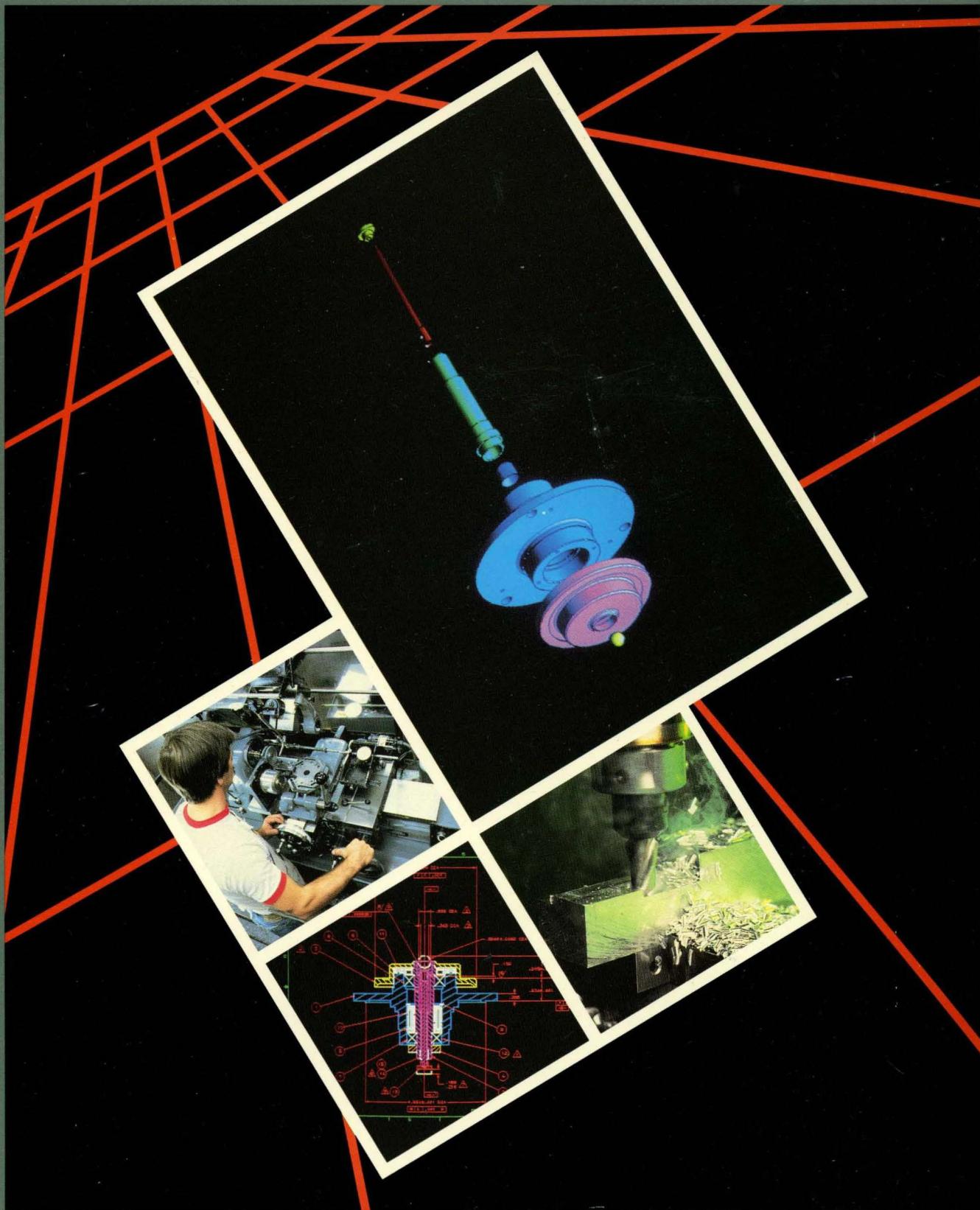


ICEM Design/Drafting

Drafting Functions for NOS



ICEM Design/Drafting

Drafting Functions

for NOS

Reference

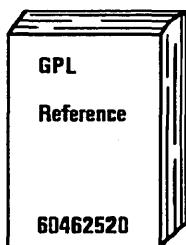
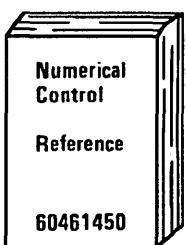
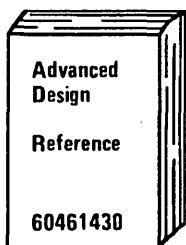
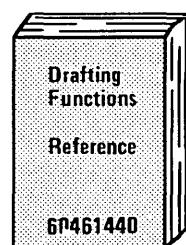
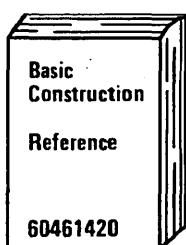
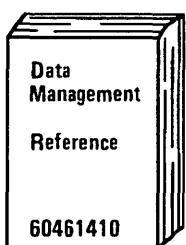
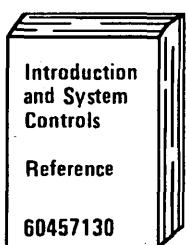
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Related Manuals

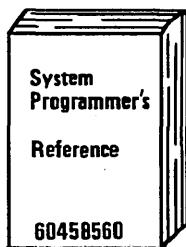
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Manual Set:



Additional:



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Manual History

Revision C documents ICEM DDN Version 1.6, printed December 1985. This revision includes enhancements and corrections to existing features or the addition of new features as reflected in the following list.

