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PREFACE

This manual describes the AD-2000 computer-aided design (CAD) software system. It contains a brief overview of the system's capabilities, information pertaining to user/graphics terminal interaction, and explanations of the messages used by the AD-2000 system to guide the user through the AD-2000 design process.

The manual is organized as follows:

- Introduction

  This section briefly describes what AD-2000 is, what it does, and the supporting hardware and software environments of AD-2000, including a short description of the AD-2000/operating system interface. This section is of interest primarily to new users of AD-2000.

- User/AD-2000 interaction

  This section describes the various methods of interacting with AD-2000. It is primarily a description of the input facilities of the graphics terminal and how AD-2000 interprets each type of input from the terminal. This section is also of interest primarily to new users of AD-2000.

- AD-2000 messages

  This section contains the messages issued by the AD-2000 system and explanations of these messages. Examples of input specification are sometimes included for input request messages. In many cases, the messages that AD-2000 issues are self-explanatory. But in the cases in which a user does not understand a message or has obtained results from an AD-2000 operation that were not what he anticipated, he can look up the message in this section and receive an extended explanation. This section is useful to both new and experienced users of AD-2000.

This manual is in a reference format and is intended to be a reference source for the AD-2000 user. It is not intended to be a tutorial guide to AD-2000. Beginners should refer to the AD-2000 User's Guide for a step-by-step introduction to the AD-2000 system.

RELATED PUBLICATIONS

The following is a list of manuals applicable to AD-2000.

Control Data Publication	Publication Number
AD-2000 User's Guide	60456940
AD-2000 Menu Summary Card	60457140
Network Products Interactive Facility Version 1 Reference Manual	60455250
Network Products Interactive Facility Version 1 User's Guide	60455260
Network Products Reference Manual, Volume 1	60435400
UNIPLOT Reference Manual	60454730

DISCLAIMER

This product is intended for use only as described in this document. Control Data cannot be responsible for the proper function of undescribed features or parameters.

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INTRODUCTION 1

AD-2000 is a computer-aided design (CAD) software package offering automated control of the design and drafting process from the development of the original design concept to the production of engineering drawings. Geometric constructions, mechanical analysis, and mechanical drafting, with a full range of drafting procedures available, including automatic and user-entered dimensioning and material identification, are features of the system. Also included are extensive drawing management capabilities and a computational capability called GRAPL for calculating parameter values used as AD-2000 input.

HARDWARE/SOFTWARE ENVIRONMENT

AD-2000 executes as an application software package on the CONTROL DATA® CYBER 170 Series; CDC® CYBER 70 models 71, 72, 73, and 74; and 6000 Series Computer Systems under control of the Network Operating System (NOS).

The AD-2000 user communicates with AD-2000 through the NOS Interactive Facilty (IAF). IAF is discussed in more detail in Operating System/AD-2000 Interface.

A Tektronix model 4012, 4014, or 4015 graphics terminal is the primary input/output (I/O) device used with AD-2000. The AD-2000 user interacts with AD-2000 via input entered at the terminal and receives output in the form of graphics displays on the terminal screen. In addition, a hardcopy printer supported by Tektronix may form part of the graphics terminal configuration.

OPERATING SYSTEM/AD-2000 INTERFACE

As mentioned previously, AD-2000 users access AD-2000 through IAF, the interactive facility of NOS. For this reason, users of AD-2000 must be familiar with elementary time-sharing procedures before they can use AD-2000. Any time the AD-2000 application software program is not executing, the Tektronix graphics terminal behaves like any other time-sharing terminal, responding to the same set of IAF commands. To use AD-2000, you must be familiar with the following time-sharing tasks.

- Logging in.
- Attaching and running AD-2000.
- Basic file manipulation for the AD-2000 data base.

The information necessary to carry out these tasks is contained in the IAF Reference Manual and IAF User's Guide. The AD-2000 User's Guide contains a step-by-step procedure for accomplishing these tasks and should be consulted, but the discussion of time-sharing procedures is limited to this information.

The AD-2000 user already familiar with time-sharing procedure needs only the following information.

- AD-2000 is contained in a direct access file named AD2000. If this file is not in the user's catalog, the user must consult site personnel for the user name to be used with the UN parameter of the ATTACH statement. Type

  ```
  ATTACH,AD2000
  ```

  if the file is in the user's catalog, or

  ```
  ATTACH,AD2000/UN=username
  ```

  if the file is in another catalog.

- AD-2000 provides a sophisticated, updatable data base whereby drawings can be stored, retrieved, and worked on from session to session. However, it is the user's responsibility to create the permanent file the data base will be contained in and to attach the data base file before executing AD-2000. AD-2000 uses a local file named TAPE3 for the data base. To create a permanent file for the data base, type

  ```
  DEFINE,TAPE3=filename
  ```

  where filename is any valid NOS file name. This is the name under which the data base is stored by NOS. Before subsequent sessions, type

  ```
  ATTACH,TAPE3=filename/M=W
  ```

  to make the permanent data base available to AD-2000. If you do not want a permanent data base, NOS will create a scratch TAPE3, which is lost at the end of the IAF session.

  The User's Guide contains more information about the AD-2000 data base file.

- Before executing AD-2000, you must change the user break 1, or interrupt key, procedure so that NOS and AD-2000 usages are compatible. To do so, press the ESC key, and then type

  ```
  B1=\
  ```

  and press the RETURN key.

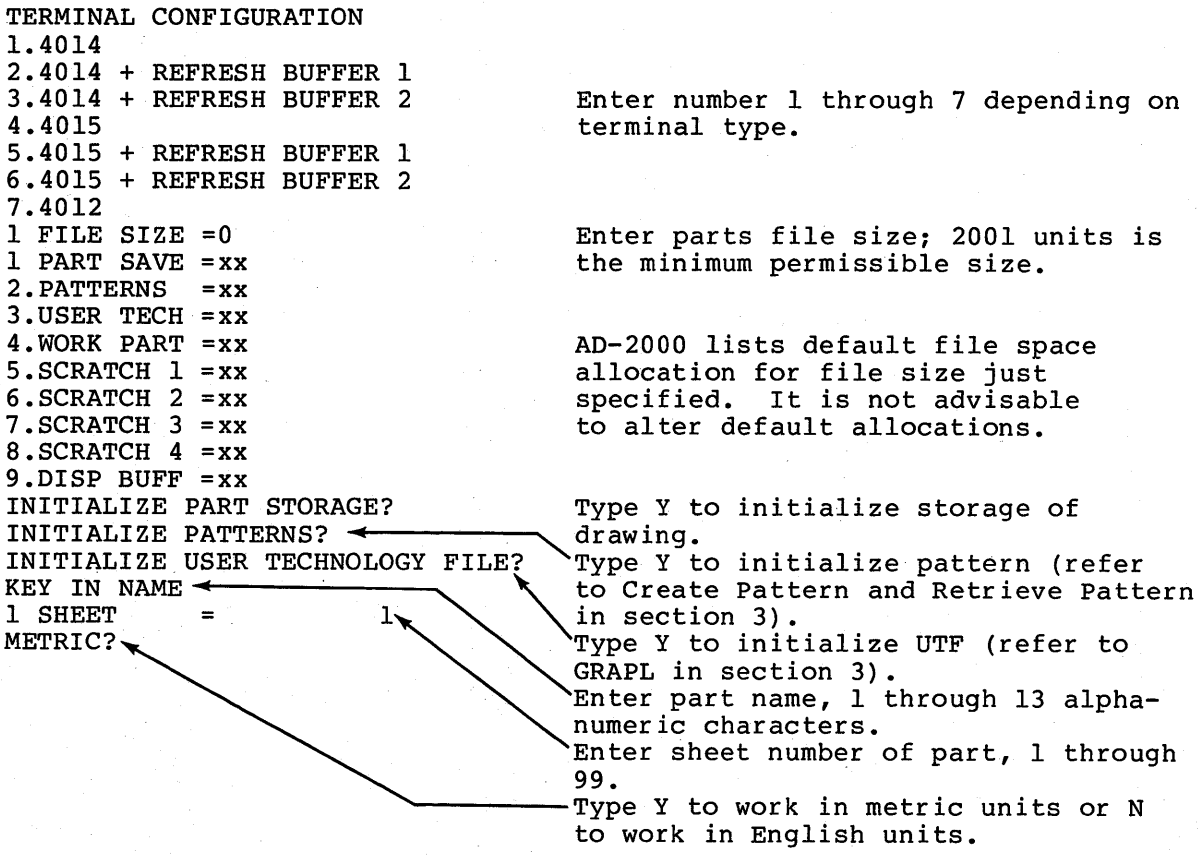
- To execute AD-2000 after you have attached the AD-2000 file and, optionally, defined or attached your data base file, type

  ```
  AD2000,x
  ```

  where x is the first digit of the baud rate of the data communication line you are using (for example, 3 for 300 baud, 1 for 1200 baud, and so on).

USER/AD-2000 INTERFACE

When AD-2000 begins execution, it asks a series of initialization questions. This process is illustrated in the AD-2000 User's Guide. The following is the list of initialization questions.



Following initialization, the user controls AD-2000 operation by selecting functions from AD-2000 menus. A menu is a list of options from which the user selects the item appropriate to his purposes. Section 2 of this manual describes menus and user/program interaction more fully. The AD-2000 User's Guide also contains a sample terminal session during which the use of menus is illustrated. The menu summary card mentioned in the preface of this manual contains all AD-2000 menus in an easily portable form.

USER/AD-2000 INTERACTION 2

This section and section 3 describe the operation of the AD-2000 system. The present section describes user procedures, command formats, and control keys used to control the execution of the system and to execute the operations and functions provided by the system. Section 3 describes each individual operation that can be selected and performed by AD-2000.

COMMAND AND REQUEST FORMAT—THE MENUS

The user requests the execution of an operation by the AD-2000 system by selecting that operation from a menu that is displayed by the AD-2000 system on the terminal screen. The system contains a number of menus (refer to appendix B). These menus are at different levels and are arranged hierarchically: entries in higher level menus point to lower level menus, and entries in the lowest level menus refer to specific operations that can be executed by the AD-2000 system. The user selects and initiates an individual operation by first obtaining the menu that contains that operation and then selecting that operation. When the user selects one of these operations, AD-2000 prompts him for any parameters needed by displaying messages. The user satisfies these requests by typing in the requested information, by using the crosshairs to indicate a position, or by selecting an entity on the screen through use of the crosshairs. The menu that contains the reference to the operation desired by the user can be obtained by either working down from higher level menus or by selecting the menu directly, using the MENU SELECT command. Appendix B contains the menus used in the AD-2000 system.

An item can only be selected from the menu currently active. To select an item in that menu, use:

- The numeric keys 1 through 9 and 0 on the keyboard to select items 1 through 9 and 10.
- The numeric keys 1 through 9 while the SHIFT key is pressed to select items 11 through 19.
- The numeric entries 10 and 11 through 19 to select menu items 10 through 19.
- Special function control keys F, control P, control L, and control A to select the top level AD-2000, POINT, LINE, and ARC/CIRCLE/FILLET menus, respectively. (Refer to table 2-1.)

A limited type-ahead facility is available to allow the user to string together a series of menu item selections while in menu selection mode. For example, if the LINE menu is active, the entry

```
F.16.4  RETURN
```

causes the system to return to the top level AD-2000 (main) menu and then go to item 4 in the DRAFTING menu (menu 16).

After an operation has been selected, the AD-2000 system responds by executing the operation when the required parameters have been entered as requested. If the parameters entered by the user are not acceptable or if the user presses the] key before all the required parameters have been entered, the system responds by either requesting the required parameters or ignoring the request to execute the operation. For example, if the user has selected the screen position operation from the POINT menu, AD-2000 waits for the user to position the crosshairs and to press the C key. If the user does so, AD-2000 creates a point at that position. If the user does not enter the required parameters (in this case, the position at which a point is to be created), but instead presses either the] or the [key, AD-2000 does not execute the operation. If, at this time (that is, before an operation has been selected), the user presses the [key, AD-2000 responds by displaying the next highest level menu, the AD-2000 menu.

Thus, in summary, AD-2000 moves through the menus in the following manner.

- If the item selected from a menu is the name of the next lower level menu, that lower level menu is displayed and is active. (That is, an item can be selected from it.)
- If the item selected from a menu is an operation, after that operation has been completed or aborted, the menu from which the operation was selected is displayed and is active.
- If no item is selected from a menu and the] key is pressed, the next higher level menu is displayed and is active.
- If the [key is pressed after an item has been selected from a menu (and if the item is an operation before the parameter requests have been satisfied), AD-2000 exits one level, redisplays the menu from which the item was selected, and leaves that menu active.
- If an invalid item is selected, the system causes a beep to be heard.

Entity creation operations (such as those that create points, lines, and arcs) may function differently depending on the setting of the construction mode modal.

- If this modal is on, the system allows the user to repeatedly execute a construction operation without reselecting it from its menu.
- If this modal (the construction modal) is off, the user must reselect a construction operation from the menu each time it is used.

This modal is described in the section on the MODALS and FONTS menu.

FUNCTION CONTROL KEYS

In some instances, the user controls the execution of AD-2000 through the use of keys on the graphics terminal keyboard that have a special and consistent use throughout the AD-2000 system. The use of these function control keys is context dependent. Under some contexts (specifically when AD-2000 is waiting for a control input such as a menu or operation selection), the keyboard may be considered to be in special function or function control mode. In other contexts (specifically when AD-2000 is waiting for alphanumeric information), the keys have their normal alphanumeric values and uses and can be used to enter data and parameter values. Table 2-1 lists and explains all function control keys.

In some cases, the terminal operator may have to press the RETURN key after pressing a function control key. Whether this is necessary depends on both the terminal configuration and the type of input expected by AD-2000 at the time.

TABLE 2-1. FUNCTION CONTROL KEYS

Key	Title	Description
[	Reject	This key tells the system to negate the previous operation, if possible, and to return control to a preceding message. This is always a valid key. There are special uses of this key in some particular applications.
]	Operation complete	This key signifies that the user is done with the current operation. If all expected inputs have been met before the operation complete key is pressed, the operation in progress is performed and control passes to the next logical message. If the necessary inputs are not given, this key returns control to a higher level menu or message, thus aborting the current operation. This is always a valid key.

TABLE 2-1. FUNCTION CONTROL KEYS (Contd)

Key	Title	Description
Y	Yes	This key signifies yes in response to a prompt question. This key is not recognized when in the key-in data mode.
N	No	This key signifies no in response to a prompt question. This key is not recognized when in the key-in data mode.
M	Change menu display	This key changes the menu display status. If the menus are off and M is selected, all subsequent menus are displayed until they are turned off again. This key is not recognized when in the key-in data mode.
R	Repaint display	This key redraws the display. After completion of the repaint, control passes back to the message active before R was selected. This key is not recognized when in the key-in data mode.
Z	Window (zoom)	This key modifies the position and size of the part on the screen. The ZOOM menu is displayed. After you select from this menu and give the new zoom constraints, the screen is redrawn and control passes back to the message active before Z was selected. This key is not recognized when in the key-in data mode.
D	Change depth	This key causes the message 1.DEPTH=0.00 to appear with the current ZT depth. Modify it as desired. After entering a value or indicating reject or operation complete, control returns to the message active before D was selected. This key is not recognized when in the key-in data mode.
F	AD-2000	This key returns control to the top level AD 2000 menu. It is not recognized when in the key-in data mode.

TABLE 2-1. FUNCTION CONTROL KEYS (Contd)

Key	Title	Description
C_P	POINT menu	This key (control P) takes system control directly to the POINT menu. It is a legal key at any time.
C_L	LINE menu	This key (control L) takes system control directly to the LINE menu. It is a legal key at any time.
C_A	ARC/CIRCLE/FILLET menu	This key (control A) takes system control directly to the ARC/CIRCLE/FILLET menu. It is a legal key at any time.
C	Read crosshair cursor	This key tells the system to read the crosshair cursor. It is a legal key only when the cursor is active. During entity selection, this is the standard key used to capture the closest entity whose type is legal for the current operation.
P	Point select	This key, used during entity selection, captures the closest point to the cursor. All other selectable entity types are temporarily locked out.
L	Line select	This key, used during entity selection, captures the closest line to the cursor. All other selectable entity types are temporarily locked out.
A	Arc, circle, or fillet select	This key, used during entity selection, captures the arc, circle, or fillet closest to the cursor. All other selectable entity types are temporarily locked out.
O	Other curves select	This key, used during entity selection, captures conics, composite curves, vectors, point set curves, three-dimensional splines, and machining curves.
S	Spline select	This key, used during entity selection, captures the closest two-dimensional spline. All other selectable entity types are temporarily locked out.

TABLE 2-1. FUNCTION CONTROL KEYS (Contd)

Key	Title	Description
T	Text select	This key, used during entity selection, captures dimensioning entities with text only. These are linear, circular, diameter, and angular dimensions; general labels; and notes. The selectable position on these entities is at the lower left of their text.
!	Data capture	This key (exclamation point) is used to capture and save the value of a parameter that has been displayed by AD-2000 and to associate 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. The exclamation point can also be used to accept the displayed value for a parameter when AD-2000 displays a prompt and a default value and then waits for a user entry. In this case, typing an exclamation point followed by RETURN has the same effect as typing in the displayed value.

ENTITY SELECTION PROCEDURE

Some operations require that the user pick or select entities. When an entity has been selected by the user, the system displays a small oval over the entity to indicate that it has been selected. In many cases throughout the AD-2000 system, the following series of prompts is displayed, and the following standard entity selection procedure is used to pick or select entities.

Prompt	User Entry
MODE 1.SINGLE 2.CHAIN 3.REGION	Indicate the manner in which entities on the screen are to be selected. Enter: 1. To use the crosshairs to select individual entities, one at a time. 2. To select a series of connected entities.

Prompt	User Entry
	3. To select all the entities inside a user-specified rectangular region or outside that region.

If 1.SINGLE was selected and the sequence number/pointer select modal (in the MODALS menu) is off, the following prompt is displayed.

Prompt	User Entry
IND ENT	Use the crosshairs to select entities. To end selection of entities, press the] key. The most recently selected entity can be rejected by pressing the [key.

If 1.SINGLE was selected and the sequence number/pointer select modal is on, the system allows the user to choose from alternative methods of selecting entities.

Prompt	User Entry
1.INDICATE 2.SEQ. NO. 3.POINTER	Enter: 1. To use the crosshairs to pick entities. 2. To select entities by entering the sequence numbers of the entities. 3. To select entities by entering the pointer numbers of the entities.

If 1.INDICATE was selected, use the crosshairs to select entities as described previously.

If 2.SEQ. NO. was selected, the system displays:

Prompt	User Entry
1.SEQ. NO. =	Enter the sequence number of the entity to be selected. The sequence numbers of entities in the current part may be determined through use of the display entity sequence number operations in the OUTPUT AND REGENERATION menu.

If 3.POINTER was selected, the system displays:

Prompt	User Entry
ENTITY POINTER =	Enter the pointer number of the entity to be selected.

If 2.CHAIN was selected in response to the MODE prompt, the user can select a chain of connected entities by using the crosshairs to pick first an entity and then a direction along the chain. The system selects for the user all the entities that are connected in series in the direction indicated, until no line or curve is found with an endpoint within 0.013 centimetre (0.005 inch) or until an entity selected by the system is connected with the first entity selected. The entities to be selected must be lines, arcs, splines, or other curves, all with common endpoints (that is, all connected). The following prompts are displayed.

Prompt	User Entry
IND ENT	Use the crosshairs to indicate the first of a series of connected entities.
IND DIR	Use the crosshairs to indicate the direction (or path) to be followed in selecting the entities.
SEL. OK?	The system marks each of the entities selected with an attention indicator (a small oval). Enter: Y — To accept the entities selected. N — To reject the entities selected. These entities are not selected and the system allows the user to select entities again.

If 3.REGION was selected, the following prompts are displayed.

Prompt	User Entry
1.INSIDE 2.OUTSIDE	Enter: 1. To select all the entities that are wholly or partially inside a rectangular region. 2. To select all the entities that are wholly or partially outside a rectangular region.
IND SCREEN POS 1	Use the crosshairs to indicate the location of one corner of a rectangular region.
IND SCREEN POS 2	Use the crosshairs to indicate the location of the diagonal corner of the rectangular region. All entities that are partially or wholly inside or outside this region are selected.

Prompt	User Entry
SEL. OK?	The system marks the entities selected with attention indicators. Enter:
	Y — To accept the entities selected.
	N — To reject the entities selected. These entities are not selected and the system allows the user to select entities again.

ENTITY SELECTION

Each operation in AD-2000 that requires the user to pick entities from those in the user's part will allow only certain types to be selected. For example, if the user creates a line by executing the JOIN TWO POINTS item in the LINE menu, the user can pick only points, not lines or arcs. Table 2-2 specifies the entities that can be selected for many of the operations in AD-2000. An X in a box in this table indicates that the entity in that column can be selected for the operation on that row.

TYPING IN DATA

When the system displays a prompt and a data value, thus requesting that the user enter a value, the user can examine the value displayed, enter a new value, and additionally define displayed values as variables. For example, in the KEY-IN LINE operation, the following prompts are displayed.

```
1.XT1 = >
2.YT1 =
3.ZT = 0.00
4.XT2 =
5.YT2 =
6.ZT = 0.00
```

The greater than symbol on the first line indicates that the system is waiting for typed input on that line. After data for line 1 has been entered and a carriage return is given, the greater than symbol on line 1 is overwritten with a less than symbol (<). A greater than symbol is then displayed on the second line. After data has been typed in for line 2, the greater than symbol is displayed on line 4. In this operation, line 3 is bypassed because it has preset data.

When data has been entered for each line on which the greater than symbol is displayed, the user can open any line for data entry by keying in that line number. The line specified for data entry or modification is crossed out on the display if the user has previously entered data on that line. When all desired entries have been made, the user should press operation complete, at which time the system processes the data entered.

TABLE 2-2. ENTITY SELECTION MATRIX

Menu Number	Point	Line	Circle	Spline	Point Set	String	Other Entities
9.8		X	X	X	X		CONIC
9.9		X	X	X			CONIC
9.12		X	X	X	X		CONIC
9.14		X	X	X	X		
9.17		X	X	X	X		
10.4			X	X			CONIC
10.11			X	X			CONIC
10.12			X	X			CONIC
10.14	X	X	X	X	X		CONIC
10.17		X					
12.2		X	X	X			
12.8		X	X	X			CONIC
13.1	X	X	X				ARRAY GROUP
13.2	X	X	X				ARRAY GROUP
13.3	X	X	X	X			CONIC, GROUP, CROSS-HATCH, NOTE, LABEL
13.4	X	X	X	X	X		CONIC, GROUP, CROSS-HATCH
13.5	X	X	X	X	X	X	
13.6	X	X	X	X	X	X	CONIC, GROUP, DIMENSION LABEL, CROSS-HATCH
13.7	X	X	X	X	X	X	
13.8	X	X	X	X	X	X	
13.10	X	X	X	X			CONIC

TABLE 2-2. ENTITY SELECTION MATRIX (Contd)

Menu Number	Point	Line	Circle	Spline	Point Set	String	Other Entities
16.2		X	X	X			CONIC
16.3		X	X	X			CONIC
16.4	X	X	X	X			CONIC
16.5	X	X	X	X			CONIC
16.6	X	X	X	X			CONIC
16.12	X		X				
16.18	X	X	X	X			CONIC

At any time the user may type in variable names in place of literal values as key-in data. These variables can be up to four characters in length. Variables may be defined and assigned values using GRAPL in the SPECIAL FUNCTIONS menu or by assigning a displayed value to a variable when in the key-in data mode. This latter method is called data capture. By keying in an exclamation point (!) followed by a character string, the value for the current line is assigned to that string. For example, when verifying two points, the following message might appear.

```
1.DXT        = 1.23
2.DYT        = 2.00
3.DZT        = 0.00
4.2-D DIST   = 2.35
5.3-D DIST   = 2.35
```

To capture the DXT value, the user can enter !DX on that line to save the value in the variable DX. Data in this example is displayed to two decimal places, but the variable DX is assigned the unrounded value, which in this case is 1.233408. To capture the 2-D DIST value in the variable D2, the user should repeatedly press the RETURN key until the greater than symbol appears on the fourth line and then enter !D2. The value of D2 would be 2.3497425. When the user has finished capturing values, he should press the] key.

COORDINATES AND AXES

Two different types of coordinate systems are used in AD-2000.

- Model coordinates are the displacements of positions along the x, y, and z axes. The x and y axes are horizontal and vertical in view 1, and the z axis is normal to the screen (or plane of definition) in view 1. The positive x, y, and z coordinate directions are to the right, upward, and out of the screen (toward the viewer), respectively.

- The xt, yt, and zt transform coordinates are 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 in some view other than view 1, especially in some view in which the part has been rotated.

The transform coordinates are useful in views other than view 1; for example, the user 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.

AD-2000 MESSAGES 3

This section is organized by menu headings. Major menu headings are found in the top level AD-2000 menu. Submenus exist for each major menu. Menu headings in this manual are characterized by a series of numbers separated by periods. The first number in each heading represents the number of the AD-2000 top level menu selection, the second number represents the item selected from the second level menu, and so on until a particular function is reached. For example, the heading 1.1 represents item 1, menu display, under the first menu selection on the top level AD-2000 menu, MODALS AND FONTS. Heading 9.3 is the third item, polar displacement, of the ninth menu selection on the AD-2000 menu, POINTS. Messages pertaining to a given menu item are listed under the appropriate menu heading.

1. MODALS AND FONTS

The selections in this menu:

- Display and set certain system parameters.
- Perform operations related to system parameters (for example, change the display font of an existing entity).
- List and display the current values for the system modals.
- Display the title block for the current part.

Other modals that affect particular areas in the system are not in this menu but can be set in other menus. Refer to the drafting modals in the DRAFTING menu.

1.1 MENU DISPLAY

Selecting this modal has the same effect as pressing the M key. When this selection is made, the mode of menu display is toggled: if on, it is turned off; if off, it is turned on. If the menu display modal is on, the name of the menu and all the items in it are displayed on the screen each time that the menu is selected. If this modal is off, only the name of the menu is displayed.

This operation has no system prompts nor user entries.

1.2 CONSTRUCTION MODAL

This modal toggles the construction modal: if it is on, it is turned off; if it is off, it is turned on. The construction modal determines the action of the system after the use of any of the construction operations (for example,

the operations in the POINT, LINE, and ARC/CIRCLE/FILLET menus). If the construction modal is off when any of the construction operations have been executed, the system returns to the menu from which that operation was selected. If the construction modal is on after the use of any of these operations, the system returns to that same operation so that it can be executed again without selecting it from the menu.

This operation has no system prompts nor user entries.

1.3 DISPLAY TOLERANCE

This modal sets the tolerance allowed in the display of curves. A curve is displayed as a series of connected straight lines. This modal sets the length of these lines. Since a smoother curve results from the use of shorter lines, setting this modal to a smaller value results in the display of smoother curves. Since displaying a curve as a smaller number of long lines is quicker, setting this modal to a larger value enables the system to display curves and to repaint displays containing curves more quickly. This modal changes the way in which the curve is displayed but not the mathematical definition of the curve in the data base. Therefore, changing the value of this modal has no effect on operations that are applied to the curve itself. For example, the operation that defines a point at the intersection of two curves places that point at the intersection of the two mathematical definitions of the curves and not necessarily where any of the straight lines used to display the curves cross.

Prompt	User Entry
1.DISP TOL = n.nnnn	Enter the desired display tolerance. Upon entry to the system, this modal has the value 0.0050.

1.4 SYSTEM DECIMAL PLACES

This modal specifies the number of decimal places (digits to the right of the decimal point) allowed when the user keys in numeric data and the number of decimal places displayed when the system displays numeric data.

Prompt	User Entry
1.DEC PLACES = nn	Enter the number of decimal places.

Upon entry to the system, this modal has the value of 4 if English units of measurement were selected. If metric units were selected, the default number of decimal places is 2.

1.5 CURVE FONT

This modal specifies the font to be used to display lines and arcs on the graphics terminal screen.

Prompt	User Entry
1.SOLID 2.DASHED 3.PHANTOM 4.CLINE	Lines and arcs are displayed in the font selected. 1. Solid lines. 2. Dashed lines. 3. Phantom lines are displayed as a long dash followed by two short dashes, and so on. 4. Centerlines are displayed as a long dash followed by a short dash, and so on.

Upon entry to AD-2000, the default font of solid is in effect. This modal changes the display font only for entities created after the modal has been selected.

1.6 MODIFY ENTITY FONT

This modal changes the display font of an existing line or curve.

Prompt	User Entry
1.SOLID 2.DASH 3.PHANTOM 4.CLINE	Select the new font for the entity selected. These fonts are the same as those described for the curve font modal.
MODE 1.SINGLE 2.CHAIN 3.REGION	Use the standard entity selection procedure to pick the entities whose fonts are to be changed.

1.7 MODIFY ENTITY LEVEL/PEN NO.

This modal changes the level number at which an entity exists. For example, if all of the entities on the display have been defined at level 0 (the default initial level), this operation can be used to change some of the entities to level 1, so that those entities can then be blanked and unblanked or deleted through use of the blank level, unblank level, and delete all of a level operations.

This operation may also be used to change the pen number (0, 1, 2, or 3) of an entity. This number is used with the PLOT feature. (Refer to the OUTPUT AND REGENERATION menu.) The default pen number is 0. Pen numbers 0 through 3 are valid.

Prompt	User Entry
IND ENTITY	Use the crosshairs to pick the entity whose level or pen number is to change.

Prompt	User Entry
1.LEVEL = nn 2.PEN NO. = nn	Enter the new level number for this entity or press ! followed by RETURN key to accept the displayed current value. Then enter the pen number or the] key.

1.8 RESERVED FOR FUTURE USE

1.9 CURSOR MODE

This modal is used to tell the system whether positional input is to be made through the crosshairs on the graphics terminal or through an attached digitizer tablet. When AD-2000 is entered, the default setting for this modal causes positional input to be accepted through the crosshairs on the graphics terminal.

Prompt	User Entry
CURSOR 1.CRT 2.TABLET	Indicate the input source. Enter: 1. To use the crosshairs on the graphics terminal. 2. To use an attached digitizer tablet.

1.10 VIEW VECTORS

This modal determines whether the view vectors will be displayed. If the view vector modal is on, making this selection turns it off; if it is off, this selection turns it on. If this modal is on, the view vectors are displayed as arrows in the lower left corner of the screen. These arrows point in the direction of the positive x, y, and z axes. If one of the x, y, or z axis is perpendicular to the display screen, the view vector for that axis is not displayed.

This operation has no system prompts nor user entries.

1.11 SEQ. NO./POINTER SELECT

This modal determines whether sequence and pointer numbers can be used to select or pick entities in the user's part. If the modal is on, each time an operation requires the user to select an entity on the screen, the user is offered the option of picking the entity by entering its sequence number or its pointer number. When this modal is on and the user is required to select an entity, the message:

1.INDICATE 2.SEQ. NO. 3.POINTER

is displayed so that the user can enter 1, 2, or 3 depending on whether he wants to use the crosshairs, enter the sequence number, or enter the pointer number, respectively, to pick an entity.

If the modal is currently off when this modal is selected, the system displays:

Prompt	User Entry
TURN SEQ. NO./POINTER SELECT ON?	Enter: Y To turn the modal on. N To leave the modal off.

If the modal is currently on, the system displays:

Prompt	User Entry
TURN SEQ. NO./POINTER SELECT OFF?	Enter: Y To turn the modal off. N To leave the modal on.

1.12 DISPLAY MODAL STATUS

This modal displays the status of each of the modals that can be set by other selections in this menu plus the number of currently defined views. Figure 3-1 gives a sample of this display.

```
DV SYSTEM MODALS
1.MENUS    OFF
2.CONSTRUCTION    ON
3.DTOL    .005
4.DEC PLACES    4
5.CURVE FONT    SOLID
6.CURVE DENSITY    NORMAL
7.(Reserved for future use.)
8.(Reserved for future use.)
9.(Reserved for future use.)
10(Reserved for future use.)
11CURSOR    CRT
12VIEW VECTORS    OFF
13SEQ #/PTR SEL    OFF
14# OF DEFINED VIEWS    4
15PEN NO.    0
```

Figure 3-1. Sample of Modal Status Display

1.13 DISPLAY TITLE BLOCK

This modal displays on the screen a title block containing information on the current part. The title block does not become a permanent part of the user's part. When the screen is repainted, the title block is not redisplayed. A sample of the title block is given in figure 3-2.

Prompt	User Entry
IND LOWER LEFT OF TITLE BLOCK	Use the crosshairs to indicate where the lower left corner of the title block is to be displayed.

2. BLANK/UNBLANK

The operations in this menu blank and unblank entities in the user's current part. Blanking an entity causes AD-2000 to leave the entity in the current part but to discontinue displaying the entity. Unblanking an entity causes AD-2000 to start displaying the entity again. Unblanking an entity that has not been blanked has no effect.

2.1 BLANK ALL OF A TYPE

This operation blanks all the entities in the current part that are of a specific type or set of types selected by the user.

Prompt	User Entry
(Refer to figure 3-3.)	Select one or more entity types as described in section 2.

When the] key is pressed, the status of all entities that are one of the types selected is set to blanked. The user must request that the screen be repainted (for example, by using the R key) to remove the entities from the display.

2.2 BLANK ALL EXCEPT A TYPE

This operation blanks all entities in the user's current part that are not of a type or set of types selected by the user.

Prompt	User Entry
(Refer to figure 3-3.)	Select one or more entity types as described in section 2.

2.3 BLANK ALL

This operation blanks all entities in the current part. This operation has no system prompts nor user entries.

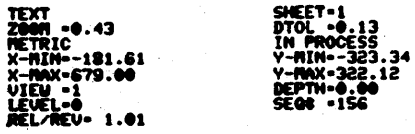


Figure 3-2. Title Block

ENTITY TYPES
1.POINTS
2.LINES AND POINT SETS
3.ARCS AND CIRCLES
4.OTHER CURVES
5.ARRAYS AND GROUPS
6.(Reserved for future use.)
7.LABELS, DIMENSIONS AND NOTES
8.CENTERLINES
9.CROSS-HATCHING

Figure 3-3. Entity Types

2.4 BLANK, SELECT FROM TYPE

This operation blanks entities that are restricted to a set of types chosen by the user and selected from the screen through use of the crosshairs.

Prompt	User Entry
(Refer to figure 3-3.)	Select one or more entity types as described in section 2.
MODE 1.SINGLE 2.CHAIN 3.REGION	Use the entity selection procedure to select the entities to be blanked.

2.5 BLANK, SELECT FROM ALL

This operation blanks entities that are of any type and selected through use of the crosshairs.

Prompt	User Entry
MODE 1.SINGLE 2.CHAIN 3.REGION	Use the entity selection procedure to select the entities to be blanked.

2.6 BLANK ALL EXCEPT n1 TO n2

This operation blanks all entities except those with sequence numbers between the user-entered values. The sequence numbers for entities in the current part can be determined by using the IDENTIFY ENTITIES n1 TO n2 operation in the OUTPUT AND REGENERATION menu.

Prompt	User Entry
1ST SEQ. # =	Enter the low sequence number.
2ND SEQ. # =	Enter the high sequence number.

The system blanks each entity in the current part whose sequence number is less than the low sequence number or greater than the high sequence number.

2.7 BLANK LEVELS

This operation blanks all entities in the current part that are defined at a single level. The user enters the level number in response to the following prompt.

Prompt	User Entry
LEVEL =	Enter the level number.

2.8 UNBLANK ALL

This operation unblanks all entities in the current part that are currently blanked. If the current part contains no blanked entities, this operation has no effect. This operation has no system prompts nor user entries.

2.9 UNBLANK ALL OF A TYPE

This operation unblanks all entities that are currently blanked and of a type or set of types selected by the user.

Prompt	User Entry
(Refer to figure 3-3.)	Select one or more entity types as described in section 2.

