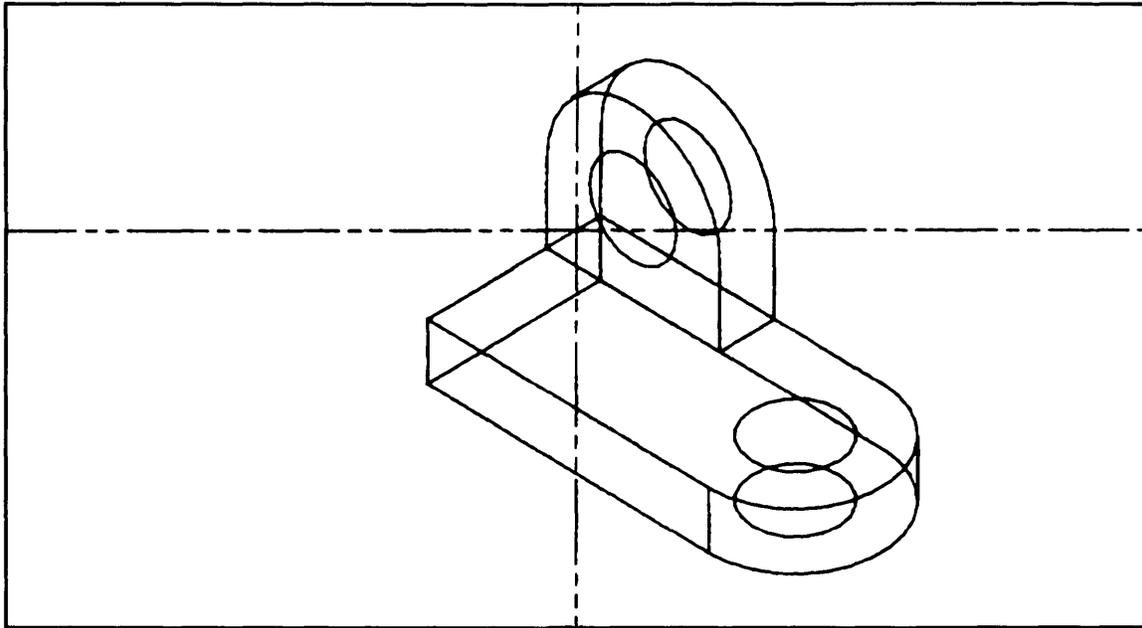
| Hidden Line Removal |  | Blanked | One End | Two End       |
|---------------------|--|---------|---------|---------------|
|                     |  |         |         |               |
|                     |  |         | Middle  | Entire Entity |

DC

**CAUTION**

It is very important when removing one end of an entity that you select the entities *on the side of the boundary that is to be removed*. You are not prompted to indicate which side of the entity is to be removed like you are with the Trim/Extend Curves function.

## HIDDEN LINE REMOVAL



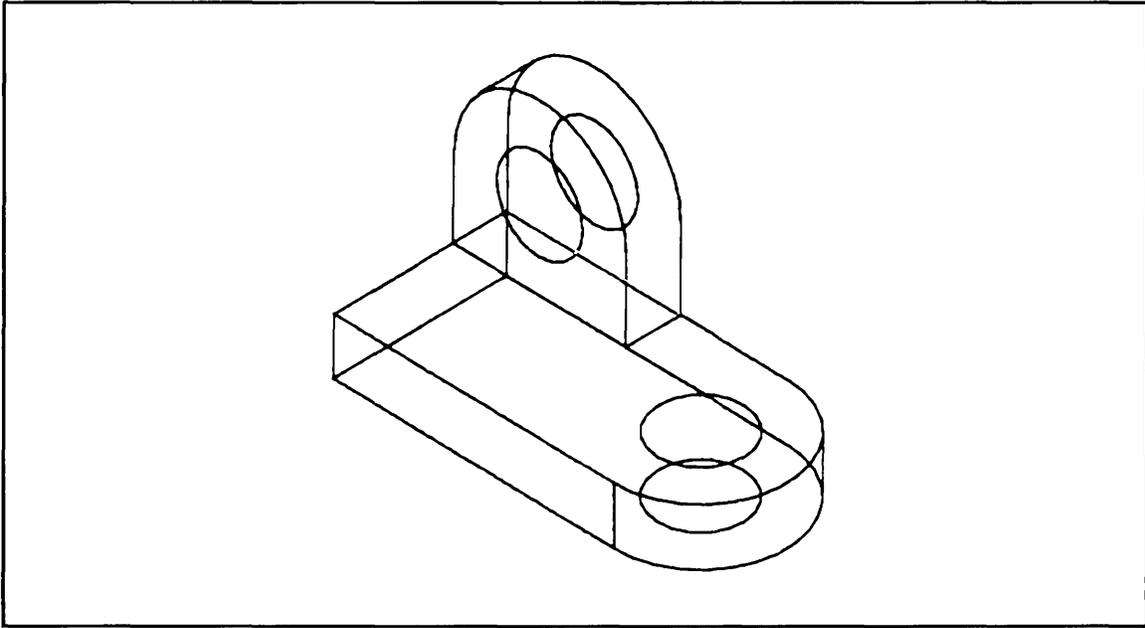
**Figure 11-20. Graphics Cursor Position for Selecting One End of a Line**

Enter ] to tell ICEM DDN that you have finished selecting entities.

### **NOTE**

Because you are working in the isometric view, it looks as though the Hidden Line Removal wasn't done. Before you did the Hidden Line Removal, there were actually two lines that overlapped. Now these two lines meet each other at the line you used as the bound for the Hidden Line Removal. Later in this practice session, you will be able to verify this Hidden Line Removal.