- Additions to 16.1 DRAFTING MODALS including dimension text prefix or suffix characters, dimension text orientation, automatic tolerances or limits, and isometric drafting.
- Addition of chamfer, arc length, and rectangular coordinate dimensions to 16.2 DIMENSION.
- Enhancements to 16.3 SECTION LINING that allow you to set modals and define the type of section lining you want to use within this menu.
- Addition of composite geometric tolerance symbols to 16.7 GEOMETRIC TOLERANCE FRAMES.
- Enhancements to 16.13 MODIFY DRAFTING ENTITY that allow you to add or remove prefix or suffix text and to indicate dimensions that are not to scale.
- Additional explanation and illustrations for 18.1 SPLINE ANALYSIS and 18.3 2-D SECTION ANALYSIS.

This revision also includes other technical and editing changes.

Previous Revision	System Version	Date
A	1.53	November 1984
B	1.57	May 1985

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About This Manual

This manual describes the CONTROL DATA® Integrated Computer-aided Engineering and Manufacturing Design/Drafting/Numerical Control (ICEM DDN) software system. It includes instructions for using ICEM DDN menus 16 DRAFTING, 18 ANALYSIS, and 19 SI/US/RESIZE.

Audience

This manual is a reference source for design engineers and drafting personnel who have already had initial training in the use of the ICEM DDN system. It is not intended to be a tutorial guide to ICEM DDN. New users should refer to the ICEM Design/Drafting User's Guide for a step-by-step introduction to the ICEM DDN system.

Organization

Chapter 1 of the manual gives instructions for using menu 16 DRAFTING to create drafting entities for mechanical line drawings. Drafting functions available in the ICEM DDN system include:

- Dimensioning
- Tolerancing
- Annotating
- Setting modals
- Section lining
- Automatic detail magnifying
- Modifying drafting entities

1982 ANSI drafting standards apply to all functions.

Chapter 2 contains menu 18 ANALYSIS. You can use this menu to calculate and display the following kinds of information about drafting entities:

- Slope and curvature of splines
- Area and perimeter of a two-dimensional, closed figure
- Two-dimensional analysis
- Three-dimensional analysis

Chapter 3 tells how to use menu 19 SI/US/RESIZE to convert parts (drawings) from SI units to U.S. customary units or vice versa and how to resize a part.

This manual is part of the ICEM DDN manual set.

The ICEM Design/Drafting Introduction and System Controls manual gives an overview of the major ICEM DDN concepts and describes menus 1 through 4 of the main menu: modals and fonts, blank/unblank operations, delete operations, and the file/exit sequence.

The ICEM Design/Drafting Data Management manual describes menus 5 through 8 of the main menu: data base management operations, input/output operations, and display control.

The ICEM Design/Drafting Basic Construction manual describes menus 9 through 14 of the main menu: point construction, line construction, arc construction, special curve construction, entity manipulation, and data verification.

The ICEM Advanced Design manual describes menu 15 ADVANCED DESIGN, which covers three-dimensional curves and surfaces.

The ICEM Numerical Control manual describes menu 17 NUMERICAL CONTROL, the numerical control programming part of ICEM DDN.

The ICEM GPL manual describes menu 5.13 GPL and the GPL programming language.

Conventions

In this manual, headings contain a series of numbers separated by periods. These numbers represent the selections available within the ICEM DDN menu hierarchy. The first number in the heading is the main menu choice, the second number is from the second-level menu, and so on. For example, menu choice 12.7.3 HEXAGON is from the third level of the menu hierarchy.

When the word *system* is used, it refers to the ICEM DDN software system. When the Network Operating System is referred to, it is called either NOS or the operating system.

All text that the system displays is shown in uppercase letters and highlighted with a special typeface, as shown below:

PEN THICKNESS
1.ON
2.OFF
3.SET PEN THICKNESS

Additional Related Publications

You can find related information in the following publications:

Manual Title	Publication Number
Network Products Interactive Facility Version 1 Reference Manual	60455250
Network Products Interactive Facility Version 1 User's Guide	60455260
NOS Version 1 Reference Manual, Volume 1	60435400
UNIPLOT Version 3 User's Guide/Reference Manual	60454730
Automatically Programmed Tooling System (APT IV) Version 2 Reference Manual	17326900
XEDIT Version 3 Reference Manual	60455730
Graphics Terminal Assist Version 1 User's Guide/Reference Manual	60476100
NOS 2 Reference Set, Volume 1, Introduction to Interactive Usage	60459660
NOS 2 Reference Set, Volume 2, Guide to System Usage	60459670
NOS 2 Reference Set, Volume 3, System Commands	60459680
ICEM Design/Drafting GRAPL Programming Language	60461460
ICEM User-Defined Tablet Overlay	60457650
ICEM Engineering Data Library Reference Manual	60459740