2.10 UNBLANK ALL EXCEPT A TYPE

This operation unblanks all entities that are currently blanked and are not in one of the types selected by the user.

Prompt	User Entry
(Refer to figure 3-3.)	Select one or more entity types as described in section 2.

When the RETURN key is pressed, the unblanked entities are displayed.

2.11 UNBLANK SEQ. NO. n1 TO n2

This operation unblanks all entities whose sequence numbers are in a range entered by the user and that are currently blanked.

Prompt	User Entry
1ST SEQ. # =	Enter the low sequence number.
2ND SEQ. # =	Enter the high sequence number.

The system displays each entity whose sequence number is greater than or equal to the low sequence number and less than or equal to the high sequence number.

2.12 UNBLANK LEVEL

This operation unblanks all entities in the current part that are defined at a level selected by the user.

Prompt	User Entry
LEVEL =	Enter the level number.

3. DELETE

The operations in this menu delete entities from the user's part and from the data base. The system marks entities that are deleted with a number sign (#). Deleted entities are erased from the screen when the display is redrawn.

The following prompts are displayed by the system if applicable during use of the operations in this menu.

Prompt	User Entry
CAUTION-DELETE ENTITY?	Indicates that the entity to be deleted is used in the definition of another entity. A reply of Y causes the entity to be made dormant (that is, the data defining the entity remains in the data base, but the entity is not displayed).

Prompt	User Entry
DELETE ENTITIES IN GROUP OR COMPOSITE?	Indicates that a group (refer to the ENTITY MANIPULATION menu), composite curve, or composite surface (composite curves and composite surfaces are not currently supported by AD-2000) has been selected for deletion. Enter: N To delete only the group, composite curve, or composite surface, but not the entities of which it is composed. Y To delete both the group, composite curve, or composite surface and the entities within it.

3.1 DELETE ALL POINTS

This operation deletes all the points in the current part.

Prompt	User Entry
DELETE ALL POINTS?	The system is asking for confirmation that the user wishes to delete all the points in the current part. Enter: N To exit this operation and to return to the DELETE menu without performing this operation. Y To delete all the points.

3.2 FROM TYPE

This operation enables the user to select entities to be deleted from one or more specified types. The operation first allows the user to select the entity types and then enables the user to pick existing entities in his part that are of those specified types.

Prompt	User Entry
(Refer to figure 3-3.)	Select one or more entity types as described in section 2.
MODE:1.SINGLE 2.CHAIN 3.REGION	Use the entity selection procedure to select the entities to be deleted.

The system then deletes the entities and returns to the DELETE menu. The deleted entities are marked with a number sign (#).

3.3 FROM ALL

This operation deletes entities of any type in the user's part. The same prompts are used for entity selection as are used in the delete from type operation. The user is not asked to select types (as in that operation), and when the user selects entities, he can select entities of any type.

3.4 LAST ENTITY

This operation deletes the last entity created. This operation cannot be used more than once without an intervening create operation. That is, if the user deletes the last entity created, the user cannot then use this operation to delete the previously created entity.

This operation has no system prompts nor user responses. The deleted entities are marked with a number sign (#).

3.5 ALL DISPLAYED

This operation deletes all the entities in the current drawing.

Prompt	User Entry	
DELETE ALL?	The system requests confirmation that all entities are to be deleted. Remember that there is no backup for the user's part unless the user has specifically created that backup (for example, through use of the FILE/TERMINATE menu selection). Enter:	
	N or [	To prevent this operation from being performed.
	Y or]	To cause all the entities in the current part to be deleted.

3.6 ALL NOT DISPLAYED

This operation deletes all entities in the current part that are not currently displayed on the screen. This operation has no effect on blanked entities.

This operation has no system prompts nor user entries.

3.7 ALL OF DISPLAYED TYPE

This operation enables the user to select one entity type and then to delete all the entities in the current part that are of that type. This operation has no effect on blanked entities.

Prompt	User Entry
(Refer to figure 3-3.)	Select one or more entity types as described in section 2.

3.8 ALL OF NOT DISPLAYED TYPE

This operation allows the user to select one entity type and then to delete all the entities in the current part that are of that type and are not currently displayed on the screen. This operation has no effect on blanked entities.

The prompts and user responses for this operation are the same as for the delete all of displayed type operation.

3.9 ALL OF A LEVEL

This operation deletes all the entities in the current part that are in a specified level.

Prompt	User Entry
LEVEL =	Enter the level number. All entities defined at that level will be deleted.

4. FILE/TERMINATE

Selecting this entry from the main AD-2000 menu enables the user to perform three tasks, depending upon the responses that the user gives to questions from the system. The user can:

- File his part in part storage so that it will be available the next time he enters AD-2000.
- Retrieve and begin work on an existing part or create and begin work on a new part.
- Exit from AD-2000.

The system first asks the user if he wishes to file the current part. During a design session with AD-2000, the user is actually working on a copy of the part in a temporary work area. The user must file his part if that part and the changes made to it are to be available and are to be accessed again during a later session with AD-2000. The user must not file his part if he wishes to avoid overlaying the original part, thereby destroying the old contents of the part. A part whose status is released or protected cannot be filed.

Prompt	User Entry
FILE?	Enter: Y To file the part. N To omit filing the part.

The part is filed under the name and sheet number that were entered in response to the prompts

KEY-IN NAME

1.SHEET # =

at the beginning of the session with that part. If the part area in the data base is full, the system displays:

TABLES FULL-ACKNOWLEDGE

The user can delete some of the parts that are not needed or can save some parts on tape and then delete them to produce free space in the file. Later the user can define a new and larger part file so that the parts in the old part file can be copied to this larger file.

The system now asks whether the user has finished working on the current part.

Prompt	User Entry
TERMINATE?	Enter: Y To exit from AD-2000 or access another part. N To continue work on the same part. The system returns to the main AD-2000 menu.

If the response to the previous prompt was Y, the system asks whether the user wants to access and begin work on an existing part in part storage or to create a new part, or whether the user wants to exit from AD-2000.

Prompt	User Entry
NEW PART WANTED?	Enter: Y To retrieve an existing part or to create a new part that the user will work on. N To exit from AD-2000 and to return control to the operating system.

If the response to this prompt was Y, the system displays the following prompts.

KEY-IN NAME

1.SHEET # =

5. SPECIAL FUNCTIONS

The operations in this menu provide miscellaneous special functions that include the following:

CANON	Displays and allows the user to modify information on individual entities stored in the data base.
GRAPL	Enables the user to define and assign values to variable names that can then be used to enter values in other AD-2000 operations and to create and modify GRAPL programs.
User defined symbols	Enable the user to define special characters.
Data graphs	Help the user to produce various types of graphs.
Level management	Helps the user in the use of levels.
Attribute management	Enables the user to assign information to an entity and later query the system for that information.
Macro management	Enables the user to define, modify, and execute macro programs composed of sequences of AD-2000 instructions.

5.1 CANON

The operations in this submenu display information on an entity stored in the data base for the current part. These operations also allow the user to modify certain pieces of that information. The format in which the information is displayed, as well as which items can be modified, are indicated below. An entity can be selected by using the crosshairs, by keying in its sequence number, or by keying in its pointer number.

Prompt	User Entry
MODE 1.IND. 2.SEQ. NO. 3.PTR.	Select the mode by which the entity is to be chosen. Enter: 1. To use the crosshairs to pick the entity. 2. To choose the entity by keying in its sequence number. 3. To choose the entity by keying in its pointer number.

If 1.IND. was entered, the following prompts are displayed.

Prompt	User Entry
IND ENT	Use the crosshairs to select the entity whose information is to be displayed.
IS SELECTION OK?	Enter: Y If the attention indicator appears over the desired entity. N If you wish to select a different entity.

If 2.SEQ NO. was selected in response to the mode prompt, the following prompt is displayed.

Prompt	User Entry
1.SEQ. NO. =	Enter the sequence number of the entity whose information is to be displayed.

If no entity with the entered sequence number exists, the following message is displayed.

SEQUENCE NUMBER NOT FOUND

If 3.PTR. was selected in response to the mode prompt, the following prompt is displayed.

Prompt	User Entry
1.POINTER =	Enter the pointer number of the entity whose information is to be displayed.

If no entity with the pointer number entered exists, the following prompt is displayed.

POINTER NOT FOUND
or
BAD PTR HIT YES TO CHECK

The information shown in figure 3-4 is displayed. Press the] key to continue to the next prompt and user entry.

TABl DATA	Explanation
1.POINTER 1038	The pointer number of the entity selected.
2.TYPE 8	The data type of the entity. The entity types and codes are listed in figure 3-5.
3.FORM 1	The data subtype of the entity. This item indicates how the entity was created.
4.ATTN ON	The attention indicator is to be displayed.
5.BLANKED? NO	NO indicates that the entity has not been blanked and that it is to be displayed.
6.DELETABLE? YES	Indicates that the entity can be deleted.
7.DORMANT? NO	A dormant entity is one that is stored in the part but is not displayed.
8.CURVE DENSITY	Indicates whether the entity is to be drawn normal or heavy.
9.FONT SOLID	Indicates the type of line used to display the entity.
10DISPLAY ALL VIEWS	The entity is to be displayed in all views requested.
11GROUP CTR 0	This counter gives the number of groups in which this entity is included.
12SEQ NO 3	The sequence number of the entity.
13"USED IN DEF" CTR 0	This counter gives the number of entities in whose definitions this entity is used.

Figure 3-4. TABl Data (Sheet 1 of 2)

	Explanation
14ATTN PT U,V 644, 2596	The position of the attention pointer. These values locate the attention point in the addressable space on the terminal in use. For example, on a Tektronix 4014, the U and V directions (horizontal and vertical) each contain 4096 addressable points.
15NO OF TAB2 WORDS 1	The number of words of memory used to hold TAB2 data for this entity.
16NO OF TAB3 WORDS 11	The number of words of memory used to hold TAB3 data for this entity.
17LEVEL 0	The level at which this entity is defined.
18VIEW NO 1	The view of definition or work view when the entity was defined.
19PEN NO. 0	The pen number associated with the entity for plotting.

Figure 3-4. TAB1 Data (Sheet 2 of 2)

Type	Name
0	Entity deleted
1	Point
2	Line
3	Circle
4	Conic
5	Spline
6	Composite curve
7	Vector
8	Point set
9	3-D spline
10	Machining curve
11	String
12	Rectangular array
13	Circular array
14	Copious data
15	Group
16	Variable
17	Data graphs
18	Plane
19	Surface of revolution
20	Tabulated cylinder
21	Ruled/developable surface
22	Curve mesh surface
23	Fillet surface
24	Sphere
25	Cylinder
26	Torus
27	Cone
28	Offset surface
29	Composite surface
30	GRAPL created entity
31	
32	Linear dimension
33	Circular dimension
34	General label
35	Diameter dimension
36	Angular dimension
37	General note
38	Centerline
39	Cross-hatching
40	True position
41	
42	
43	
44	
45	Point-to-point
46	Lathe/profile/pocket/3-axis/5-axis
47	
48	
49	Tool
50	
51	
52	Hexahedron
53	Spheroid or spherical shell
54	Circular rod or cylindrical shell
55	Toroid
56	Ellipsoid or elliptical shell
57	Projected solid
58	Rotated solid or shell
59	Composite solid or shell
60	
61	
62	Drawing ID block
63	Forms
64	Path or block (level 6)
65	Grid or parcel/lot (level 2)
66	Node or street (level 5)
67	Symbol or bearing/dist (level 3)
68	Interconnect or arc length/radius (level 4)
69	Tick mark (level 7)

Figure 3-5. Entity Types and Codes

The system then displays:

Prompt	User Entry
DISPLAY TAB2?	Enter: Y To display the information shown in figure 3-6. N To skip display of this information.

The meaning of the information in TAB2 depends on the way in which the entity was defined.

	Explanation
1.VALUE 1 = 4	The meaning of this entry depends on the entity selected.

Figure 3-6. TAB2 Data

If Y was entered and the information does not exist, the system displays:

```
NO TAB2 DATA
```

Press the [key to exit from this operation. Press the] key to continue to the next prompt and user entry.

Prompt	User Entry
DISPLAY TAB3?	Enter: Y To display the information shown in figure 3-7. N To skip display of this information.

	Explanation
1.VALUE 1 = 1.00 2.VALUE 2 = 4.00 . . . 9.VALUE 9 = 237.85 0.VALUE 10 = 56.79	Entity information. The meaning of these items depends on the entity selected.
(Enter the] key to display additional words of data.)	
1.VALUE 11 = 233.86	Additional TAB3 entity information.

Figure 3-7. TAB3 Data

If Y was entered and this information does not exist, the system displays:

NO TAB3 DATA

The system then enables the user to modify selected items (words) within these blocks.

Prompt	User Entry
INSERTIONS AND DELETIONS 1.INSERT TAB2 2.INSERT TAB3 3.DELETE TAB2 4.DELETE TAB3 5.REDISPLAY TAB2 AND TAB3	These prompts are only displayed if the menu display modal is on. Enter: 1. To insert information in the TAB2 block. 2. To insert information in the TAB3 block. 3. To delete information from the TAB2 block. 4. To delete information from the TAB3 block. 5. To redisplay information in either of these blocks.

If either 1 or 2 was selected, the following prompts are displayed.

Prompt	User Entry
INSERT LOC =	Enter the location of the word after which additional data is to be inserted. For example, enter 1 in order to specify that the additional data is to be inserted between the current first and second words of data.
1.VALUE =	Key in the values for one or more consecutive words to be inserted after the specified location. The format for the entry and the allowable range of values depends on the particular entry selected. Pressing the] key before entering a value for any given word terminates entry.

If either 3 or 4 was selected, the following prompts are displayed.

Prompt	User Entry
1.START = 2.END =	Enter the starting word number and the ending word number of the words to be deleted. For example, START = 1 and END = 3 indicate that the current first three words of data are to be deleted.

If either the starting or the ending word number is omitted, the system displays:

```
START > END--REENTER VALUES!
```

If 5.REDISPLAY TAB2 & TAB3 was selected, the system displays

```
DISPLAY TAB2?
```

so that the user can display TAB2 data and TAB3 data or make insertions and deletions as described previously.

5.2 GRAPL

The operations in this submenu enable the user to:

- Define and calculate values for variable expressions.
- Create GRAPL programs that can later be executed to perform certain calculations for the user.
- Run GRAPL programs.
- Move GRAPL variables from the user technology file (UTF) to the run time library (RTL).
- Move GRAPL variables from the RTL to the UTF.
- List the names of variables in the UTF.
- List the names of variables in the RTL.

5.2.1 VARIABLE CALCULATION

This operation enables the user to define a variable and to assign a value to that variable. The name of the variable must be from one to four characters, the first of which must be alphabetic. The value can be specified as either a constant or an expression. Any variables in the right-hand part of the statement must have been previously defined and must exist in the RTL.

Prompt	User Entry
ENTER EXPRESSION (4 CHAR NAME)	Enter the assignment statement in the form: variable = expression

Then:

- The right-hand side of the statement is evaluated.
- The calculated value is assigned to the variable on the left-hand side of the equation.

- The variable and its value are saved in the RTL.
- The variable name and calculated value are displayed in the following format.

```
1.variable = value
```

This variable and its value are then available for use in other variable definitions, in GRAPL programs, as the entry for a keyed-in value for any AD-2000 operation, or during macro execution.

5.2.2 CREATE GRAPL PROGRAM

This operation creates a GRAPL program. A GRAPL program is a sequence of statements assigned to a program name. The program can be executed by invoking the name. (Refer to the run GRAPL program operation in this submenu.)

With the help of GRAPL the user can write FORTRAN-like programs that perform calculations. The results of these calculations can be saved in the RTL as the values of variables to be used later (for example, in generating entities).

The following statements are allowed in a GRAPL program.

- The SIZE statement specifies the size (number of elements) in a subscripted variable (that is, in a one-dimensional array). For example:

```
SIZE XCORD(20),YCORD(20),ZCORD(20)
```

- The assignment statement calculates the value of the expression on the right-hand side of an equal sign (the assignment operator) and stores that value in the variable on the left-hand side of an equal sign. For example:

```
XCORD=30.40
YCORD=XCORD/2.0
```

- The IF statement causes the execution of a single statement if and only if a logical expression is true. For example:

```
IF I.GT.0 THEN GOTO 100
IF I.LT.J THEN I = J
```

- The FOR statement sets a value of a variable and causes repeated execution of a range of statements until a condition specified by a logical expression becomes true. For example, the statements

```
FOR I=0, UNTIL I.GT.10;
    A(I) = 0;
    B(I)=0;
    END ITERATION;
```

cause I to be set to 0, after which the two assignments to A(I) and B(I) are executed repeatedly. After each execution of this range of statements, I is incremented by 1, and the condition specified (I.GT.10) is tested. If the condition is true, then the next statement after the END ITERATION statement is executed. If the condition is false, then the variable (in this case, I) is incremented, and the statements in the range of the FOR statement are executed again.

- The END statement must be the last statement of every GRAPL program.

When this operation is selected, the system displays the following prompt.

Prompt	User Entry
ENTER FOUR CHARACTER NAME	Enter one to four alphanumeric characters.

If no GRAPL program exists in the UTF with this name, the program to be created and entered is assigned to this name. If a GRAPL program exists with that name, that program is retrieved and can be modified. The following text editing commands can be used to create and modify GRAPL programs.

Dn1,n2
Dn1
Instructs AD-2000 to delete lines n1 through n2 in the GRAPL program. In the second form shown, only the line numbered n1 is deleted. If both n1 and n2 are omitted, the command is ignored.

F
Instructs AD-2000:

- To file the updated copy of the current GRAPL program in the UTF.
- To generate object code.
- If there are no compilation errors, to store this object code in the UTF.

If errors are found during compilation, the object code is not filed.

In1
Instructs AD-2000 to accept insertions. Lines typed by the user after this command are inserted after line n1. Terminate entry of each line by pressing RETURN. Terminate insertions with the] key. Use this command to begin entry into and to create a new GRAPL program.

Q
Terminates editing the file without updating the GRAPL program in the RTL. All work done since entering the editing program is lost, unless the additions and changes were filed through use of the F command.

Rn1,n2
Instructs AD-2000 to replace lines n1 through n2 with the lines typed by the user following this command. This command is equivalent to the following commands.

```
Dn1,n2
In1-1
```

Terminate entry of each line by pressing the RETURN key. Terminate insertions with the] key.

Tn1,n2 Tn1 T	Instructs AD-2000 to type out (display) lines n1 through n2 of the GRAPL program. If the second form shown is entered, only the line numbered n1 is typed out. If both n1 and n2 are omitted (the third form shown), all lines are typed out.

5.2.3 RUN GRAPL PROGRAM

This operation initiates the execution of a GRAPL program.

Prompt	User Entry
ENTER FOUR CHARACTER NAME	Enter the name of the program to be executed.

The four-character name entered should be the name of a GRAPL program that has been placed in the UTF through use of the F command in the create GRAPL program operation.

If the object code for the program is not in the UTF, the following message is displayed.

GRAPL OBJECT NOT FOUND-ACKNOWLEDGE

5.2.4 MOVE VARIABLES FROM UTF TO RTL

This operation moves the variables that are currently in the UTF along with their values to the RTL. Thus, this operation provides a way of retrieving variables and values that were saved in the UTF during a previous session with AD-2000. After execution of this operation, the variables requested are available for use (for example, in GRAPL programs).

Prompt	User Entry
ENTER VARIABLE NAME(S)	Enter the names of the variables to be retrieved. The names should be separated by commas. A maximum of 34 characters can be entered.

5.2.5 MOVE VARIABLES FROM RTL TO UTF

This operation saves variables and their values in the UTF. The variables and their values in the RTL are not altered. The user can use this operation to produce backup for his variables in the UTF or to save his variables for use in a later session with AD-2000. Since the UTF is common to all parts in the data base, this operation can be used to move variables from one part to the common UTF and then, if desired, to another part.

Prompt	User Entry
ENTER VARIABLE NAME(S)	Enter the names of the variables to be saved. The variable names should be separated by commas. A maximum of 34 characters can be entered.

5.2.6 LIST TECHNOLOGY FILE VARIABLES

This operation clears the screen and displays the name and (for subscripted variables) the dimension of each variable in the UTF. This operation has no system prompts nor user entries.

5.2.7 LIST RUN TIME LIBRARY VARIABLES

This operation clears the screen and displays the name and (for subscripted variables) the dimension of each variable in the RTL. This operation has no system prompts nor user entries.

5.3 USER-DEFINED SYMBOLS

The operations in this submenu enable the user to define a special set of characters or symbols. These characters or symbols can be used in dimensions, notes, and labels created by operations in the DRAFTING menu, if the character set drafting modal is set for the user-defined character set. When this menu item is selected and before any of the items in this submenu can be executed, the following prompts are displayed.

Prompt	User Entry
KEY-IN SYMBOL-SET NAME	Enter the name of the symbol set to be created or modified.

If a symbol set with this name does not currently exist, a symbol set is created. If a symbol set with this name does exist, that symbol set is retrieved from the UTF and made available so that the user can modify it. The following prompt is displayed.

RETRIEVED

Press the] key to continue to the next prompt.

The user can define a new symbol by creating lines that connect points within a grid. AD-2000 requests the density of the grid that is to be used.

Prompt	User Entry
GRID:1.9 2.17 3.33 4.65	Select the density of the grid. Enter 1, 2, 3, or 4 to use a 9 by 9, 17 by 17, 33 by 33, or 65 by 65 grid, respectively.

The system displays a grid as chosen above. Figure 3-8 gives samples of each grid.

The system then requests the type of operation to be performed.

Prompt	User Entry
MODE:1.DEF/REP 2.DEL 3.DISP 4.LIST	Enter:
	] To be allowed to file the symbol set and to exit from this operation.
	1. To define a new symbol or to replace an existing symbol.
	2. To delete an existing symbol.
	3. To display a symbol.
	4. To list the characters in the symbol set.

If the] key was entered, the system displays:

Prompt	User Entry
DONE?	Enter:
	Y To be allowed to file the part and to exit.
	N To redisplay the mode prompt.

If Y was entered, the system displays:

Prompt	User Entry
FILE?	Enter:
	Y To file the symbol set in the UTF and to exit from this operation. If an old copy of a symbol set with this name exists in the UTF, that old copy is overwritten.
	N To exit from this operation without filing the symbol set.

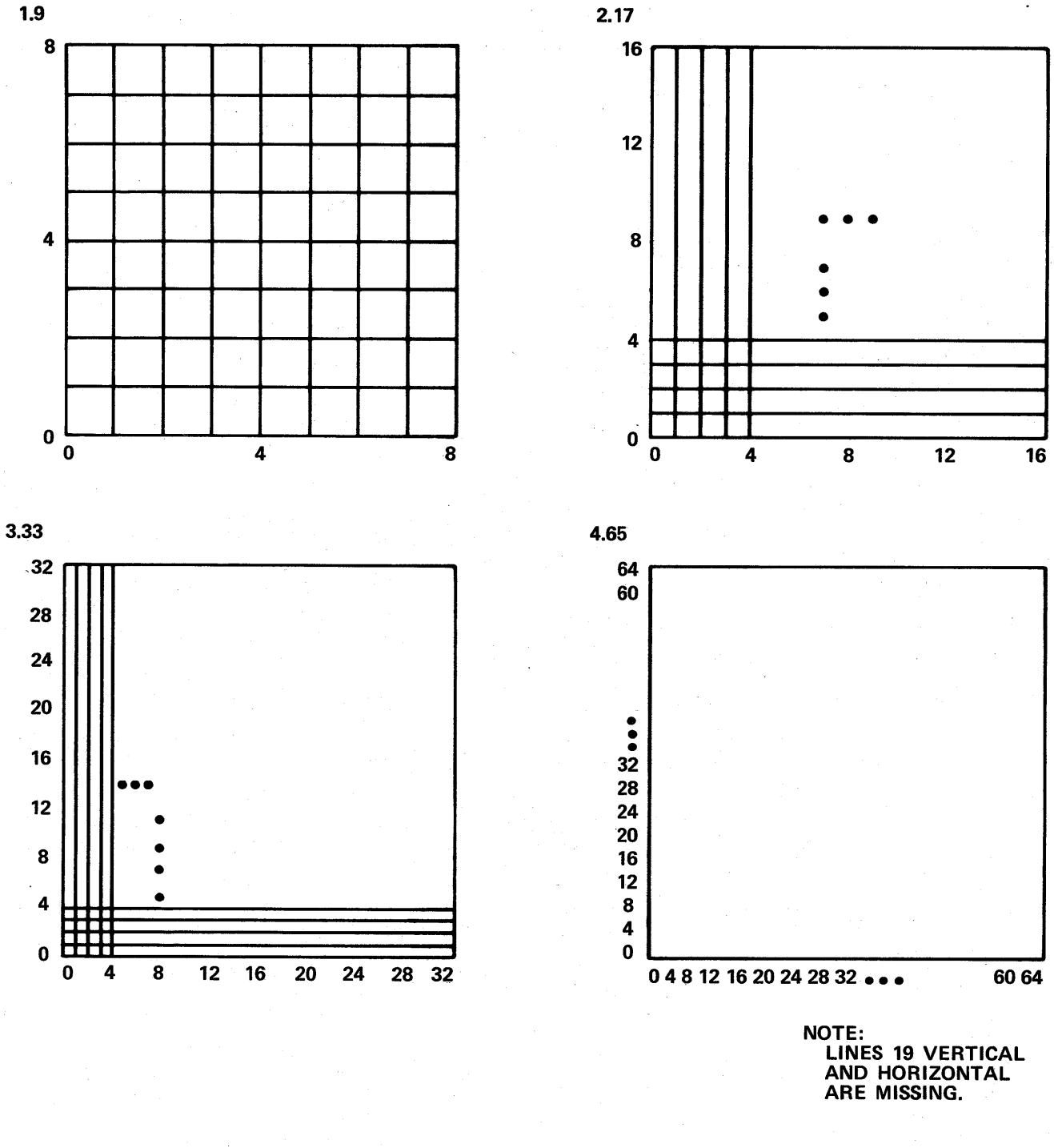


Figure 3-8. Grids for User-Defined Symbols

The system then requests that the user indicate which key on the terminal keyboard is to be associated with the symbol to be defined, replaced, deleted, or displayed. Keys which are not defined by the user retain their standard meaning.

Prompt	User Entry
GIVE KEY CORRESPONDENCE	Enter a single key. This key will be used as the name for the symbol.

Press the [key or the] key to return to the MODE prompt.

5.3.1 DEFINE OR REPLACE SYMBOL

This operation creates a new symbol or character or replaces an existing symbol in the symbol set being modified. The user draws the symbol in a displayed grid. Optionally, the user can begin with one or two existing symbols and can proceed by adding lines to these.

Prompt	User Entry
EXISTING BASE?	Enter: Y To create a symbol by modifying an existing symbol (without altering that existing symbol). N To create a symbol from scratch.

If Y was entered in response to the previous prompt, the following prompt is displayed.

Prompt	User Entry
KEY-IN EXISTING BASE	Press the key associated with the symbol that is to be used as a basis for the new symbol.
START PT	Use the crosshairs to indicate the square on the grid where building the symbol is to begin or press: [To return to the prompt EXISTING BASE?] To return to the MODE prompt.

The system then requests that the user build the symbol.

Prompt	User Entry
DRAW	Use the crosshairs to indicate the next square to be connected with the previous one.

The user can continue using the crosshairs to indicate additional lines in the symbol or he may press:

[To erase the previous line. Any number of lines can be erased consecutively. Then use the crosshairs to indicate the point on the grid where the process of building the symbol is to be continued.

] To indicate that he is finished constructing this symbol.

If the user has entered], the system displays:

Prompt	User Entry
SYMBOL COMPLETE?	Enter: Y To store the symbol in the UTF. N To continue to the next prompt.
CONTINUE?	Enter: Y To continue drawing and to go to the DRAW prompt. N To return to the MODE prompt and erase symbol.

If there is no more room in the work area, the following message is displayed.

TABLES ARE FULL

Enter Y or N to exit from the operation.

5.3.2 DELETE SYMBOL

This operation deletes a symbol from the character set currently being modified.

Prompt	User Entry
GIVE KEY CORRESPONDENCE	Press the key associated with the symbol to be deleted.

The system displays:

DELETED

5.3.3 DISPLAY SYMBOL

This operation displays an existing symbol in the character set currently being modified.

Prompt	User Entry
GIVE KEY CORRESPONDENCE	Press the key associated with the symbol to be displayed.

If no symbol in the current character set is associated with the key, the following message is displayed.

NOT FOUND

Press the] key to continue.

5.3.4 LIST SYMBOLS

This operation lists the symbols in the character set being modified. The system displays the keys that correspond to the special symbols defined in the symbol set. This operation has no system prompts nor user entries.

5.4 LEVEL MANAGEMENT

The operations in this submenu help the user take advantage of the level facilities provided by AD-2000. These operations:

- Change the current level.
- Define a level name.
- Delete the name associated with a level number.
- List the level names.
- Initialize the level table.

5.4.1 CHANGE LEVEL/PEN NO.

This operation changes the current level. After execution of this operation, entities created by the user are at the level specified. When AD-2000 is entered, entities are defined at level 0.

Prompt	User Entry
1.LEVEL = n 2.PEN NO = n	Enter the new level and pen numbers.

5.4.2 DEFINE LEVEL

This operation enables the user to define a name to be associated with a level.

Prompt	User Entry
ENTER LEVEL n NAME	The system asks for the lowest level number which has still no name associated with it.

5.4.3 LIST LEVELS

This operation lists all existing level names. This operation has no system prompts nor user entries.

5.4.4 DELETE LEVELS

This operation deletes a name associated with a level. Only the names associated with the levels are deleted; the entities defined at each level are not deleted.

Prompt	User Entry
1.LEVEL =	Enter the number of the level whose name is to be deleted.

5.4.5 INITIALIZE LEVELS

This operation deletes all current level names. Use this operation with care.

Prompt	User Entry
INITIALIZE?	Enter: Y To delete all names currently associated with levels. N To avoid deleting all defined level names.

If Y was entered and a level table was already defined, the system displays:

Prompt	User Entry
DELETE LEVEL TABLE NAMED xxxx	The system displays the name of each level table to be deleted. Enter: Y To delete the level (but not the entities in it).

Prompt	User Entry
	N To avoid deleting the specified level table.
KEY IN LEVEL TABLE NAME	Enter the four-character level table name.

5.5 ATTRIBUTE MANAGEMENT

The operations in this submenu enable the user to associate attributes with individual entities in his part. Each attribute has an attribute name and an associated value. In addition, the user can attach subattribute character strings to any attribute name. Then the user can, using operations in this submenu, search the entities in his part for:

- The number of entities having a specified attribute name.
- The entities that have a specified attribute name.
- The entity that has a specified attribute name and the minimum or maximum value for that attribute.
- The number of entities that have a specified attribute and which satisfy a specified constraint.
- The sum of the values of a specified attribute.

5.5.1 ATTRIBUTE CREATION

This operation creates an attribute: it associates a name and value pair, entered by the user, with an entity selected by the user.

Prompt	User Entry
IND ENT	Use the crosshairs to pick the entity with which the attribute is to be associated.
IS SELECTION OK?	The system marks the entity selected with an attention indicator. Enter: Y To accept the entity indicated. N To reject the entity indicated and to make another selection.
ENTER ATTRIBUTE NAME	Enter the name for the attribute. The name can be from 1 to 32 alphanumeric characters, the first of which must be alphabetic.
1.VALUE 01 =	Enter the numeric value for this attribute. The value may be entered with or without a decimal point but is stored as a real number.

The system then requests that the user enter values 02, 03, and so on until the user presses the] key. The system then requests that the user enter subattribute strings.

Prompt	User Entry
ENTER SUB-ATTRIBUTE STRING NO. 1 2 . . . 8	Enter a 1- to 32-character text string to be associated with the attribute name. From zero to eight subattribute strings can be entered. To exit from this operation, press the] key as the first character of a string.

5.5.2 INTERROGATE ATTRIBUTES

This operation searches the current part for those entities that have a specified attribute or for those entities that have a specified attribute that satisfies a given relation or constraint.

Prompt	User Entry
1ST SEARCH FORM 1.ATTRIBUTE 2.SUBATTRIBUTE 3.BOTH	Select the level at which the search is to be performed. Enter: 1. To search for those entities having a specified attribute name. 2. To search for those entities having a specified subattribute text string. 3. To search for both those entities having a specified attribute name and a subattribute text string.
SELECT MODE 1.RETRIEVE 2.IDENTIFY MINIMUM 3.IDENTIFY MAXIMUM 4.FIND TOTAL 5.CONSTRAINED RETRIEVE 6.DISPLAY	Indicate the type of search to be made. Enter: 1. To find the number of entities in the part having the attribute to be specified. 2. To find the entity that has the smallest value associated with the attribute (to be specified) for all entities in the part having that attribute. 3. To find the entity that has the greatest value associated with the attribute (to be specified) for all entities in the part having that attribute.

Prompt	User Entry
	4. To find the sum of the values associated with an attribute for all entities in the current part.
	5. To find the number of entities that have a given attribute and have a value associated with that attribute that satisfies a user-specified condition.
	6. To display all existing attributes in the part.

If 1.RETRIEVE, 2.IDENTIFY MINIMUM, 3.IDENTIFY MAXIMUM, 4.FIND TOTAL, or 6.DISPLAY was selected, the system displays:

Prompt	User Entry
ENTER DESCRIPTOR	Enter the attribute, and optionally, the subattribute string for which the system is to search.

If 1.RETRIEVE was selected, the system displays:

```
nn   xxxx    FOUND
```

nn Number of entries found.

xxxx Attribute or substring typed in response to the ENTER DESCRIPTOR prompt.

If 2.IDENTIFY MINIMUM, 3.IDENTIFY MAXIMUM, or 4.FIND TOTAL was selected, the system displays

```
MINIMUM xxxx = n.nnnn

        or

MAXIMUM xxxx = n.nnnn

        or

TOTAL xxxx = n.nnnn
```

depending on the option selected.

xxxx Attribute or substring that was typed in response to the prompt ENTER DESCRIPTOR.

n.nnnn Minimum, maximum, or total.

If 5.CONSTRAINED RETRIEVE was selected in response to the previous prompt, the system requests the condition that an attribute value must satisfy in order that the entity be retrieved.

Prompt	User Entry
CONSTRAINT RELATIONS 1.LESS THAN 2.LESS THAN OR EQUAL 3.EQUAL 4.NOT EQUAL 5.GREATER THAN OR EQUAL 6.GREATER THAN	Select one of the displayed relations.
1.VALUE =	Enter the value.

An entity will be retrieved only if the relation

atrval relop entrval

is true, where:

atrval Value associated with the attribute associated with the entity.

relop Relational operator selected in response to the CONSTRAINT RELATIONS prompt.

entrval Value entered in response to the previous prompt.

The system then requests the attribute or subattribute name.

Prompt	User Entry
ENTER DESCRIPTOR	Enter the attribute name or (if the search is to be conducted for a subattribute) a subattribute text string.

If no value satisfying the constrained relation is found, the system displays:

NONE FOUND

If values that satisfy the relation are found, the system displays:

nn XXXXs FOUND

nn Number of values satisfying the relation.

XXXX Attribute or subattribute string.

5.5.3 ATTRIBUTE DELETE

This operation is not implemented.

5.6 MACRO MANAGEMENT

The operations in this submenu enable the user to create, modify, and execute macro programs. AD-2000 macro programs are strings of AD-2000 entity creation commands. These commands either can be generated automatically by the system once the user is in macro construction mode or can be typed in by the user through use of the AD-2000 editor. When the macro is executed, the AD-2000 commands create entities.

The user can also enter this submenu to turn off macro construction mode. Therefore, if the user is in macro construction mode, the following prompt is displayed before the MACRO MANAGEMENT submenu is displayed.