Apply what you've just learned to remove the end of the line highlighted in figure 11-21.



**Figure 11-21. Hidden Line Removal of Another End of a Line**

## Removing Entire Entities

The next step is to remove the entire entities highlighted in figure 11-22.

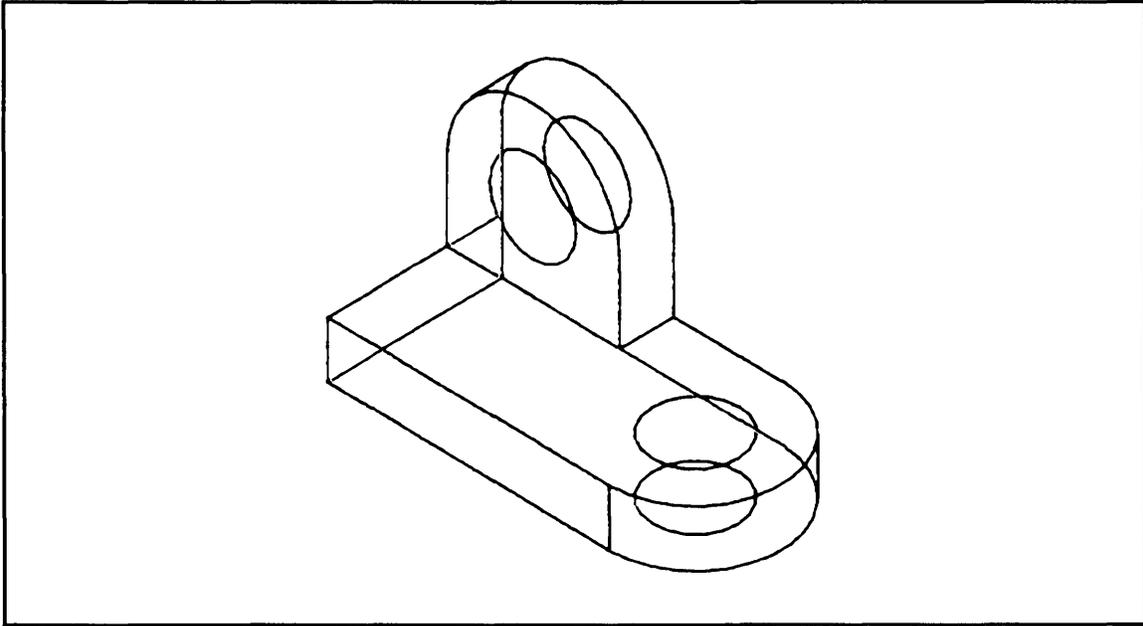


Figure 11-22. Removing Entire Entities from the Isometric View

### Keyboard Selection

Select 4.ENTIRE ENTITY from the Hidden Line Removal Mode menu.

### Tablet Selection

| Hidden Line Removal |  |         |         |               |
|---------------------|--|---------|---------|---------------|
|                     |  | Blanked | One End | Two End       |
|                     |  |         | Middle  | Entire Entity |

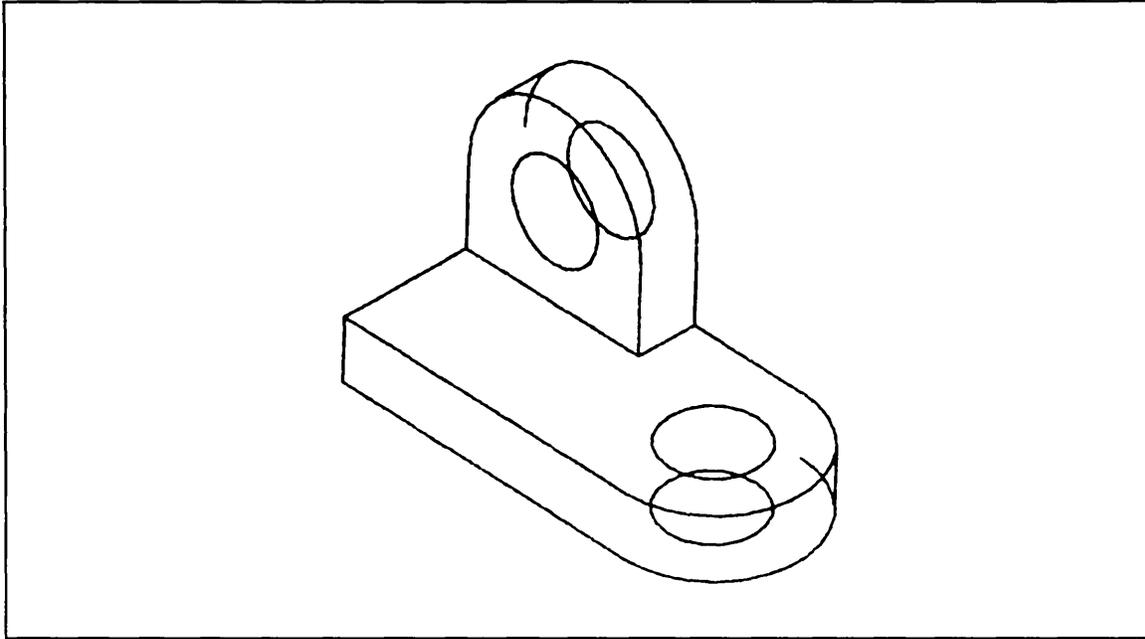
DC

- Position the graphics cursor on each of the lines highlighted in figure 11-22 and select them using the L key or the tablet pen.
- Enter ] to tell ICEM DDN that you have finished selecting entities.