Ordering Manuals

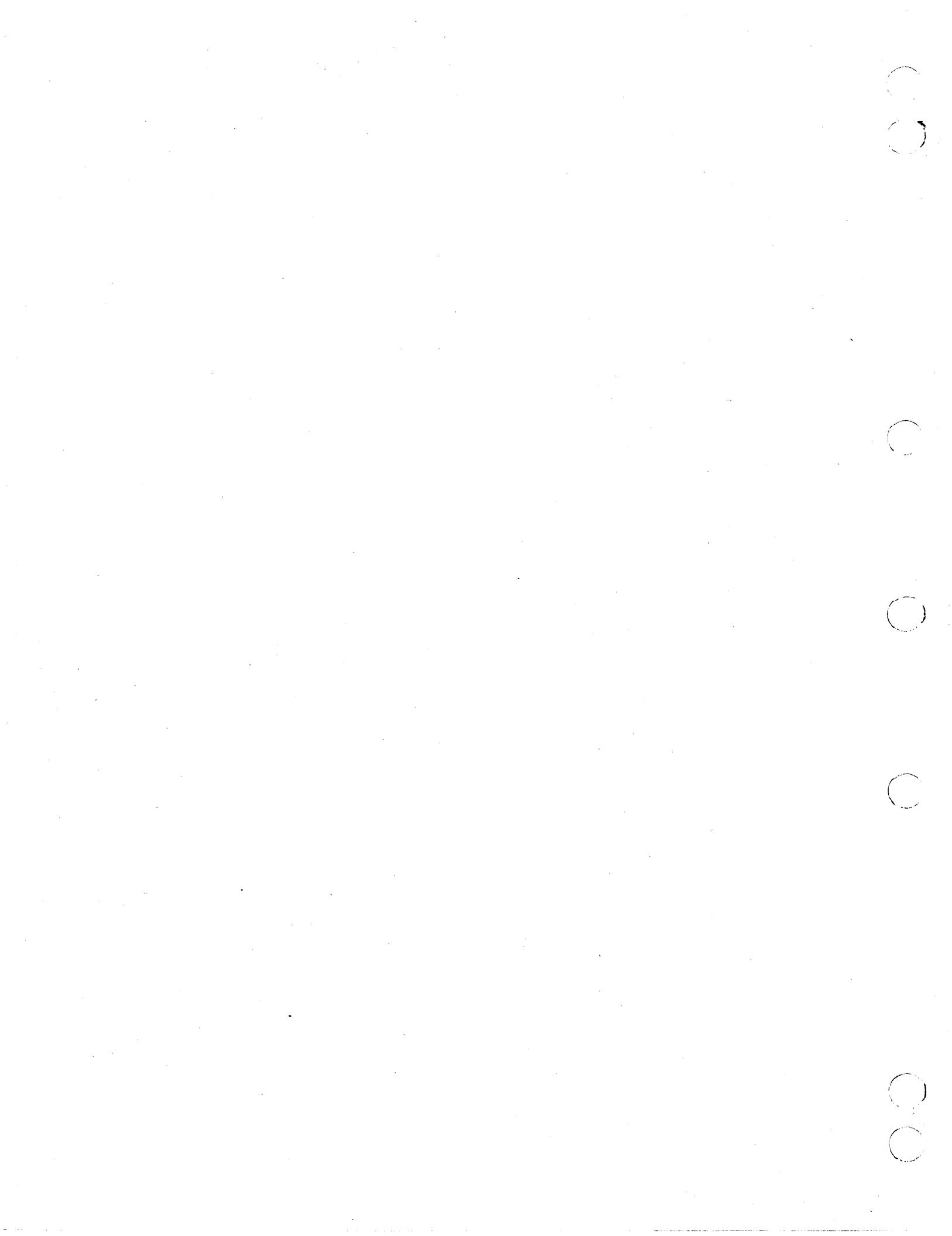
Control Data manuals are available through Control Data sales offices or through Control Data Corporation Literature Distribution Services (308 North Dale Street, St. Paul, Minnesota 55103).

Submitting Comments

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

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Menu 16: Drafting

1

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Menu 16: Drafting

1

Using Menu 16

You can use this menu to create the following drafting entities required for mechanical line drawings:

- Dimensions
- Extension lines and arrows
- Section lining
- Notes and labels
- Geometric tolerance symbols
- Centerlines
- Detail magnifications
- References to drawings in balloons
- Surface texture-basic symbol

All drafting entities are created at the current depth.

The Drafting menu is:

```

DRAFTING
1.DRAFTING MODALS
2.DIMENSION
3.SECTION LINING
4.BALLOON
5.DATUM TARGET
6.DATUM FEATURE
7.GEOMETRIC TOLERANCE FRAMES
8.SURFACE TEXTURE-BASIC SYMBOL
9.ARROW
10.NOTE
11.LABEL
12.CENTERLINE
13.MODIFY DRAFTING ENTITY
14.DETAIL MAGNIFY

```

The drafting functions give you the following capabilities:

Menu Title	Description
16.1 DRAFTING MODALS	Sets values and options needed for the available drafting functions.
16.2 DIMENSION	Generates horizontal, vertical, parallel, thickness, angular, radius, diameter, taper, chamfer, arc length, and rectangular coordinate dimensions.