Prompt	User Entry
CONTINUE MACRO CONSTRUCTION MODE?	Enter: Y To return to a higher level menu and to continue entering commands into the previously referenced macro by interactively executing AD-2000 commands. N To turn off the macro construction mode, to end construction of the macro program that was started by the START MACRO CONSTRUCTION MODE operation in this submenu, and to begin a new operation within this submenu.

5.6.1 CREATE POSTPROCESSOR INSERT MACRO

This operation is not implemented.

5.6.2 CREATE MACRO COMMAND STRING

This operation creates a macro program or allows the user to modify a macro program.

Prompt	User Entry
ENTER FOUR CHARACTER NAME	Enter a one- to four-character name.

The on-line AD-2000 command editor is invoked. Item 5.2.2 (CREATE GRAPL PROGRAM) in this menu (special functions) describes the editor commands. The format for strings within the macro body is described in figure 3-9.

Macro A graphic entity definition language (created as a graphic sequence and invoked after changing if required or entered as a COMMAND list).

Commands and Lexicon (each is considered an atom)

A	Current angle (data identifier)
AEND	End angle (data identifier)
AGO	Start angle (data identifier)
C	Circle (COMMAND)
CIRnn	Circle entity name (CIR05=CIR5=CR5=CR05)
DS	Circle (data identifier)
DX	Delta X (data identifier)
DY	Delta Y (data identifier)
DZ	Delta Z (data identifier)
L	Line (COMMAND)
LINnn	Line entity name (LIN02=LIN2=LN2=LN02)
P	Point (COMMAND)
PTnn	Point entity name (PT07=PT7)
R	Radius (data identifier)
X	X-coordinte (data identifier)
XL	XLARGE (position indicator)
XS	XSMALL (position indicator)
Y	Y-coordinate (data identifier)
YL	YLARGE (position indicator)
YS	YSMALL (position indicator)
V	Miscellaneous value (data identifier)
Z	Z-coordinate (data identifier)

All atoms (names and values) are separated by commas.

A COMMAND string causes the definition of one entity.

A COMMAND is the first entry in every COMMAND string.

A semicolon separates COMMAND strings.

The number following a COMMAND indicates the menu choice for that COMMAND.

Names are assigned in sequential order (PT1, PT2, PT3, LN1, and so on). The default name assignment can be overridden by putting the desired name in parentheses after the COMMAND; for example, P8(PT11) assigns the name PT11 to the defined curve endpoint. If no name is entered, the entity is assigned the next sequential name for the entity type. The numeric name sequence continues from the last name used, either default or user-entered.

Example:

```
P2,X,XORG,Y,YORG,Z,0.0;
P4,PT1,DX,DX1,DY,0.0000,DZ,0.0000;
P4,PT1,DX,DX1,DY,DY1,DZ,0.0;
P4,PT1,DX,0.0000,DY,DY1,DZ,0.0000;
P4,PT4,DX,DX3,DY,DY2,DZ,0.0;
P4,PT3,DX,DX2,DY,DY2,DZ,0.0;
L3,PT1,PT2;
L3,PT6,PT5;
C7,PT5,PT3,PT2;
C7,PT1,PT4,PT6;
```

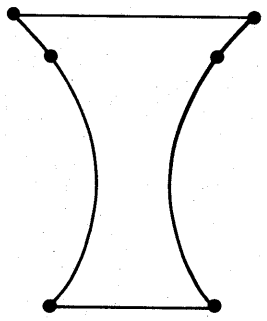


Figure 3-9. Macro Lexicon

5.6.3 EXECUTE MACRO

This operation initiates execution of a macro. If the macro commands use variables for entity definition, the current values of these variables in the RTL will be used.

Prompt	User Entry
ENTER FOUR CHARACTER NAME	Enter a one- to four-character name. This string should name a program in the UTF.

If a macro program by this name does not exist in the UTF, the following message is displayed.

```
xxxx DOES NOT EXIST
```

xxxx Name of the macro program that could not be located.

5.6.4 START MACRO CONSTRUCTION MODE

Once the user is in macro construction mode, for each command that he executes and that creates a basic entity (a point, line, arc, or circle), the system generates a command and saves that command in the macro program.

Prompt	User Entry
ENTER FOUR CHARACTER NAME	Enter a one- to four-character name.

After this command has been executed, any AD-2000 commands that create points, lines, arcs, and circles and that are now entered by the user are collected to form the executable body of the macro program. This process continues until the user turns off the macro construction mode by entering the MACRO MANAGEMENT submenu of the SPECIAL FUNCTIONS menu and responds N to the prompt:

```
CONTINUE MACRO CONSTRUCTION MODE?
```

If variables have been used to specify the values of coordinates or characteristics in the entity creation commands, then the current value of each variable is used each time the macro is executed. Therefore, the user can control the entities created by changing the values of the variables referenced. The GRAPL submenu in the SPECIAL FUNCTIONS menu can be used to make these changes.

Commands may be added to a macro program either by entering macro construction mode and then executing AD-2000 entity creation commands or by using the AD-2000 command editor described in GRAPL. Once macro construction mode has been terminated for a particular macro program, then only the editor can be used to add to or modify that program. The user should be aware that if an entity is deleted while in macro construction mode, the command generated in the macro program by the creation of that entity is not removed from that program. Therefore, the entity will be created when the macro program is executed. This applies as well to basic entity operations such as modify/replace, which in effect delete and create an entity in a single operation. When the macro program is executed, both the original entity and its modified replacement are created.

The format for commands in macro programs is described in figure 3-9.

5.7 RESERVED FOR FUTURE USE

5.8 DATA BASE DUMP

This operation enables the user to display the contents of the arrays used by the system and stored in COMMON. The user can also modify the contents of these arrays.

Prompt	User Entry
ALLOW COMMON TO BE MODIFIED?	Enter: Y To modify the contents of the array words displayed. N To cause the system to protect the contents of the arrays from modification by the user.
DUMP 1.INTEGER 2.REAL	Choose the type of array whose contents are to be displayed. Enter: 1. To display the contents of an integer array. 2. To display the contents of a real array.

If 1.INTEGER was selected, the system displays:

Prompt	User Entry
INTEGER ARRAYS 1.TAB1 2.TAB2 3.TAB3 4.DBUF 5.EC 6.GCA 7.IMODE 8.GC 9.GOSW 10GI 11MADD 12MMP 13PAGE 14PRTNA 15IVIEW 16TEMI 17STAB1	Select the integer array whose contents are to be displayed.

Prompt		User Entry
1.xxxx 2.xxxx	FROM = TO =	Enter the starting and ending word numbers. The system displays the contents of all words between and including these two addresses. If the user presses] with no entry, the system displays the first 10 words, and subsequently, the next 10 words. xxxx is the name of the requested array. A sample of this display is shown in figure 3-10.

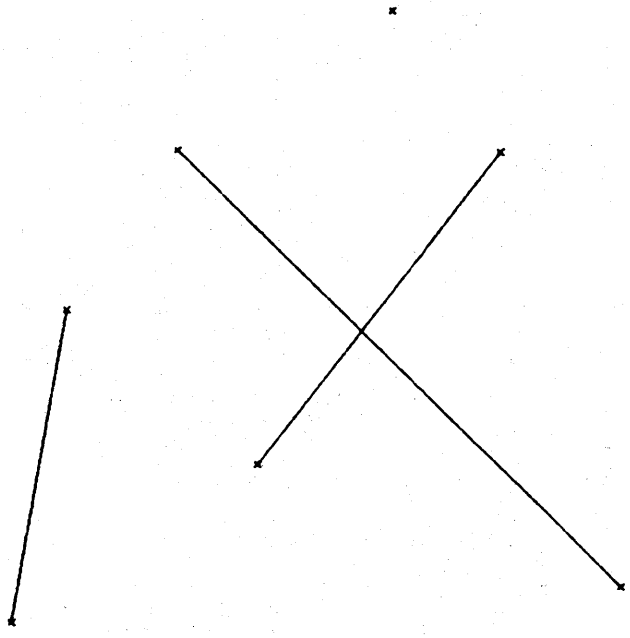
If 2.REAL was selected in response to the DUMP prompt, the system displays:

Prompt	User Entry
REAL ARRAYS 1.TAB3 2.TAB4 3.DEPTH 4.ECURV 5.ESURF 6.GR 7.MATH 8.RMODE 9.ZOOM 10TEMR 11STAB2	Select the real array whose contents are to be displayed.
1.xxxx FROM = 2.xxxx TO =	Enter the starting and ending word numbers. The system displays the contents of all words between and including these two addresses. If the user presses] with no entry, the system displays the first 10 words, and subsequently, the next 10 words. xxxx is the name of the requested array. A sample of this display is shown in figure 3-11.

5.9 DATA GRAPHS

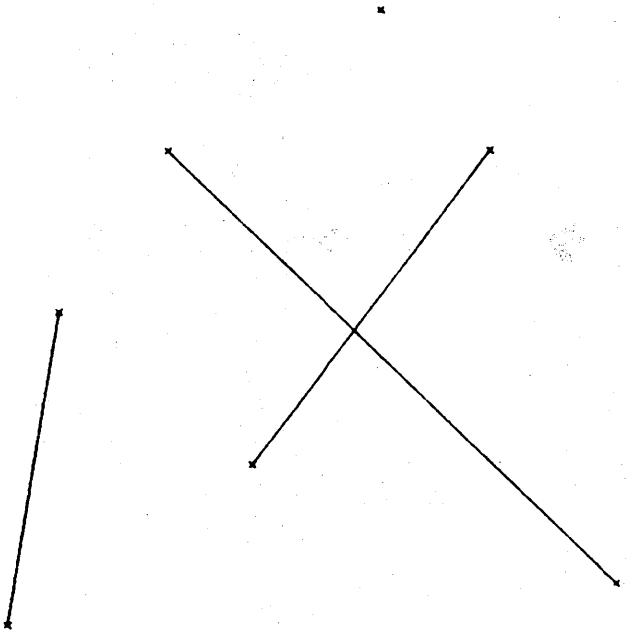
The operations in this submenu help the user to produce log, linear, line, point plot, bar, and circular graphs.

Prompt	User Entry
1.RETRIEVE FROM UTF 2.SELECT FROM SCREEN 3.CREATE NEW TEMPLATE	Select the method to be used to produce a border template. Enter: 1. To retrieve a template created earlier and filed in the UTF. 2. To select a template on the screen. 3. To create a new template.



```
END-ENT, AD-0000 ? 5.8
SPECIAL FUNCTIONS
ALLOW COMMON TO BE MODIFIED?? N
DUMP 1-INTEGER 2-REAL? 1
INTEGER ARRAYS? M
1.TAB1
2.TAB2
3.TAB5
4.DBUF
5.EC
6.GCA
7.IMODE
8.GC
9.GOSU
10GI
11MADD
12MMP
13PAGE
14PRTMA
15IVIEW
16TEMI
17STAB1? 1
1ITAB1  FROM=1          ? 1
2ITAB1  TO  =1024       ? 10
1ITAB1(1)    =41        ? 3
2.TAB1(2)    =5
3.TAB1(3)    =3
4.TAB1(4)    =0
5.TAB1(5)    =274877972222
6.TAB1(6)    =4462222222222
7.TAB1(7)    =0
8.TAB1(8)    =274877972750
9.TAB1(9)    =4462222222222
0.TAB1(10)   =0
1)TAB1  FROM=11         ? E
2.TAB1  TO  =1024
INTEGER ARRAYS
1.TAB1
2.TAB2
3.TAB5
4.DBUF
5.EC
6.GCA
7.IMODE
8.GC
9.GOSU
10GI
11MADD
12MMP
13PAGE
14PRTMA
15IVIEW
16TEMI
17STAB1? M
```

Figure 3-10. Data Base Dump - Integer Array



```
AD-2000 SPECIAL FUNCTIONS
ALLOW COMMON TO BE MODIFIED? Y
DUMP  1.INTEGER 2.REAL? 1
INTEGER ARRAYS? 1
1)TAB1   FROM=1                ? ]
2.TAB1   TO   =1024
1)TAB1(1)    =41               ? ]
2.TAB1(2)    =5
3.TAB1(3)    =3
4.TAB1(4)    =0
5.TAB1(5)    =.274877972222
6.TAB1(6)    =.4462222222222
7.TAB1(7)    =0
8.TAB1(8)    =.274877972750
9.TAB1(9)    =.4462222222222
0.TAB1(10)   =0
1)TAB1(11)   =.131329          ? [
2.TAB1(12)   =.4462222222222
3.TAB1(13)   =.5653688
4.TAB1(14)   =.196865
5.TAB1(15)   =.4462222222222
6.TAB1(16)   =.9439084
7.TAB1(17)   =.262401
8.TAB1(18)   =.4462222222222
9.TAB1(19)   =.9421760
0.TAB1(20)   =.327937
INTEGER ARRAYS? [
DUMP  1.INTEGER 2.REAL? 2
REAL ARRAYS ? M
1.TAB3
2.TAB4
3.DEPTH
4.ECURV
5.ESURF
6.GR
7.MATH
8.RMODE
9.ZOOM
10TEMP
1)STAB2? M.1
1)TAB3   FROM=1                ? ]
2.TAB3   TO   =1024
1)TAB3(1)    =-47.00           ?
2.TAB3(2)    =-107.95
3.TAB3(3)    =-123.55
4.TAB3(4)    =0.00
5.TAB3(5)    =-170.72
6.TAB3(6)    =-207.37
7.TAB3(7)    =0.00
8.TAB3(8)    =-95.45
9.TAB3(9)    =-207.01
0.TAB3(10)   =0.00
```

Figure 3-11. Data Base Dump - Real Array

If 3.CREATE NEW TEMPLATE was selected, the following prompts are displayed.

Prompt	User Entry
TYPE:1.LINEAR/LOG 2.PIE(CIRCULAR)	Choose the type of graph to be produced. Enter: 1. To produce a log/log, log/linear, linear/log, or linear/linear graph. 2. To produce a circular graph.
GRAPH PHYSICAL CHARACTERISTICS 1.HEIGHT = 2.WIDTH = 3.BORDER DX = 4.BORDER DY =	Enter the size of the paper on which the graph is to be drawn. Enter the width of the left and right margin (DX) and the width of the top and bottom margin (DY). A border will be drawn at these distances from the edge of the paper.

The system then requests scale information to be used in producing the graph. Up to four scales may be entered, but only the most recently entered scale is used.

Prompt	User Entry
SCALE:1.HORIZONTAL 2.VERTICAL	Indicate whether the information about to be entered is to be used for horizontal or vertical scaling. Enter: 1. To enter information for a horizontal scale. 2. To enter information for a vertical scale.
SCALE 1 X-ORIGIN =	Enter the origin (with respect to the left border) of the scale being entered.
Y-ORIGIN =	Enter the origin (with respect to the bottom border) of the scale being entered.
LENGTH =	Enter the length of the scale.
INTERVAL =	Enter the interval. There are two tick marks per interval.
DELTA/INT =	Enter the change per interval.
MIN. VALUE =	Enter the minimum value. This is the value of the left side of a horizontal scale or the bottom of a vertical scale.
TICK MARK =	Enter the length of the tick marks to be drawn.

Prompt	User Entry
CHAR. HT. =	Specify the height of the characters to be drawn.
SCALE TYPE:1 LINEAR 2 LOG	Specify whether the scale just entered is to be used as a linear or a logarithmic scale.

The system then requests information to be used to draw the title on the graph.

Prompt	User Entry
TITLE:1 TOP/RIGHT 2 BOTTOM/LEFT	Indicate the location and orientation of the title. Enter: 1. If the scale is horizontal and the title goes on top or if the scale is vertical and the title is to the right of the scale. 2. If the scale is horizontal and the title goes on the bottom or if the scale is vertical and the title is to the bottom of the scale.
ENTER TITLE	Enter a title consisting of 0 to 34 alphanumeric characters.

The system now requests information to be used to produce labels in the graph.

Prompt	User Entry
LABEL PARAMETERS 1.TEXT HT =	Enter the text height.
TEXT ROT. =	Enter the angle of rotation. The lines of text are drawn at this angle. The angle is measured in a counterclockwise direction from the positive x axis.
X-ORIGIN = Y-ORIGIN =	Enter the origin. The lower left corner of the first line of text will be placed at this location.
ENTER LABEL	Enter a label consisting of 0 to 34 alphanumeric characters.

The system then requests information used to construct the grid lines on the graph.

Prompt	User Entry
REFERENCE GRIDS 1.NO. OF X = 2.NO. OF Y =	Enter the number of vertical and horizontal grid lines to be drawn.

Prompt	User Entry
X-GRIDS FONT:1.SOLID 2.DASHED 3.DOTTED	Select the font for the vertical grid lines. Enter: 1. For solid lines. 2. For dashed lines. 3. For dotted lines.
X-MIN =	Enter the location (with respect to the left border) of the leftmost vertical grid line.
Y-MIN =	Enter the location (with respect to the bottom border) of the bottommost horizontal grid line.
DELTA =	Enter the displacement between grid lines.
LENGTH =	Enter the length of the grid lines.
Y-GRIDS FONT:1.SOLID 2.DASHED 3.DOTTED	Select the font for the horizontal grid lines.
X-MIN =	Enter the x-minimum value.
Y-MIN =	Enter the location (with respect to the bottom border) of the bottommost horizontal grid line.
DELTA =	Enter the displacement between grid lines.
LENGTH =	Enter the length of the grid lines.

The system then requests information used to produce a point or line plot, a histogram, or a bar graph. The user may repeat this set of prompts so that up to four separate plots can be constructed on the same graph.

Prompt	User Entry
PLOT 1 1.POINT PLOT 2.LINE PLOT 3.FUNCTION 4.HISTOGRAM 5.HORIZONTAL BAR 6.VERTICAL BAR	Select the style of graph to be drawn. Enter: 1. To produce a plot composed of individual points. 2. To produce a plot composed of a single smooth line. 3. To produce a plot produced by a function. 4. To produce a histogram.

Prompt	User Entry
	5. To produce a bar graph with horizontal bars.
	6. To produce a bar graph with vertical bars.

If 1.POINT PLOT was selected in response to the previous prompt, the following prompt is displayed.

Prompt	User Entry
SYMBOL:1 0 2 + 3 # 4 * 5 X 6 .	Select the symbol to be used to mark the location of individual points in a point plot.
	1. Zero (0)
	2. Plus (+)
	3. Number sign (#)
	4. Asterisk (*)
	5. The letter X
	6. Period (.)
PLOT 1 DATA MODE:1 LITERALS 2 VARIABLE	Indicate the mode in which the location of individual points is to be entered. Enter:
	1. To enter x and y coordinates through the keyboard.
	2. To specify a subscripted variable whose values indicate point plot locations.

5.10 THROUGH 5.12 RESERVED FOR FUTURE USE

5.13 USER TEXT ENTRY

This operation enables the user to enter and edit files containing lines of text in the UTF. The user first enters the text file name, after which the system calls the editor so that the user can enter and modify text.

Prompt	User Entry
ENTER FOUR CHARACTER NAME	Enter the name of the text file.

The user can now use the editor to make additions and changes to the file, to display the file, and to file it in the UTF.

6. PART/PATTERN MANAGEMENT

Operations in this menu provide backup services for the user's parts. These operations:

- Store a part on a sequential file.
- Retrieve a part from a sequential file.
- List the user's parts.
- Delete a part.
- Change the status of a part.

This menu also contains operations that enable the user to employ patterns in the construction of parts. These operations:

- Create a pattern.
- Retrieve a pattern and place a copy of it in the user's part.
- Delete a pattern.
- List the patterns in the user's pattern library.
- Initialize the pattern library.
- Save the user's patterns on a sequential file.
- Retrieve patterns stored on a sequential file.

In addition to the services provided for parts and patterns, this menu has operations:

- To manage the UTF.
- To provide data base information.

6.1 SAVE PARTS ON TAPE

This operation stores a user's part on local file TAPE1. After finishing the session with AD-2000, the user can make this local file permanent on mass storage or can copy it to a magnetic tape file.

Prompt	User Entry
PART SAVE	No entry is required.

The system requests that the user indicate which parts and sheets are to be saved.

Prompt	User Entry
MODE: 1.SINGLE 2.ALL	Enter: 1. To be allowed to specify the name and sheet number of each part to be saved and then to save one part at a time. 2. To save all of the parts currently in the data base file.

If 1.SINGLE was chosen in response to the MODE prompt, the system asks the user to enter the name and sheet number of the part to be saved.

Prompt	User Entry
KEY IN NAME	Enter the name of the part.
SHEET # =	Enter the sheet number of the part to be saved.

If the save operation is completed successfully, the system displays:

```
xxxxxxxxxxxxx, SAVED
```

xxxxxxxxxxxxx is the part name. If the requested part cannot be found, the system displays:

```
xxxxxxxxxxxxx NOT FOUND
```

If 2.ALL was requested in response to the MODE prompt, after all parts in the data base have been saved, the system displays:

```
ALL PARTS SAVED.
```

6.2 RESTORE/MERGE PARTS FROM TAPE

This operation retrieves a part from a local file, TAPE1, and places it in the data base file in use. After executing this operation, the part will look just as it did when the user last stored it on a sequential file or tape. The part to be restored must be on the local file named TAPE1 before starting the session with AD-2000.

Prompt	User Entry
PART RESTORE	No entry is required.

Then the system requests that the user indicate which parts on the tape are to be restored.

Prompt	User Entry
MODE 1.SINGLE 2.ALL	Enter: 1. To specify individual parts and sheets to be restored.

Prompt	User Entry
	2. To restore all the parts on the local file TAPE1.

If 1.SINGLE was selected, the system requests the name and sheet number of the part to be retrieved.

Prompt	User Entry
KEY IN NAME	Enter the name of the part.
SHEET # =	Enter the sheet number of the part to be saved.

If a part with the same name is already in the data base, the system displays:

Prompt	User Entry
xxxxxxxxxxxxx DUPL CONT	xxxxxxxxxxxxx is the part name. Enter:
	Y To delete the part in the data base and to copy the part from tape in its place.
	N To avoid deleting the part in the data base and to omit copying the part from the storage file.

If the local file TAPE1 contains no parts, the system displays:

```
NO PARTS ON TAPE
```

If the area in the data base used for part storage is full, the system displays:

```
NO ROOM ON DISC
```

If 2.ALL was selected in response to the MODE prompt, when the operation has been completed successfully, the system displays:

```
ALL PARTS RESTORED
```

6.3 LIST PARTS

This operation displays a list of the parts currently in the data base. The display includes: the part name and sheet number, the version (date and time of day) of the latest filing of the part on mass storage, the number of words required for storage, and the location of the part on mass storage. Figure 3-12 shows an example of this display. There are no prompts nor user entries for this operation.

HIT YES OR NO TO GO ON?

PART LIST

* 1520 7/ 9/79

NAME	SHEET	TIME	DATE	LOCATION	SIZE	REL/REV
S	1	* 1543	7/ 5/79	0	54	1.02
CDCPATT	1	* 848	7/ 9/79	1346	182	1.02
AXIS	2	* 1700	5/24/79	54	74	1.01
UP HAU2	2	* 1305	7/ 3/79	128	122	1.02
CHESS	2	* 1202	6/11/79	250	82	1.01
CHESS	3	* 1300	7/ 6/79	804	202	1.02
SURFACE	1	* 1507	5/30/79	332	74	1.01
A	1	* 1814	7/ 6/79	1272	74	1.02
UNISTRUC	1	* 1623	6/21/79	406	74	1.01
UP HAU2	1	* 1036	6/18/79	480	122	1.01
UNISTRUC	2	* 1637	6/21/79	602	66	1.01
TOOL	1	* 907	7/ 3/79	668	70	1.02
Q	1	* 1446	7/ 9/79	1528	74	1.02
CDC1	1	* 1351	7/ 6/79	1006	82	1.02
U	1	* 1111	7/ 5/79	738	66	1.02
PLATT	1	* 1455	7/ 6/79	1088	66	1.02
BOTVINNIK	1	* 1542	7/ 6/79	1154	118	1.02

Figure 3-12. Part List

6.4 COPY PART UNDER NEW NAME

This operation creates a new part that is identical to the part on which the user is currently working. The new part has a name and sheet number specified by the user.

Prompt	User Entry
KEY IN NEW NAME	Enter the name for the new part.
SHEET # =	Enter the sheet number for the new part.

The new part name and sheet number exist only in the local working part space until a file command is given. (Refer to the TERMINATE menu.) The system deletes an existing part that has the entered name and sheet number. Therefore, the user should be careful when using this operation.

6.5 DELETE PART

This operation deletes a part stored in the data base.

Prompt	User Entry
KEY IN NAME	Enter the name of the part to be deleted.
SHEET # =	Enter the number of the sheet to be deleted.

The system asks the user to verify that the part is to be deleted.

Prompt	User Entry
DELETE	Enter:
	Y To delete the part.
	N To avoid deleting the part.

When the part has been successfully deleted, the system displays:

```
xxxxxxxxxxxxx DELETED
```

xxxxxxxxxxxxx is the name of the part. If the part cannot be found in the data base, the system displays:

```
xxxxxxxxxxxxx      /n  NOT FOUND
```

xxxxxxxxxxxxx is the name of the part and n is the sheet number.

6.6 CHANGE PART STATUS

This operation changes the status of a part from in-process to released. A part whose status is released cannot be changed or filed. It is also impossible to change the status of a part from released to in-process.

Prompt	User Entry
DO YOU WANT TO RELEASE PART?	The system requests verification that the status is to be changed. Enter: Y To change the status to release. N To avoid changing the status.

After the status has been successfully changed, the following message appears in the part legend in the title block.

```
RELEASED
```

Refer to the DISPLAY TITLE BLOCK modal in the MODALS AND FONTS menu.

6.7 CREATE PATTERN

This operation combines a set of entities into a unit called a pattern so that the pattern can be reproduced at any location in the user's part. The original entities that formed the pattern remain unchanged.

Prompt	User Entry
4-CHAR. NAME	Enter a four-character name for the pattern to be created. The name should be unique within the user's pattern area.

If there is not enough room in the pattern area in the data base, the system displays:

Prompt	User Entry
NO ROOM TO FILE	Enter either Y or N to exit from this operation.

If a pattern already exists in the user's pattern area with this same name, the system displays:

Prompt	User Entry
PATTERN EXISTS--CONTINUE	Enter: Y To delete the old pattern with this name and to create a new one. N To avoid deleting the old pattern and to exit from this operation.

The system then requests that the user pick the entities to be included in the pattern.

Prompt	User Entry
1.SCREEN SELECT 2.ALL DISPLAY	Enter: 1. To select the individual entities on the screen that are to be included in the pattern. 2. To form a pattern that contains all the entities on the display.

If 1.SCREEN SELECT was chosen, the system asks the user to pick the entities to be included in the pattern.

Prompt	User Entry
MODE:1.SINGLE 2.CHAIN 3.REGION	Use the standard entity selection procedure to pick the entities that are to be included in the pattern.

The system now asks the user to indicate the position of the origin for this pattern. Later, when the pattern is retrieved and reproduced in the user's part, the origin is the position within the pattern that is placed at the position entered in response to the ORIGIN prompt for the retrieve pattern operation.

Prompt	User Entry
ORGN:1.LOW LFT 2.POS 3.PT 4.KEY-IN	Select the manner in which the pattern origin will be indicated. Enter: 1. To use the lower leftmost point of the pattern as the origin. 2. To use the crosshairs to indicate the position of the origin within the pattern. 3. To pick an existing point as the origin. 4. To type in the transform coordinates of the origin.

If 2.POS was entered, the system requests the position:

Prompt	User Entry
IND POS	Use the crosshairs to indicate the position of the origin.

If 3.PT was selected, the system displays:

Prompt	User Entry
IND PT	Use the crosshairs to pick an existing point on the screen.

If 4.KEY-IN was selected, the system displays:

Prompt	User Entry
1.XT = 2.YT =	Enter the transform coordinates of the location that is to serve as the origin.

6.8 RETRIEVE PATTERN

This operation copies the entities in a pattern into the user's part. The effect of this operation is the same as if the user creates each of the entities in the pattern individually. The origin defined in the pattern is placed at a position in the part indicated by the user. The entities created can be defined as a group or as individual entities. The user can also select the scale at which the pattern is reproduced.

Prompt	User Entry
4-CHAR. NAME	Enter the name of the pattern to be retrieved.
GROUP DESIRED?	Enter: Y If the entities in the pattern are to be defined as a group. (Refer to the group operation in the ENTITY MANIPULATION menu.) N To define the entities as separate entities and not as part of a group.
ORIGIN: 1.SCREEN POS 2.KEY-IN 3.PT	Select the manner in which the origin position is to be indicated. Enter: 1. To use the crosshairs to indicate a position. 2. To type in the transform coordinates of a position. 3. To use the crosshairs to pick a point at the position desired.

If 1.SCREEN POS was selected, the system displays:

Prompt	User Entry
IND POS	Use the crosshairs to indicate the position of the origin.

If 2.KEY-IN was selected in response to the ORIGIN prompt, the system displays:

Prompt	User Entry
1.XT = 2.YT = 3.ZT =	Enter the transform coordinates of the origin. The default value of the z transform coordinate is the current depth. The z transform coordinate must be opened (by typing a 3 after the x and y values have been entered) before an entry can be made.

If 3.PT was selected in response to the ORIGIN prompt, the system displays:

Prompt	User Entry
IND POINT	Use the crosshairs to pick an existing point as the origin.

The system then requests that the user enter the scale at which the entities in the pattern are to be reproduced. A scale of one causes the entities to be reproduced at the same size as those selected when the pattern was created. A scale less than one causes the entities to be reproduced smaller than the original entities. A scale larger than one causes the entities to be reproduced larger than the originals. A scale of zero causes the system to produce a single, bright dot.

Prompt	User Entry
1.SCALE =	Enter the scale factor.

6.9 DELETE PATTERN

This operation deletes a single pattern in the user's pattern area.

Prompt	User Entry
ENTER A NAME	Enter the name of the pattern to be deleted.

If the pattern is found and successfully deleted, the system displays:

Prompt	User Entry
xxxx DELETED	xxxx is the name of the pattern. Enter Y or N to continue.

If the pattern cannot be found, the system displays:

Prompt	User Entry
xxxx DOESN'T EXIST	Enter Y or N to continue.

6.10 LIST PATTERNS

This operation displays a list of all the patterns in the user's pattern area. The display includes the pattern name, the date and time of day of the last filing (version), the number of words required for storage, and the location on mass storage. Figure 3-13 gives an example of this display.

If there are more patterns than fit on a single screen, the system displays:

Prompt	User Entry
CONTINUE?	Enter: Y To display a list of additional patterns. N To redisplay the user's part.

The user's current part is redisplayed when the R key is pressed.

6.11 INIT PATTERN LIBRARY

This operation initializes the pattern area. All existing patterns stored there are deleted.

Prompt	User Entry
INITIALIZE PATTERN LIBRARY?	Enter: Y To delete all existing patterns. N To avoid deleting all existing patterns.

6.12 MANAGE USER TECH FILE

This set of operations enables the user to perform various file operations on the UTF.

Prompt	User Entry
1.LIST 2.DEL 3.SAVE 4.RSTOR 5.INIT	Select the operation to be performed. Enter: 1. To list the names of the current files in the UTF.

Prompt	User Entry
	2. To delete a file from the UTF.
	3. To copy the files currently in the UTF to the local file TAPE1.
	4. To retrieve the UTF files previously saved on TAPE1 and copy them into the UTF.
	5. To initialize the UTF, deleting all current files.

If 1.LIST was selected, the system displays a list of file names. A sample of this display is given in figure 3-14.

If 2.DEL was selected, the system displays:

Prompt	User Entry
KEY-IN NAME	Enter the name of the file to be deleted.
1.TYPE =	Enter the file type. The file type code can be obtained by using the list operation described previously.

If the file is not present in the UTF, the system displays:

Prompt	User Entry
NOT FOUND HIT YES TO CONTINUE	Enter Y to acknowledge this prompt and to return to the PART/PATTERN MANAGEMENT menu.

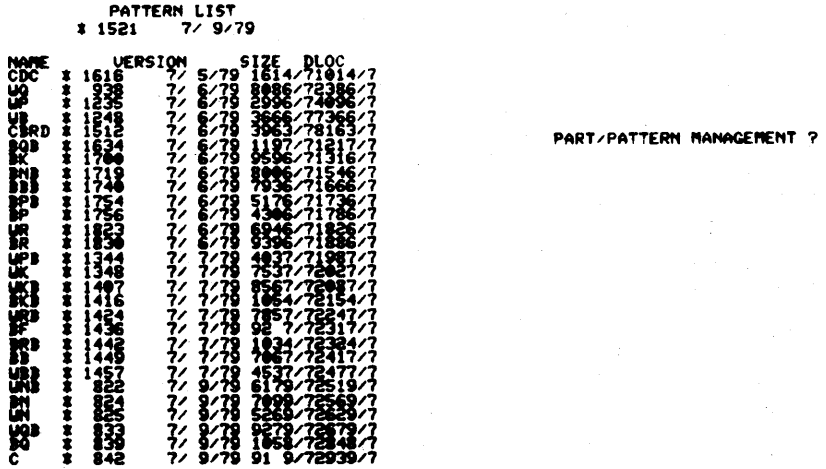


Figure 3-13. Pattern List

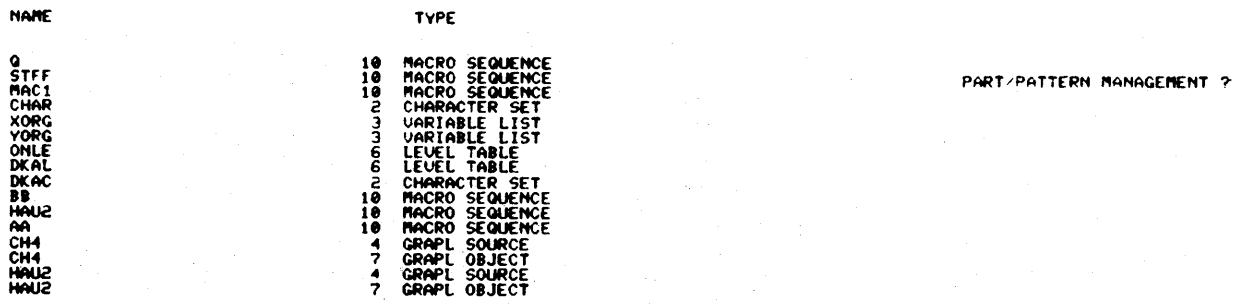


Figure 3-14. List UTF

If 3.SAVE was selected, the system displays:

Prompt	User Entry
MODE:1.SAVE ALL 2.SAVE BY NAME	Enter: 1. To copy all the files in the UTF to the local file. 2. To enter the names of individual files to be copied to the local file.

If 1.SAVE ALL was selected, all the files in the UTF are copied to the local file.

If 2.SAVE BY NAME was selected, the system displays:

Prompt	User Entry
KEY-IN NAME	Enter the name of the file to be saved.
IND TYPE 1.TOOL CYCLE 2.CHARACTER SET 3.VARIABLE LIST 4.GRAPL SOURCE 5.GRAPH TEMPLATE 6.LEVEL TABLE 7.GRAPL OBJECT 8.FORM 9.USER TEXT STATEMENTS	Enter 1 through 9 to indicate the type of file to be saved.

If the file to be saved does not exist in the UTF, the system displays:

Prompt	User Entry
NOT FOUND	Enter Y to acknowledge this prompt and to continue.

If 4.RSTOR was selected, the system displays:

Prompt	User Entry
MODE:1.RSTALL 2.RST BY NAME 3.LIST	Enter: 1. To copy all the files in the local file into the UTF. 2. To enter the names of individual files to be copied from the local file into the UTF. 3. To list the names of the items on the local file.