### NOTE

Note that the entities you just selected still appear on the screen. With Hidden Line Removal, a host repaint is needed to remove the entities from the screen.

The display on your screen should now match the display shown in figure 11-23.



**Figure 11-23. Removing Entire Entities from the Display**

## Removing One End of Two Arcs

The next step is to remove the ends of the arcs highlighted in figure 11-24.

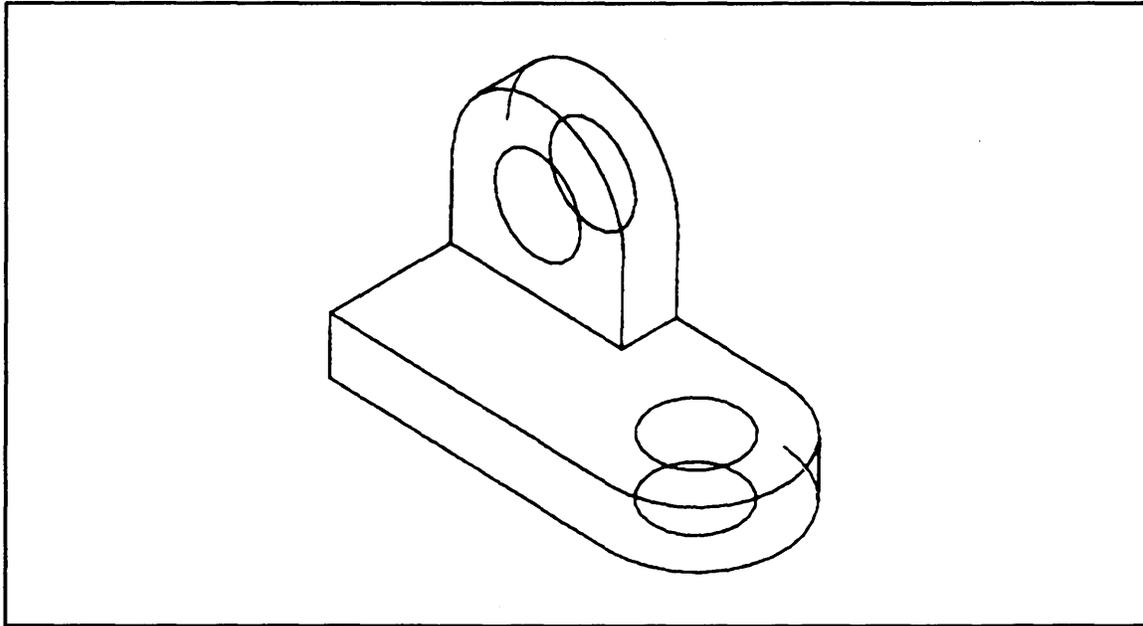


Figure 11-24. Removing One End of Two Arcs

### Keyboard Selection

Select 1.ONE END from the Hidden Line Removal Mode menu.

### Tablet Selection

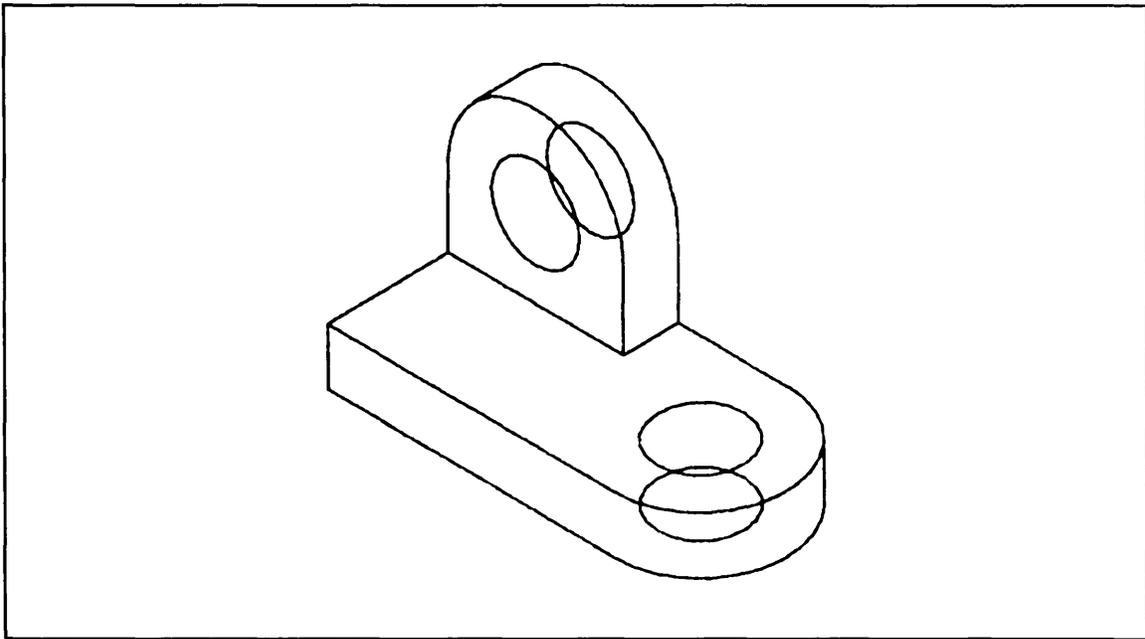
| Hidden Line Removal |  |         |         |               |
|---------------------|--|---------|---------|---------------|
|                     |  | Blanked | One End | Two End       |
|                     |  |         | Middle  | Entire Entity |

DC

- Position the graphics cursor on the top tangent line and select it as the bound using the L key or the tablet pen.
- Position the graphics cursor on the top arc *near the end that is to be removed* and select it using the A key or the tablet pen.
- Enter ] to tell ICEM DDN that you have finished selecting entities.