Menu Title	Description
16.3 SECTION LINING	Produces section lining (formerly called crosshatching) within specified boundaries.
16.4 BALLOON	Draws a balloon with an arrow pointing to an entity in the current drawing.
16.5 DATUM TARGET	Defines a datum target symbol with a leader pointing to a target area or location.
16.6 DATUM FEATURE	Displays a datum feature symbol and associated text within a feature frame on a drawing.
16.7 GEOMETRIC TOLERANCE FRAMES	Displays geometric tolerance or composite geometric tolerance symbols and associated text within a feature frame on a drawing.
16.8 SURFACE TEXTURE-BASIC SYMBOL	Displays the surface texture-basic symbol either displaced from an entity or entities and joined to the entities by arrows or attached directly to the entity.
16.9 ARROW	Draws an arrowhead along a curve or creates cross-section arrows for declaring details or cutouts.
16.10 NOTE	Writes text on a drawing and specifies its location.
16.11 LABEL	Writes a label on a drawing, including text and an arrow from the text to an entity on the drawing.
16.12 CENTERLINE	Draws a centerline between points, circles, or a circular array of points or circles.
16.13 MODIFY DRAFTING ENTITY	Modifies various drafting entities without redefining them.
16.14 DETAIL MAGNIFY	Magnifies splines, point sets, rectangular arrays, circular arrays, and section lining material.

After you respond to the drafting functions prompts, the system displays the newly created entity and then takes one of two actions:

- If 1.2.1 CONTINUE OPERATION MODE is on, the system redisplays the prompts and allows you to define more entities. Entering] terminates this option and returns you to the next higher menu.
- If 1.2.1 CONTINUE OPERATION MODE is off, the system returns to the next higher menu.

1.2.1 CONTINUE OPERATION MODE is a construction modal that is explained in the ICEM Design/Drafting Introduction and System Controls manual.

The system uses a general procedure for the dimensioning and label functions that specifies the location of the text written by the operation. This general procedure is described in 16.1.12 DIMENSION ORIGIN.

You can select text on the screen by using the graphics cursor and entity select character T. To select section lining, use entity select character H. All functions in this menu use U.S. ANSI standard dimensioning.

Using Isometric Drafting

When using Isometric Drafting, (menu 16.1.14), drafting entities are created in the current workspace. Then, if the current workspace is not aligned with the work view, the drafting entities created appear appropriately skewed.

For example, you can view a model in a three-dimensional or isometric view but have the front plane as your work plane. Drafting entities created then appear in that plane and are skewed. The work plane and view plane do not need to be aligned. Isometric drafting allows the work plane and view plane to be independent of each other.

To use isometric drafting effectively you should be familiar with these five Design/Drafting features:

1. Depth control (D command key or 8.1.1 DEPTH ENTRY MODE)
2. Implicit points (1.11.5 IMPLICIT POINT MODE)
3. Changing workspace (8.11 CHANGE WORK SPACE)
4. Defining a new workspace (8.12 DEFINE WORK SPACE)
5. Selective view blanking (1.10 BLANK/UNBLANK VIEW SELECT)

Controlling Drafting Entity Placement in Three-Dimensional Space

To control drafting entity placement while in isometric drafting, you can do two things. The simplest is to change the depth of the current workspace using the D or Depth command key. This moves you along the zt-axis of the workspace. The second method is to change the workspace itself. Use menu 8.11 CHANGE WORK SPACE to change from front to top, for example. This works for all views created. This operation automatically resets the depth to zero. You must then change the depth to the appropriate value.

Use menu 8.12 DEFINE NEW WORK SPACE when a new workspace is required. You may need to do this for models that have skewed or diagonal lines that do not fall on orthogonal planes. This operation also resets depth to zero.

Joint Use of Depth Control and Implicit Points

The default mode for 8.1.1 DEPTH ENTRY MODE requires you to enter a value for the new depth (1.ENTER DEPTH VALUE). In cases where you do not know what new depth value to enter, use 2.INDICATE POINT or 3.DELTA FROM A CURVE END. Indicating a point changes the depth to the zt value of that point. This lets you change depth by screen selecting points.

1.11.5 IMPLICIT POINT MODE can be used to further enhance this capability by automatically defining points at curve ends, middles of curves, and circle centers. Using depth control and implicit points together allows you to screen select curves and points to reset depth.

Changing Workspace

Most isometric drafting is done on the three model axes: front, right, and top. A combination of changing between these three axes and setting correct depth is the most common method for controlling the placement of drafting entities.

Defining New Workspace

You must define a new workspace if you have entities that do not exist on one of the standard eight workspaces automatically defined by the system. Use menu 8.12 DEFINE WORK SPACE.

Selective View Blanking

Selective View Blanking (1.10 BLANK/UNBLANK VIEW SELECT) is the fifth feature you will find helpful. After creating entities with isometric drafting modals on, you can selectively view blank to manipulate the viewing of these or other entities. If you turn this modal on, you can blank or unblank entities according to their view. If the entities created were set to be displayed only in their view of definition, you can turn the modal off to unblank those entities while viewing them from some other view. If entities created were set to be displayed in all views, you can turn the modal off to blank selected entities while in any view.

16.1 Drafting Modals

You can use menu choice 16.1 DRAFTING MODALS to set the values and options needed for the available drafting functions. To see the preset values for the drafting modals, select function 16.1.16 DISPLAY DRAFTING MODALS when you enter ICEM DDN.