If 1.RSTALL was selected, the system copies all files on the local file into the UTF. If a UTF file on the local file already exists in the UTF, the system displays:

Prompt	User Entry
xxxx/nn EXISTS RESTORE ANYWAY	xxxx is the name of the UTF file and nn is the type code for the file (refer to figure 3-5 for entity types and codes). Enter: Y To delete the version of the file in the UTF and to copy the UTF file on the local file into the UTF. N To avoid copying this file to the UTF.

If 5.INIT was selected, the system displays:

Prompt	User Entry
INITIALIZE?	Since this operation deletes the entire contents of the UTF, the user should be cautious in responding to this prompt. Enter: Y To initialize the UTF, deleting all contents. N To avoid initializing the UTF.

6.13 DATA BASE INFO

This set of operations enables the user to obtain information about the contents of the data base as described below.

Prompt	User Entry
DATA BASE INFO 1.ENTITY INFO 2.CURRENT PART SPACE 3.PART FILE SPACE 4.PATTERN FILE SPACE 5.USER TECH SPACE	Select the type of information to be displayed. Enter: 1. To display information on the number, types, and statuses of entities in a selected view. 2. To display information on the number of entities, integer words, reals, views, and on the percentage of disk space used in the part space for the current part. 3. To display information on the number of parts, the number of sectors used and available, and the percentage of disk space used in the part file. 4. To display information on the number of patterns and the amount of disk space used to store these patterns. 5. To display information on the number of files in the UTF and the amount of disk space used to store these files.

6.13.1 ENTITY INFORMATION

This operation displays information on the entities in the current part. The system first asks whether the user wants a display of the number of entities defined at each level.

Prompt	User Entry
DISP. NO. OF ENTS. ON VIEWS/LEVELS?	Enter: Y To display a list of the number of entities on each level used and on each view defined. A sample of this display is given in figure 3-15. N To avoid this display.
1.VIEW NO. =	Enter a view number to display information on a single view (figure 3-16), or press the] key to obtain information on all existing views.

The system displays the information shown in figure 3-17 if] is pressed.

The system then asks whether a display of the contents of the view transformation matrix for the selected view is desired.

```
THE NUMBER OF ENTITIES DEFINED IN EACH LEVEL USED

   LEVEL   ENTITIES
     0        161
     1        157
     2         17

                                        HIT YES OR NO TO CONTINUE ?
```

Figure 3-15. Entities Defined at Each Level

```
THE NUMBER OF ENTITIES DEFINED IN EACH VIEW USED

   VIEW   ENTITIES
     1        335

                                        HIT YES OR NO TO CONTINUE ? Y
                                        )VIEW NO.  0          ?
```

Figure 3-16. Entities Defined in Each View

```
COUNT OF ENTITY TYPES  FOR ALL VIEWS AND ALL LEVELS

   ENTITY TYPE   COUNT
     1             92
     2            122
     3              8
    14              3
    16              1
    33             38
    34              4
    35              5
    36              1
    37              6
    38             14
    39              4

                                        HIT YES OR NO TO CONTINUE ?
```

Figure 3-17. Entities Defined at All Levels in All Views

Prompt	User Entry
DISPLAY VIEW MATRIX?	Enter:
	Y To display the matrix.
	N To omit displaying the matrix.

A sample of this display is given in figure 3-18.

The system then asks whether a display is desired of the entity types used.

Prompt	User Entry
DISPLAY ENTITY TYPES USED?	Enter:
	Y To produce the display.
	N To omit the display.

6.13.2 CURRENT PART SPACE

The operation displays information on the current part space. A sample of this display is given in figure 3-19. There are no system prompts nor user entries for this operation.

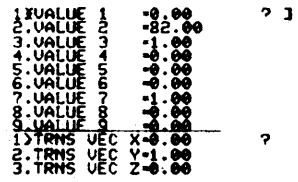


Figure 3-18. View Transformation Matrix

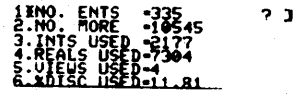


Figure 3-19. Current Part Space Display

6.13.3 PART FILE SPACE

This operation displays information on the part file space. This information includes the number of parts files, the number of disk sectors used and available, and the percentage of disk space used. A sample of this display is given in figure 3-20. There are no system prompts nor user entries for this operation.

6.13.4 PATTERN FILE SPACE

This operation displays information on the pattern file space. This information includes the number of patterns filed, the number of disk sectors used and available, and the percentage of disk sectors used. A sample of this display is given in figure 3-21. There are no system prompts nor user entries for this operation.

6.13.5 USER TECHNOLOGY FILE SPACE

This operation displays information on the UTF space. This information includes the number of files in the UTF, the number of disk sectors used and available, and the percentage of disk space used. A sample of this display is given in figure 3-22. There are no system prompts nor user entries for this operation.

7. OUTPUT AND REGENERATION

The operations in this menu:

- Produce plotter output.
- Display and identify the sequence numbers of entities in the user's part.
- Update and regenerate entities in the user's part.

Figure 3-20. Part File Space Display

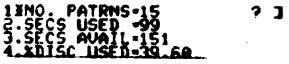


Figure 3-21. Pattern File Space Display

Figure 3-22. UTF Space Display

7.1 OUTPUT CL-FILE/CLPRINT/CLEDIT

Not provided with this release.

7.2 PLOT

This operation produces a plot file, that is, a file that can be sent to a plotter to produce a hardcopy drawing. The plot file is stored on a local file named TAPE9. After the session with AD-2000 has ended, the user can make this a permanent file or can copy it to tape. To interface with UNIPLOT, the local file TAPE9 must be made permanent. This file may then be attached from a terminal with a plotter to generate a plot of the drawing. (Refer to the NOS manuals and UNIPLOT Reference Manual listed in the preface.)

Prompt	User Entry
1.PLOT 2.CONTINUE PLOT 3.TERM PLOT	Enter: 1. To begin creation of a new plot file. 2. To continue creating or editing an existing plot file. 3. To terminate the creation of a plot file.
1.PAPER XMAX = n.nnnn 2.PAPER YMAX = n.nnnn	Enter the size of the paper on which the plot is to be produced.

Prompt	User Entry
PLOT SCALE = n.nnnn	Enter the plot scale. The default value shown is the largest value that produces a plot that fits on the size of paper entered.
XT OFFSET = n.nnnn YT OFFSET = n.nnnn	Enter the offset from the lower left corner of the paper to the location of the origin. (The user indicates the origin in response to the next prompt.)
ORIGIN:1.POS 2.KEY-IN 3.PT 4.SCREEN	Select the method by which the origin is to be specified. The origin is the location within the part to be plotted that is located on the paper by the offsets entered in response to the previous prompts. Enter: 1. To use the crosshairs to enter a position. 2. To type in the coordinates of a position. 3. To pick a point at a position. 4. To select the lower left corner of the screen as the position.

If 1.POS was selected, the system displays:

Prompt	User Entry
INDICATE ORIGIN	Use the crosshairs to indicate a position.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.ORIGIN XT = 2.ORIGIN YT =	Type in the transform coordinates of the origin.

If 3.PT was selected, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to pick a point as the location of the origin.

If 4.SCREEN was selected, the location of the lower left corner of the screen is automatically taken as the origin.

After the plot file has been written, the system displays:

Prompt	User Entry
TERMINATE PLOT	Enter: Y To terminate the plot file. N To inhibit termination of the plot file.

7.3 DISPLAY LAST SEQ. NO. USED

This operation displays the sequence number of the last entity created. Sequence numbers are assigned in order (generally one to each entity) as each entity is created. (All dormant entities also have sequence numbers associated with them, although the user may not know when a dormant entity has been created.)

Prompt	User Entry
1.LAST SEQ. # = nn	The last sequence number assigned is displayed in this prompt. No user entry is required. Press the] key to return to the OUTPUT AND REGENERATION menu.

7.4 DISPLAY ENTITY SEQ. NO.

This operation causes the system to display the sequence number of a selected entity next to the entity's attention point.

Prompt	User Entry
SELECT ENTITY	Use the crosshairs to pick an entity.

7.5 IDENTIFY ENTITY # n

This operation identifies an entity whose sequence number is typed in by the user. The system identifies the entity by displaying the sequence number of the entity next to it on the screen.

Prompt	User Entry
1.1ST SEQ. # = nn	Enter the sequence number of the entity which is to be identified.

7.6 IDENTIFY ENTITIES n1 TO n2

This operation identifies a set of entities whose sequence numbers fall within a range specified by the user. The system identifies the entities by displaying the sequence number of each entity next to that entity on the screen.

Prompt	User Entry
1.1ST SEQ. # = nn 2.2ND SEQ. # = nn	Type in a first (low) and last (high) sequence number. The entities whose sequence numbers fall in this range are identified.

7.7 REGEN ENTITY # n

Regeneration modifies an entity so that it has the original relationships to any entities used in its definition. For example, if a second point has been defined at a distance and angle from a first point, and the first point has been moved since that definition, regenerating the second point causes that point to be moved so that it is at the original distance and angle from the first point. If no entities were used in the definition of the entity selected for regeneration or none of the entities used in the definition of the entity selected for regeneration have been modified, this operation has no effect.

Prompt	User Entry
1.1ST SEQ. # = nn	Enter the sequence number of the entity to be regenerated.

An attention indicator (small oval) is displayed on the entity selected, and the sequence number of the entity is displayed near the entity's attention point, after which the entity is regenerated.

7.8 REGEN FROM n1 TO n2

This selection performs the same operation as the REGENERATE ENTITY NUMBER n operation, except that a set of entities whose sequence numbers fall within a user-specified range is regenerated.

Prompt	User Entry
1.1ST SEQ. # = nn 2.2ND SEQ. # = nn	Type in a low and a high sequence number. All the entities whose sequence numbers fall within this range are regenerated.

Attention indicators (small ovals) are displayed on the entities selected, and the sequence numbers of the entities are displayed near the entities, after which the entities are regenerated.

7.9 REGEN ALL

This selection performs the same operation as the REGENERATE ENTITY NUMBER n operation, except that all the entities in the part are regenerated. There are no system prompts nor user entries for this operation.

7.10 GENERATE SYMBOLIC APT OUTPUT

Not provided with this release.

8. ZOOM/DEPTH/VIEW CONTROL

The operations in this menu:

- Control the scale of the display of the part (zoom) and control which portion of the part is displayed (windowing).
- Change the depth of the plane of definition in the current working view.
- Control which of the currently defined views is to be displayed.
- Select a different (existing) view to serve as the working view.
- Define new views of the part.

8.1 ZOOM

The operations in this submenu change the scale factor by which the part is displayed and change the position in the part that is displayed at the center of the screen.

A scale factor of 1 is normal in the sense that measurements on the screen are reasonably accurate. A line that has been defined as an inch long is approximately 1 inch long on the screen.

Changing the scale factor changes the size at which a part is displayed but does not alter the information on the part in the data base or the sizes at which entities are defined.

This submenu can be entered directly by pressing the Z key.

8.1.1 RETURN TO ORIGINAL SCALE

This operation replaces the current scale at which the part is displayed on the terminal screen with the scale that was in effect immediately after the last auto rescale, the last auto max-min, or the last execution of save zoom status in the ZOOM menu. The screen is redrawn. There are no system prompts nor user entries for this operation.

8.1.2 IND NEW CENTER

This operation causes the system to redisplay the part with the same scale but with a new location, which has been chosen by the user, at the center of the screen.

Prompt	User Entry
IND SCREEN CENTER	Use the crosshairs to indicate a location that is to be moved to the center of the screen. Placing the crosshairs to the left shifts the display to the right; placing the crosshairs high on the screen shifts the display downward, and so on.

8.1.3 IND NEW CENTER, DOUBLE SCALE

This operation repaints the display with a scale factor that is twice the current scale and with a new location selected by the user at the center of the screen. The effect of this operation is to make the part appear twice as large as it was and to shift the part horizontally or vertically on the display.

Prompt	User Entry
IND SCREEN CENTER	Use the crosshairs to indicate a location that will be at the center of the screen.

8.1.4 IND NEW CENTER, HALF SCALE

This operation repaints the display with a scale that is half the current scale and with a new location selected by the user at the center of the screen.

Prompt	User Entry
IND SCREEN CENTER	Use the crosshairs to indicate a location that will be at the center of the screen.

8.1.5 IND NEW CENTER, KEY-IN SCALE

This operation repaints the display with a scale that is keyed in by the user and with a new location selected by the user at the center of the screen.

Prompt	User Entry
IND SCREEN CENTER	Use the crosshairs to indicate a location that is to be at the center of the screen.
1.ZOOM = n.nnnn	Enter the new scale at which the display is to be repainted.

The system repaints the display at the keyed-in scale and with the new center.

8.1.6 DOUBLE SCALE

This operation repaints the display at twice the current scale but with the same center. The current scale is multiplied by two.

There are no system prompts nor user entries for this operation.

8.1.7 HALF SCALE

This operation repaints the display at half the current scale but with the same center. The current scale factor is divided by two.

There are no system prompts nor user entries for this operation.

8.1.8 KEY-IN SCALE

This operation repaints the display at a scale keyed in by the user but with the same center.

Prompt	User Entry
1.ZOOM = n.nnnn	Enter the new scale at which the display is to be repainted.

8.1.9 ZOOM WITH DIAGONAL POINTS

This operation repaints the display, filling as much of the screen as possible with a rectangular area of the part indicated by the user with diagonal corners of an imaginary rectangle. The operation also changes the scale factor.

Prompt	User Entry
IND SCREEN POS 1	Use the crosshairs to indicate the location of one corner of an imaginary rectangle.
IND SCREEN POS 2	Use the crosshairs to indicate the corner diagonal to the first.

8.1.10 KEY-IN MAX-MINS

This operation repaints the display in such a way that two locations whose transform coordinates are entered by the user appear in the lower leftmost corner and the upper rightmost corners of the display. This operation sets the scale factor to that of the resulting display.

Prompt	User Entry
1.XMIN = n.nnnn 2.YMIN = n.nnnn 3.XMAX = n.nnnn 4.YMAX = n.nnnn	Enter the transform coordinates of the location that is to become the lower left corner of the display (XMIN and YMIN) and the upper right corner of the display (XMAX and YMAX). The displayed values are the existing values for the lower left and upper right corners of the display.

To reset the max/min values to the standard values used at entry to AD-2000, enter the following values.

Parameter	English Units	Metric Units
XMIN	0	0
YMIN	0	0
XMAX	14.5	368.30
YMAX	10.875	276.22

8.1.11 AUTOMATIC MAX-MINS GENERATION

This operation redisplays the part so that it fills the terminal screen. The system updates the current max-min values and the current scale factor. This operation also resets the permanent scale factor which will be returned to if the return to original scale operation is used. There are no system prompts nor user entries for this operation.

8.1.12 SAVE ZOOM STATUS

This operation saves the current scale factor as a permanent scale factor. The return to original scale operation in this menu causes the system to return to the scale factor set by the SAVE ZOOM STATUS operation. There are no prompts nor user entries for this operation.

8.2 CHANGE DEPTH

This operation changes the current depth. The current depth is used as the value for the z transform coordinate, the value that specifies the position along the axis that is normal to the screen in any view (for example, when an object is defined using screen position selected by crosshairs). The value entered stays in effect until the next use of this command or until the current depth is changed through use of the D key. The value for the current depth stays the same when changed to a different view, even though this value is used for a different model coordinate. For example, if you move from view 1 (top or plan) to view 2 (front), the depth value, which was used as the z model coordinate in view 1, is used for the negative y model coordinate value in view 2.

Prompt	User Entry
1.DEPTH = n.nnnn	Enter the new depth. A positive entry is measured from the screen toward the user, and a negative entry is measured behind the screen away from the user.

8.3 CHANGE VIEW(S)

This operation changes the views displayed on the screen. The user can choose to see:

- A single view.
- Two views, one above the other.
- Two views, side by side.
- Four views in the four quadrants of the screen.

The views displayed can be any of the standard views (views 1 through 4: plan, front, right, and isometric) or any other views that have been defined by the user (through use of the DEFINE AUXILIARY VIEW submenu in the ZOOM/DEPTH/VIEW CONTROL menu).

Prompt	User Entry
1.SINGLE 2.T-B 3.L-R 4.FOUR VIEWS	Select: 1. To display a single view. 2. To display two views, one above the other. 3. To display two views, side by side. 4. To display four views, each in a separate quadrant on the screen.

If 1.SINGLE was selected, the system displays:

Prompt	User Entry
1.VIEW NO. =	Enter the number of the view to be displayed.
AUTO MAX-MIN DESIRED?	Enter: Y To cause the system to redisplay the part so that it fills the screen. N To prevent the display from being rescaled.

If 2.T-B was selected, the system displays:

Prompt	User Entry
1.TOP VIEW = 2.BOT. VIEW =	Enter the view numbers of the views to be displayed on the top and bottom portions of the screen.

If 3.L-R was selected, the system displays:

Prompt	User Entry
1.LEFT VIEW = 2.RGHT VIEW =	Enter the view numbers of the views to be displayed on the left and right portions of the screen.

If 4.FOUR VIEWS was selected, the system displays:

Prompt	User Entry
1.UPPER LEFT = 2.LOWER LEFT = 3.UPPER RGHT = 4.LOWER RGHT =	Enter the view numbers of the views to be displayed in each of the four quadrants of the screen. Entering the values 1, 2, 3, and 4 produces a display of the standard plan, front, side, and isometric views.

In some cases, when multiple views are displayed, a view may be clipped or the zoom factor may be different from what the user requires. To remedy this, display each individual view as a single view and use the operations in the ZOOM submenu in the ZOOM/DEPTH/VIEW CONTROL menu to adjust each view. Then redisplay the multiple view display.

8.4 CHANGE WORK VIEW

This operation changes the view in which the user is working, the work view. This operation is needed only when more than one view is displayed on the screen. The work view is the view in which the cursor can be used to select entities and to indicate positions. Only one view can be the active work view at any time. The work view before this operation is used is:

- The view displayed, if only one view is displayed.
- The top left view, if four views are on the screen.
- The top view, if top and bottom views are on the screen.
- The left view, if left and right views are on the screen.

After executing this operation, the crosshairs can be used to indicate positions and to pick objects in the view selected.

Prompt	User Entry
SELECT WORK AREA	Select the view that is to be the work view by positioning the crosshairs in that view and pressing the C key.

8.5 DEFINE AUXILIARY VIEW

With the operations in this submenu, the user can rotate and tilt an existing view and then define the resulting view as a permanent view under a new view number. After the new view has been defined and given a view number, the user can display that view (using the CHANGE VIEW(s) submenu in the ZOOM/DEPTH/VIEW CONTROL menu), work in that view, and define objects in that view. This submenu contains operations that rotate and tilt the current view, display the rotated and tilted object, and make the view permanent.

After the appropriate entry has been made for any of the operations in this submenu that tilt or rotate a view, pressing the] key causes the system to perform the same operation that the SAVE AS NEW VIEW operation in this submenu does. That is, the system displays the part as tilted and rotated and displays the following prompt.

Prompt	User Entry
MAKE VIEW nn PERMANENT?	nn is the next sequential unused view number. The view numbers are assigned in ascending sequential order beginning with nn = 5. Views 1 through 4 are the standard predefined views. Enter: Y To assign the view number nn to the view as it is displayed so that it can be called, displayed, and used. N To continue using the operations in the DEFINE AUXILIARY VIEW submenu and to continue tilting and rotating the view.

To quit use of the DEFINE AUXILIARY VIEW submenu without making a new view permanent, use the [key.

8.5.1 NORMAL AXIS CW

This operation rotates the displayed part around an imaginary line that is normal or perpendicular to and in the center of the screen. Entering a positive angle causes a clockwise rotation.

Prompt	User Entry
1.ROTATION = n.nn	Enter the angle in degrees through which the part is to be rotated. A positive angle causes a clockwise rotation.

8.5.2 NORMAL AXIS CCW

This operation rotates the part about a line that is normal to and in the center of the screen. Entering a positive angle causes a counterclockwise rotation.

Prompt	User Entry
1.ROTATION = n.nn	Enter the angle in degrees through which the part is to be rotated.

8.5.3 HORIZONTAL AXIS TOP OUT

This operation tilts the top of the view toward the user. Its effect is to rotate the part around an imaginary horizontal line in the plane of definition and at the bottom of the screen.

Prompt	User Entry
1.ROTATION = n.nn	Enter the angle in degrees through which the view is to be rotated. A positive entry causes the top of the view to tilt toward the user.

8.5.4 HORIZONTAL AXIS TOP IN

Executing this operation tilts the top of the view away from the user. The part rotates around an imaginary horizontal line in the plane of and at the bottom of the screen.

Prompt	User Entry
1.ROTATION = n.nn	Enter the angle in degrees through which the part is to be rotated. A positive entry causes the top of the view to be tilted away from the user.

8.5.5 VERTICAL AXIS RIGHT OUT

This operation swings the right side of the view out toward the user. The effect is to rotate the view around an imaginary line in the plane of and at the left edge of the screen.

Prompt	User Entry
1.ROTATION = n.nn	Enter the angle in degrees through which the view is to be rotated. A positive entry causes the right-hand side of the view to swing toward the user.

8.5.6 VERTICAL AXIS RIGHT IN

This operation causes the right side of the view to swing away from the user toward the back of the terminal. The effect is to rotate the view around an imaginary vertical line in the plane of and at the left side of the screen.

Prompt	User Entry
1.ROTATION = n.nn	Enter the angle in degrees through which the view is to be rotated. A positive entry causes the right side of the view to swing away from the user.

8.5.7 ROTATE ABOUT ANY LINE

This operation rotates the view around any existing line in the display. The user selects the line, picks the direction of rotation, and enters the amount of rotation.

Prompt	User Entry
1.ROTATION = n.nn	Enter the angle in degrees through which the view is to be rotated. A positive entry causes the view to rotate in a counterclockwise direction.
IND ENT	Use the crosshairs to pick a line around which the view is to be rotated.
DIR OF VEC	Use the crosshairs to pick one end or the other of the line selected in response to the previous prompt. The view is rotated about the line in a direction that is counterclockwise to a viewer looking down the line from a position at the end indicated.

8.5.8 PARALLEL TO A PLANE

Not provided with this release.

8.5.9 KEY-IN MATRIX

The user keys in the nine values of a rotation matrix. The effect is to multiply each x, y, z coordinate triple by this rotation matrix.

Prompt	User Entry
1.VALUE #1 = 2.VALUE #2 = 3.VALUE #3 = 4.VALUE #4 = 5.VALUE #5 = 6.VALUE #6 = 7.VALUE #7 = 8.VALUE #8 = 9.VALUE #9 =	Key in the entries in the rotation matrix. Every model coordinate triple is multiplied by this matrix to produce the new view.

9. POINT

The operations in this menu create points in the user's part.

9.1 SCREEN POS

This operation creates a point at a position on the screen indicated by the user through use of the crosshairs and at the current depth.

Prompt	User Entry
IND POS.	Use the crosshairs to indicate the position of the point to be created.

9.2 KEY-IN

This operation creates a point at a location specified by the entry of transform coordinates.

Prompt	User Entry
1.XT = n.nnnn 2.YT = n.nnnn 3.ZT = n.nnnn	Enter the x and y transform coordinates of the location at which the point is to be created. The z transform coordinate value is assumed to be the current depth unless, after entering the x and y values or entering ! RETURN to accept the current values, the user enters a 3 to open the z coordinate parameter for entry.

9.3 POLAR

This operation creates a point at a user-specified distance and angle from a user-selected base point.

Prompt	User Entry
IND. PT.	Use the crosshairs to pick an existing point that is to serve as a base point.
1.ANGLE = n.nnnn	Enter the angle between an imaginary line parallel to the x transform axis, through the selected point, and extending to the right and another imaginary line through the selected point measured in a counterclockwise direction. The new point is defined on this second line. A positive angle is measured in a counterclockwise direction; a negative angle is measured in a clockwise direction.
2.RADIUS = n.nnnn	Enter the distance from the base point to the point to be defined.

The new point is defined at the entered angle and distance from the base point.

9.4 DELTA

This operation defines a point at user-specified displacements in the x, y, and z directions from a selected base point. The user first picks the base point and can then define multiple points from that same base point without reselecting the same base point.

Prompt	User Entry
IND. PT.	Use the crosshairs to select an existing point for the base point.
1.DELTA X = n.nnnn 2.DELTA Y = n.nnnn 3.DELTA Z = n.nnnn	Enter the displacement from the base point in the directions of the x, y, and z transform axes.

The system creates the point. If the transform coordinates of the base point are XT, YT, ZT and the entered displacements are DXT, DYT, DZT, the new point is at transform coordinates XT+DXT, YT+DYT, ZT+DZT. Then, if the construction modal is on, AD-2000 redisplays the DELTA X, DELTA Y, and DELTA Z prompts so that the user can define another point relative to the same base point.

9.5 VECTORED

This operation defines a point at a user-specified distance from a user-selected base point. The direction from the base point to the point to be defined is specified by selecting an existing line. If a positive distance is entered, the point is defined in the direction parallel to the line and toward the positive x direction; if a negative distance is specified, the point is defined toward the negative x direction. If the line is vertical, a positive distance specifies the up direction and a negative distance specifies the down direction on the screen.

Prompt	User Entry
IND. PT.	Select the point to be used as a base point.
IND LINE	Select a line that is to indicate the direction from the base point to the point to be defined.
1.DISTANCE = n.nnnn	Enter the distance from the base point to the point to be defined. A positive value causes the system to measure along an imaginary line parallel to the selected line and through the base point in the direction closest to the positive x direction. A negative number causes the system to measure in the opposite direction.

9.6 CIRCLE CENTER

This operation creates a point at the center of an existing arc or circle.

Prompt	User Entry
IND CIR.	Use the crosshairs to select the arc or circle at whose center a point is to be defined.

The point is defined at the same depth at which the circle is defined.

9.7 ON A CIRCLE

This operation creates a point at a user-entered angle on the edge of a selected circle. The user first picks the circle and then can define multiple points around the edge of that same circle without reselecting it.

Prompt	User Entry
IND CIR.	Use the crosshairs to pick the circle around whose edge the points are to be defined.

Prompt	User Entry
1.ANGLE = n.nnnn	Enter the angle at which the point is to be defined. A positive angle is measured counterclockwise from a line through the center of the circle, parallel to the x transform axis, and extending to the right.

The point is defined at the same depth at which the circle is defined.

9.8 CURVE END

This operation creates a point at the end of a line, arc, or curve closest to the position of the crosshairs when the curve was selected.

Prompt	User Entry
IND CURV	Use the crosshairs to select a curve near the end at which the point is to be defined.

9.9 INTERSECT 2 CURVES

This operation creates a point at the location of the intersection of two curves. If the curves intersect at more than one location, the point is defined at the intersection closest to the position of the crosshairs when the second curve is selected. Lines, arcs, or conics may be extended to create the intersection.

Prompt	User Entry
IND CURV	Use the crosshairs to select two entities. Either selection can be a line, circle, arc, conic, or spline.

The point is defined at the current depth.

NOTES

This operation will not work correctly and the position of the created point may not be at the intersection of the entities if these entities are not lines and were defined in some view other than the current workview.

If associative regeneration or symbolic APT output is to be used, then the second entity must be indicated in such a way that a vector from the intersection to the position of indication will define a unique XLARGE, YLARGE, XSMALL, or YSMALL position indicator.

9.10 SPLINE POINTS

This operation creates points along a spline at the positions originally used to define the spline.

Prompt	User Entry
IND SPLINE	Use the crosshairs to pick a spline.

9.11 ON A LINE

This operation creates a point on a line at a given displacement along the x, y, or z transform axis. A displacement along only one axis can be used. If the displacement entered does not allow a point to be defined on the existing line segment, the point is defined on an imaginary extension of the line.

Prompt	User Entry
IND LINE	Use the crosshairs to pick a line.
1.XT = n.nnnn 2.YT = n.nnnn 3.ZT = n.nnnn	Enter a displacement along one and only one of the three transform axes. To enter a y or z transform value, use the data capture sequence (! followed by RETURN) for the parameters that precede it.

9.12 CURVE NORMAL

This operation defines a point on a curve at the location of the intersection of the curve and an imaginary line that is through a selected existing point and is normal to the curve.

Prompt	User Entry
IND. PT.	Use the crosshairs to select an existing point.
IND CURV	Use the crosshairs to select an existing curve.

9.13 BEARING/DISTANCE

This operation defines a point at an angle and distance from a selected existing point and at the current depth.

Prompt	User Entry
IND. PT.	Use the crosshairs to select an existing point that is to serve as the base point.
1.NORTH 2. SOUTH	Indicate whether the point to be defined is to be above (north of) or below (south of) the base point.
1.DEGREES = n.nnnn 2.MINUTES = n.nnnn 3.SECONDS = n.nnnn	Enter the angle from the base point to the new point. The angle is measured from an imaginary line parallel to the y transform axis and through the base point.
4.DISTANCE = n.nnnn	Enter the distance from the base point to the new point.
1.EAST 2. WEST	Indicate whether the new point is to be to the right (east) or to the left (west) of the base point.

The new point is defined in the quadrant selected through the responses to the prompts 1.NORTH 2.SOUTH and 1.EAST 2.WEST. The new point will be at the entered angle (from the y transform axis) and distance.

9.14 ON A CURVE

This operation creates a point on a curve.

Prompt	User Entry
IND CURV	Select a curve.
1.ST. PARAM = n.nnnn 2.END PARAM = n.nnnn	AD-2000 displays the parameter values of the start and end of the curve. No user entries are made in response to these prompts.
3.PARAMETER = n.nnnn	Enter a value between the two values displayed to indicate where the point is to be created. Entering a value equal to either of the displayed values causes the system to create a point at one or the other end of the curve. Values between these two values cause points along the curve. If the curve selected is a line, arc, or conic, then values greater or less than those displayed can be entered, and AD-2000 will extrapolate the curve to define the point.

9.15 SURFACE NORMAL/PIERCE

Not provided with this release.

9.16 SPHERICAL

This operation creates a point at a distance and angle from the origin (that is, the location with coordinates x = 0, y = 0, and z = 0). The angle is specified with respect to both the x transform and z transform axes, and therefore, the new point can be created outside the plane of the display screen.

Prompt	User Entry
1.RADIUS = n.nnnn	Enter the distance from the origin to the new point. This distance is a straight-line distance in three-dimensional space.
2.Z-AXIS ANG = n.nnnn 3.X-AXIS ANG = n.nnnn	Enter the angle from the origin. These angles are measured from imaginary lines through the origin and parallel to the positive z and x transform axes, respectively.

9.17 FAN POINTS

This operation creates one or more points along a curve. The first of this series of points is at a specified angle from a horizontal line through a base point. The second and subsequent points are at an increment angle from the first and subsequent points.

Prompt	User Entry
IND. PT.	Use the crosshairs to select a base point. This base point must not lie on the curve to be selected.
IND CURV	Use the crosshairs to select a curve. The new points are defined along this curve.
1.START ANG. = n.nnnn	Enter the angle to the first new point measured in a counterclockwise direction from an imaginary line through the base point, extending to the right, and parallel to the x transform axis. The angles are measured from right or left of this line, depending on the position of the crosshairs when the curve was selected.
2.DELTA ANG. = n.nnnn	Enter the angle between each new point and each subsequent new point measured in a counterclockwise direction with the base point as a vertex.
3.NO. OF PTS = n	Enter the number of new points to be created.

9.18 INC. POINTS

This operation creates points along a curve at increments in:

- The x transform axis direction, or
- The y transform axis direction, or
- Along the curve.

Prompt	User Entry
IND CURV	Use the crosshairs to pick the curve along which the new points are to be created.
DELTA: 1.X 2.Y 3.T	Select the type of increment. Enter: 1. To define points at increments of a specified length in the x transform axis direction. 2. To define points at increments of a specified length in the y transform axis direction. 3. To define points at increments specified as parameter values along the curve itself. This selection determines how the start and delta values are used.
1.START VAL. = n.nnnn	Enter the starting value.
2.DELTA VAL. = n.nnnn	Enter the increment.
3.NO. OF PTS = n	Enter the number of new points to be created.

9.19 MODIFY/REPLACE

This operation replaces a point with a new point. Its effect is to delete the existing point and to create a new point at a new location. This operation does not apply to some point-construction forms.

Prompt	User Entry
IND. PT.	Use the crosshairs to select the point to be replaced. The attention indicator for the point selected will be turned on. (In some cases, the attention indicators for any entities used in the original definition of the point will also be turned on.)

Prompt	User Entry
MODE:1.SCREEN POS 2.KEY-IN	Choose the mode to be used to specify the new location. Select: 1. To use the crosshairs to indicate a position on the screen. 2. To key-in the transform coordinates.

If 1.SCREEN POS was selected, the system displays:

Prompt	User Entry
IND POS	Use the crosshairs to indicate the new position of the point.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.XT = n.nnnn 2.YT = n.nnnn 3.ZT = n.nnnn	Enter the transform coordinates of the new location of the point.

10. LINE

The operations in this menu create lines and modify existing lines in the user's part.

10.1 SCREEN POS

This operation defines a straight line whose endpoints are locations specified through use of the crosshairs.

Prompt	User Entry
IND POS.	Use the crosshairs to indicate the location of the first endpoint of the line to be created.
IND POS.	Use the crosshairs to indicate the location of the other endpoint of the line.

10.2 KEY-IN

This operation creates a line whose endpoints are specified by coordinates.

Prompt	User Entry
1.XT1 = 2.YT1 = 3.ZT1 = n.nnnn	Enter the coordinates of the first endpoint of the line to be created.
4.XT2 = 5.YT2 = 6.ZT2 = n.nnnn	Enter the coordinates of the other endpoint of the line.

In order to enter the z coordinates, the user must first make or accept (by entering ! RETURN) all other entries, and then open the z coordinate parameters by typing 3 or 6.

10.3 JOIN 2 POINTS

This operation creates a line whose endpoints are two existing points selected by the user.

Prompt	User Entry
IND. PT.	Use the crosshairs to pick existing points at the locations of the endpoints of the line to be created.

10.4 TANTO 2 CURVES

This operation defines a line that is tangent to two arcs, circles, or other curves and whose endpoints are the points of tangency. If an arc is selected, the point of tangency may be on the imaginary circle of which the arc is a part and not on the arc at all.

Prompt	User Entry
IND CURV	Use the crosshairs to indicate first one and then the other curve. The line is drawn from the first curve to the second. When more than one point of tangency is possible on a given curve, select the curve nearest the desired point of tangency.

10.5 POINT—HORIZ OR VERT

This operation creates a line that passes through a selected point, is either horizontal or vertical, and is, in effect, infinite (that is, it is bounded by the edges of the screen at any zoom factor).

Prompt	User Entry
IND. PT.	Use the crosshairs to pick an existing point through which the new line is to pass.
1.HORIZONTAL 2.VERTICAL	Enter: 1. To create a horizontal line. 2. To create a vertical line.

10.6 POINT—TANTO CURVE

This operation creates a line from an existing point to a point of tangency on an arc, circle, or other curve. If an arc is selected, the point of tangency may be on the imaginary circle of which the arc is a part and not on the arc at all.

Prompt	User Entry
IND. PT.	Use the crosshairs to select the existing point at one endpoint of the new line. The point must not lie within the circle or within the imaginary circle of which the arc is a part.
IND CURV	Use the crosshairs to select the curve to which the new line is to be tangent. The other endpoint is this point of tangency. Since there are two possible points of tangency on some curves, the user should select the curve closest to the desired point of tangency.

10.7 POLAR

This operation creates a line whose first endpoint is an existing point on the screen and which extends in a specified direction for a specified distance.

Prompt	User Entry
IND. PT.	Use the crosshairs to pick the existing point that is to be the first endpoint of the new line.

Prompt	User Entry
IND LINE	Pick a reference line from which the angle (or slope) of the line to be created is to be measured. The angle is measured in the counterclockwise direction with the lowest point on the reference line (least x coordinate) as the vertex.
1.ANGLE = n.nnnn	Enter the angle between the reference line and the new line. A positive angle is measured in a counterclockwise direction.
2.DISTANCE = n.nnnn	Enter the length of the new line.