In the same way that you just removed the end of the top arc, remove the end of the bottom arc.

After you repaint the screen using the R key, the display on your screen should match the display shown in figure 11-25.



**Figure 11-25. Hidden Line Removal of One End of Two Arcs**

## Removing Two Ends of a Circle

The last step in this process is to remove the two ends of the hidden circles that form the drill holes.

New users are often confused about how to remove the ends of a circle.

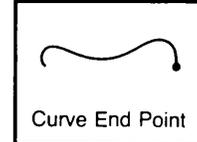
Whenever you need to use Hidden Line Removal on a circle, always remember that the beginning of a circle is at 0 degrees and the end of a circle is at 360 degrees. If you're not sure where the ends of the circle meet, you can use the Point menu to place a point at the ends of the circle as follows:

### Keyboard Selection

**(CTRL) P + (RETURN)**

Select 8.CURVE END from the Point menu.

### Tablet Selection



- Position the graphics cursor on the top hidden circle and select it using the A key or the tablet pen.
- Position the graphics cursor on the bottom hidden circle and select it using the A key or the tablet pen.

A point appears at the end points of each of the hidden circles.

Enter ] to return to the Point menu.

Enter F to return to the main menu.

Now that you know where the two ends of the hidden circles meet, you can remove the ends as follows:

### Keyboard Selection

Select 12.OTHER CURVES from the main menu.

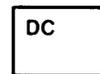
Select 11.HIDDEN LINE REMOVAL from the Other Curves menu.

Select 5.BLANKED from the Line Font menu.

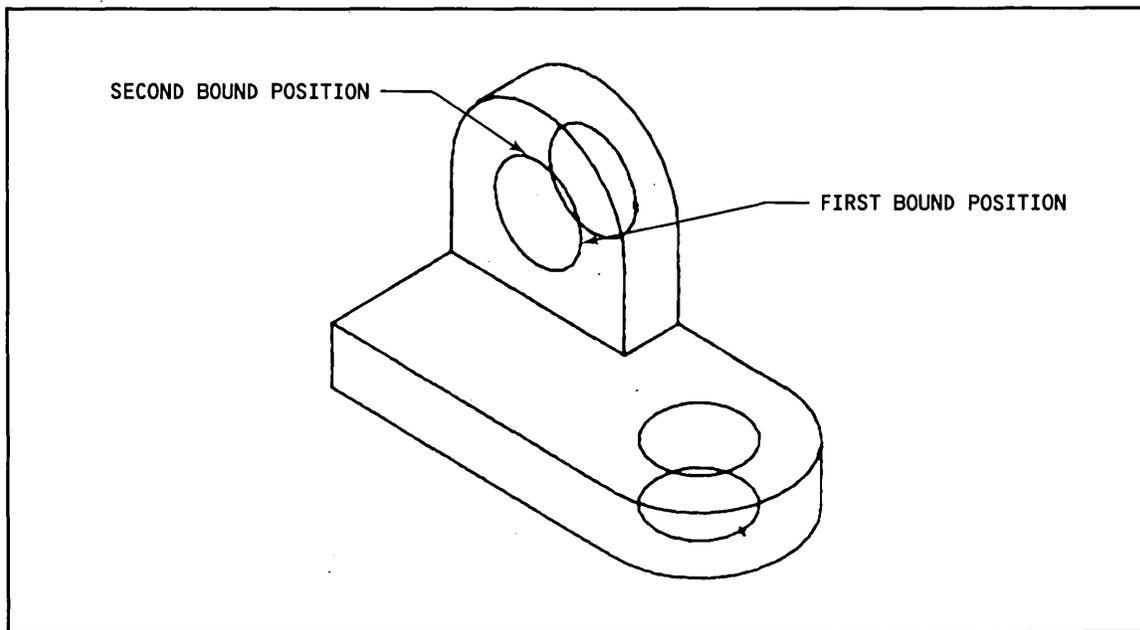
Select 2.TWO ENDS from the Hidden Line Removal Mode menu.

### Tablet Selection

| Hidden Line Removal |  |         |         |               |
|---------------------|--|---------|---------|---------------|
|                     |  | Blanked | One End | Two End       |
|                     |  |         | Middle  | Entire Entity |



- Position the graphics cursor first as shown in figure 11-26 using the front circle as both bounds.



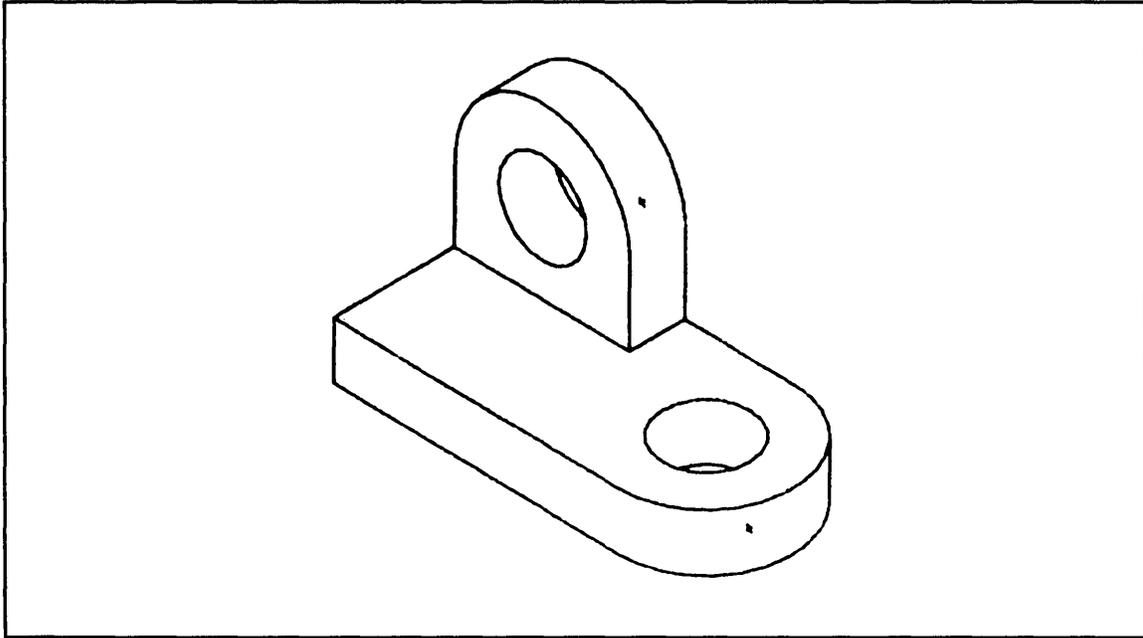
**Figure 11-26. Selecting the Bounds for Removing Two Ends of a Circle**