The Drafting Modals menu is:

- DRAFTING MODALS
 - 1.CHARACTER CONTROL
 - 2.TEXT JUSTIFICATION
 - 3.TEXT ORIGIN METHOD
 - 4.DIMENSION TEXT
 - 5.TEXT ANGLE CONTROL
 - 6.DRAFTING SCALE FACTOR
 - 7.ARROWHEAD LENGTH
 - 8.EXTENSION/DIMENSION LINE CONTROL
 - 9.ARROW/LEADER CONTROL
 - 10.ARROW ALIGNMENT
 - 11.DIMENSION OFFSET DISTANCES
 - 12.DIMENSION ORIGIN
 - 13.AUTOMATIC TAIL LOCATION
 - 14.ISOMETRIC DRAFTING
 - 15.DRAFTING STANDARDS
 - 16.DISPLAY DRAFTING MODALS

The following sections describe each modal individually.

16.1.1 Character Control

With this modal, you can select symbol characteristics used when defining dimensions, labels, and notes. The menu for this modal is:

- CHARACTER CONTROL
 - 1.CHARACTER SIZE
 - 2.CHARACTER SET TYPE
 - 3.CHARACTER SLANT
 - 4.DISPLAY RATIOS
 - 5.SPECIAL SET PREFIX
 - 6.OUTPUT MODE

The following pages describe these choices and their resulting prompts.

16.1.1.1 Character Size

With this modal, you can specify the height of the characters written by the drafting dimension, balloon, datum target, datum feature, geometric tolerance, surface texture-basic symbol, note, and label operations. Existing dimensions are not affected. If the characters are too small for the screen, the system draws a straight line, even through blank spaces.

CHARACTER SIZE = n.nnnn

Enter the height of the characters to be displayed. This entry specifies the size of characters drawn in a full-scale plot.

16.1.1.2 Character Set Type

With this modal, you can select the character set used for defining text. This modal does not affect text in existing dimensions, labels, or notes.

CHARACTER SET TYPE

- 1. STANDARD
- 2. USER-GENERATED

Enter:

- 1 To use the standard character set. The system returns to 16.1.1 CHARACTER CONTROL.
- 2 To access a user-generated character set. The system displays the following prompt:

ENTER CHARACTER SET NAME

Enter the four-character name of the user-generated character set you want to access.

If the character set you specified is not found, the system displays:

CHARACTER SET cccc NOT FOUND

The ICEM Design/Drafting Data Management manual explains how to create a user-defined character set (see 5.4 USER-DEFINED SYMBOLS). The system displays standard characters for any characters not defined in the set.

16.1.1.3 Character Slant

With this modal, you can select vertically oriented or slanted characters. This modal does not affect existing text. The default setting for character slant is off.

CHARACTER SLANT

- 1.ON
- 2.OFF

Enter:

- 1 To produce slanted characters.
- 2 To produce vertical characters.

16.1.1.4 Display Ratios

With this modal, you can modify relationships between characters. This modal does not change existing dimensions, only those created after changing the modal.

DISPLAY RATIOS

1.SPACING = n.nnnn
 2.ASPECT = n.nnnn
 3.DOWN SPACE = n.nnnn
 4.TOLERANCE RATIO = n.nnnn

Enter the spacing factor. This value, multiplied by the character height (set by 16.1.1.1 CHARACTER SIZE), gives the spacing between the centers of consecutive characters. When you enter the system, this value is preset to 1.1.

Enter the ratio between the character width and the character height. This value is preset to 1.

Enter the line spacing factor. This value, multiplied by the character height, gives the distance from the bottom of one text line to the bottom of another text line. This value is preset to 1.5.

Enter the tolerance or fractional ratio between the character size for tolerance characters and the character size for main characters in dimensions. This value is preset to 1.0.

16.1.1.5 Special Set Prefix

With this modal, you can change the character used to prefix a special symbol character set.

CURRENT PREFIX CHARACTER IS c
 ENTER NEW PREFIX CHARACTER

Enter the new single character used to prefix special symbols.

The previous prefix character is c. Reverse slant (\) is the default prefix character.

Entering the prefix character, followed by certain preassigned characters, allows you to include drafting symbols in notes, labels, and dimensions. See table 1-1, Special Character Symbols, for the list of predefined characters and associated symbols.

Table 1-1. Special Character Symbols

Character Entry	Symbol	Symbol Name
\D	▼	Depth
\L	[Left bracket
\O	Ø	Diameter
\R]	Right bracket
\S	□	Square
\B	□	Counterbore
\C	▽	Countersink
_	°	Degree
\@	±	Plus or minus
\	\	Prefix character
\c	c	All other characters

16.1.1.6 Output Mode

With this modal, you can select an output mode for standard set type characters. This modal affects all existing standard characters generated. Lowercase characters appear as uppercase in coarse mode. Figure 1-1 shows examples of coarse and fine characters. The bold-faced characters are user-generated.

OUTPUT MODE

- 1.FINE
- 2.COARSE

Enter:

- 1 To use characters generated with multiple strokes.
- 2 To use standard characters generated with few strokes.

COARSE CHARACTERS.

THESE ARE STANDARD VERTICAL CHARACTERS.
THESE ARE USER VERTICAL CHARACTERS.
THESE ARE STANDARD SLANTED CHARACTERS.
THESE ARE USER SLANTED CHARACTERS.

Fine characters.