10.8 POINT—PARALLEL TO LINE

This operation defines a line that passes through an existing point, is parallel to an existing line, and is, in effect, infinite.

Prompt	User Entry
IND. PT.	Use the crosshairs to indicate the point through which the new line is to pass.
IND LINE	Use the crosshairs to indicate the line to which the new line is to be parallel.

10.9 POINT—PERPTO LINE

This operation defines a line that passes through an existing point, is perpendicular to an existing line, and is, in effect, infinite.

Prompt	User Entry
IND. PT.	Use the crosshairs to indicate the point through which the new line is to pass.
IND LINE	Use the crosshairs to indicate the line to which the new line is to be perpendicular.

10.10 PARALLEL TO LINE

This operation creates a line that is parallel to a selected existing line, is at a specified distance from that existing line, and has the same length as that existing line. The endpoints of the new line are located on imaginary lines perpendicular to the existing line and passing through the endpoints of that existing line.

Prompt	User Entry
IND LINE	Use the crosshairs to indicate the line to which the new line is to be parallel.

Prompt	User Entry
1.DISTANCE = n.nnnn	Enter the distance between the existing line and the new line.
IND SIDE	Use the crosshairs to indicate a position on one side or the other of the existing line. The new line is defined on that side of the existing line.

10.11 PARALLEL TO LINE, TANTO CURVE

This operation creates a line that is parallel to an existing line and tangent to an existing curve.

Prompt	User Entry
IND LINE	Use the crosshairs to pick the existing line to which the new line is to be parallel.
IND CURV	Use the crosshairs to pick the arc, circle, conic, or spline to which the new line is to be tangent. If more than one possible point of tangency exists, select the curve near the desired point of tangency.

10.12 PERPTO LINE, TANTO CURVE

This operation creates a line that is perpendicular to a selected existing line and tangent to a selected existing curve. The endpoints of the new line are its point of intersection with the line to which it is perpendicular and its point of tangency to the curve.

Prompt	User Entry
IND LINE	Use the crosshairs to pick the line to which the new line is to be perpendicular.
IND CURV	Use the crosshairs to pick the arc, circle, conic, or spline to which the new line is to be tangent. If more than one possible point of tangency exists, select the curve by positioning the crosshairs nearer to the desired point of tangency.

10.13 n SEGMENT LINE

This operation divides an existing line into a user-specified number of segments. The effect of the operation is to delete the existing line and to create the specified number of new lines that are end to end, equal to each other in length, and lie in the same position as the old line.

Prompt	User Entry
IND LINE	Use the crosshairs to select the existing line that is to be replaced by new lines.
1.NO. SEGS =	Enter the number of new lines or segments.

10.14 JOIN 2 CURVES

This operation creates a line that connects the endpoints of two curves (for example, lines, arcs, circles, conics, and splines). This operation can also create a line that connects two points.

Prompt	User Entry
IND CURV	Use the crosshairs to indicate first one and then the other curve whose endpoints are to be connected. Select the curve by positioning the crosshairs nearer to the endpoint to which the line is to be extended.

10.15 MODIFY INFINITE STATUS

This operation changes a finite line into an infinite line or an infinite line into a finite line. An infinite line is one that extends to the edge of the screen regardless of the display scale. When an infinite line is changed into a finite line, the endpoints of the new finite line are where the line touches the current edges of the screen.

Prompt	User Entry
IND LINE	Indicate the line that is to have its infinite status changed.

10.16 DEFINE AXIS

This operation creates an axis in the plane of definition. If 1.HORIZ is selected, the axis is the x axis (line y = 0 and z = 0). If 2.VERTICAL is selected, the axis is the y transform axis (line xt = 0 and zt = 0).

Prompt	User Entry
AXIS:1.HORIZ 2.VERTICAL 3.BOTH	Enter: 1. To create a horizontal axis along the x transform axis. 2. To create a vertical axis along the y transform axis. 3. To create both a horizontal and a vertical axis.

The axis is displayed as a center line (long and short dashes). The point at which the line passes through the origin is marked with a square.

10.17 CHAMFER

This operation produces a chamfer joining two lines. The lines connected by the chamfer may optionally be trimmed. If requested, the trimming may be performed automatically. Multiple chamfers may be produced during a single operation if the lines between which the chamfer is created are connected.

Prompt	User Entry
1.AUTO TRIM 2.SELECTIVE 3.NONE	Enter the trimming option. 1. To cause both entities connected by the chamfer line to be trimmed automatically. 2. If the user desires to specify which of the entities connected by the chamfer are to be trimmed and which end of the entity is to be removed. 3. If no trimming is to be done.
MODE: 1.SINGLE 2.MULTI-(LNS ONLY)	Indicate whether multiple chamfer lines are to be created. Enter: 1. If only a single chamfer line is to be created each time this operation is performed. 2. If multiple chamfer lines are to be created between a connected series of lines during a single execution of this operation.

If 2.MULTI was selected, the following prompt is displayed.

Prompt	User Entry
MODE: 1.SINGLE 2.CHAIN	Use the entity selection procedure to pick a set of connected entities to be connected by multiple chamfer lines.

The system then requests the dimensions of the chamfer.

Prompt	User Entry
1.CHAMFER DS = n.nnnn	Enter the distance between the intersection of the chamfer and the first entity selected and the intersection of the first and second entities.

Prompt	User Entry
2.ANGLE = n.nnnn	Enter the angle between the chamfer and the first entity. This angle is measured from the first entity in a counterclockwise direction.
IND CHAMFER LOCATION	Use the crosshairs to indicate the approximate chamfer location. This resolves the ambiguity as to which side of the entities the chamfer is to be located.

If the manual trim option was selected (that is, 1 was entered in response to the prompt 1.AUTO TRIM 2.SELECTIVE 3.NONE), the system displays the following prompts.

Prompt	User Entry
TRIM 1ST?	Enter Y to trim the first entity selected or N to omit trimming.
IND TRIM END	Use the crosshairs to indicate the end of the entity to be removed. The portion from the point of intersection of the entity and the chamfer line to the end of the entity is removed.
TRIM 2ND? IND TRIM END	Indicate whether the second entity is to be trimmed using the same manner as that used for the first.

The system then redisplays the prompt

MODE:1.SINGLE 2.MULTI-(LNS ONLY)

so that the user can produce more chamfers using the same option for trimming. To change the trim option, first exit to the line menu and then reenter the operation.

10.18 MODIFY/REPLACE

This operation changes one or more of the parameters defining an existing line. For example, any of the following can be changed.

- The location of one endpoint.
- The location of both endpoints.
- A single coordinate of one of the endpoints.

Each of these changes may be made without entering the rest of the information needed to define a new line and without having to delete the old line. This operation does not apply to some line-construction forms.

Prompt	User Entry
IND LINE	Use the crosshairs to pick the line to be modified. The attention indicator for the line selected (in some cases, for other entities used in the original definition of the selected line) is displayed.
MODE:1.SCREEN POS. 2.KEY-IN	Enter: 1. To use the crosshairs to indicate a new location for either endpoint of the line. 2. To key in new coordinate values for either endpoint of the line.

If 1.SCREEN POS was selected, the system displays:

Prompt	User Entry
IND POS.	Use the crosshairs to indicate the position of each endpoint of the line.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.XT1 = 2.YT1 = 3.ZT1 = n.nnnn 4.XT2 = 5.YT2 = 6.ZT2 = n.nnnn	Enter new values for the coordinates of the first (X1, Y1, Z1) or second (X2, Y2, Z2) endpoint of the line.

11. ARC/CIRCLE/FILLET

11.1 SCREEN POS AND RADIUS

This operation defines a circle with a center at a location specified through use of the crosshairs.

Prompt	User Entry
INDICATE POSITION	Enter the position of the center of the new circle through use of the crosshairs.
1.RADIUS = n.nnnn	Enter the radius of the new circle.
2.STRT ANGLE = n.nnnn	Enter the starting angle of the new circle ısured from the positive x axis.
3.END ANGLE = n.nnnn	Enter the ending angle of the new arc.

The arc is defined in a counterclockwise direction beginning at the first endpoint and ending at the second endpoint.

11.2 KEY-IN CENTER AND RADIUS

This operation defines a circle with center at a position specified by keying in the coordinates of the center in transform space.

Prompt	User Entry
1.XT = n.nnnn 2.YT = n.nnnn 3.ZT = n.nnnn	Enter the transform coordinates of the center for the circle under construction. The value for the z transform coordinate is preset to the current depth. (To exit from this operation, press the [key.)
1.RADIUS = .nnnn 2.STRT ANGLE = .nnnn 3.END ANGLE = .nnnn	Enter the radius and the starting and ending angles. Both starting and ending angles are measured in a counterclockwise direction from the positive x transform axis.

The] key causes a circle to be defined with a center at the location in transform space given by the coordinates entered.

11.3 CENTER AND RADIUS

This operation defines a circle or arc with a center at a location given by an existing point.

Prompt	User Entry
IND POINT	Use the crosshairs to indicate a point whose location will be the center of the circle.
1.RADIUS = n.nnnn 2.STRT ANGLE = n.nnnn 3.END ANGLE = n.nnnn	Enter the radius and the starting and ending angles. Both starting and ending angles are measured in a counterclockwise direction from the positive x transform axis.

11.4 CENTER AND TAN LINE

This operation defines an arc or circle with a center at a selected existing point and with a radius such that the arc or circle is tangent to an existing selected line.

Prompt	User Entry
INDICATE POINT	Use the crosshairs to pick the point to be used as the center of the circle.

Prompt	User Entry
INDICATE LINE	Indicate the line to be used to determine the radius of the circle. The radius is defined as the length of the perpendicular from the point selected to the line selected. If necessary, an imaginary extension of the existing line segment is used. The arc does not necessarily touch the existing line segment.
1.STRT ANGLE = n.nnnn 2.END ANGLE = n.nnnn	Enter the start and end angles. Both angles are measured in a counterclockwise direction from the positive x transform axis.

11.5 CENTER AND TAN CIRCLE

This operation defines a circle with center at an existing point and with a radius such that the arc or circle is tangent to an existing selected arc or circle.

Prompt	User Entry
INDICATE POINT	Use the crosshairs to select an existing point to be used as the center of the arc or circle being constructed.
INDICATE CIRCLE	Indicate the circle to be used to determine the radius of the circle under construction. Since there are two possible tangent points, choose the circle on the half of the circle that contains the point of tangency desired. The location of the attention indicator does not indicate which half was selected.
1.STRT ANGLE = n.nnnn 2.END ANGLE = n.nnnn	Enter the starting and ending angles. Both angles are measured in a counterclockwise direction from the positive x transform axis.

11.6 CENTER AND EDGE POINT

This operation defines an arc or circle with center at a selected existing point and with a radius given by the distance between that point and another selected point.

Prompt	User Entry
INDICATE POINT	Use the crosshairs to select the point to be used as the center of the circle.

Prompt	User Entry
INDICATE POINT	Use the crosshairs to select a second point. The radius of the circle created is the distance between the first and this second point. The arc created need not pass through this second point.
1.STRT ANGLE = .nnnn 2.END ANGLE = .nnnn	Enter the starting and ending angles. Both angles are measured in a counter-clockwise direction from the positive x transform axis.

11.7 THROUGH 3 POINTS

This operation constructs an arc or circle through three selected existing points.

Prompt	User Entry
INDICATE POINT	Use the crosshairs to pick three points to be used in the construction of an arc. An arc is defined on an imaginary circle that goes through the three points starting with the first point indicated and ending with the third point indicated. The arc is defined in an counterclockwise direction. The arc contains the second point indicated only if the points were selected in a counterclockwise manner.

11.8 MODIFY ANGLES

This operation produces another arc with the same center and radius as an existing arc. The existing arc is deleted. The new arc can partially or completely overlay the existing arc.

Prompt	User Entry
INDICATE CIRCLE	Use the crosshairs to select the circle whose center and radius are to be used.
1.STRT ANGLE = n.nnnn 2.END ANGLE = n.nnnn	Enter the starting and ending angles. Both angles are measured in a counterclockwise direction from the positive x transform axis.

11.9 FILLET

The fillet operation produces a fillet between two objects that can be points, lines, arcs, conics, or splines. More than one fillet can be produced by a single execution of this operation if the multiple option is selected. The multiple fillets to be produced can be selected using the chain mode if the lines, arcs, conics, or splines are connected. Either end of either or both of the lines, arcs, conics, or splines connected to the fillet can be trimmed (deleted). The ends to be trimmed can be selected automatically or the user can select the end of the line to be trimmed manually.

Prompt	User Entry
1.AUTO 2.MANUAL 3.NONE 4.FIXED LNS	Indicate the type of trim option desired or press the] key to select the mode selected on a previous iteration. Select: 1. To automatically trim lines, curves, conics, or splines on the convex side of a fillet. 2. To manually select which lines and which ends of these lines are to be trimmed. 3. To perform no trimming. 4. To produce fillets between fixed lines.

The system then requests that the user select the entities between which the fillets are to be constructed. If 1.AUTO was selected, the system displays:

Prompt	User Entry
MODE:1.SINGLE 2.MULTIPLE	Select the mode by which the entities to be filleted are to be picked. Select: 1. To pick entities one at a time. 2. To select more than one connected line, spline, or curve. The user picks a line, spline, or curve and then indicates a direction. AD-2000 automatically picks all the entities along a connected path from the entity selected. This option is valid only if either 1.AUTO or 3.NONE was selected.

If 1.SINGLE is selected, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to pick the objects between which the fillet is to be produced. These objects must be points, lines, arcs, conics, or splines. The fillet is constructed in a counterclockwise direction from the first object selected to the second object selected. Therefore, the order of selection is significant.

If 2.MULTIPLE was selected, the system displays:

Prompt	User Entry
MODE: 1.SINGLE 2.CHAIN	Select the mode by which the entities connected by multiple fillets are to be picked. Enter: 1. To pick individual entities. Up to 200 objects can be selected. These objects must be connected, and the order of selection is significant. 2. To pick a set of connected entities by indicating one of the entities and a direction along the connected entities by which the system is to pick them.

If 2.MULTIPLE and 2.CHAIN were selected, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to select the first of a series of connected lines, arcs, conics, or splines.
IND DIR	Use the crosshairs to indicate the direction in which the system should choose entities by indicating a location to one side or the other of the center of the object chosen. The system picks all lines, arcs, conics, and splines along a path of connected entities. If the path bifurcates, the route of selection is unpredictable.

Prompt	User Entry
SEL. OK?	The system marks the entities selected with attention indicators. Answer Y to accept this selection or N to reject it.

The system then requests that the user enter the radius.

Prompt	User Entry
1.RADIUS = n.nnnn	Enter the radius of the fillet.

The system asks the user to indicate the approximate location of the center of the fillet.

Prompt	User Entry
CENTER	Use the crosshairs to indicate on which side of the entities selected the center of the fillet arc is to be. This may be necessary in order to remove ambiguity, since, in some cases, as many as four possible arcs could be defined between the same two entities. Press the] key to define the fillets.

The following prompts appear if 2.MANUAL was selected from the first prompt.

Prompt	User Entry
TRIM 1ST?	Enter Y to trim the first of the two entities that were selected.
IND TRIM END	Indicate which end of the entity is to be trimmed by using the crosshairs to indicate a position on one side or the other of the attention indicator position for the entity being trimmed. The segment of the entity on that side of the intersection of the entity and the fillet is deleted.
TRIM 2ND	Enter Y to trim the second entity selected.
IND TRIM END	Indicate the end to be trimmed as described above.

11.10 INSCRIBE IN 3 LINES

This operation inscribes an arc so as to be inside and tangent to three lines.

Prompt	User Entry
INDICATE LINE	Use the crosshairs to pick the three lines in a counterclockwise direction.

The arc will be a complete circle if and only if the line segments (when extended if necessary) form a triangle that encloses the arc to be constructed.

11.11 NORMAL TO VIEW

This operation defines an arc with its center at an existing point, normal to the view in which it is defined, and having another existing point on the arc (or its full-circle extension).

Prompt	User Entry
INDICATE POINT	Use the crosshairs to select the point to serve as the center of the circle.
INDICATE POINT	Use the crosshairs to select the point which will be on the arc (or its full-circle extension).
1.STRT ANGLE = n.nnnn 2.END ANGLE = n.nnnn	Enter the starting and ending angle for the arc to be defined. The arc is defined in a counterclockwise direction with both starting and ending angles measured in a counterclockwise direction as seen from the viewer's right.

11.12 MODIFY/REPLACE

This operation enables the user to create an arc or circle based on the characteristics of some existing arc or circle. The user selects an arc or circle, and after the system displays the data for that object, the user can change the following:

- The position or coordinates of the center.
- The radius.
- The starting or ending angles.

This operation does not apply to some arc-construction forms.

Prompt	User Entry
INDICATE CIRCLE	Select the circle whose characteristics are to be modified.
CENTER: 1.SCREEN POS 2.KEY-IN	Select the mode by which the new location of the center is to be entered.

If 1.SCREEN POS was selected, the system displays:

Prompt	User Entry
INDICATE POSITION	Use the crosshairs to indicate the new location of the center of the arc or circle.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.XT = n.nnnn 2.YT = n.nnnn 3.ZT = n.nnnn	Indicate the transform coordinates of the new location of the center for the arc or circle.
1.RADIUS = n.nnnn 2.STRT ANGLE = n.nnnn 3.ENDING ANGLE = n.nnnn	Key-in the new values for the radius, the starting angle, and the ending angle.

The old circle is deleted and the new circle created with any new and entered characteristics and with any unmodified characteristics unchanged. To define a new arc or circle with the same center location, the user can select 2.KEY-IN and then enter the displayed values.

12. OTHER CURVES

The operations in this menu create entities such as:

- Splines.
- Offset curves (a curve at a distance from another curve).
- Conics such as ellipses, parabolas, and hyperbolas.
- Strings (a connected set of lines and arcs).
- N-gons such as triangles, rectangles, and hexagons.

Other selections in this menu can be used to:

- Transform strings into lines and arcs.
- Transform a connected set of lines and arcs into a string.
- Trim lines and arcs at intersections with other entities.
- Transform strings into point-set curves.

12.1 SPLINE

This operation creates a spline curve. AD-2000 constructs the spline by interpolating or smoothing a curve over a set of points or positions indicated by the user. The user can optionally specify the slope of the spline at the beginning and endpoints. If the user specifies tolerances within which the system is allowed to move the positions entered, the system attempts to construct a curve which contains the start and end positions but which is as smooth as possible while passing within the tolerance distance from the other positions indicated. The spline constructed need not be monotonic; that is, it need not be constantly increasing along any line.

The spline may turn back on itself and may even intersect itself. The angle between any three consecutive points used in the definition of the spline should not be less than 90 degrees.

Prompt	User Entry
1.POS 2.KEY-IN 3.PTS 4.RAD, ANGLE	Select the method for specifying positions used to construct the spline. Enter: 1. To use the crosshairs to indicate positions on the screen. 2. To key-in the coordinates of positions. 3. To use the crosshairs to pick existing points that indicate positions. 4. To enter a distance and angle from the origin (0,0) that specifies another position.

After one of these methods has been chosen, multiple positions can be specified. The user can change from one method of specifying positions along the spline to another method during the definition of the same spline by pressing the] key to redisplay the prompt

1.POS 2.KEY-IN 3.PTS 4.RAD, ANGLE

and then selecting another option. The most recent position entered can be deleted by pressing the [key.

If 1.POS was selected, the system displays:

Prompt	User Entry
IND POS	Use the crosshairs to indicate a position for a point on the spline.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.XT = n.nnnn 2.YT = n.nnnn	Type in the transform coordinates of the next point on the spline.

If 3.PTS was selected, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to pick the next point that is to be on the spline.

If 4.RAD,ANGLE was selected, the system displays:

Prompt	User Entry
1.RADIUS = n.nnnn	Enter the distance from the origin and enter the angle to the next point measured from the positive x transform axis. These numbers are the polar transform coordinates of the point.

Terminate entry of positions by pressing the] key twice. The system then requests information used to determine smoothing properties and starting and ending slopes.

Prompt	User Entry
1.POINT MOVE = n.nnnn	Enter the maximum distance that the system is to be allowed to move a position in order to construct the smoothest possible spline. The spline constructed contains the start and end positions but can be within the entered distance from any of the other points.
2.1ST SL,DEG = n.nn 3.END SL,DEG = n.nn	Enter angles (positive numbers representing degrees) that are the maximum slopes of the spline at the start and end positions with respect to the positive x transform axis. The slopes are measured in the counterclockwise direction. If a slope entry is omitted, the slope at the start or endpoint is the slope of an arc through the first three points.

If a nonzero entry was made in response to the 1.POINT MOVE prompt, the system requests entry of the constraints on position movement.

Prompt	User Entry
MOVE:1.XT,YT 2.XT ONLY 3.YT ONLY	Indicate how position movement is to be constrained. Enter: 1. To allow movement along both x and y transform axes. 2. To allow movement in the x transform direction only. 3. To allow movement in the y transform direction only.

12.2 OFFSET CURVE

This operation creates a curve that is constructed so that it is a specified distance from an existing curve and has the same slope along its length as the existing curve. This is not a rigid translation but is a normal offset. For example, a curve offset from an arc would be another arc with the same center but a different radius.

Prompt	User Entry
MODE:1.SINGLE 2.CHAIN	Select the method to be used to select the entity or set of entities from which the new curve is to be offset. Enter: 1. To use the crosshairs to pick individual entities. 2. To use the crosshairs to pick one of a connected set of entities and a direction along which the system is to pick the entities connected to it.

If 1.SINGLE was selected, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to pick one or more entities. These entities can be any connected lines, arcs, or curves.

If 2.CHAIN was selected, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to pick the first of a connected set of lines, arcs, and curves.
IND DIR	Use the crosshairs to indicate the direction along the connected set of entities used to select the chained entities.
SELECTION OK?	Enter: Y If the desired entities have been selected. N To redisplay the IND ENT prompt and to try again.

The system now requests information to be used to determine the offset and location of the new curve.

Prompt	User Entry
1.OFFSET DST = n.nnnn	Enter the distance between the existing set of entities and the curve to be constructed.
IND SIDE	Use the crosshairs to indicate on which side of the existing set of entities the new curve is to be constructed.

12.3 CONICS

The operations in this submenu create the following types of curves.

- Ellipse.
- Hyperbola.
- Parabola.
- General conic.

12.3.1 ELLIPSE

This operation constructs an ellipse in the plane of definition.

Prompt	User Entry
1.EXISTING POINT 2.KEY-IN ORIGIN	Indicate the method to be used to indicate the center point of the ellipse. The center is the midpoint along the segment that connects the two foci. Enter: 1. To pick an existing point as the center. 2. To key-in the coordinates of the center.

If 1.EXISTING POINT was selected, the following prompt is displayed.

Prompt	User Entry
PICK CENTER (OR VERTEX) POINT	Use the crosshairs to pick an existing point as the center.

If 2.KEY-IN ORIGIN was selected, the following prompt is displayed.

Prompt	User Entry
1.CENTER XT = n.nnnn 2.CENTER YT = n.nnnn	Enter the transform coordinates of the center.

The system then requests additional information used to determine the portion of the curve to be created, the dimensions of the major and minor axes, and the orientation of the ellipse.

Prompt	User Entry
1.STRT ANGLE = n.nnnn 2.END ANGLE = n.nnnn	Enter the starting and ending angle for the curve. The ending angle should be a larger number than the starting angle. Only the portion of the ellipse that falls in the area swept out by a line anchored at the center of the ellipse and rotated from the first to the second of these two angles in the positive direction is constructed. The angles are measured in a counterclockwise direction from the positive x axis.
1..5 MAJ AX = n.nnnn 1..5 MIN AX = n.nnnn	Enter half the length of half of the major and minor axes of the ellipse.
1.ROT. ANGLE =	Enter the angle of rotation. The ellipse is constructed with this angle between the major axis of the ellipse and the positive x transform axis.

12.3.2 HYPERBOLA

This operation constructs one branch of a hyberbola.

Prompt	User Entry
1.EXISTING POINT 2.KEY-IN ORIGIN	Indicate the method to be used to indicate the center point of the hyberbola. The center is the midpoint of the segment that connects the foci of both branches of the hyperbola. Enter: 1. To pick an existing point as the center. 2. To key-in the coordinates of the center.

If 1.EXISTING POINT was selected, the following prompt is displayed.

Prompt	User Entry
PICK CENTER (OR VERTEX) POINT	Use the crosshairs to pick an existing point as the center.

If 2.KEY-IN ORIGIN was selected, the following prompt is displayed.

Prompt	User Entry
1.CENTER XT = n.nnnn 2.CENTER YT = n.nnnn	Enter the transform coordinates of the center.

The system then requests additional information used to determine the portion of the curve to be created, the dimensions of the focal radius, and the orientation of the hyberbola.

Prompt	User Entry
1.STRT ANGLE = n.nnnn 2.END ANGLE = n.nnnn	Enter the starting and ending angle for the curve. Only the portion of the hyberbola that falls in the area swept out by a line anchored at the center of the hyberbola and rotated in a positive direction from the first to the second of these two angles is constructed. The angles are measured in a counterclockwise direction from the positive x axis.
1..5 TRAN AX = n.nnnn 2..5 CONJ AX = n.nnnn	Enter half the length of the transverse and conjugate axes.
1.ROT ANGLE = n.nnnn	Enter the angle of rotation. The hyberbola is constructed with this angle between the transverse axis and the positive x transform axis.

12.3.3 PARABOLA

This operation constructs a parabola.

Prompt	User Entry
1.EXISTING POINT 2.KEY-IN ORIGIN	Indicate the method to be used to indicate the center point of the parabola. The center is the vertex or the point of intersection of the parabola and the axis of symmetry. Enter: 1. To pick an existing point as the center. 2. To key-in the coordinates of the center.

If 1.EXISTING POINT was selected, the following prompt is displayed.

Prompt	User Entry
PICK CENTER (OR VERTEX) POINT.	Use the crosshairs to pick an existing point as the center.

If 2.KEY-IN ORIGIN was selected, the following prompt is displayed.

Prompt	User Entry
1.CENTER XT = n.nnnn 2.CENTER YT = n.nnnn	Enter the transform coordinates of the center.

The system then requests additional information used to determine the portion of the curve to be created, the dimensions of the focal radius, and the orientation of the parabola.

Prompt	User Entry
1.STRT ANGLE = n.nnnn 2.END ANGLE = n.nnnn	Enter the starting and ending angle for the curve. Only the portion of the parabola that falls in the area swept out by a line anchored at the center of the parabola and rotated in a positive direction from the first to the second of these two angles is constructed. The angles are measured in a counterclockwise direction from the positive x transform axis.
1.MIN Y = n.nnnn 2.MAX Y = n.nnnn	Enter the bounds for the parabola. The parabola is constructed between and does not extend beyond these two imaginary lines. These lines are parallel to the transverse axis and are the entered distances below and above that axis (before rotation).
1.FOCUS LNGT = n.nnnn	Enter the length of the segment from the vertex to the focus of the parabola.
1.ROT. ANGLE = n.nnnn	Enter the angle of rotation. The parabola is constructed with this angle between the axis of symmetry and the positive x transform axis.

12.3.4 GENERAL CONIC

This operation defines and creates any second order curve that can be described by the equation:

$$AX^2 + BXY + CY^2 + DX + EY + F = 0$$

This operation constructs a conic that is the set of all coordinate values (X,Y) that satisfy the user-entered coefficients for this equation.

Prompt	User Entry
GENERAL CONIC 1.FIRST COEF = 1.0000 2.SECND COEF = .0000 3.THIRD COEF = 1.0000 4.FORTH COEF = .0000 5.FIFTH COEF = .0000 6.SIXTH COEF = -1.0000	Input each of the coeficients A, B, C, D, E, and F as the first through sixth coefficients, respectively.

12.3.5 CYLINDER SLICE

Not provided with this release.

12.4 STRING

The operations in this submenu create strings. A string is a single entity that appears on the display as though it were a connected set of lines and arcs. Strings have several advantages.

- They can be constructed quickly and easily using the operations in this submenu.
- They save space in the data base, because little information need be saved for the definition of a string.

Strings are not lines and arcs; strings cannot be used in place of lines and arcs. However, a string can be easily converted to lines and arcs through use of the MAKE LINES/ARCS FROM STRING operation in this menu (OTHER CURVES).

To use the operations in this submenu, first define the starting point for the string by selecting one of the following

SCREEN POSITION

KEY-IN COORDINATES

EXISTING POINTS

CONNECT TO CURVE

from the string submenu. Define the starting point. The following options are available.

- Continue defining additional segments of the string using the same operation.
- Press the] key (a single time) to exit from that operation. The system redisplays the STRING submenu so that you can select another operation from it and continue adding segments to the same string.
- Press the [key to delete the most recently defined segment.
- Press the] key twice to exit from the operation and the STRING submenu, therefore terminating construction of the current string.

12.4.1 SCREEN POSITION

This operation enables the user to use the crosshairs to indicate the location of the starting point or the endpoint of the next segment of the string. The operation can be used in either constrained or unconstrained mode. When used in constrained mode, only horizontal and vertical line segments can be defined: a horizontal segment is defined if the endpoint of that segment is closer to the indicated position than a vertical line; otherwise, a vertical line segment is defined.

Prompt	User Entry
CONSTRAINED?	Enter: Y To define only horizontal or vertical line segments. N To define line segments with any slope.
IND POS	Use the crosshairs to indicate the starting position of the string (for the first entry for this string) or the endpoint of the next segment to be defined.

12.4.2 KEY-IN COORDINATES

This operation indicates the starting point of the string or adds to the string a line segment whose endpoint is specified by keyed-in coordinates.

Prompt	User Entry
1.XT = 2.YT =	Enter the transform coordinates of the starting point or of the endpoint of the next line segment to be defined.

12.4.3 EXISTING POINT

This operation defines the starting point or adds to the string a line segment whose endpoint is an existing point on the screen.

Prompt	User Entry
IND ENT	Use the crosshairs to pick a point.

12.4.4 DELTA POSITION

This operation adds to the string a line segment whose endpoint is specified in terms of displacements from the current endpoint along the x and y transform axes.

Prompt	User Entry
1.DELTA-XT = n.nnnn 2.DELTA-YT = n.nnnn	Enter the displacement to the new endpoint along the x and y transform axes.

12.4.5 POLAR POSITION

This operation adds to the string a line segment whose endpoint is specified in terms of an angle and a distance from the endpoint of the last segment.

Prompt	User Entry
1.ANGLE = n.nnnn	Enter the angle from the current endpoint to the new endpoint. The angle is measured in a counterclockwise direction from the positive x transform axis.
2.RADIUS = n.nnnn	Enter the distance from the current endpoint to the new endpoint.

12.4.6 BEARING

This operation adds a line segment to the string that extends at an angle for a distance from the current endpoint.

Prompt	User Entry
N00 00 00E N00 00 00E S00 00 00W	Enter the angle from the current endpoint; enter 1, 2, or 3 to select one of the displayed bearings; or enter -1, -2, or -3 to reverse one of the displayed bearings. If a bearing in degrees is entered, proceed the entry with N or S (north or south) and end the entry with E or W (east or west) to indicate whether the angle is to be measured from the positive or negative y transform axis and whether the angle is to be deflected in the direction of the positive or negative x transform axis. For example, an entry of N60 00 00E represents an angle of 60 degrees, 0 minutes, and 0 seconds to the east of north, or 2 o'clock.
DISTANCE = n.nnnn	Enter the distance from the current endpoint to the new endpoint.

12.4.7 CW ARC

This operation adds an arc to the string that begins at the endpoint of the string and extends in a clockwise direction.

Prompt	User Entry
1.RADIUS = n.nnnn	Enter the radius of the arc to be defined.
2.ARC LNGTH = n.nnnn	Enter the length of the arc, or
3.DELTA ANG = n.nnnn	Enter the angle through which the arc is to be inscribed. Do not enter both the length and angle for the arc.

12.4.8 CCW ARC

This operation adds an arc to the string that begins at the endpoint of the string and extends in a counterclockwise direction.

The prompts and user entries for this command are the same as those for the CW ARC command in this submenu.

12.4.9 CONNECT TO CURVE

This operation adds a line segment to the string that connects the current endpoint with an endpoint of an existing line, arc, conic, or spline or the junction of two segments of a string or an existing point.

Prompt	User Entry
IND ENT	Use the crosshairs to pick a point, line, arc, conic, spline, or string. If a line, arc, conic, or spline is selected, a line segment is extended to the endpoint of that entity that is nearest the position of the crosshairs. If a point is selected, a line segment is extended to that point. If a string is selected, a line segment is extended to the junction of two segments of the string closest to the position of the crosshairs.

12.4.10 INDICATE ARC

This operation adds a portion of an arc in an existing string to the string under construction.

Prompt	User Entry
IND ENT	Use the crosshairs to pick an arc segment in a completed string.
2.ARC LNGTH = n.nnnn	Enter the length of the arc, or
3.DELTA ANG = n.nnnn	Enter the angle through which the arc is to be drawn. Do not enter both the length and angle for the arc.

12.4.11 CLOSE

This operation adds to the string one or two line segments that connect the endpoint of the current segment with the starting point of the string. The effect of this operation is to make a closed figure of the string.

Prompt	User Entry
1.HORIZONTAL 2.VERTICAL 3.FREE	Select the close option. Enter: 1. To define a horizontal segment followed by a vertical segment to the starting point. 2. To define a vertical segment followed by a horizontal segment to the starting point. 3. To define a direct line segment to the starting point.

12.5 MAKE STRINGS FROM LINES AND ARCS

This operation converts a set of connected lines and arcs into a single string. After execution of this operation, the lines and arcs that are converted no longer exist in the user's part (although they can be recreated by using the converse of this operation: the MAKE LINES/ARCS FROM STRING operation). The entities to be converted must be connected and must form a single linear chain in which the endpoint of one entity is at the same location as an endpoint of the next. The command is rejected and the system returns to the OTHER CURVES menu if this condition is not satisfied.

Prompt	User Entry
MODE:1.SINGLE 2.CHAIN	Choose the way in which entities are to be selected. Enter: 1. To use the crosshairs to pick each individual entity. 2. To use the crosshairs to pick a connected set of entities by indicating an entity and a direction along which the system is to pick the entities.

If 1.SINGLE was selected, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to pick one or more connected lines and arcs to be converted into a string. When the] key is pressed, the set of entities is converted to a string.

If 2.CHAIN was selected, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to pick the first of a set of connected entities.
IND DIR	Use the crosshairs to indicate the direction along the chain of entities that the system is to follow in picking entities.
SEL OK?	Enter: Y If the entities marked with attention indicators are the entities desired. The entities selected are converted to a single string. N To reject the entities marked with attention indicators and to retry selecting entities.

12.6 MAKE LINES/ARCS FROM STRING

This operation converts a string into individual lines and arcs. The lines and arcs are defined in the user's part and are connected, that is, they have endpoints in common. The string selected is deleted from the user's part (although it can be redefined by using the converse of this operation, the MAKE STRING FROM LINES/ARCS operation).

Prompt	User Entry
IND ENT	Use the crosshairs to pick the string to be converted.

12.7 N-GON

The operations in this submenu can be used to define:

- Triangles.
- Rectangles.
- Hexagons.

The objects defined by these operations are single (point set) entities and are not composed of individual lines that can be picked or selected separately.

12.7.1 TRIANGLE

This operation defines a triangle in the plane of definition, that is, in the current view plane and at the current depth. The user can specify the triangle either by entering locations of three vertices or by picking an existing line to be used as one side and a point to be used as a vertex.

Prompt	User Entry
1.POS 2.KEY-IN 3.PT 4.LINE	Pick the method to be used to specify the triangle. Enter: 1. To use the crosshairs to indicate the positions of the three vertices. 2. To type in the coordinates of the three vertices. 3. To use the crosshairs to pick existing points as the vertices of the triangle. 4. To use the crosshairs to pick a line that will become one side of the triangle and then to use the crosshairs to indicate the position of the vertex not on that line.