- Position the graphics cursor on the hidden circle and select it as the curve to be removed using the A key or the tablet pen.
- Enter ] to tell ICEM DDN that you have finished selecting entities.

Apply what you just learned to remove the two ends of the bottom hidden circle.

HIDDEN LINE REMOVAL

After you repaint the screen using the R key, the display on your screen should match the display shown in figure 11-27.



**Figure 11-27. Completed Isometric View**

## Deleting All Points

Delete all of the points in your part as follows:

### Keyboard Selection

Enter F to return to the main menu.

Select 3.DELETE from the main menu.

Select 3.ALL POINTS from the Delete menu.

### Tablet Selection

| Delete    |            |            |       |
|-----------|------------|------------|-------|
| Select    |            | Screen     | Chain |
| Region In | Region Out | All Points | Level |

Next, you are asked if you are sure that you want to delete all of the points in your part:

DELETE ALL POINTS?

Enter Y in response to this prompt.

All of the points in your part are deleted.

The display on your screen should now match figure 11-28.

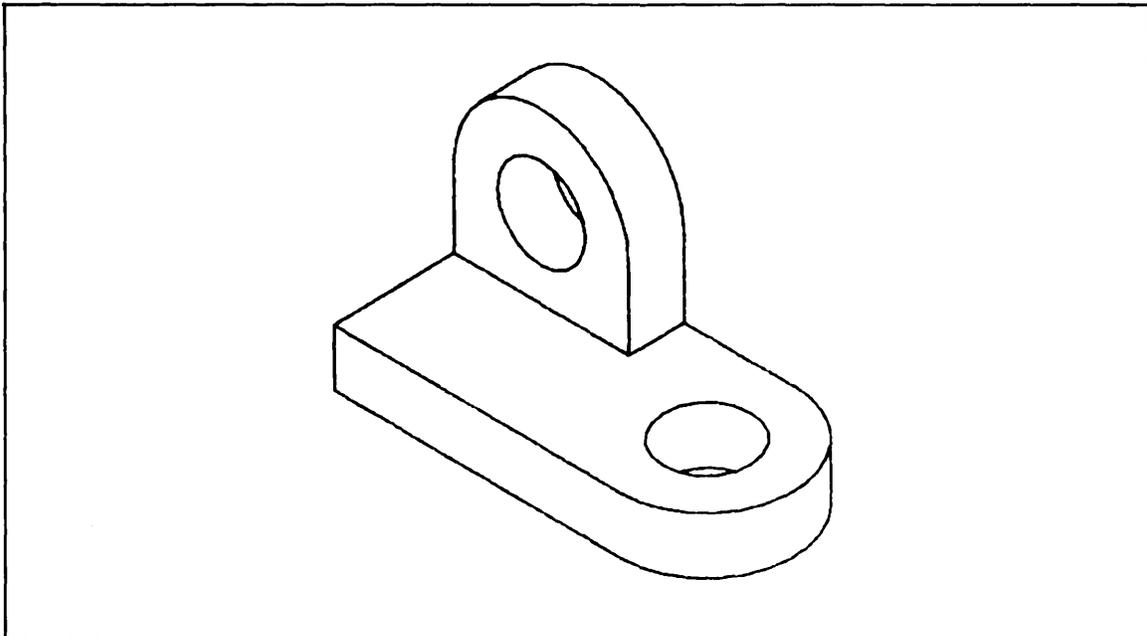


Figure 11-28. Deleting All Points

## Displaying the Four Standard Views of the Bracket

Display the four completed views of the bracket as follows:

### Keyboard Selection

Select 8.DISPLAY CONTROL from the main menu.

Select 4.DISPLAY MULTIPLE VIEWS from the Display Control menu.

Select 9.VIEWS 6, 1, 3, 8 (TOP/FRONT/RIGHT/ISO 1) to display the bracket in the four standard views.

Next, you are asked to select the zoom scale for the display:

#### ZOOM SCALE FOR MULTIPLE VIEWS

- 1.LAST SCALE USED
- 2.BASE SCALE
- 3.SAVED SCALE
- 4.ENTER SCALE
- 5.AUTOMATICALLY MAXIMIZED

Select 5.AUTOMATICALLY MAXIMIZED as the zoom scale.

The four completed views of your bracket should match the display shown in figure 11-29.