THESE ARE STANDARD VERTICAL CHARACTERS.
THESE ARE USER VERTICAL CHARACTERS.
THESE ARE STANDARD SLANTED CHARACTERS.
THESE ARE USER SLANTED CHARACTERS.

Figure 1-1. Character Display Sets

16.1.2 Text Justification

Use this modal to specify the text origin (in relation to the text) and text justification. You can specify the text origin as the lower left corner of the first character, the center of the full text, or the lower right corner of the last character of the first line. You can justify text on the left, right, or center of the screen. Changing this modal does not affect existing text.

TEXT JUSTIFICATION

- 1.LOWER LEFT
- 2.CENTER
- 3.LOWER RIGHT

Enter:

- 1 To use the lower left corner of the first character as the text origin and to left-justify the text.
- 2 To use the center of the full text as the text origin and to center-justify the text.
- 3 To use the lower right corner of the last character of the first text line as the text origin and to right-justify the text.

16.1.3 Text Origin Method

With this modal, you can preset the method of indicating the position of text. The dimension, note, and label functions automatically use the method selected in this modal to request this information. (Refer to 16.1.2 TEXT JUSTIFICATION for specifying the text origin in relation to the text.)

TEXT ORIGIN METHOD

1. SCREEN POSITION
2. ENTER COORDINATES
3. DELTA
4. AUTOMATIC

Enter:

- 1 To use the graphics cursor to indicate a screen position. The following prompt then appears when the text origin is required:

INDICATE TEXT ORIGIN

Use the graphics cursor to indicate the screen position.

- 2 To enter the coordinates of a position. The following prompt then appears when the text origin is required:

TEXT COORDINATES

1. ORIGIN XT = n.nnnn
2. YT = n.nnnn

Enter the coordinates of the position.

- 3 To enter horizontal and vertical displacements from an existing point, note, label, or dimension. The following prompt then appears when the text origin is required:

INDICATE ENTITY

Select the entity from which to measure the displacements to the origin of the dimension text.

TEXT DISPLACEMENTS

1. DELTA XT = n.nnnn
2. YT = n.nnnn

Enter the displacements in the x and y directions.

- 4 To automatically center horizontal, vertical, parallel, and thickness dimensions or to join the leader line extension of a circular dimension at the center of an arc. For diameter and angular dimensions and for the label and note functions, this selection has the same effect as choosing 1.SCREEN POSITION.

When you select any dimension or label functions, the system requests that you indicate the text origin.

16.1.4 Dimension Text

With these modals, you can change the way dimension text is generated. The menu varies, according to choices you made when you signed on to the system. Menu choice 3 varies, depending on whether you chose 1982 or 1973 ANSI standards. Menu choice 7 appears only when you set the Units of Measure to 3.ENGLISH (FOOT/INCH).

The menu for this section is as follows:

```
DIMENSION TEXT
1.AUTOMATIC DIMENSION TEXT
2.DECIMAL/FRACTIONS
3.DIMENSION TEXT PREFIX1
4.DIMENSIONING UNITS
5.AUTOMATIC TOLERANCE/LIMITS
6.DIMENSION TEXT ORIENTATION
7.FOOT SYMBOL ON VALUE
```

16.1.4.1 Automatic Dimension Text

With this modal, you can automatically calculate the text length or angle, then insert dimension text specifying that length or angle in the drafting standard format you are currently using.

AUTOMATIC DIMENSION TEXT
 1.ON
 2.OFF

Enter:

- 1 To have the system generate dimension text automatically.
- 2 To have the system prompt you to enter dimension text manually.

16.1.4.2 Decimal/Fractions

With this modal, you can set the number of decimal places in the dimension display. This modal does not affect existing dimensions. The menu for this section is:

DECIMAL/FRACTIONS
 1.DECIMAL
 2.FRACTIONS

These choices and their resulting prompts are described in the following sections.

1. When you are using 1973 ANSI standards, menu choice 3 is DIMENSION TEXT SUFFIX.

16.1.4.2.1 Decimal

With this modal, you can change the number of decimal places used for automatically generated dimension text. Dimensions are rounded according to the number of decimal places you specify. This modal takes the system out of fraction mode (if set with 16.1.4.2.2 FRACTIONS) and does not affect existing dimensions.

DECIMAL PLACES = n

Enter the number of decimal places to be displayed. Allowable entries are 0 through 9.

If 16.1.4.4 DIMENSIONING UNITS is set to dual dimensioning, the system requests the number of decimal places to use in the alternate units text.

SECONDARY DECIMAL PLACES

DECIMAL PLACES = n

Enter the number of decimal places to be used for the alternate units text. Allowable entries are 0 through 7.

16.1.4.2.2 Fractions

With this modal, you can use fractions for automatically generated dimension text.

If Units of Measure is set to 2.ENGLISH (INCH), the system writes dimensions as common or mixed fractions displayed with an accuracy of 1/64. There are no prompts or actions.