If 1.POS was selected, the system displays:

Prompt	User Entry
IND. POS.	Use the crosshairs to indicate the position of each vertex of the triangle to be created.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.XT = 2.YT =	Type in the coordinates of a vertex. This same prompt is redisplayed for entry of the other two vertices.

If 3.PT was selected, the system displays:

Prompt	User Entry
IND. POINT	Use the crosshairs to select three points at the locations of the vertices of the triangle to be created.

If 4.LINE was selected, the system displays:

Prompt	User Entry
IND. LINE	Use the crosshairs to pick an existing line that will become one side of the triangle. After the triangle has been created, the line still exists as a separate entity. The endpoints of this line become two of the vertices of the triangle.
1.POS 2.KEY-IN 3.PT	Choose the manner in which the location of the third vertex is to be specified. Enter: 1. To use the crosshairs to indicate a position. 2. To type in the coordinates. 3. To use the crosshairs to select an existing point.

If 1.POS was selected, the system displays:

Prompt	User Entry
IND. POS.	Use the crosshairs to indicate the position of the third vertex.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.XT = n.nnnn 2.YT = n.nnnn	Enter the transform coordinates of the third vertex.

If 3.PT was selected, the system displays:

Prompt	User Entry
IND. POINT	Use the crosshairs to pick an existing point to indicate the location of the third vertex.

After the triangle is created, the line and point still exist as separate entities.

12.7.2 RECTANGLE

This operation defines a rectangle in the plane of definition. The sides of the rectangle are parallel to the x and y transform axes. The position and dimensions of the rectangle are determined by the user-entered positions of two diagonal vertices.

Prompt	User Entry
1.SCREEN POS PTS 2.KEY-IN PTS 3.EXISTING PTS 4.SCREEN POS + DX,DY 5.KEY-IN + DX,DY 6.EXISTING PT + DX,DY	Choose the manner in which two diagonal vertices of the rectangle are to be specified. Enter: 1. To use the crosshairs to indicate a position. 2. To type in transform coordinates. 3. To use the crosshairs to pick existing points. 4. To use the crosshairs to indicate a position and then type in x and y transform displacements to the diagonal point. 5. To type in the coordinates of a position and then type in x and y transform displacements to the diagonal point. 6. To use the crosshairs to pick an existing point and then type in x and y transform displacements to the diagonal point.

If 1.SCREEN POS PTS was selected, the system displays:

Prompt	User Entry
IND. POS. 1	Use the crosshairs to indicate the position of the vertex.
IND. POS. 2	Use the crosshairs to indicate the position of the diagonal vertex of the rectangle to be created.

If 2.KEY-IN PTS was selected, the system displays:

Prompt	User Entry
1.X1 = 2.Y1 = 3.X2 = 4.Y2 =	Enter the transform coordinates of the two diagonal vertices of the rectangle to be defined.

If 3.EXISTING PTS was selected, the system displays:

Prompt	User Entry
IND. POINTS	Use the crosshairs to pick two existing points that are at the locations of the two diagonal vertices of the rectangle to be created.

If 4.SCREEN POS + DX,DY was selected, the system displays:

Prompt	User Entry
IND. POS	Use the crosshairs to indicate the position of a vertex.
1.DX = n.nnnn 2.DY = n.nnnn	Enter the displacement along the x and y transform axes to the position of the diagonal vertex.

If 5.KEY-IN + DX,DY was selected, the system displays:

Prompt	User Entry
1.X1 = 2.Y1 =	Enter the coordinates of a vertex.
1.DX = n.nnnn 2.DY = n.nnnn	Enter the displacement along the x and y transform axes to the position of the diagonal vertex.

If 6.EXISTING PT + DX,DY was selected, the system displays:

Prompt	User Entry
IND. POINT	Use the crosshairs to pick an existing point at the location of a vertex.
1.DX = n.nnnn 2.DY = n.nnnn	Enter the displacement along the x and y transform axes to the position of the diagonal vertex.

12.7.3 HEXAGON

This operation creates a hexagon in the plane of definition. The hexagon is defined so that two of its (opposite) sides are parallel to either the x transform axis or the y transform axis as specified by the user.

Prompt	User Entry
ORIGIN PLUS:1.IN RAD 2.FLAT 3.DIA	Choose the manner by which the dimensions of the hexagon are to be specified. Enter: 1. To enter the radius of an imaginary circle which is inscribed in the hexagon. 2. To type in the length of a side of the hexagon. 3. To enter the diameter of an imaginary circle in which the hexagon is to be inscribed.

If 1.IN RAD was selected, the system displays:

Prompt	User Entry
1.SCREEN POS 2.KEY-IN 3.EXIST PT	Select the method to be used to indicate the location of the center of the hexagon. Enter: 1. To use the crosshairs to indicate the position of the center. 2. To type in the transform coordinates of the center. 3. To use the crosshairs to pick an existing point to be used as the center.

If 1.SCREEN POS was selected, the system displays:

Prompt	User Entry
IND POS	Use the crosshairs to indicate the position of the center of the hexagon.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.XT = 2.YT =	Enter the transform coordinates of the center of the hexagon.

If 3.EXIST PT was selected, the system displays:

Prompt	User Entry
IND PT	Use the crosshairs to pick an existing point at the location of the center of the hexagon.

After the user has indicated the location of the center of the hexagon, the system requests the dimension of the hexagon.

Prompt	User Entry
1.IN RADIUS = n.nnnn	Enter the radius of the imaginary circle that is inscribed in the hexagon.
SIDE OF FLAT: 1.X-AXIS 2.Y-AXIS	Specify the orientation of the hexagon. Enter: 1. To produce a hexagon with two sides parallel to the x transform axis. 2. To produce a hexagon with two sides parallel to the y transform axis.

If 2.FLAT was selected in response to the ORIGIN PLUS prompt, the system displays:

Prompt	User Entry
1.SCREEN POS 2.KEY-IN 3.EXIST PT	Select the method to be used to indicate the location of the center of the hexagon and indicate the center of the hexagon as described previously.
1.FLAT = n.nnnn	Enter the length of a side of the hexagon.

If 3.DIA was selected in response to the prompt ORIGIN PLUS, the system displays:

Prompt	User Entry
1.SCREEN POS 2.KEY-IN 3.EXIST PT	Select the method to be used to indicate the location of the center of the hexagon and indicate the center of the hexagon as described previously.
1.DIAMETER =	Enter the diameter of an imaginary circle in which the hexagon is to be inscribed.

The system then requests that the user indicate the orientation of the hexagon.

Prompt	User Entry
SIDE OF FLAT: 1.X-AXIS 2.Y-AXIS	Indicate whether two sides of the hexagon are to be parallel to the x transform axis or the y transform axis.

12.8 TRIM CURVES

This operation trims one or more existing lines, arcs, or curves at a point or along a curve specified by the user. The trim operation deletes a segment of an entity and redefines one or both endpoints of the entity. The entities trimmed can be lines, arcs, or curves. The entities that serve as the boundaries against which trimming is performed can be lines, arcs, or curves that cross the entities to be trimmed or points that lie on the entity to be trimmed.

Prompt	User Entry
1.1 END 2.2 ENDS 3.MID 4.2 CURVES	Select the type of trim operation that is to be performed. Enter: 1. To trim one end of one or more lines, arcs, or curves. 2. To trim both ends.

Prompt	User Entry
	3. To trim or remove the middle of one or more lines, arcs, or curves.
	4. To trim one or both of two curves, one against the other.

If 1.1 END was selected, the system requests that the user indicate the boundary point, line, or curve against which the lines, arcs, or curves are to be trimmed.

Prompt	User Entry
IND BND(S)	Use the crosshairs to pick a single point, line, arc, or curve that is to serve as the boundary against which trimming is to be performed.
IND CURVE(S)	Use the crosshairs to pick the lines and curves that are to be trimmed.
IND SIDE	With the crosshairs, indicate one side or the other of the boundary point, line, arc, or curve. The segment of the lines, arcs, and curves to be trimmed that lie on this side of the boundary will be removed.

If 2.2 ENDS was selected, the system displays:

Prompt	User Entry
IND BND(S)	Use the crosshairs to pick two lines, arcs, or curves that are to serve as the boundaries against which trimming is to be performed.
IND CURVE(S)	Use the crosshairs to pick the lines and curves that are to be trimmed.

The segments of the lines, arcs, and curves selected that lie outside the two boundary lines are deleted.

If 3.MID was selected, the same set of prompts are displayed as those displayed when 2.2 ENDS is selected. However, the segments of the lines, arcs, and curves that lie between the two boundary lines are deleted, rather than the segments that lie outside the boundary lines.

If 4.2 CURVES was selected, the system displays:

Prompt	User Entry
IND CURVE(S)	Select two lines, arcs, or curves, either or both of which are to be trimmed.

Prompt	User Entry
TRIM END 1	Use the crosshairs to indicate which end of the first curve selected above is to be trimmed (deleted). Do this by positioning the crosshairs on one side or the other of the other (second) curve selected. The end of the first curve on that side of the second curve will be deleted.
TRIM END 2	In the same way, use the crosshairs to indicate which end of the second curve selected is to be trimmed.

12.9 CONVERT STRING TO POINT-SET

This operation converts a string (constructed using the string operations in this menu) into a point-set. The string to be converted must not contain any arcs.

Prompt	User Entry
INDICATE STRING	Use the crosshairs to pick the string that is to be converted.

If the string selected contains any arcs, the system displays:

Prompt	User Entry
CANNOT CONVERT--STRING HAS ARCS	Enter Y or N to acknowledge this message and to continue.

13. ENTITY MANIPULATION

The operations in this menu:

- Generate arrays of objects in the user's drawing.
- Gather a set of existing objects into a group so that that set of objects can be referred to as a single unit.
- Produce a copy of a set of objects that is a mirror image of the objects on the other side of a line.
- Move (translate) a set of objects to a new location.
- Rotate a set of objects around a position to a new location.
- Produce duplicate copies of a set of existing objects at other locations in the drawing by either translating or rotating the objects.

13.1 RECT ARRAY

This operation produces copies of an object. The copies are arranged in rows and columns. The entire array may be tilted with respect to the screen.

Prompt	User Entry
SELECT BASE ENTITY	Use the crosshairs to pick the entity that is to be duplicated to produce each element of the array. The following types of entities may be selected: group, point, line, arc, and array. An array may be selected as a base entity only if it does not also use an array as its base entity.
1.XT ORIGIN = n.nnnn 2.YT ORIGIN = n.nnnn	Enter the transform coordinates of the start point for the reproduction. If the entity to be duplicated is a point, the first point is at this location. If the first entity is a line, the start position of the line is at this location.
1.HORIZ # = n 2.VERTICAL # = n	Enter the number of columns (horizontal number) and the number of rows (vertical number) to be produced.
3.HOR. DELTA = n.nnnn 4.VER. DELTA = n.nnnn	Enter the displacement between columns (horizontal increment) and the displacement between rows (vertical increment).
5.TILT = n.nnnn	Enter the angle of rotation about the origin of the array measured in the counterclockwise direction from the positive x transform axis to an imaginary line along a row of the array.
1.DO 2.DONT	If the entire array is to be created, press the] key. If less than half of the elements of the array are to be created, press 1, and in response to the next prompt, specify the elements that are to be created. If more than half of the elements of the array are to be created, press 2, and in response to the next prompt, specify the elements that are not to be created.

If 1.DO or if 2.DONT was selected, the system asks the user to specify the elements of the array that are or are not to be created.

Prompt	User Entry
1.X POSITION = nn 2.Y POSITION = nn	Enter the position of each element in the array -- To be created if 1.DO was selected, and -- Not to be created if 2.DONT was selected. For example, an x position of 2 and a y position of 1 designates the element which is in the second column and the first row of the array.

The maximum size of the array is governed by the restriction:

ROWS * COLUMNS is less than or equal to 100

ROWS Number of rows.

COLUMNS Number of columns.

13.2 CIRC ARRAY

This operation produces much the same effect as the rectangular array operation, except that the objects are created around the edge of an imaginary circle whose center and radius are specified by the user.

Prompt	User Entry
SELECT BASE ENTITY	Use the crosshairs to pick the entity that is to be duplicated to produce each element of the array. The following types of entities may be selected: group, point, line, arc, and array. An array may be selected as a base entity only if it does not also use an array as its base entity.
1.XT ORIGIN = n.nnnn 2.YT ORIGIN = n.nnnn	Enter the transform coordinates of the center of the circle around which the elements of the circular array are to be produced.
1.RADIUS = n.nnnn	Enter the radius of the imaginary circle around which the elements of the array are to be produced.
2.STRT ANGLE = n.nnnn 3.END ANGLE = n.nnnn	Enter the angle of the first and last elements to be produced. The angles are measured in a counterclockwise direction from an imaginary line starting at the center of the circle, parallel to the x transform axis, and extending to the right.

The first of the two values displayed next is the angular increment between successive elements of the circular array. A positive angle is measured in a counterclockwise direction with the center of the circle as the vertex. The second value is the number of elements to be produced in the array.

Prompt	User Entry
1.DELTA ANG = n.nnnn 2.NUMBER = nn	Enter the] key if these displayed values are satisfactory. Otherwise enter a new value for one of the quantities. (A new second value can be input without affecting the first by entering ! followed by RETURN, then the value for the second parameter, and then RETURN). AD-2000 adjusts the other value so that strt-angle + delta-angle * (number - 1) = end-angle and then redisplays the two values.
1.DO 2.DONT	If the entire array is to be created, press the] key. If less than half of the elements of the array are to be created, press 1, and in response to the next prompt, specify the elements that are to be created. If more than half of the elements of the array are to be created, press 2, and in response to the next prompt, specify the elements that are not to be created.

If 1.DO or if 2.DONT was selected, the system asks the user to specify the elements that are or are not to be created.

Prompt	User Entry
1.X POSITION = n.nnnn 2.Y POSITION = n.nnnn	Enter the position of each element in the array -- To be created if 1.DO was selected, and -- Not to be created if 2.DONT was selected.

The maximum number of elements in a circular array is 100.

13.3 GROUP

This operation creates a group and defines a set of existing entities as belonging to that group. Subsequently, the entities in the group cannot be referred to individually but can only be referred to as a whole group.

Prompt	User Entry
1.SCREEN SELECT 2.ALL	Select the mode by which you wish to choose the members of the group. Enter: 1. To pick entities on the screen. 2. To select all the entities on the screen.

If 1.SCREEN SELECT was chosen, the following prompt appears:

Prompt	User Entry
1.SINGLE 2.CHAIN 3.REGION	Select the entities that are to be members of the group using the standard entity selection procedure.

The following facts concern groups under AD-2000.

- An entity other than a group can be a member in up to seven different groups.
- A group can be a member of up to six different groups.
- A group can have a maximum of 240 members.
- A group can be selected (for example, using the standard entity selection procedure), but the individuals in it cannot be selected.
- A group can be deleted without deleting the individuals in it. (You can delete the grouping without deleting the objects that are grouped.) Refer to the subsection on the DELETE menu.

13.4 MIRROR

This operation produces a single copy of a set of entities. That copy is a mirror image of the original set of entities. The entities are mirrored on the other side of an existing line selected by the user.

Prompt	User Entry
SEL MIRROR AXIS	Use the crosshairs to indicate a line to serve as the mirror axis. The mirror operation produces a copy that appears to have been rotated 180 degrees around this line.
MODE:1.SINGLE 2.CHAIN 3.REGION	Use the standard entity selection procedure to select the entities from which the copy is to be made.

13.5 TRANSLATE

This operation moves a set of existing entities in the user's part to a new location. After the operation has been performed, the entities no longer exist at the old location. The size of the entities may also be changed by entering a scale factor.

Prompt	User Entry
1.SINGLE 2.CHAIN 3.REGION	Use the standard entity selection procedure to pick the entities that are to be moved.
1.SCALE FAC = n.nnnn	Enter the scale factor at which the entities are to be reproduced.

The system then requests that the user indicate the direction and distance that the objects are to be moved. The user indicates a base position and a move position. The imaginary line or vector that begins at the base position and ends at (points to) the move position gives the direction of distance for the move.

Prompt	User Entry
1.POS 2.KEY-IN 3.PT 4.LOW-LEFT	Select the manner in which the base position is to be indicated. Enter: 1. To use the crosshairs to indicate the base position. 2. To type in the transform coordinates of the position. 3. To select an existing point as the position. 4. To use the position at the lower left corner of the smallest rectangle (with horizontal and vertical sides) that encloses the entities selected.

If 1.POS was selected, the system displays:

Prompt	User Entry
IND BASE POS	Use the crosshairs to indicate the base position.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.BASE-XT = n.nnnn 2.BASE-YT = n.nnnn 3.BASE-ZT = n.nnnn	Type in the transform coordinates of the base position.

If 3.PT was selected, the system displays:

Prompt	User Entry
SEL BASE PT	Use the crosshairs to pick an existing point as the location of the base position.

If 4.LOW-LEFT was selected, the system automatically uses the position at the lower left corner of the entities selected as the base position.

The system then requests the move position, that is, the location of the head of the vector that defines the direction and distance of the move.

Prompt	User Entry
1.POS 2.KEY-IN 3.PT 4.DELTA	Select the manner in which the move position is to be indicated. Enter: 1. To use the crosshairs to indicate the move position. 2. To type in the transform coordinates of the position. 3. To select an existing point at the position. 4. To enter a displacement from the base position in the x, y, and z transform directions.

If 1.POS was selected, the system displays:

Prompt	User Entry
IND MOVE POS	Use the crosshairs to indicate the move position.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.MOVE-XT = n.nnnn 2.MOVE-YT = n.nnnn 3.MOVE-ZT = n.nnnn	Type in the transform coordinates of the move position.

If 3.PT was selected, the system displays:

Prompt	User Entry
SEL MOVE PT	Use the crosshairs to pick an existing point as the move position.

If 4.DELTA was selected, the system displays:

Prompt	User Entry
1.DELTA-XT = n.nnnn 2.DELTA-YT = n.nnnn 3.DELTA-ZT = n.nnnn	Enter the displacement along the x, y, and z transform axes from the base position to the move position.

After the entities have been translated, the system allows the user to exit or to repeat certain parts of this operation.

Prompt	User Entry
1.TERM 2.RPT 3.RPT/SCL 4.NEW ENT	Select the option. 1. To terminate and exit from this operation. 2. To repeat the operation and to enter a new move position. 3. To repeat the operation and to key-in a new scale factor and move position. 4. To repeat the operation beginning with entity selection.

13.6 ROTATE

This operation rotates or moves a set of existing entities around a selected base point to a new location. After the operation has been performed, the entities no longer exist at the old location. The size of the entities may also be changed by entering a scale factor.

Prompt	User Entry
1.SINGLE 2.CHAIN 3.REGION	Use the standard entity selection procedure to pick the entities that are to be moved.
1.SCALE FAC = n.nnnn	Enter the scale factor at which the entities are to be reproduced.

Prompt	User Entry
1.2-D 2.3-D ROTATION	Select: 1. To rotate the entities in the plane of definition. 2. To rotate the entities from the plane of definition. This option is currently limited to lines.

If 2.3-D ROTATION was selected, the system displays:

Prompt	User Entry
ROTATION AXIS 1.NORMAL 2.HORIZONTAL 3.VERTICAL 4.LINE	Select the axis about which the rotation will take place. Enter: 1. To use an axis normal (perpendicular) to the screen. 2. To use an axis which is in the plane of definition and is horizontal. 3. To use an axis which is in the plane of definition and is vertical. 4. To use an axis that is in the plane of definition and is parallel to a given line.

If 4.LINE was selected, the system displays:

Prompt	User Entry
IND ENT	Select the existing line which will, together with the base point, define the axis.

The system then requests that the user indicate a base position around which the entities are to be moved and a rotation angle through which the entities are to be moved.

Prompt	User Entry
1.POS 2.KEY-IN 3.PT 4.LOW-LEFT	Select the method by which the base position around which the entities are to be rotated is to be indicated. Enter: 1. To use the crosshairs to indicate the base position. 2. To type in the transform coordinates of the base position. 3. To select an existing point at the base position.

Prompt	User Entry
	4. To use the position at the lower left corner of the smallest rectangle (with horizontal and vertical sides) which encloses the set of entities selected.

If 1.POS was selected, the system displays:

Prompt	User Entry
IND BASE POS	Use the crosshairs to indicate the base position.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.BASE-XT = n.nnnn 2.BASE-YT = n.nnnn 3.BASE-ZT = n.nnnn	Type in the transform coordinates of the base position.

If 3.PT was selected, the system displays:

Prompt	User Entry
SEL BASE PT	Use the crosshairs to pick an existing point as the location of the base position.

If 4.LOW-LEFT was selected, the system automatically uses the position at the lower left of the entities involved as the base position.

The system then asks for the angle through which the entities are to be rotated around the base position.

Prompt	User Entry
1.ROT. ANGLE = n.nnnn	Enter the rotation angle. A positive entry causes rotation in the counterclockwise direction; a negative entry causes rotation in the clockwise direction.

After the entities have been moved (rotated), the system allows the user to exit from or to repeat certain parts of this operation.

Prompt	User Entry
1.TERM 2.RPT 3.RPT/SCL 4.NEW ENT	Select the option. 1. To terminate and exit from this operation. 2. To repeat the operation and to enter a new move position.

Prompt	User Entry
	3. To repeat the operation and to key-in a new scale factor and move position. 4. To repeat the operation beginning with entity selection.

If 2.RPT was selected, the system asks whether the user wishes to change the rotation angle.

Prompt	User Entry
1.MOVE 2.CHG ANG	Enter: 1. To cause the same amount of rotation to take place again. 2. To cause the prompt 1.ROT. ANGLE = to be redisplayed and to enter a new rotation angle.

13.7 DUPL AND TRANSLATE

This operation performs the same operation that the translate operation does, except that the entities in their original positions are not deleted. In addition, the set of entities can be duplicated and translated more than once so that multiple copies can be made in a single execution of this operation.

Prompt	User Entry
1.SINGLE 2.CHAIN 3.REGION	Use the standard entity selection procedure to pick the entities that are to be moved.
1.SCALE FAC = n.nnnn	Enter the scale factor at which the entities are to be reproduced.

The system then requests that the user indicate the direction and distance that the objects are to be moved. The user indicates a base position and a move position. The imaginary line or vector that begins at the base position and ends at (points to) the move position gives the direction and distance for the move.

Prompt	User Entry
1.POS 2.KEY-IN 3.PT 4.LOW-LEFT	Select the manner in which the base position is to be indicated. Enter: 1. To use the crosshairs to indicate the base position. 2. To type in the transform coordinates of the position. 3. To select an existing point at the position. 4. To use the position at the lower left corner of the smallest rectangle (with horizontal and vertical sides) that encloses the set of entities selected.

If 1.POS was selected, the system displays:

Prompt	User Entry
IND BASE POS	Use the crosshairs to indicate the base position.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.BASE-XT = n.nnnn 2.BASE-YT = n.nnnn 3.BASE-ZT = n.nnnn	Type in the transform coordinates of the base position.

If 3.PT was selected, the system displays:

Prompt	User Entry
SEL BASE PT	Use the crosshairs to pick an existing point as the location of the base position.

If 4.LOW-LEFT was selected, the system automatically uses the position at the lower left corner of the set of entities selected as the base position.

The system then requests the move position, that is, the location of the head of the vector that defines the direction and distance of the move.

Prompt	User Entry
1.POS 2.KEY-IN 3.PT 4.DELTA	Select the manner in which the move position is to be indicated. Enter: 1. To use the crosshairs to indicate the move position. 2. To type in the transform coordinates of the position.

Prompt	User Entry
	3. To select an existing point at the position.
	4. To enter a displacement from the base position in the x, y, and z transform directions.

If 1.POS was selected, the system displays:

Prompt	User Entry
IND MOVE POS	Use the crosshairs to indicate the move position.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.MOVE-XT = n.nnnn 2.MOVE-YT = n.nnnn 3.MOVE-ZT = n.nnnn	Type in the transform coordinates of the move position.

If 3.PT was selected, the system displays:

Prompt	User Entry
SEL MOVE PT	Use the crosshairs to pick an existing point as the move position.

If 4.DELTA was selected, the system displays:

Prompt	User Entry
1.DELTA-XT = n.nnnn 2.DELTA-YT = n.nnnn 3.DELTA-ZT = n.nnnn	Enter the displacement along the x, y, and z transform axes from the base position to the move position.

The system then asks how many copies are to be made. If the 1 is entered, the system makes one translation, scale, and copy. If a value greater than 1 is entered, the system makes additional copies by repeatedly translating the set of entities the specified direction and distance from the most recent copy. The scale factor is applied each time to the most recent copy. The effect is to translate along a straight line whose total length is the number of copies times the distance that the entities are to be translated.

Prompt	User Entry
1.COPIES = nn	Enter the number of translations to be performed and copies to be made.

The system then allows the user to select from among several options that enable him to repeat this operation.

Prompt	User Entry
1.TERM 2.RPT 3.RPT/SCL 4.NEW ENT	Enter: 1. To terminate and exit from this operation. 2. To repeat the operation and to enter a new move position. 3. To repeat the operation and to key-in a new scale factor and move position. 4. To repeat the operation beginning with entity selection.

13.8 DUPL AND ROTATE

This operation performs the same operation that the rotate operation does, except that the entities in their original positions are not deleted. In addition, the set of entities can be duplicated and rotated more than once so that multiple copies can be made in a single execution of this operation.

Prompt	User Entry
1.SINGLE 2.CHAIN 3.REGION	Use the standard entity selection procedure to pick the entities that are to be moved (rotated).
1.SCALE FAC = n.nnnn	Enter the scale factor at which the entities are to be reproduced.
1.2-D 2.3-D ROTATION	Select: 1. To rotate the entities in the plane of definition. 2. To rotate the entities from the plane of definition. This option is currently limited to lines.

If 2.3-D ROTATION was selected, the system displays:

Prompt	User Entry
ROTATION AXIS 1.NORMAL 2.HORIZONTAL 3.VERTICAL 4.LINE	Select the axis about which the rotation will take place. Enter: 1. To use an axis normal (perpendicular) to the screen. 2. To use an axis which is in the plane of definition and is horizontal.

Prompt	User Entry
	3. To use an axis which is in the plane of definition and is vertical. 4. To use an axis that is in the plane of definition and is parallel to a given line.

If 4.LINE was selected, the system displays:

Prompt	User Entry
IND ENT	Select the existing line which will, together with the base point, define the axis.

The system then requests that the user indicate a base position around which the entities are to be moved and a rotation angle through which the entities are to be moved.

Prompt	User Entry
1.POS 2.KEY-IN 3.PT 4.LOW-LEFT	Select the method by which the base position around which the entities are to be rotated is to be indicated. Enter: 1. To use the crosshairs to indicate the base position. 2. To type in the coordinates of the base position. 3. To select an existing point at the base position. 4. To use the lower left corner of the set of entities selected as the base position.

If 1.POS was selected, the system displays:

Prompt	User Entry
IND BASE POS	Use the crosshairs to indicate the base position.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.BASE-XT = n.nnnn 2.BASE-YT = n.nnnn 3.BASE-ZT = n.nnnn	Type in the coordinates of the base position.

If 3.PT was selected, the system displays:

Prompt	User Entry
SEL BASE PT	Use the crosshairs to pick an existing point at the location of the base position.

If 4.low-LEFT was selected, the system automatically uses the position at the lower left of the entities selected as the base position.

The system then asks for the angle through which the entities are to be rotated around the base position.

Prompt	User Entry
1.ROT. ANGLE = n.nnnn	Enter the rotation angle. A positive entry causes rotation in the counterclockwise direction; a negative entry causes rotation in the clockwise direction.

The system then asks how many copies are to be made. If the 1 is entered, the system makes one rotation, scale, and copy. If a value greater than 1 is entered, the system makes additional copies by repeatedly rotating the set of entities the same specified direction and angle from the most recent copy. The scale factor is applied each time to the most recent copy. The effect is to rotate and duplicate the same set of entities around a base point a multiple number of times.

Prompt	User Entry
1.COPIES = nn	Enter the number of rotations to be performed and copies to be made.

The system then allows the user to select from among several options that enable him to repeat this operation.

Prompt	User Entry
1.TERM 2.RPT 3.RPT/SCL 4.NEW ENT	Select: 1. To terminate and exit from this operation. 2. To repeat the operation and to enter a new move position. 3. To repeat the operation and to key-in a new scale factor and move position. 4. To repeat the operation beginning with entity selection.

If 2.RPT was selected, the system asks whether the user wishes to change the rotation angle.

Prompt	User Entry
1.MOVE 2.CHG ANG	Enter: 1. To cause the same amount of rotation to take place again. 2. To cause the prompt 1.ROT. ANGLE = to be redisplayed and to enter a new rotation angle.

13.9 ARRAY EXPLODE

This operation enables the user to remove an element from an array that was created using either the rectangular array or the circular array operations in this same menu. The element removed becomes an individual element with the type and characteristics of the base entity used in the creation of the array.

Prompt	User Entry
IND ENT	Use the crosshairs to pick the array from which an element is to be removed.
INDICATE ELEMENT	Use the crosshairs to pick the element of the array that is to be removed.

If there is no element of the array in the area of the display that the user indicated in response to the INDICATE ELEMENT prompt, the system displays:

NO ELEMENT DISPLAYED IN THAT AREA

13.10 STRETCH

This operation enables the user to move a proper subset of a set of connected entities while allowing entities connected to those in the subset to stretch. For example, we can move sides B and C in figure 3-23 and allow sides A and D to stretch so as to form the object displayed in figure 3-24.

Prompt	User Entry
IND ENT	Use the crosshairs to pick the entities that are to be moved. These entities must be connected and contiguous. They must be connected to the entities that are to be stretched. The system automatically selects lines in response to use of the C key. Therefore, to select a point to be moved, use the crosshairs and the P key. Up to 20 entities can be selected.

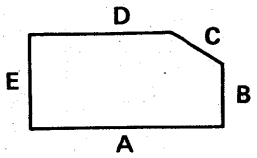


Figure 3-23. Before Stretch

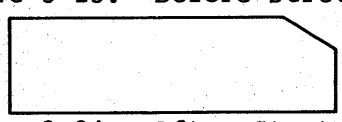


Figure 3-24. After Stretch

Prompt	User Entry
STRETCH LINES 1.AUTO 2.SELECT	Indicate the mode by which the entities to be stretched are to be determined. Enter: 1. To allow the system to select the lines (a maximum of 30) to be stretched. Usually the system picks the closest (straight) lines on either side of the entities to be moved. 2. To use the crosshairs to pick the lines to be stretched.

If 2.SELECT was indicated, the system asks the user to pick the lines.

Prompt	User Entry
IND STRETCH LINES	Use the crosshairs to pick the lines to be stretched. Up to 30 lines can be chosen. If the figure is not to be broken, at least one line on each side of the entities to be moved must be selected to be stretched.

The system now asks the user to indicate two positions, a from position and a to position. The entities to be moved are moved from their current position in the direction and distance of a vector whose tail is at the from position and whose (arrow) head is at the to position.

Prompt	User Entry
STRCH:1.POS 2.KEY-IN 3.PT 4.DELTA	Indicate the method by which the to position is to be indicated. Enter: 1. To use the crosshairs to indicate the position. 2. To type in the transform coordinates of the position. 3. To use the crosshairs to pick a point as the position.

Prompt	User Entry
	4. To indicate the position by entering displacements along the transform coordinate axes from the from position.
FROM:1.POS 2.KEY-IN 3.PT 4.LOW-LEFT	Indicate the method by which the from position is to be indicated. Enter:
	1. To use the crosshairs to indicate the position.
	2. To type in the transform coordinates of the position.
	3. To use the crosshairs to pick a point as the position.
	4. To use the lower leftmost corner of the smallest rectangle (with horizontal and vertical sides) that encloses the set of entities selected as the position.

The system then requests that the user enter both the from position and the to position by displaying prompts in accordance with the user's answers to the two previous prompts. First, the from position is requested. If 1.POS was selected in response to the FROM prompt, the system displays:

Prompt	User Entry
IND BASE POS	Use the crosshairs to indicate the from position.

If 2.KEY-IN was selected in response to the FROM prompt, the system displays:

Prompt	User Entry
1.BASE-XT = n.nnnn 2.BASE-YT = n.nnnn 3.BASE-ZT = n.nnnn	Type in the transform coordinates of the from position.

If 3.PT was selected in response to the FROM prompt, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to pick a point at the location of the from position.

The system then asks the user to enter the to position. If 1.POS was entered in response to the STRCH prompt, the system displays:

Prompt	User Entry
IND TRANS POS	Use the crosshairs to indicate the to position.

If 2.KEY-IN was selected in response to the STRCH prompt, the system displays:

Prompt	User Entry
1.TRANS-XT = n.nnnn 2.TRANS-YT = n.nnnn 3.TRANS-ZT = n.nnnn	Type in the transform coordinates of the to position.

If 3.PT was selected in response to the STRCH prompt, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to pick a point as the location of the to position.

If 4.DELTA was selected in response to the STRCH prompt, the system displays:

Prompt	User Entry
1.DELTA-XT = n.nnnn 2.DELTA-YT = n.nnnn 3.DELTA-ZT = n.nnnn	Type in the displacements along the transform coordinate axes from the from position to the to position.

14. DATA VERIFY

This submenu describes the information that can be obtained through use of the DATA VERIFY menu. To obtain information on a particular entity, select the item from the DATA VERIFY menu for that entity type and then use the crosshairs to select that entity on the display. Interpret the information displayed with the help of the following descriptions. The system gives numerical information in the units of measure (inches or millimetres) chosen for the part. In some cases, the] key can be used to display more information. To return to the DATA VERIFY menu, press the [key.

While using the operations in this menu, press:

- The [key to exit from an operation.
- The] key to continue with the same operation.

14.1 POINTS

Prompt	User Entry
1.ONE PT 2.TWO PTS 3.PT TO LINE	Select the type of information to be displayed. Select: 1. To display information on a single point. 2. To display information on the displacement between two points along the transform coordinate axes and the distance between two points. 3. To display information on the displacement between a point and a line and the distance between a point and a line.

14.1.1 POINT—ONE POINT

Prompt	User Entry
IND ENT	Use the crosshairs to select the point for which information is to be displayed.
1.X = n.nnnn 2.Y = n.nnnn 3.Z = n.nnnn 4.XT = n.nnnn 5.YT = n.nnnn 6.ZT = n.nnnn	The model and transform coordinates of the point are displayed.

14.1.2 POINT—TWO POINTS

Prompt	User Entry
IND ENT	Use the crosshairs to select the points for which information is to be displayed.
1.DXT = n.nnnn 2.DYT = n.nnnn 3.DZT = n.nnnn	The displacements between the points in the directions of the x, y, and z transform axes are displayed.
4.2-D DIST = n.nnnn	The distance between the points along a line projected on the plane of the display screen is displayed.
5.3-D DIST = n.nnnn	The straight-line distance between the points is displayed.

14.1.3 POINT—POINT T LINE

Prompt	User Entry
IND ENT	Use the crosshairs to pick a point.
IND ENT	Use the crosshairs to pick a line.
1.DXT = n.nnnn 2.DYT = n.nnnn 3.DZT = n.nnnn	The displacements in the direction of the x, y, and z transform axes between the point and a position on the line and normal to the point are displayed.
4.2-D DIST = n.nnnn	The distance between the points along a line projected on the plane of the display screen is displayed.
5.3-D DIST = n.nnnn	The straight-line distance between the two points is displayed.

14.2 LINES

Prompt	User Entry
1.ONE LINE 2.TWO LINES	Choose the type of information to be displayed. Select: 1. To obtain information on a single line. 2. To obtain information on the relationship between two lines.