### Tablet Selection

| Display Views |              |          |           |                 |
|---------------|--------------|----------|-----------|-----------------|
| Top-6         | Front-1      | ISO1-8   | One       | Last Layout     |
| Front-1       | Right-3      |          |           |                 |
| Two Views     |              | Std Four | Std Eight | Last Multi View |
| Above         | Side by Side |          |           |                 |
| Below         |              |          |           |                 |

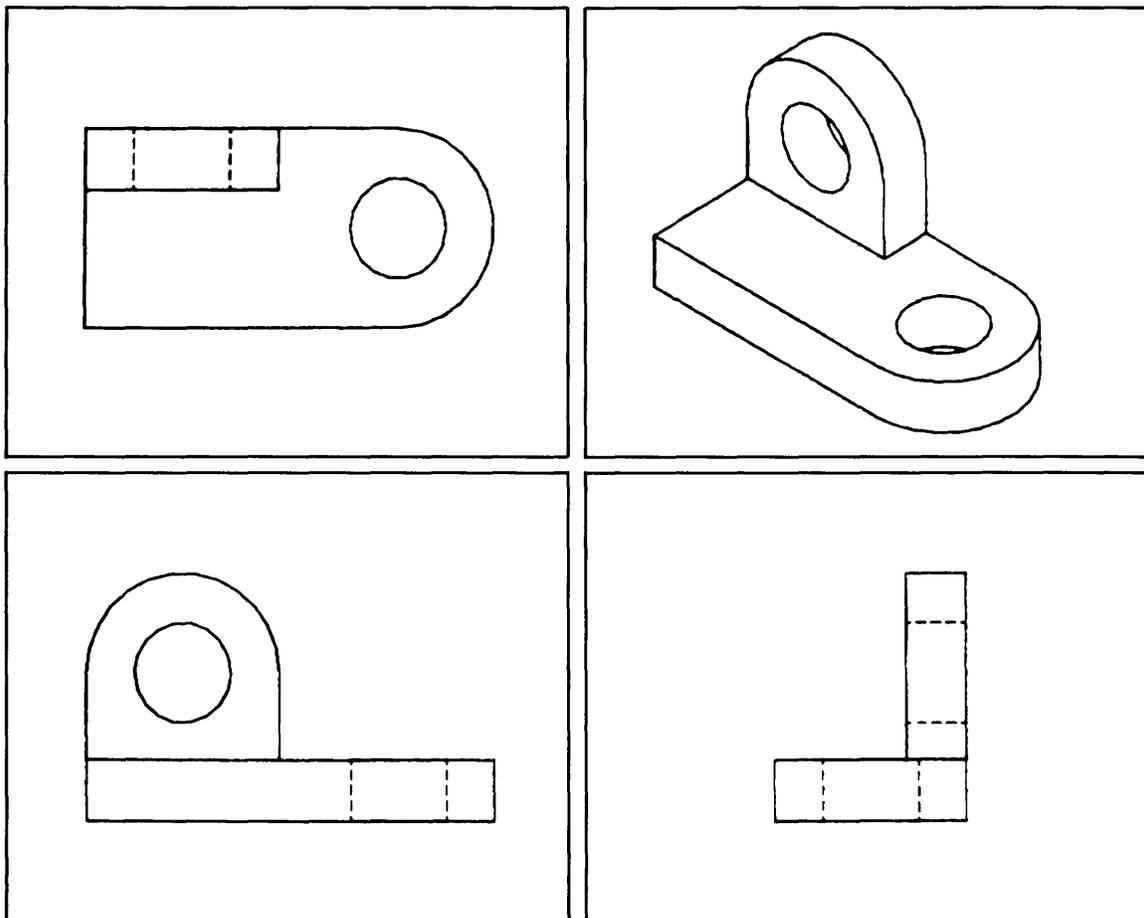


Figure 11-29. Four Completed Views of the Bracket

# Filing and Exiting

Figure 11-30 outlines the steps to end this terminal session and save your part.

| Terminal Session                                                                                                                                                                                                                                                                                                                                               | Comments                                                                                                                                                                                                                                 |
|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ICEMDDN L=0 V=1 D=0.00 P=0 C=1 4                                                                                                                                                                                                                                                                                                                               | If you are using the keyboard, enter 4 to select the File Current Part/Exit ICEM DDN menu.                                                                                                                                               |
| <p>FILE CURRENT PART/EXIT ICEM DDN</p> <ol style="list-style-type: none"> <li>1.FILE - CONTINUE CURRENT PART</li> <li>2. - GET DIFFERENT PART</li> <li>3. - QUIT SESSION</li> <li>4. - SUSPEND SESSION</li> <li>5.DO NOT FILE - CONTINUE FROM LAST FILE</li> <li>6. - GET DIFFERENT PART</li> <li>7. - QUIT SESSION</li> <li>8. - SUSPEND SESSION 3</li> </ol> | <p>If you are using the tablet, you can file your part and exit from ICEM DDN in a single step by selecting the FILE/QUIT square with your tablet pen.</p> <p>Select 3.FILE - QUIT SESSION to save your part and exit from ICEM DDN.</p> |
| ICEMDDN TERMINATED.                                                                                                                                                                                                                                                                                                                                            |                                                                                                                                                                                                                                          |
| /                                                                                                                                                                                                                                                                                                                                                              |                                                                                                                                                                                                                                          |

Figure 11-30. Sample File/Exit Sequence



# Appendixes

---

Glossary ..... A-1

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---

## A

### **Alphanumeric**

The letters of the alphabet (A through Z) and the digits (0 through 9).

### **Angular Dimension**

A measure, in degrees, of the angle between two lines.

### **Attach**

To establish user communication with a direct access permanent file.

### **Attention Indicator**

A small oval that the system displays on an entity, usually to indicate that the entity has been selected.