If Units of Measure is set to 3.ENGLISH (FOOT/INCH), the system asks how you want to round the dimensions:

FEET/INCHES ROUNDING
 1.1/64 INCH
 2.1/32 INCH
 3.1/16 INCH
 4.1/8 INCH
 5.1/4 INCH
 6.1/2 INCH
 7.1 INCH
 8.1 FOOT

Enter:

- 1 To display automatic dimensions to the nearest 1/64 inch.
- 2 To display automatic dimensions to the nearest 1/32 inch.
- 3 To display automatic dimensions to the nearest 1/16 inch.
- 4 To display automatic dimensions to the nearest 1/8 inch.
- 5 To display automatic dimensions to the nearest 1/4 inch.
- 6 To display automatic dimensions to the nearest 1/2 inch.
- 7 To display automatic dimensions to the nearest inch.
- 8 To display automatic dimensions to the nearest foot.

When you sign on, this modal is set to 1/16 inch. When you change the modal, existing dimensions are affected.

16.1.4.3 Dimension Text Prefix

16.1.4.3 Dimension Text Suffix

With this modal, you can control the addition of certain symbols to automatically generated dimension text. You can add the prefix S to automatically generated spherical radius and diameter dimension text. You can indicate repetitive features and dimensions by a prefix consisting of the number of features or dimensions, followed by a times (X) symbol. You can add the diameter or radius symbol to linear dimensions. The placement of the text depends on whether you are using 1982 (prefix) or 1973 (suffix) ANSI standards. Here is the menu for this section:

DIMENSION TEXT PREFIX

- 1.AUTOMATIC LINEAR DIMENSION SYMBOL
- 2.SPHERICAL DIAMETER/RADIUS SYMBOL
- 3.REPETITIVE FEATURES

or

DIMENSION TEXT SUFFIX

- 1.AUTOMATIC LINEAR DIMENSION SYMBOL
- 2.SPHERICAL DIAMETER/RADIUS SYMBOL
- 3.REPETITIVE FEATURES

16.1.4.3.1 Automatic Linear Dimension Symbol

This modal allows you to control the automatic insertion of a diameter or radius symbol within automatically generated text. The automatic diameter or radius symbol applies to horizontal, vertical, parallel, and thickness dimensioning. Note that automatically generated diameter dimensions always have diameter symbols and automatically generated radius dimensions always have radius symbols. This modal only affects dimensions generated after you set it.

AUTOMATIC LINEAR DIMENSION SYMBOL

- 1.NONE
- 2.DIAMETER SYMBOL
- 3.RADIUS SYMBOL

Enter:

- 1 To generate text without diameter or radius symbols. This is the system default value.
- 2 To include diameter symbols with automatically generated linear dimension text.
- 3 To include radius symbols with automatically generated linear dimension text.

When you set this modal using 1973 ANSI standards, the letters DIA or R are appended to the dimension text. When you use 1982 ANSI standards, the dimension text is preceded by a diameter symbol (\emptyset) or R. The default setting for this modal is NONE.

16.1.4.3.2 Spherical Diameter/Radius Symbol

This modal allows you to control the automatic insertion of spherical diameter or radius symbols within automatically generated diameter or radius dimension text. It only affects dimensions generated after being set.

SPHERICAL DIAMETER/RADIUS SYMBOL

- 1.ON
- 2.OFF

Enter:

- 1 To include spherical diameter or radius symbols with automatically generated dimension text.
- 2 To generate text without spherical diameter or radius symbols. This is the system default value.

When you use 1973 ANSI standards, the words SPHER DIA or SPHER R are appended to the dimension text. When you use 1982 ANSI standards, SØ or SR precedes the dimension text. The default setting for this modal is off (normal diameters and radii).

16.1.4.3.3 Repetitive Features

This modal allows you to control the automatic insertion of nX within automatically generated dimension text, where n is a positive integer representing the number of features identical to the one being dimensioned. Figure 1-2 shows an example of repetitive features. This modal affects dimensions generated after you set it and is available only with 1982 ANSI standards.

REPETITIVE FEATURES

- 1.ON
- 2.OFF

Enter:

- 1 To add repetitive feature text to automatically generated dimension text.
- 2 To generate dimensions without repetitive feature text. This is the system default value.

When this modal is on, the following prompt is displayed immediately before each dimension is created:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

Use 16.13.6.3 REPLACE STRING to add repetitive feature text to existing 1973 ANSI dimension text. The default setting for this modal is off.