14.2.1 LINE—ONE LINE

Prompt	User Entry
IND ENT	Use the crosshairs to pick the line for which information is to be displayed.
1.PT1 X = n.nnnn 2.PT1 Y = n.nnnn 3.PT1 Z = n.nnnn 4.PT2 X = n.nnnn 5.PT2 Y = n.nnnn 6.PT2 Z = n.nnnn 7.3-D LENGTH = n.nnnn	The model coordinates of the endpoints of the line are displayed along with the three-dimensional length.

Press the [key if this information is sufficient. Press the] key to display additional information.

Prompt	User Entry
1.N63 26 OSE =nn.nnnn 2.PT1 XT = n.nnnn 3.PT1 YT = n.nnnn 4.PT1 ZT = n.nnnn 5.PT2 XT = n.nnnn 6.PT2 ZT = n.nnnn 7.PT2 ZT = n.nnnn 8.2-D LENGTH = n.nnnn	The bearing and angle, the transform coordinates of the endpoints of the line, and the length of the projection of the line on the display screen are displayed.

14.2.2 LINE—TWO LINES

Prompt	User Entry
IND ENT	Use the crosshairs to pick the first of two lines for which information is to be displayed.
IND ENT	Use the crosshairs to pick the other line.
1.ANGLE-1 = nnn.nnnn 2.ANGLE-2 = nnn.nnnn	The angles between the lines are displayed.
3.INT-X = n.nnnn 4.INT-Y = n.nnnn	The coordinates of the position of the intersection of the two lines are displayed.
1.DISTANCE = n.nnnn 2.SLOPE DGE = n.nnnn	The distances between the two lines and their slope are displayed when the lines are parallel.

14.3 ARCS AND CIRCLES

Prompt	User Entry
IND ENT	Use the crosshairs to pick the arc or circle for which information is to be displayed.
	The following information is displayed.
1.START ANG. = n.nnnn 2.END ANG. = n.nnnn	Start and end angles. The angle is measured from the positive x axis to an imaginary line from the arc center through the start or end of the arc.
3.RADIUS = n.nnnn	Radius.
4.XT-CENTER = n.nnnn 5.YT-CENTER = n.nnnn 6.ZT-CENTER = n.nnnn 7.X-CENTER = n.nnnn 8.Y-CENTER = n.nnnn 9.Z-CENTER = n.nnnn	Transform and model coordinates of the center of the arc or circle.

Press the [key if this information is sufficient. Press the] key to display additional information.

Prompt	User Entry
	The following information is displayed.
1.END-1 XT = n.nnnn 2.END-1 YT = n.nnnn 3.END-1 ZT = n.nnnn 4.END-2 XT = n.nnnn 5.END-2 YT = n.nnnn 6.END-2 ZT = n.nnnn	Transform coordinates of the start and end points.
7.STRT SLOPE = n.nnnn 8.END SLOPE = n.nnnn	The slope of the arc or circle at the start and end points.
9.ARC LENGTH = n.nnnn	The length of the arc.

14.4 SPLINES

Prompt	User Entry
IND ENT	Use the crosshairs to pick the spline for which information is to be displayed.
1.SEGMENT # = n	The number of segments in the spline is displayed. The segments are the curves between the points used to define the spline. Segment 1 is the curve between the first two points used in the definition; segment 2 is between the second two points, and so on.

Prompt	User Entry
2.MIN XT = n.nnnn 3.MIN YT = n.nnnn 4.MAX XT = n.nnnn 5.MAX YT = n.nnnn	The minimum and maximum values of the spline along the x and y transform axes are displayed. For example, the maximum y transform value is the y transform coordinate value of the position on the segment that of all the positions on the segment has the largest y transform coordinate value.

Press the [key to return to the DATA VERIFY menu. Press the] key to display additional information.

Prompt	User Entry
1.STRT SLOPE = n.nnnn 2.END SLOPE = n.nnnn 3.XT-SEG PT1 = n.nnnn 4.YT-SEG PT1 = n.nnnn 5.ZT-SEG PT1 = n.nnnn 6.XT-SEG PT2 = n.nnnn 7.TY-SEG PT2 = n.nnnn 8.ZT-SEG PT2 = n.nnnn	The slope of the spline at the endpoints and the transform coordinates for the endpoints of the spline segment are displayed.

Press the] key to display additional information. Press the [key to return to the DATA VERIFY menu.

Prompt	User Entry
1.PT1 X = n.nnnn 2.PT1 Y = n.nnnn 3.PT1 Z = n.nnnn 4.PT2 X = n.nnnn 5.PT2 Y = n.nnnn 6.PT2 Z = n.nnnn	The model coordinates of the endpoints of the spline segment are displayed.

14.5 ELLIPSES

Prompt	User Entry
IND ENT	Use the crosshairs to pick the ellipse for which information is to be displayed. The following information is displayed.
1.STRT ANG. = n.nnnn 2.END ANG. = n.nnnn	Start and end angles measured with respect to the positive x transform axis to an imaginary line from the center of the ellipse through the start or end of the curve.

Prompt	User Entry
3..5 MAJ AX = n.nnnn 4..5 MIN AX = n.nnnn	Length of half of the major and minor axes.
5.ROTATION = n.nnnn	Angle of rotation of the ellipse about its center measured with respect to the positive x transform axis.
1.CENTER X = n.nnnn 2. Y = n.nnnn 3. Z = n.nnnn 4.CENTER XT = n.nnnn 5. YT = n.nnnn 6. ZT = n.nnnn 7.STRT SLOPE = n.nnnn 8.END SLOPE = n.nnnn	Model and transform coordinates of the center of the ellipse as well as the slope at the start and end points of the curve are displayed.

14.6 HYPERBOLAS

Prompt	User Entry
SELECT HYPERBOLA	Use the crosshairs to pick the hyperbola for which information is to be displayed. The following information is displayed.
1.STRT ANG. = n.nnnn 2.END ANG. = n.nnnn	Start and end angles measured with respect to the positive x axis to an imaginary line from the center of the ellipse through the start or end of the curve.
3..5 TRAN AX = n.nnnn 4..5 CONJ AX = n.nnnn	Length of half of the transverse and conjugate axes.
5.ROTATION = n.nnnn	Rotation angle of the hyperbola about its center measured with respect to the positive x transform axis.

Press the [key if this information is sufficient. Press the] key to display additional information.

Prompt	User Entry
1.CENTER X = n.nnnn 2. Y = n.nnnn 3. Z = n.nnnn 4.CENTER XT = n.nnnn 5. YT = n.nnnn 6. ZT = n.nnnn 7.STRT SLOPE = n.nnnn 8.END SLOPE = n.nnnn	The model coordinates and the transform coordinates of the center of the conic and the slope of the curve at the start and end-points are displayed.

14.7 PARABOLAS

Prompt	User Entry
IND ENT	Use the crosshairs to pick the parabola for which information is to be displayed.
1.Y-MIN = n.nnnn 2.Y-MAX = n.nnnn	The minimum and maximum y values for the lower and upper bounds of the parabola are displayed. These values give the maximum distance that the parabola extends from the main axis of the parabola.
3.FOC LNGTH = n.nnnn 4.ROTATION = n.nnnn	The focal length (length of segment from vertex to focal point) and the rotation angle of the parabola about its vertex are displayed.

Press the [key to exit from this operation. Press the] key to display additional information.

Prompt	User Entry
1.CENTER X = n.nnnn 2. Y = n.nnnn 3. Z = n.nnnn 4.CENTER XT = n.nnnn 5. YT = n.nnnn 6. ZT = n.nnnn 7.STRT SLOPE = n.nnnn 8.END SLOPE = n.nnnn	The model and transform coordinates of the center of the conic and the slope at the start and endpoints are displayed.

14.8 ARRAYS

Prompt	User Entry
1.RECT ARRAY 2.CIRC ARRAY	Select the type of array for which information is to be displayed.

14.8.1 RECTANGULAR ARRAY

Prompt	User Entry
IND ENT	Use the crosshairs to pick the rectangular array for which information is to be displayed.
1.X-POS = n.nnnn 2.Y-POS = n.nnnn 3.DELTA-X = n.nnnn 4.DELTA-Y = n.nnnn 5.TILT ANGLE = n.nnnn	The number of array element positions in both horizonal and vertical directions, the delta coordinates, and the tilt angle of the rectangular array are displayed.

Press the] key to display additional information. Press the [key to return to the DATA VERIFY menu.

Prompt	User Entry
1.BASE-X = n.nnnn 2. Y = n.nnnn 3. Z = n.nnnn 4.BASE-XT = n.nnnn 5. YT = n.nnnn 6. ZT = n.nnnn	The model and the transform coordinates of the base point of the rectangular array are displayed.

14.8.2 CIRCULAR ARRAY

Prompt	User Entry
IND ENT	Use the crosshairs to pick the array for which information is to be displayed.
	The following information is displayed.
1.# OF POS = nn	Number of array positions.
2.RADIUS = n.nnnn	Radius of the imaginary arc around which the entities are defined.
3.START ANG = n.nnnn 4.DELTA ANG = n.nnnn	Start and increment angles measured with respect to the positive x axis.
5.BASE-X = n.nnnn 6. Y = n.nnnn 7. Z = n.nnnn	Model coordinates of the base point.
8.BASE-XT = n.nnnn 9. YT = n.nnnn 10. ZT = n.nnnn	Transform coordinates of the base point.

14.9 GROUPS

Prompt	User Entry
SELECT GROUP	Use the crosshairs to pick the group for which information is to be displayed.
# OF ENT. = nn	The number of entities in the group is displayed.

14.10 MEASUREMENTS

Prompt	User Entry
SELECT ENTITY	Use the crosshairs to pick the first entity to be verified at a corner, endpoint, or midpoint. Only entities of the following types may be selected: points, lines, triangles, rectangles, and hexagons.
SELECT ENTITY	Use the crosshairs to pick the second entity at a corner, endpoint, or midpoint.
1.PT1 X = n.nnnn 2.PT1 Y = n.nnnn 3.PT1 Z = n.nnnn 4.PT2 X = n.nnnn 5.PT2 Y = n.nnnn 6.PT2 Z = n.nnnn	The model coordinates of the corner, endpoint, or midpoint of the selected entity are displayed.
1.PT1 XT = n.nnnn 2.PT1 YT = n.nnnn 3.PT1 ZT = n.nnnn 4.PT2 XT = n.nnnn 5.PT2 YT = n.nnnn 6.PT2 ZT = n.nnnn	The transform coordinates of the corner, endpoint, or midpoint of the selected entity are displayed.
1.DX = n.nnnn 2.DY = n.nnnn 3.DZ = n.nnnn 4.DISTANCE = n.nnnn	The distance between the points in the directions of the x, y, and z axes and the distance between the points in model space are displayed.

14.11 DRAFTING ENTITIES

Prompt	User Entry
IND ENT	Use the crosshairs to pick the label or dimension for which information is to be displayed.
1.ORIGIN XT = n.nnnn 2.ORIGIN YT = n.nnnn	The transform coordinates of origin of the dimension or label are displayed.
3.CHAR SIZE = n.nnnn	The character size is displayed.
4.LEADER ANG = n.nnnn	The leader line angle is displayed for labels only.

14.12 TRIANGLES, RECTANGLES, HEXAGONS

14.12.1 TRIANGLES

Prompt	User Entry
SELECT ENTITY	Use the crosshairs to pick the triangle for which information is to be provided.
1.X1 = n.nnnn 2.Y1 = n.nnnn 3.Z1 = n.nnnn 4.X2 = n.nnnn 5.Y2 = n.nnnn 6.Z2 = n.nnnn 7.X3 = n.nnnn 8.Y3 = n.nnnn 9.Z3 = n.nnnn	The model coordinates of the vertices of the triangle are displayed.
1.XT1 = n.nnnn 2.YT1 = n.nnnn 3.ZT1 = n.nnnn 4.XT2 = n.nnnn 5.YT2 = n.nnnn 6.ZT2 = n.nnnn 7.XT3 = n.nnnn 8.YT3 = n.nnnn 9.ZT3 = n.nnnn	The transform coordinates of the vertices of the triangle are displayed.

14.12.2 RECTANGLES

Prompt	User Entry
IND ENT	Use the crosshairs to pick the rectangle for which information is to be provided. The following information is displayed.
1.X = n.nnnn 2.Y = n.nnnn 3.Z = n.nnnn	Model coordinates of origin.
4.DX = n.nnnn 5.DY = n.nnnn 6.DZ = n.nnnn	Distance between the diagonal corners along the x and y model axes.
7.PERIMETER = n.nnnn	Perimeter.
8.AREA = n.nnnn	Area.

Prompt	User Entry
	The following information is displayed.
1.XT = n.nnnn 2.YT = n.nnnn 3.ZT = n.nnnn	Transform coordinates of the origin.
4.DXT = n.nnnn 5.DYT = n.nnnn 6.DZT = n.nnnn	Distance between the diagonal corners of the rectangle along the x and y transform axes.

14.12.3 HEXAGONS

Prompt	User Entry
IND ENT	Use the crosshairs to pick the hexagon for which information is to be displayed.
	The following information is displayed.
1.CENTER X = n.nnnn 2.CENTER Y = n.nnnn 3.CENTER Z = n.nnnn	Model coordinates of the center.
4.CENTER XT = n.nnnn 5.CENTER YT = n.nnnn 6.CENTER ZT = n.nnnn	Transform coordinates of the center.
7.SIDE LNGTH = n.nnnn	Length of the side.

16. DRAFTING FUNCTIONS

The operations in this menu enable the user to perform the drafting tasks required to complete mechanical line drawings. Some of the facilities include those that produce:

- Dimensions, including horizontal, vertical, parallel, angular, radius, diameter, and thickness.
- Witness (extension) lines and arrows.
- Cross-hatching.
- Notes and labels.
- Centerlines.
- Detail magnifications.
- References to drawings in balloons.

A general procedure is used in the dimensioning and the label operations to specify the location of the text written by the operation. This general procedure is described in the subsection on the LABEL AND DIMEN ORIGIN modal in the DRAFTING MODALS menu.

16.1 DRAFTING MODALS

The following modals set values and select options that are used by the operations in the DRAFTING FUNCTIONS menu. The modal submenu also includes an operation that displays the current setting for each drafting modal. For example, executing that operation upon entry to AD-2000 displays the preset values for the drafting modals on the user's system.

16.1.1 CHAR SIZE

This modal specifies the height of the characters that are written by the dimensioning, labeling, and note operations in the DRAFTING FUNCTIONS menu. Existing dimensions are not affected.

Prompt	User Entry
1.CHAR. SIZE = n.nnnn	Enter the height of the characters to be displayed. This entry specifies the size of characters to be drawn in a full-scale plot.

16.1.2 WITNESS LINES

This modal controls the generation of witness (extension) lines. The witness line is the line that extends from the entity to which the dimension is measured; the tip of the dimension arrow touches the witness line. Either or both witness lines associated with a dimension can be suppressed and not drawn. Existing dimensions are not affected by this operation.

Prompt	User Entry
WITNESS LINE CONTROL 1.NO SUPPRESSION 2.SUPPRESS FIRST 3.SUPPRESS SECOND 4.SUPPRESS BOTH	Enter: 1. To generate witness lines to both entities. 2. To suppress the witness line that extends from the first entity selected. 3. To suppress the witness line that extends from the second entity selected. 4. To suppress both witness lines.

The numbered prompting lines for this modal are displayed only if the menu modal is on.

16.1.3 TEXT-ARROW

This modal determines the placement of the text and arrows generated by the dimensioning operations in the DRAFTING FUNCTIONS menu. Existing dimensions are not affected.

Prompt	User Entry
1.TEXT IN, ARROWS IN 2.TEXT IN, ARROWS OUT 3.TEXT OUT, ARROWS OUT 4.TEXT OUT, ARROWS IN	Enter: 1. To generate dimensions in which both the text and the arrows are inside (between) the entities or witness lines to which the dimension extends. 2. To generate dimensions in which the text is inside and the arrows are outside the entities or witness lines to which the dimension extends. 3. To generate dimensions in which the text and the arrows are outside. 4. To generate dimensions in which the text is outside and the arrows are inside.

The numbered prompting lines for this modal are displayed only if the menu modal is on. Figures 3-25 through 3-28 give examples of these four types of dimensions.

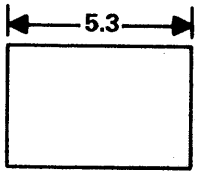


Figure 3-25. Text In, Arrows In

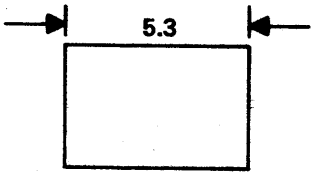


Figure 3-26. Text in, Arrows Out

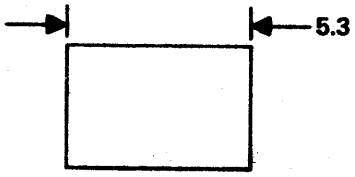


Figure 3-27. Text Out, Arrows Out

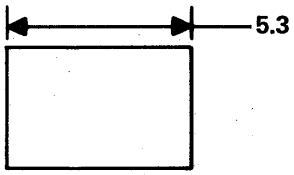


Figure 3-28. Text Out, Arrows In

16.1.4 AUTO DIMENS

This modal specifies that the dimensioning operations in the DRAFTING FUNCTIONS menu are to automatically calculate the length or angle and to insert the text specifying that length or angle in the dimension. This menu selection reverses the effect of the KEY-IN DIMENS modal.

There are no system prompts nor user entries for this modal.

16.1.5 KEY-IN DIMENS

This modal specifies that the text for dimensions drawn by the dimensioning operations of the DRAFTING FUNCTIONS menu is to be typed in by the user. Picking this menu selection reverses the effect of the automatic dimension calculation modal.

There are no system prompts nor user entries for this modal selection.

16.1.6 CROSS HATCH MATERIAL

This modal specifies the font of cross-hatching generated by the cross-hatching operation in the DRAFTING FUNCTIONS menu.

Prompt	User Entry
CROSS-HATCHING MATERIAL 1.IRON 2.STEEL 3.BRASS/COPPER 4.RUBBER/PLASTIC 5.REFRACTORY 6.GLASS/SLATE 7.LEAD 8.ALUMINUM/MAG	Select the type of cross-hatching to be generated.

The numbered prompting lines for this modal are displayed only if the menu modal is on. Figure 3-29 shows a sample of each type.

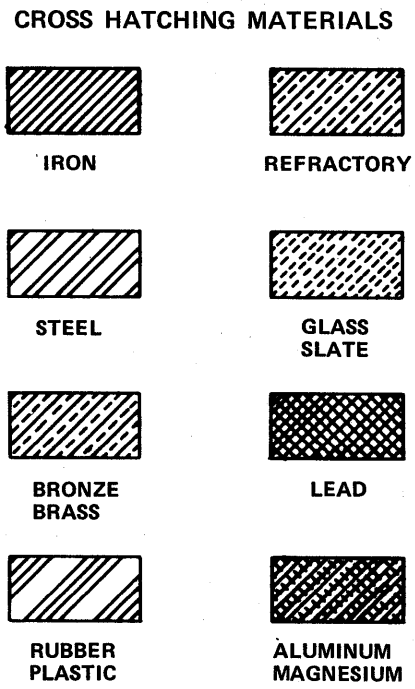


Figure 3-29. Cross-Hatching Material

16.1.7 DECIMAL PLACES

This modal specifies the number of decimal places that the system is to write after the decimal point in dimensions generated by the dimensioning operations in the DRAFTING FUNCTIONS menu. This modal takes the system from the fraction mode set by the fraction modal. Existing dimensions are not affected by this operation.

Prompt	User Entry
1.DEC PLACES = n	Enter the number of decimal places to be displayed.

16.1.8 FRACTION

This modal places the system in fraction mode and causes the system to write dimensions as fractions.

There are no system prompts nor user entries for this modal.

16.1.9 LABEL AND DIMEN ORIGIN

This modal presets the method that will be used to determine the position of the lower left corner of the first character of labels and dimensions. The dimensioning, note, and label operations in the DRAFTING FUNCTIONS menu automatically use the method selected in this modal to request this position information.

Prompt	User Entry
1.IND POS 2.KEY-IN 3.DELTA 4.AUTO	Enter: 1. To use the crosshairs to indicate a position. 2. To type in the coordinates of a position. 3. To enter horizontal and vertical displacements from an existing point, note, label, or dimension. 4. To automatically center horizontal, vertical, and parallel dimensions, and to join the leader line and leader line extension of a circular dimension at the center of an arc. For diameter and angular dimensions and for the label and note operations, making this selection has the same effect as selecting 1.IND POS.

When any of the dimensioning or label operations in the DRAFTING FUNCTIONS menu are selected, the system requests that the user indicate the location of the lower left corner of the first character of text. One of the following three prompts is displayed, depending on the setting of this modal.

Prompt	User Entry
IND. ORIGIN	Use the crosshairs to indicate the position.
or	
1.X ORIGIN = n.nnnn 2.Y ORIGIN = n.nnnn	Enter the coordinates of the position.
or	
IND ENT	Pick the entity from which the displacements to the origin of the dimension text are to be measured.
1.DELTA X = n.nnnn 2.DELTA Y = n.nnnn	Enter the displacements in the x and y directions from the selected entity to the position.

Indicate the position of the lower left corner of the first character in the manner indicated by the prompt, or press the [key to display the prompt

1.IND POS 2.KEY-IN 3.DELTA 4.AUTO

and to choose an alternative method.

16.1.10 ARROW ALIGNMENT

This modal causes any of the dimension-creating operations of the DRAFTING FUNCTIONS menu to align an arrowhead with the arrowhead of the last created or modified dimension of the same type (for example, vertical or horizontal).

Prompt	User Entry
ASK FOR ARROWHEAD ALIGNMENT?	Answer: Y To cause alignment. N To inhibit alignment.

16.1.11 DRAFTING SCALE FACTOR

The system uses the scale factor set through use of this modal in generating dimensions, labels, and notes. The following are affected.

- The arrowhead size of all dimensions and labels.
- The distance from the text of a dimension to the dimension line.
- The distance that the witness line (extension line) is offset from a reference point, line, and so on.
- The distance that the witness line extends past the dimension line.
- The spacing and dash size for centerlines.
- The preset balloon radius.

This operation affects all existing dimensions and labels.

Prompt	User Entry
1.DRAFT SCLE = n.nnnn	Enter the drafting scale to be used. At entry to the system, this value is preset to 1.

16.1.12 CHAR SET

This modal selects the character set to be used for dimensioning, labels, and notes. This operation also causes the text in all existing dimensions, labels, and notes to be modified.

Prompt	User Entry
1.FAST 2.STANDARD 3.USER GENERATED	Select the character set: 1. Fast script. 2. Standard character set. 3. User-generated character set.

The numbered prompting lines for this modal are displayed only if the menu modal is on. Figure 3-30 gives samples of the standard and the fast character sets. If 3.USER GENERATED was selected, the following prompt appears.

Prompt	User Entry
KEY-IN CHARACTER SET NAME	Enter the name of the user-generated character set.

Refer to the USER DEFINED SYMBOLS submenu of the SPECIAL FUNCTIONS menu for an explanation of how to produce a user-generated character set.

16.1.13 CHAR SLANT

This modal determines whether the system will produce vertical or slanted characters in dimensions, labels, and notes. This selection causes existing dimensions and labels to be rewritten.

Prompt	User Entry
SLANT:1.OFF 2.ON	Select: 1. To produce vertical characters. 2. To produce slanted characters.

Figure 3-30 also shows slanted and unslanted characters.

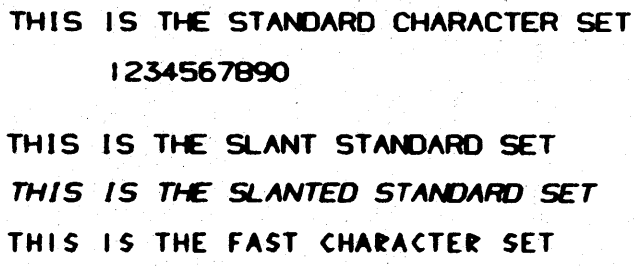


Figure 3-30. Display Character Sets

16.1.14 CHARACTER DISPLAY RATIOS

This modal sets

- The distance between consecutive characters,
- The relative height with respect to the width of characters, and
- The spacing between lines of characters

for text generated in dimensions, labels, and notes. All existing dimensions, labels, and notes in the current part are modified.

Prompt	User Entry
1.SPACING = n.nnnn	Enter the spacing factor. This value multiplied by the character height (set by the character size modal in the DRAFTING MODALS menu) gives the spacing between the centers of consecutive characters. Upon entry to the system, this value is preset to 1.1.
2.ASPECT R = n.nnnn	Enter the ratio between the character width and the character height. This value is preset to 1.
3.DOWN SPACE = n.nnnn	Enter the line spacing factor. This entry multiplied by the character height gives the distance between the bottoms of lines of text. This value is preset to 1.5.
4.TOL RATIO = n.nnnn	Enter the tolerance ratio. This is the ratio between the character size for tolerance characters and the character size for main characters in dimensions. This value is preset to 0.75.

16.1.15 ARROWHEAD LENGTH

This modal sets the length of arrowheads generated by operations in the DRAFTING FUNCTIONS menu. The width of arrowheads is one third the length. Modifying this modal setting causes the system to change all existing label and dimension arrowheads in the part displayed. This modal does not affect existing arrowheads defined at the ends of lines.

Prompt	User Entry
1.ARROWHEAD SIZE = n.nnnn	Enter the arrowhead length.

16.1.16 DIMENSION OFFSET DISTANCES

This modal sets the following distances used in generation of dimensions.

- The distance from the text in a dimension to its dimension lines.
- The distance (space) between a witness line (extension line) and the entity from which it extends.
- The distance that a witness line extends past a dimension line.

Modifying this modal setting does not cause the system to change existing dimensions.

Prompt	User Entry
1.TEXT-DIMEN = n.nnnn	Enter the distance from the text to the dimension line. Upon entry to the system, this value is preset to 2.54 millimetres (0.1 inch).
2.WITNESS-PT = n.nnnn	Enter the distance from the witness line to the entity. This value is preset to 1.59 millimetres (0.0625 inch).
3.WITNESS-EX = n.nnnn	Enter the distance that the witness line extends past the dimension line. This value is preset to 3.17 millimetres (0.125 inch).

16.1.17 NOTE PARALLEL TO LINE/ARC

This modal on causes the general note operation in the DRAFTING FUNCTIONS menu to allow the user to create notes that are parallel to lines or arcs selected by the user.

Prompt	User Entry
ASK FOR PARALLEL LINE/ARC IN NOTE?	Select: Y To be given the option of creating notes that are parallel to a selected line or arc. N To create notes that are horizontal. The prompt PARALLEL TO LINE OR ARC? is not displayed in the general note operation.

16.1.18 DUAL DIMENSIONING

This modal determines whether the dimensioning operations in the DRAFTING FUNCTIONS menu are to write both metric and English units or only the units in effect for the part. The alternate units are displayed inside square brackets.

Prompt	User Entry
1.STANDARD 2.DUAL	Enter: 1. To produce dimensions containing only the units in effect for the part. 2. To produce dimensions containing both metric and English units.

16.1.19 DISPLAY DRAFTING MODALS

This modal causes the system to display the current setting of the drafting modals. A sample display is given in figure 3-31.

```
DV DRAFTING MODALS
1.CHARACTER SIZE  n.nnn
2.WITNESS LINE  NO SUPPRESSION
3.TEXT ARROW  TEXT IN, ARROWS IN
4.AUTO DIMENSION  ON
5.KEY-IN DIMENSION  OFF
6.CROSS-HATCH  IRON
7.DECIMAL PLACES  4
8.FRACTIONS  DECIMAL
9.L+D ORIGIN  SCREEN POS
10.CHARACTER SET  STANDARD
11.CHARACTER SLANT  OFF
```

Figure 3-31. Sample Display

There are no system prompts nor user entries for this modal.

16.2 PROJECTED ENTITIES

This operation creates a copy of a set of entities in a direction from the entities that is perpendicular to the plane of definition and at a distance specified by the user. The operation also connects the original to the copy with lines so that this new combined object appears as a three-dimensional object.

Prompt	User Entry
1.DELTA ZT = n.nnnn	Enter the depth relative to the current depth to which the object is to be projected. The depth is measured in the z transform direction. A positive value causes the objects to be projected toward the viewer; a negative value causes the objects to be projected away from the viewer.
1.SINGLE 2.CHAIN	Use the entity selection procedure to select the set of entities to be projected.

16.3 CROSS HATCH

This operation produces cross-hatching lines within boundaries specified by the user. The user specifies the angle of the cross-hatching lines, the distance between cross-hatching lines, and the tolerance between the end of the cross-hatching lines and the boundaries. The user can also specify the boundaries of areas or islands within the main boundary that are not to be cross-hatched. The font of the cross-hatching lines is specified through use of the CROSS-HATCHING MATERIAL modal in the DRAFTING MODALS menu.

Prompt	User Entry
1.ANGLE = nn.nnnn	Enter the angle between the cross-hatch lines to be created and the positive x transform axis.
2.DISTANCE = .nnnn	Enter the distance between the cross-hatch lines to be created. The preset value is 6.35 millimetres (0.2500 inch). The minimum distance is 0.127 millimetre (0.005 inch).
3.INTOL = n.nnnn 4.OUTOL = n.nnnn	Specify the maximum distance that the cross-hatch lines drawn by the system may be short of or extend across a boundary line. The preset value is 0.127 millimetre (0.005 inch).
CROSS-HATCHING MAIN BOUNDARY MODE:1.SINGLE 2.CHAIN	Use the entity selection procedure to select a set of lines, arcs, and curves that define the outside boundary of the area to be cross-hatched.
CROSS-HATCHING ISLAND MODE:1.SINGLE 2.CHAIN	Use the entity selection procedure to select lines, arcs, and curves that define areas (islands) to be omitted from cross-hatching. Press the] key to end entry of each island. Press the] key twice to end entry of all islands.

Press the] key once to specify additional island areas. Press the] key twice to cause the system to create and display the cross-hatching.

16.4 HORIZONTAL DIMEN

This operation produces a dimension between two entities. The dimension line produced is horizontal. The dimension includes extension (witness) lines, arrowheads, dimension lines, and text. The text in the dimension is automatically calculated or typed in by the user depending on the setting of the automatic dimensioning modal.

Prompt	User Entry
IND ENT	Select the first reference entity.
IND ENT	Select the second reference entity.

If the ARROW ALIGNMENT modal in the DRAFTING MODALS menu is off, the dimension is created. If that modal is on, the system displays:

Prompt	User Entry
ARROWHEAD ALIGNMENT DESIRED? Enter:	
	Y To align the arrowheads to be produced with the arrowheads of the last dimension of this type created.
	N To suppress arrowhead alignment.

The system then requests that the user indicate the location of the lower left corner of the first character of text to be written by this operation. This procedure is described in the subsection on the LABEL AND DIMEN ORIGIN modal in the DRAFTING MODALS menu.

If the AUTO DIMENS modal is on, the system calculates the distance between the two entities selected and writes that distance in the dimension. If the AUTO DIMENS modal is off, the system requests that the user enter the text for the dimension.

Prompt	User Entry
KEY IN TEXT	Enter the text to be written in the dimension.

16.5 VERTICAL DIMEN

This operation produces a dimension between two entities. The dimension lines produced are vertical. The dimension includes extension (witness) lines, arrowheads, dimension lines, and text. The text in the dimension is automatically calculated or typed in by the user depending on the setting of the AUTO DIMENS modal.

Prompt	User Entry
IND ENT	Select the first reference entity.
IND ENT	Select the second reference entity.

If the ARROW ALIGNMENT modal in the DRAFTING MODALS menu is off, the dimension is created. If that modal is on, the system displays:

Prompt	User Entry
ARROWHEAD ALIGNMENT DESIRED? Enter:	
	Y To align the arrowheads to be produced with the arrowheads of the last dimension of this type created.
	N To suppress arrowhead alignment.

The system then requests that the user indicate the location of the lower left corner of the first character of text to be written by this operation. This procedure is described in the subsection on the LABEL AND DIMEN ORIGIN modal in the DRAFTING MODALS menu.

If the AUTO DIMENS modal is on, the system calculates the distance between the two entities selected and writes that distance in the dimension. If the AUTO DIMENS modal is off, the system requests that the user enter the text for the dimension.

Prompt	User Entry
KEY IN TEXT	Enter the text to be written in the dimension.

16.6 PARALLEL DIMEN

This operation produces a dimension between two entities. If the entities selected are two parallel lines, the dimension lines produced are perpendicular to two parallel lines. The dimension includes extension lines, arrowheads, dimension lines, and text. The distance is measured perpendicular to the two parallel lines. The text in the dimension is automatically calculated or is typed in by the user depending on the setting of the AUTO DIMENS modal.

Prompt	User Entry
IND ENT	Select the first reference entity.
IND ENT	Select the second reference entity.

The system then requests that the user indicate the location of the lower left corner of the first character of text to be written by this operation. This procedure is described in the subsection on the LABEL AND DIMEN ORIGIN modal in the DRAFTING MODALS menu.

If the AUTO DIMENS modal is on, the system calculates the distance between the two entities selected and writes that distance in the dimension. If the AUTO DIMENS modal is off, the system requests that the user enter the text for the dimension.

Prompt	User Entry
KEY IN TEXT	Enter the text to be written in the dimension.

16.7 ANGULAR DIMEN

This operation produces a dimension that gives the angle between two lines.

Prompt	User Entry
IND ENT	Use the crosshairs to select the first line.
IND ENT	Use the crosshairs to select the second line.

In selecting these two lines, the following affect the function of the angular dimension operation.

- The order in which the lines are selected is significant because the angle is measured from the first line to the second line in a counterclockwise direction.
- The end nearest which the line is selected is significant because the system measures the angle from or to the line (extended if necessary) on the side of the point of intersection corresponding to the end of the line.

Figure 3-32 show examples of the control of the angular dimension operation through the order of selection and the end selected.

The system then requests that the user indicate the location of the lower left corner of the first character of text to be written by this operation. This procedure is described in the subsection on the LABEL AND DIMEN ORIGIN modal in the DRAFTING MODALS menu.

If the AUTO DIMENS modal is on, the system calculates the distance between the two entities selected and writes that distance in the dimension. If the AUTO DIMENS modal is off, the system requests that the user enter the text for the dimension.

Prompt	User Entry
KEY IN TEXT	Enter the text to be written in the dimension.

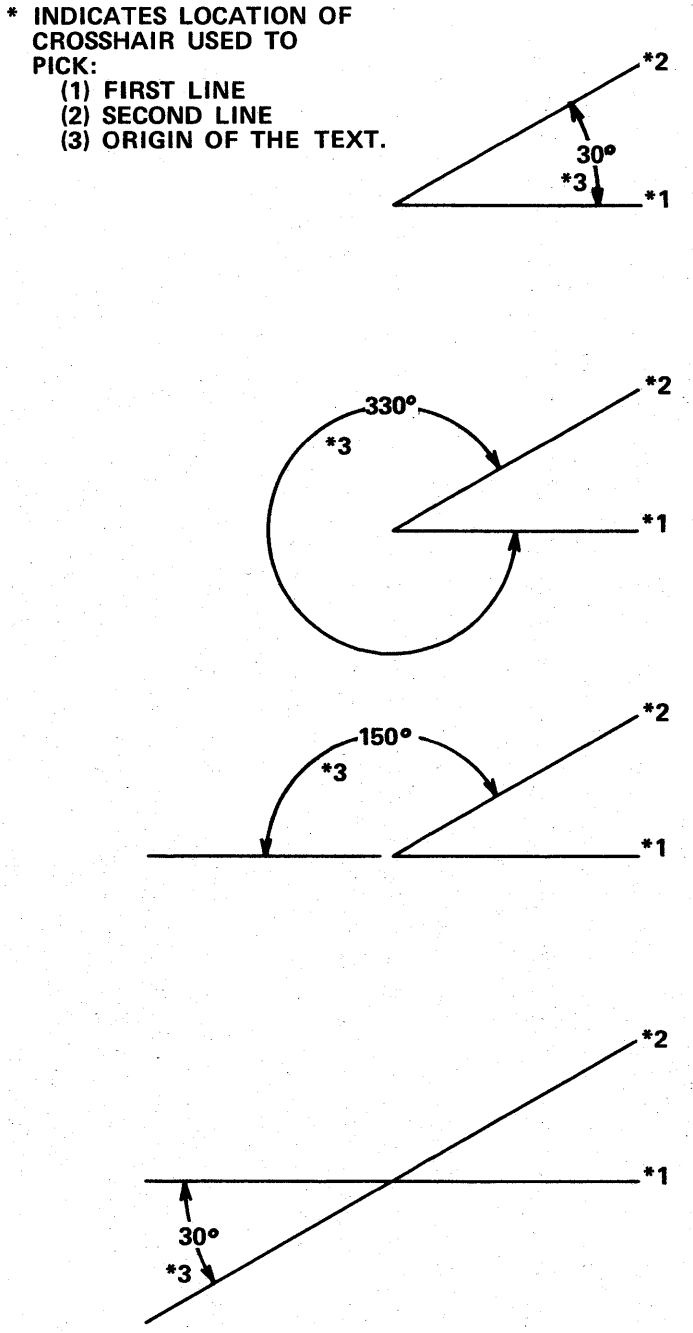


Figure 3-32. Angular Dimension Operation Examples

16.8 CIRCULAR DIMEN

This operation draws the dimension of the radius of an arc or circle.