### **Attribute**

A quality assigned to an entity such as the length, depth, name, or number.

### **Auxiliary View**

Views that you can define in addition to the standard views (top, front, side, and isometric) defined by the system. You create auxiliary views by rotating and tilting the part using Display Control operations.

---

## B

### **Batch**

A mode of data processing in which a job is submitted as a complete unit and without further user intervention during processing. In the Interactive Facility, a batch mode is available for processing control statements one at a time with each control statement treated as a complete job and bringing an immediate response as the job output.

### **Baud Rate**

The rate at which data passes over the communication line between the terminal and the computer.

### **Blanking**

A process of discontinuing the display of an entity, but retaining it in the data base for access or future unblinking. Contrast with Deleting.

---

## C

### **Chain Select Mode**

A process of selecting a series of connected entities in a single step.

### **Computer System**

Hardware configuration of data processing units and peripheral equipment.

### **Contiguous Curves**

Adjacent curves that share a common endpoint or that have endpoints within a short distance of each other.

**Coordinates**

The values of a spatial condition measured against the right-handed, three-dimensional Cartesian coordinate axes.

**Curve**

A line, arc, conic, or two-dimensional spline. In some cases, it also includes points, three-dimensional splines, strings, composite curves, machining curves, and surface intersection curves.

**D**

---

**Default**

A value used by the system if you enter ] in response to a data entry prompt. It is also the initial value assigned to a modal until you change it.

**Deleting**

A process of removing entities in the current part so that the entities are no longer accessible for display or manipulation. If a deleted entity is used in the definition of another entity, that entity can still access the necessary parameters from the deleted entity. Contrast with Blanking.

**Direct Access File**

A permanent file that can be attached directly to a job without any local copy being generated. All changes to this file are made to the file itself.

**E**

---

**Entity**

An object that ICEM DDN creates and stores. Entities are the primary data objects of the ICEM DDN data base. Examples of entities are points, lines, arcs, spheres, and so on.

**F**

---

**Family**

Designation that an installation may give to a grouping of mass storage devices.

**File**

A collection of data that can be accessed in its entirety with a single name. All information contained in a data processing system is organized into files.

**Fillet**

An arc that joins two lines, arcs, or curves. It begins and ends at a point of tangency on each of the two lines.

**Font**

A form of display representation for curves and surfaces. Examples of fonts are the various solid and dashed lines that can be used in displaying a curve.

**Function Control Key**

A keyboard character (or equivalent tablet square) that has a special function in ICEM DDN.

---

**G**

---

**Graphics Cursor**

The cursor symbol used to locate or define entities by screen position. The graphics cursor can be the crosshairs or some other cursor symbol.

**Graphics Terminal**

A terminal that can display and manipulate graphic information as well as alphanumeric characters.

**Group**

A collection of entities that you select to be defined as a group in the data base.

---

**I**

---

**Interactive Facility (IAF)**

System software product that contains the time-sharing subsystems for the Network Operating System. These subsystems enable a user to enter commands and control statements at a terminal and interact with the operating system.

---

**L**

---

**Level Management**

A method of assigning entities to different levels for management purposes.

**Local File**

Any file that is currently associated with a job. It can be a copy of a permanent file, or one that is read in from a batch job, or one that the user is entering one statement at a time at a terminal.

---

**M**

---

**Mass Storage**

Magnetic disk or extended memory.

**Menu**

A list of options from which you can select to perform operations using ICEM DDN.

**Modal**

A number or status that controls the execution and operation of ICEM DDN.

**Model Coordinates**

The displacements of positions along the x-, y-, and z-axes (also called absolute coordinates). The x- and y-axes are horizontal and vertical in view 1; the z-axis is normal to the screen (plane of definition) in view 1. The positive coordinate directions are to the right, upward, and out of the screen, respectively. Refer also to Transform Coordinates.

---

**N**

---

**Normal**

A line or vector that is perpendicular to a point of tangency on a curve or surface.

---

**O**

**Operating System**

Master collection of software routines that is installed in a computer to supervise the processing of user input.

---

**P**

**Parameter**

Range of options a user may select when entering a command or control statement.

**Part**

The division of space in the parts file in which you can create a drawing. Also, the drawing which is created. Each part in the parts file can have 1 through 99 sheets, each of which can have a drawing.

**Pattern**

A set of entities that you selected and named that can be copied into a position specified. You can use operations in 6.1 PART MANAGEMENT and 6.2 PATTERN MANAGEMENT to create patterns and copy them into the part.

**Permanent File**

A mass storage file saved by the system between terminal sessions.

---

**R**

**Rectangular Array**

A set of objects produced by repeatedly creating the same object at equally spaced intervals in rows and columns. Menu selection 13.1 RECT ARRAY creates rectangular arrays.

**Region Select Mode**

A process of selecting all the entities inside or outside a specified region.

---

**S**

**Session**

Refer to Time-Sharing Session.

**Sheet**

A subdivision of a part.

**Single Select Mode**

A process of selecting individual entities, one at a time.

**System**

The ICEM DDN software system.

**T**

---

**TAPE3**

A local file that serves as a data base in which to store and retrieve the parameters that define your drawings.

**Temporary File**

A nonpermanent file associated with a job only as long as the job is being processed. The job can be in either batch or time-sharing mode. A temporary file is a subset of the local file classification.

**Time-Sharing**

System software routine that allows a number of terminals to use a computer simultaneously. In time-sharing, each command brings a response as soon as entered; this is in contrast to ordinary batch in which an entire job must be submitted as a unit before the computer responds.

**Time-Sharing Job**

Consists of all input entered during a time-sharing session.

**Time-Sharing Session**

All the interaction of user and system from the time of login to the time of logout.

**Transform Coordinates**

Displacements along the xt, yt, and zt transform coordinate axes. The xt, yt, and zt transform coordinate axes are those that are horizontal, vertical, and normal to the screen (plane of definition) in the current work view. The transform coordinates are useful in views that have been rotated from view 1; for example, you can place an object to the left or right of a known coordinate position by decreasing or increasing the xt transform coordinate value. In a view other than view 1, an increase or decrease in the x model coordinate value does not necessarily move the position to the left or right if the view has been rotated. Refer also to Model Coordinates.

**Trimming**

The process of dividing a line, arc, or curve by a boundary consisting of another line, arc, or curve and deleting or extending one portion or the other with 12.8 TRIM CURVES. Two boundaries are used to delete or extend either the middle or both ends of the line, arc, or curve.

**U**

---

**User Name**

An alphanumeric code, assigned to a user or system library by the installation, to establish validation for system use. Defines the resources the user may employ and keeps a record of what is actually used.



# Control Card Parameters

B

As you become familiar with the use of ICEM DDN, you may wish to use a short cut method to enter information that you are prompted for at the beginning of every session. For example, you can set your terminal configuration to the correct setting, activate your tablet, and select the unit of measure that you wish to use—all in a single step. To do this, you simply add parameters to the ICEMDDN call statement using the following format:

```
ICEMDDN,[baud rate], p1,p2 ... pn
```

The most frequently used parameters appear in table B-1 below. Refer to the ICEM Design/Drafting Introduction and System Controls manual for more information on parameters.

**Table B-1. Control Card Parameters**

| Parameter | Description                                                                                                                |
|-----------|----------------------------------------------------------------------------------------------------------------------------|
|           | <b>Drafting Standard†</b>                                                                                                  |
|           | If you do not set this parameter, you are prompted for the drafting standard each time you create a new part.              |
| DS=A82    | Sets the drafting standard to ANSI 1982. Any new parts that you create during the session are made to 1982 ANSI standards. |
| DS=A73    | Sets the drafting standard to ANSI 1973. Any new parts that you create during the session are made to 1973 ANSI standards. |
|           | <b>Local Assist</b>                                                                                                        |
|           | The default depends on terminal type.                                                                                      |
| LA        | Sets the local assist to the on position.                                                                                  |
| LA=N      | Sets the local assist to the off position.                                                                                 |
| LA=Y      | Sets the local assist to the on position.                                                                                  |
|           | <b>LDF (Local Display File)</b>                                                                                            |
|           | The default depends on terminal type.                                                                                      |
| LD        | Sets the LDF to the on position if the selected terminal supports a local display file. If not, no action is taken.        |
| LD=N      | Sets the LDF to the off position regardless of terminal type.                                                              |
| LD=Y      | Sets the LDF to the on position if the selected terminal supports a local display file.                                    |

† If you retrieve an existing part from your TAPE3 file, the drafting standard with which the part was originally created is used, regardless of the current DS parameter setting.

(Continued)

**Table B-1. Control Card Parameters (Continued)**

| <b>Parameter</b> | <b>Description</b>                                                                                                                                        |
|------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------|
|                  | <b>Menu Area</b>                                                                                                                                          |
|                  | The default sends menu text to the graphics terminal.                                                                                                     |
| MA               | Sends menu text to the graphics terminal.                                                                                                                 |
| MA=GRAF          | Sets the menu text area according to terminal specifications.                                                                                             |
| MA=A722          | Selects a microprocessor assist designer station or Tektronix Model 4114 terminal with a CDC 722 terminal as the alphanumeric terminal device.            |
|                  | <b>Tablet Control</b>                                                                                                                                     |
|                  | The default setting is off.                                                                                                                               |
| TB               | Sets the tablet to the on position.                                                                                                                       |
| TB=N             | Sets the tablet to the off position.                                                                                                                      |
| TB=Y             | Sets the tablet to the on position.                                                                                                                       |
|                  | <b>Terminal Type</b>                                                                                                                                      |
|                  | If the TT parameter has been selected, the graphics terminal type menu is not displayed.                                                                  |
| TT=T4014         | Selects the Tektronix Model 4014 graphics terminal with alpha assist on.                                                                                  |
| TT=T4105         | Selects the Tektronix Model 4105 graphics terminal with alpha assist on.                                                                                  |
| TT=T4107         | Selects the Tektronix Model 4107 graphics terminal with alpha assist on.                                                                                  |
| TT=T4109         | Selects the Tektronix Model 4109 graphics terminal with alpha assist on.                                                                                  |
| TT=T4113         | Selects the Tektronix Model 4113 with alpha assist, local assist, and LDF all set to the on position.                                                     |
| TT=T4114         | Selects the Tektronix Model 4114 with alpha assist, local assist, and LDF all set to the on position.                                                     |
| TT=T4115         | Selects the Tektronix Model 4115 with alpha assist, local assist, and LDF all set to the on position.                                                     |
| TT=C721          | Selects the CDC Model 721 terminal with alpha assist on.                                                                                                  |
| TT=C790T         | Selects the CDC IEW 790 running as a Tektronix 4115 with menu text on the separate alphanumeric screen, local assist, and LDF all set to the on position. |
|                  | <b>Units of Measure</b>                                                                                                                                   |
|                  | You are prompted to enter the units of measure unless you specify one of the parameters below.                                                            |
| UM=E             | Selects U.S. customary measurement units.                                                                                                                 |
| UM=FI            | Selects U.S. measurement units (feet, inches).                                                                                                            |
| UM=M             | Selects the metric measurement units (centimeter, millimeter).                                                                                            |

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