Prompt	User Entry
IND ENT	Use the crosshairs to pick the arc or circle whose radius is to be dimensioned.
TAIL FROM TEXT: 1.START 2.END	Indicate to which end of the text the extension line will be drawn. Select: 1. To produce an extension line leading to the left of the text. 2. To produce an extension line leading to the right of the text.
ARROW FROM CIRCLE CENTER?	Determine the placement of the arrow. Select: Y To produce an arrow on the inside of the circle pointing outwards. N To produce an arrow on the outside of the circle pointing inwards.

The system then requests that the user indicate the location of the lower left corner of the first character of text to be written by this operation. This procedure is described in the subsection on the LABEL AND DIMEN ORIGIN modal in the DRAFTING MODALS menu.

If the AUTO DIMENS modal is on, the system calculates the radius of the arc or circle and writes that radius in the dimension. If the AUTO DIMENS modal is off, the system requests that the user enter the text for the dimension.

Prompt	User Entry
KEY IN TEXT	Enter the text to be written in the dimension.

The dimension is drawn with a letter R following the text to indicate a radius dimension.

16.9 DIAMETER DIMEN

This operation draws the dimension of the diameter of an arc or circle.

Prompt	User Entry
IND ENT	Use the crosshairs to select the arc or circle whose diameter is to be dimensioned.

The system then requests that the user indicate the location of the lower left corner of the first character of text to be written by this operation. This procedure is described in the subsection on the LABEL AND DIMEN ORIGIN modal in the DRAFTING MODALS menu.

If the AUTO DIMENS modal is on, the system calculates the diameter of the arc or circle and writes that diameter in the dimension. If the AUTO DIMENS modal is off, the system requests that the user enter the text for the dimension.

Prompt	User Entry
KEY IN TEXT	Enter the text to be written in the dimension.

The diameter dimension is drawn with a double-headed arrow extending across the circle.

16.10 NOTE

This operation writes text on the drawing. The user selects the location of the note and either the angle at which the text will be written or a line or arc that the text is to follow.

The system then requests that the user indicate the location of the lower left corner of the first character of text to be written by this operation. This procedure is described in the subsection on the LABEL AND DIMEN ORIGIN modal in the DRAFTING MODALS menu.

If the NOTE PARALLEL TO LINE/ARC modal is set to yes, the system displays:

Prompt	User Entry
PARALLEL TO LINE OR ARC?	Enter: Y To produce a line of text that follows an existing line or arc. N To produce a line at a chosen angle.

If Y was entered in response to this prompt, the system displays:

Prompt	User Entry
IND ENT	Use the crosshairs to pick the line or arc that the line of text is to follow.

If the entity selected is an arc, the system displays:

Prompt	User Entry
DIR:1.CW 2.CCW	Select the direction that the text is to follow by selecting: 1. To write text in a clockwise direction along the arc. 2. To write text in a counterclockwise direction along the arc.

If the entity selected is a line, the system displays:

Prompt	User Entry
DIR:1.POS X 2.NEG X	Indicate the direction that the note is to be written by selecting: 1. To write the note from left to right. 2. To write the note from right to left.

If the NOTE PARALLEL TO LINE/ARC modal is off or if N was entered in response to the prompt PARALLEL TO LINE OR ARC?, the system displays:

Prompt	User Entry
1.ANGLE = n.nnnn	Enter the angle at which the text is to be displayed. An angle equal to 0 produces horizontal text.

The system then requests that the user enter the text for the note.

Prompt	User Entry
KEY IN NOTE	Key in the text. Press RETURN to end a line. Press the] key to end entry of the note. The maximum number of characters that can be entered is 500 - 2x where x is the number of lines entered.

16.11 LABEL

This operation writes a label on the drawing to include user-entered text and an arrow from the text and pointing at an entity on the drawing.

Prompt	User Entry
IND ENT	Pick the entity at which the arrow is to point.
TAIL FROM TEXT 1.START 2.END	Enter: 1. To cause the arrow to originate at the left-hand side of the text. 2. To cause the arrow to originate at the right-hand side of the text
LEADER LINE:1.TO SEL PT 2.SLOPE	Indicate where on the entity the arrow is to point. Enter:

Prompt	User Entry
	1. To draw an arrow that points to the position on the entity where the entity was selected. 2. To be allowed to enter the slope (angle) of the arrow.

If 2.SLOPE was entered, the system requests the angle of the arrow.

Prompt	User Entry
1.SLOPE = n.nnnn	Enter the angle. The angle is measured in a counterclockwise direction from the positive x transform axis.

The system then requests the angle and location of the text.

Prompt	User Entry
1.TEXT ANGLE = n.nnnn	Enter the angle at which the lines of text in the label are to be written. The angle is measured in a counterclockwise direction from the positive x transform axis to an imaginary line under each line of text.
KEY IN LABEL	Enter the label. More than one line can be entered. Press RETURN to end a line. Press the] key to terminate entry of the label.

16.12 CENTERLINE

This operation draws a centerline between points, between circles, or between bolt circles (a circular array of points or circles).

Prompt	User Entry
1.POINT 2.CIRCLE 3.BOLT CIRCLE	Select the type of centerline. Enter: 1. To draw a centerline between points. 2. To draw a centerline between circles. 3. To draw a centerline through bolt circles.

If 1.POINT was selected, the system requests that the user pick the points.

Prompt	User Entry
PICK PTS	Use the crosshairs to pick existing points through which the centerline is to be drawn. Up to 12 points may be selected. If more than two points are indicated, all the points must be collinear (in a straight line).

If two points are selected, the centerline that is constructed begins and ends at the two points and has a short dash at the midpoint of the centerline. If more than two points are selected, the centerline begins and ends at the outermost points and has a short dash through each of the inner points.

If 2.CIRCLE was selected, the system requests that the user pick the circles.

Prompt	User Entry
PICK CIRCS	Use the crosshairs to pick the circles through whose centers the centerline is to be drawn. Up to 12 circles may be selected. If more than two circles are indicated, the centers of the circles must be collinear.

If a single circle is selected, a horizontal and a vertical centerline are drawn through the center. If more than one circle is selected, one common centerline is drawn through all the centers, and a separate centerline is drawn through each center and perpendicular to the common centerline.

If 3.BOLT CIRCLE was selected, the system requests that the user select the bolt holes.

Prompt	User Entry
IND BOLT CIRCLE	Use the crosshairs to pick a circular array of points or circles.

A circular centerline is drawn through all the points or centers, and a separate centerline is drawn through each center and normal to the circular centerline. If the array of points or circles selected was defined in a view other than the current plane of definition, the following error message is displayed.

Prompt	User Entry
THIS VIEW ONLY	Enter Y or N to continue.

16.13 MODIFY DRAFTING ENTITY

This operation modifies or alters various drafting entities without requiring the user to redefine the entity. The following modifications can be made.

- Change the origin of any label, note, or dimensioning text.
- Add or delete a rectangle around the text in a dimension.
- Add or delete parentheses around the text in a dimension.
- Add a tolerance to a dimension.
- Change the character size of a note.
- Change the text of a note.
- Alter the angle at which a note is written.
- Replace an existing tolerance.

Prompt	User Entry
MODIFICATION TYPE 1.NEW ORIGIN 2.BASIC 3.REFERENCE 4.ADD TOLERANCE 5.NEW CHAR. SIZE 6.MODIFY TEXT 7.MODIFY SLANT STATUS 8.MODIFY ANGLE 9.CHANGE TOLERANCE	Select the type of modification. Enter: 1. To change the origin of text written in label, note, or dimension. 2. To add a rectangle around the text in a dimension. 3. To add parentheses around the text in a dimension. 4. To add a tolerance to the text in a dimension. 5. To change the character size of a note or label. 6. To change the text in a note. 7. To change the slant status of the individual characters in a note. 8. To change the angle of the imaginary line along which the text of a note is written. 9. To replace an existing tolerance in a dimension.
IND ENTITY	Select the entity to be modified. The type of entities that can be selected depends on the type of modification requested.

If 1.NEW ORIGIN was selected, the system requests the location of the new origin using the method selected in the LABEL AND DIMEN ORIGIN modal in the DRAFTING MODALS menu. Indicate the location or press the [key to select a different method of indicating the location. The system draws the drafting entity with its origin at the new location.

If 2.BASIC was selected, a box is drawn around the text in the dimension. To remove the box (but not the dimension), use this operation a second time.

If 3.REFERENCE was selected, parentheses are written around the text in the dimension. To remove the parentheses (but not the dimension), use this operation a second time.

If 4.ADD TOLERANCE was selected, the tolerance information is added to the text in the dimension.

Prompt	User Entry
KEY-IN TOLERANCE	Type in the tolerance.

The tolerance should be entered in any of the following formats.

+tolerance1-tolerance2

or

@tolerance1

or

+tolerance1

or

-tolerance1

tolerance1 and tolerance2 are of the form n.nnnn . If both plus and minus tolerances are entered, the tolerances are written one above the other immediately following the text in the dimension and in 3/4-size characters. (Refer also to the description of the CHARACTER DISPLAY RATIOS under the DRAFTING MODALS menu.) If a single tolerance is entered, it is written in full-size characters. Entering the tolerance in the form using the commercial at sign (@)produces a tolerance containing a plus or minus symbol. Figure 3-33 gives examples of dimensions containing tolerances.

Prompt	User Entry
1.TOLERANCE 2.LIMITS	Indicate whether tolerances or limits are desired. Enter: 1. To write tolerances as described previously. 2. To write limits in full character size in square brackets. The upper limit is the dimension and the positive value. The lower limit is the remainder from subtracting the entered negative value from the dimension.

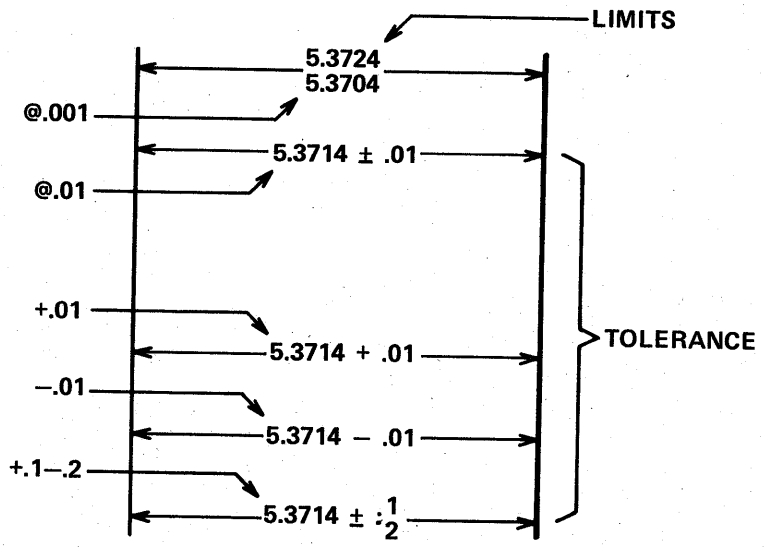


Figure 3-33. Tolerances and Limits

If 5.NEW CHAR. SIZE was selected, the system displays the current size of the characters and requests the size of characters desired.

Prompt	User Entry
CHAR. SIZE = n.nnnn	Type in the size at which the characters are to be rewritten.

If 6.MODIFY TEXT was selected, the system requests that the user specify the type of change to be made.

Prompt	User Entry
TEXT MOD. TYPE 1.DELETE LINE 2.INSERT LINE 3.REPLACE STRING	Select the type of change. Enter: 1. To delete an entire line of text. 2. To insert a new line of text. 3. To replace a part of a line.

If 1.DELETE LINE was selected, the system displays:

Prompt	User Entry
ENTER LINE NO.	Enter the number of the line to be deleted. In any group of lines, the lines are numbered consecutively beginning with 1, 2, 3,..., and so on.

If 2.INSERT LINE was selected, the system displays:

Prompt	User Entry
ENTER LINE NO.	Enter a line number. The user-entered line is inserted after this line. In any group of lines, the lines are numbered consecutively beginning with 1, 2, 3,..., and so on.
ENTER NEW STRING	Type in the new line.

If 3.REPLACE STRING was selected, the system displays:

Prompt	User Entry
ENTER LINE NO.	Enter the number of the line in which a string of characters is to be replaced. In any group of lines, the lines are numbered consecutively beginning with 1, 2, 3,..., and so on.
ENTER OLD STRING	Type in the string to be replaced. The system searches the line selected for this string of characters; this string is deleted.
ENTER NEW STRING	Type in the new string. These characters replace the old deleted string.

If 7.MODIFY SLANT STATUS was selected in response to the MODIFICATION TYPE prompt, the text selected is rewritten with slant if it was normal and rewritten normally if it was slanted.

If 8.MODIFY ANGLE was selected in response to the MODIFICATION TYPE prompt, the system requests the new angle.

Prompt	User Entry
TEXT ANGLE = n.nnnn	Enter the new angle. This angle is measured in a counterclockwise direction from the positive x transform axis to an imaginary line under the line of text.

If 9.CHANGE TOLERANCE was selected in response to the MODIFICATION TYPE prompt, the system requests the new tolerance information. The prompts and user entries for this subcommand are the same as for the 4.ADD TOLERANCE subcommand.

16.14 DETAIL MAG

This operation produces a magnified drawing of an area in a circle.

Prompt	User Entry
EXISTING CIRCLE?	Indicate whether the area to be magnified is within an existing circle or whether a circle must be constructed. Enter: Y To use an existing circle. N To create a new circle around the area to be magnified.

If Y was entered, the system asks the user to indicate the existing circle.

Prompt	User Entry
IND CIRCLE	Use the crosshairs to pick an existing circle.

If N was entered, the system asks the user to indicate the position of the center and edge of an imaginary circle that encloses and indicates the area to be magnified.

Prompt	User Entry
IND CENTER	Use the crosshairs to indicate the position of the center of the imaginary circle.
IND EDGE	Use the crosshairs to indicate a position on the edge of the circle to be constructed.

The system then requests that the user indicate the location and (optionally) the size of the circle in which the magnified reproduction is to be drawn.

Prompt	User Entry
MAG 1.EXISTS 2.MID/EDGE 3.MID/MAG	Select the method to be used to enter the location and size of the circle. Enter: 1. To pick an existing circle. 2. To create a new circle by picking center and edge positions. 3. To create a new circle by indicating a center position and a magnification size that the system will use to determine the size of the circle needed.

If 1.EXISTING CIRCLE was selected, the system displays:

Prompt	User Entry
IND CIRCLE	Use the crosshairs to pick an existing circle.
OK?	Enter: Y If the circle selected is correct. N If the wrong circle has been selected. The MAG prompt is redisplayed so that another circle can be selected.

If 2.CENTER AND EDGE was selected in response to the MAG prompt, the system asks whether the magnified drawing is to be bordered, where the center is to be, and how big it is to be.

Prompt	User Entry
BORDERED?	Enter: Y If a circle is to be drawn around the magnified drawing. N If no visible circle is to be drawn.
IND CENTER	Use the crosshairs to indicate the position of the center of the circle.
IND EDGE	Indicate the size of the circle by using the crosshairs to indicate a position on the edge of the circle.

If 3.CENTER AND MAGNIFICATION was selected in response to the MAG prompt, the system asks whether the magnified drawing is to be bordered, where the center is to be, and how much the detailed drawing is to be magnified. If the user asks for a border, a circle large enough to contain the magnified drawing is created.

Prompt	User Entry
BORDERED?	Enter: Y If a circle is to be drawn around the magnified drawing. N If no visible circle is to be drawn.
IND CENTER	Use the crosshairs to indicate the position of the center. The center of the magnified drawing is placed here.

Prompt	User Entry
1.MAGNIFCATN = n.nn	Type in the magnification factor desired. The size of the new drawing is the size of the old drawing magnified by the magnification factor. For example, a magnification factor of 1 causes no magnification or reduction; a factor of 2 causes a drawing twice as large as the original, and so on.
OK?	Enter: Y To continue. N To return to the MAGNIFIED AREA prompt and to specify a new circle.

16.15 BALLOON

This operation draws a balloon with an arrow pointing to an entity in the current drawing. The system writes a user-entered drawing and part number in the balloon.

Prompt	User Entry
IND ENTITY	Use the crosshairs to pick an entity in the current drawing. The arrow from the balloon points at this entity.
IND BALLOON CENTER	Use the crosshairs to indicate a position for the center of the balloon.
BALLOON RAD = n.nnnn	Enter the radius of the balloon.
ENTER DETAIL NO ENTER SHEET NO	Enter the part number and the sheet number that are to be written in the balloon.

16.16 TRUE POSITION SYMBOLS

This operation enables the user to draw true position symbols and associated user-entered text in his drawing.

Prompt	User Entry
ORIGIN 1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT 4.BELOW FEATURE CONTROL BOX 5.ABOVE FEATURE CONTROL BOX	Select the method to be used to indicate the position of the lower left corner of the feature control box. Enter: 1. To use the crosshairs to indicate a position. 2. To type in the transform coordinates of a position.

Prompt	User Entry
	3. To use the crosshairs to pick an existing point at the position.
	4. To place the feature control box directly below an existing box.
	5. To place the feature control box directly above an existing feature control box.

If 1.SCREEN POSITION was selected, the system displays:

Prompt	User Entry
IND POS	Use the crosshairs to indicate the location of the lower left corner of the feature control box.

If 2.KEY-IN was selected, the system displays:

Prompt	User Entry
1.XT = n.nnnn 2.YT = n.nnnn	Type in the transform coordinates for the lower left corner of the feature control box.

If 3.EXISTING POINT was selected, the system displays:

Prompt	User Entry
IND PT	Use the crosshairs to pick a point at the position of the lower left corner of the feature control box.

If either 4.BELOW FEATURE CONTROL BOX or 5.ABOVE FEATURE CONTROL BOX was selected, the system displays:

Prompt	User Entry
IND BOX	Use the crosshairs to pick an existing feature control box.

The system then asks whether the box is to be connected to an entity and how that connection is to be drawn.

Prompt	User Entry
CONNECTION DESIRED?	Enter:
	Y To produce a connecting line.
	N To avoid producing a connecting line.

If Y was selected, the system displays the following series of prompts.

Prompt	User Entry
IND ENTITY AT END	Use the crosshairs to pick the datum reference entity. Pick a point, line, arc, conic, or spline near the end of the entity to which the connection is desired.
CONNECTION TO FEATURE CONTROL BOX 1.LEFT 2.RIGHT 3.LL 4.LR 5.UR 6.UL	Indicate where the connection line should join the feature control box. These selections have the following meanings. 1. Midpoint of left edge of box. 2. Midpoint of right edge of box. 3. Lower left corner of box. 4. Lower right corner of box. 5. Upper right corner of box. 6. Upper left corner of box.
CONNECTION METHOD 1.JOG 2.DIRECT 3.DIRECT-MOD ORIGIN	Specify the type of connection line to be drawn. Enter: 1. To produce a horizontal line followed by a vertical line or vice versa. 2. To produce a straight line from the datum to the feature control box. 3. To cause the system to modify the position of the box so that a horizontal or vertical straight line connection can be drawn.
ARROWHEAD DESIRED?	Enter: Y To produce an arrowhead at the end on the connection line. N To omit the arrowhead.

The system then requests that the user select the geometric characteristic and the symbols.

Prompt	User Entry
GEOMETRIC CHARACTERISTIC 1.STRAIGHTNESS 2.FLATNESS 3.ROUNDNESS 4.CYLINDRICITY 5.PROFILE TO A LINE 6.PROFILE TO A SURFACE 7.ANGULARITY 8.PERPENDICULARITY 9.PARALLELISM 10.POSITION 11.CONCENTRICITY 12.SYMMETRY 13.CIRCULAR RUNOUT 14.TOTAL RUNOUT	Enter 1 through 14 to select the geometric characteristic. The symbol associated with the characteristic selected is drawn in the leftmost section of the box.
ENTER DATUM REFERENCE	Key in a datum reference. Each reference can consist of one to eight alphanumeric characters. Up to five datum references can be included in a single box.
SELECT SYMBOL 1.MMC (MAXIMUM MATERIAL CONDITION) 2.RFS (REGARDLESS OF FEATURE SIZE) 3.DIA (DIAMETER) 4.TOL ZONE PROJ	Enter 1 through 4 to select a symbol. Figure 3-34 shows these symbols.
ENTER TOLERANCE	Enter the tolerance. This entry may contain up to six characters to include the decimal point.
SELECT SYMBOL 1.MMC (MAXIMUM MATERIAL CONDITION) 2.RFS (REGARDLESS OF FEATURE SIZE) 3.DIA (DIAMETER) 4.TOL ZONE PROJ	Enter 1 through 4 to select a symbol.

16.17 ARROWHEAD AT END OF LINE

This operation draws an arrowhead at the end of an existing line.

Prompt	User Entry
IND ENT	Use the crosshairs to pick a line. An arrowhead is drawn at the end of the line nearest the crosshairs.

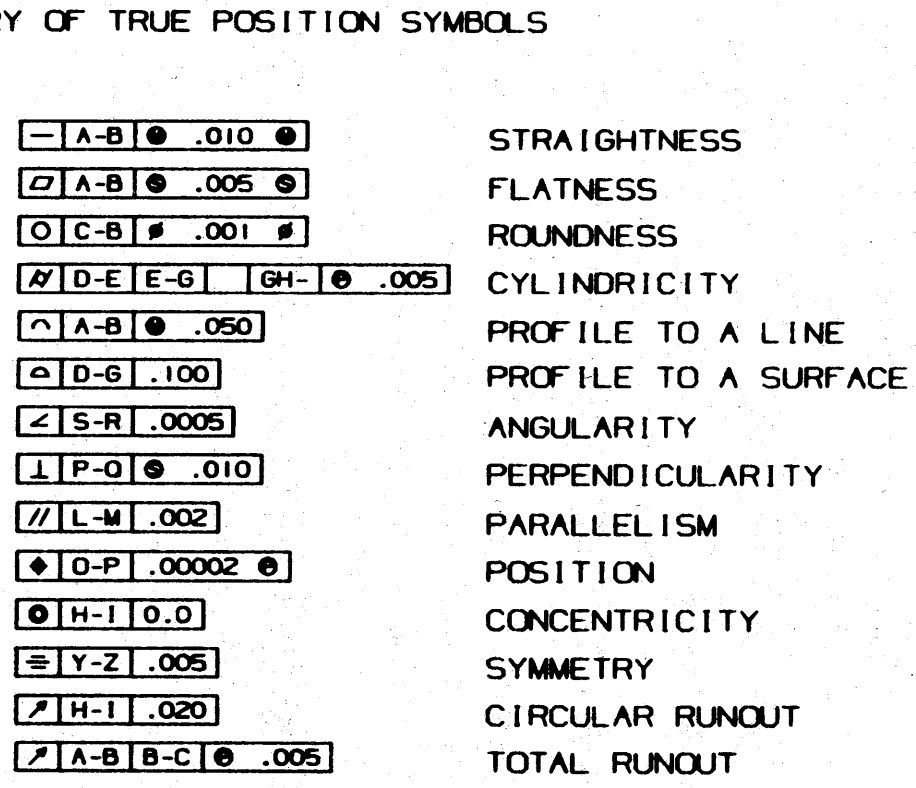


Figure 3-34. True Position Symbols

16.18 THICKNESS DIMENSION

This operation produces a dimension between two curves. The dimension measures the distance between the two curves from a point selected on the first curve along an imaginary line extending from that position and normal to the second curve to the position at which this imaginary line intersects the second curve.

Prompt	User Entry
LABELED (1ST) CURVE	Use the crosshairs to pick the first curve. The placement of the crosshairs along the curve is significant, since the measurement is made from the point on the curve nearest this position.
2ND CURVE	Use the crosshairs to pick the other curve.

The system then requests that the user indicate the location of the lower left corner of the first character of text to be written by this operation. This procedure is described in the subsection on the LABEL AND DIMEN ORIGIN modal in the DRAFTING FUNCTIONS modals.

18. ANALYSIS

The operations in this menu calculate and display information on the properties of entities in the current part. Any of the following can be calculated and displayed.

- The slopes and curvatures of splines.
- The area and perimeter of a two-dimensional closed figure.
- Two-dimensional analysis to include:

  The perimeter and area.

  Center of gravity.

  The first moments of mass with respect to each coordinate axis.

  The radii of gyration with respect to each coordinate axis and the origin.

  The moment of inertia with respect to each coordinate axis and the origin.

18.1 SPLINE ANALYSIS

This operation determines and displays a graph showing:

- The slope at each of the points of definition along the spline.
- The curvature of the points of definition along the spline.
- The radius of curvature of the points of definition along the spline.

The positions with respect to the x transform axis of the points on the graph and the points of definition along the spline are the same. The heights (or positions with respect to the y axis) of the points on the graph show the slope, the amount of curvature, and radii of curvature for each point of definition of the spline.

Prompt	User Entry
IND ENT	Use the crosshairs to pick a spline.
SELECT ANALYSIS 1.SLOPE 2.CURVATURE 3.RAD OF CURV 4.XT 5.YT	Choose which type of information is to be displayed. Enter:

Prompt	User Entry
	1. To display a graph showing the slope of the spline with respect to the x axis. 2. To display a graph of the curvature along the spline. 3. To display a graph of the radii of curvature along the spline. 4. To display a graph of the XT. 5. To display a graph of the YT.

If the extent of the plot in the x transform direction (horizontally across the screen) is less than half of the available display, the following message is displayed.

Prompt	User Entry
XSCL = YSCL?	Enter: N To cause the graph to be stretched in the x transform direction and redisplayed to fill the screen. Y To cause the graph to remain unchanged.

18.2 ANALYTIC AREA/PERIMETER

This operation calculates and displays the area and the length of the perimeter of a single closed figure.

Prompt	User Entry
MODE:1.SINGLE 2.CHAIN	Use the entity selection procedure to pick a set of entities that form a single closed figure.

If the selected figure contains segments that are not connected, the following message is displayed.

Prompt	User Entry
DOESN'T CONNECT, TRY AGAIN	Enter: Y To try again to select the figure. N To exit from this operation.

If a connected figure has been selected, the system displays the perimeter and area in the following format.

```
1.PERIMETER = n.nn
2.AREA = n.nn
```

Press the] key to display the area in acres as follows:

1.ACRES = n.nnnn

18.3 2-D SECTION ANALYSIS

This operation calculates any of the following on any two-dimensional figure.

- Length of the perimeter.
- Area.
- Center of gravity.
- First moment.
- Moment of inertia.
- Radius of gyration.
- Polar moment of inertia.
- Polar radius of gyration.
- Minimum and maximum x and y.

The user specifies the location of the origin to be used in the analysis, selects the figure on which analysis is to be performed, and chooses the type of analysis to be performed.

Prompt	User Entry
ORIGIN:1.0,0 2.KEY-IN 3.CG OF FIG	Select the method to be used to specify the location of the origin of a coordinate system. This coordinate system is used in the analysis to be performed. Enter: 1. To use the origin (0,0) in the coordinate system. 2. To type in the coordinates of the origin. 3. To use the computed location of the center of gravity of the figure that will be selected.

If 2.KEY-IN was selected, the following prompt is displayed.

Prompt	User Entry
1.Y-AXIS X = 2.X-AXIS Y =	Type in the coordinates of the origin.

The system then requests that the user select the figure for which information is to be computed.

Prompt	User Entry
1.SINGLE 2.CHAIN	Use the entity selection procedure to pick a set of entities that form a single closed figure.

The system then requests that the user select the type of analysis to be performed.

Prompt	User Entry
SELECT ANALYSIS 1.PERIMETER 2.AREA 3.CENTER OF GRAVITY 4.FIRST MOMENTS 5.MOMENT OF INERTIA 6.RADII OF GYRATION 7.POLAR INERTIA 8.POLAR RADIUS	Select the type of analysis. Enter: 1. To determine the perimeter of the figure. 2. To determine the area. 3. To determine the center of gravity of the figure when the figure is considered as a sheet of even thickness. 4. To determine the first moments of mass with respect to each coordinate axis. 5. To determine the moments of inertia with respect to each coordinate axis. 6. To determine the radii of gyration with respect to each coordinate axis. 7. To determine the moment of inertia with respect to the origin. 8. To determine the radius of gyration with respect to the origin.

The origin used in response to selections 3 through 8 is the origin indicated in response to the prompt:

```
ORIGIN:1.0,0 2.KEY-IN 3.CG OF FIG
```

The coordinate axes used in response to selections 4, 5, and 6 are those that pass through this same user-entered origin and are parallel to the bottom and left edge of the screen.

18.4 3-D ANALYSIS

Not provided with this release.

19. SIU/ENGLISH/RESIZE

Prompt	User Entry
MODE:1.METRIC/ENGLISH 2.RESIZE	Indicate 1 for conversion from metric to English. Indicate 2 to resize the part to a different scale.
MODE:1.ENGLISH/METRIC 2.RESIZE	Indicate 1 for conversion from English to metric. Indicate 2 to resize the part to a different scale.
DO YOU WANT TEXT CONVERTED?	This question is asked if there are any automatically created dimensions in the PART. Indicate YES or NO in response to this question. (Keyed-in dimensions are not converted.)
1.DEC PLACES = 2	If the answer to the above question is YES, this message appears. The user should indicate the number of places he wishes to appear after the decimal point in the converted dimensions. The preset value is 2 for metric and 4 for English.
1.RESIZE FAC = 1.00	Indicate the desired rescale factor.

GLOSSARY A

Angular Dimension	This type of dimension is a measure of the angle between two lines in degrees. It is produced by the ANGULAR DIMEN operation in the DRAFTING FUNCTIONS menu.
Automatic Dimension Calculation	The facility by which AD-2000 calculates the distance, or in the case of angular dimensions, the angle for a dimensioning operation and automatically writes the text specifying that distance or angle in the dimensions. If the modal that controls automatic dimension calculation is off, the user must type in the text to be written in a dimension.
Auxiliary View	These views may be defined by the user in addition to the standard views defined by the system: plan, front, right, and isometric. The user creates auxiliary views by rotating and tilting the part using operations in the ZOOM/DEPTH/VIEW CONTROL menu.
Blank	Entities in the user's part that the user has specified as blanked that remain in the part but are not displayed on the screen.
Chamfer	Two lines, arcs, or curves may be joined by a straight line, called a chamfer, usually used to change a sharp corner into a blunt one.
Circular Array	A set of objects produced by repeatedly creating the same object at equal spacings along an imaginary circle.
Data Capture	The facility that enables the user to capture the default value for a parameter or the displayed value which was previously typed in by the user for a parameter and to save that value in a variable. The variable name can later be entered instead of a numerical value.
Fillet	Two lines, arcs, or curves may be joined by an arc, called a fillet, that begins and ends at a point of tangency on each of the two lines.

Group	The user can specify a collection of entities to be in a group; for example, all the entities in the group can be picked by a single use of the crosshairs and the C key.
Macro	A sequence of operations selected from the POINT, LINE, and ARC/CIRCLE/FILLET menus that can be referred to by name and can be executed. If variables have been used in place of typing in numeric values and if the values of the variables have been updated, the latest value of the variable is used when the macro is executed.
Max/Min	These values specify the lines perpendicular to the xt and yt transform axes along which the view of the part is clipped. The user can control the extent of the part displayed on the screen by entering a maximum and minimum xt value and a maximum and minimum yt value. These values are entered through the KEY-IN MAX-MINS operation in the ZOOM submenu in the ZOOM/DEPTH/VIEW CONTROL menu. The rectangular portion of the part that falls between these maximum and minimum values is displayed filling as much of the screen as possible. If the rectangle is not geometrically similar to the screen, additional portions of the part may also be displayed.
Mirror	This operation enables the user to produce a copy of a set of entities in the part by rotating them 180 degrees around a selected line.
Modal	These switches and values control the execution and operation of AD-2000. The user can change the settings and values of modals through use of the MODALS AND FONTS menu and the DRAFTING MODALS submenu in the DRAFTING FUNCTIONS menu.
Model Coordinates	Model coordinates (also absolute coordinates) are the displacements of positions along the x, y, and z axes. The x and y axes are horizontal and vertical in view 1, and the z axis is normal to the screen (or 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.

Pattern	A set of entities, selected and given a name by the user, can be copied into a position specified by the user. Operations in the PART/PATTERN MANAGEMENT menu enable the user to create patterns and to copy them into the part.
Rectangular Array	A set of objects produced by repeatedly creating the same object at equal spacings along rows and columns. The RECT ARRAY operation in the ENTITY MANIPULATION menu creates rectangular arrays.
Released Part Status	A part whose status is released cannot be modified. The status of a part can be changed to released through use of the CHANGE PART STATUS operation.
Run-Time Library (RTL)	The RTL is the storage area for all variables defined through variable calculation and GRAPL program runs in the GRAPL submenu in the SPECIAL FUNCTIONS menu and through data capture. Operations in the GRAPL submenu enable the user to move variables between the RTL and the UTF. Changes and additions made to the RTL are lost if the current part is not filed. Each part has its own RTL.
Transform Coordinates	The xt, yt, and zt transform coordinates are 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 (or plane of definition) in the current working view. The transform coordinates are useful in views that have been rotated from view 1; for example, the user can place an object to the left or right of a known coordinate position by decreasing or increasing the xt transform coordinate value. Notice that, 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.
Trim	A line, arc, or curve can be divided by a boundary consisting of another line, arc, or curve and one portion or the other deleted. This process is called trimming and is done with the TRIM CURVES operation in the OTHER CURVES menu. Also, two boundaries may be used to delete either the middle or both ends of the line, arc, or curve.

User Technology File (UTF)	The UTF is the storage area for various products of the SPECIAL FUNCTIONS menu, including user-defined character sets, source and object forms of GRAPL programs, level tables, user text statements, and macro sequences. Also, operations in the GRAPL submenu of the SPECIAL FUNCTIONS menu enable the user to move variables between the UTF and the RTL. Status of changes and additions made to the UTF are independent of whether the current part is filed. There is only one UTF, which is shared by all parts.
Variable Calculation	This operation in the GRAPL submenu in the SPECIAL FUNCTIONS menu calculates the value of a user-entered arithmetic expression and assigns that value to a user-specified variable. The variable can be used where typed-in numeric values are required.
Witness Lines	These extension lines indicate the entities between which dimensions are drawn and the limits of the distance being measured.
Work View	If more than one view is displayed on the screen, the work view is that view in which the crosshair can be used to select entities and to indicate positions; it is the view in which the position of the crosshairs relative to the objects in that view is what it appears to be.

AD-2000 MENUS B

(This information will be supplied at a later date.)

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