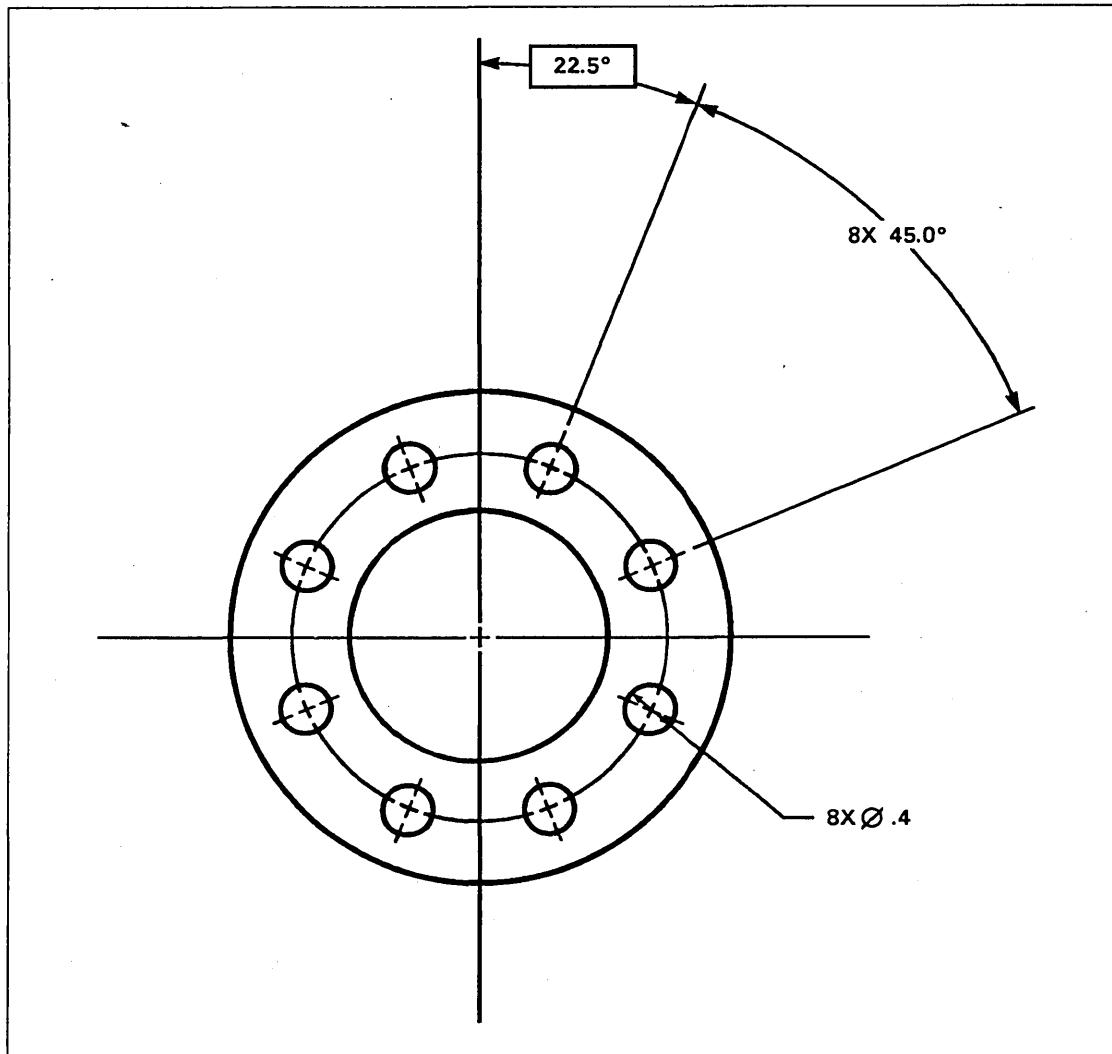


Figure 1-2. Repetitive Features

16.1.4.4 Dimensioning Units

With this modal, you can determine whether the dimension functions are in both SI and U.S. customary units or only in units in effect for the part. Alternate units text is displayed below standard units text, according to the dual dimensioning method.

DIMENSIONING UNITS

1. STANDARD
2. DUAL

Enter:

- 1 To produce dimensions containing only the units in effect for the part.
- 2 To produce dimensions in both SI and U.S. customary units. For this choice you may choose a dual dimensioning method from the following prompt:

DUAL DIMENSIONING METHOD

- 1.BRACKET AND POSITION
- 2.BRACKET
- 3.POSITION

Enter:

- 1 To produce dual dimensions with the alternate dimensions inside square brackets and a line separating upper and lower text. (See figure 1-3.)
- 2 To produce dual dimensions with the alternate dimensions inside square brackets and no line separating upper and lower text. (See figure 1-3.)
- 3 To produce dual dimensions with a line separating upper and lower text and no brackets around the alternate dimensions. (See figure 1-3.)

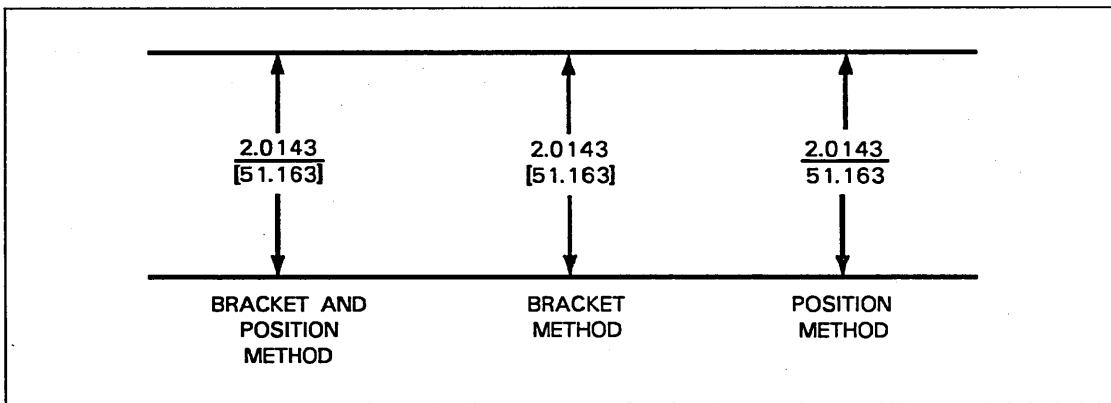


Figure 1-3. Dual Dimensioning Methods

16.1.4.5 Automatic Tolerance/Limits

With this modal, you can control the addition of the tolerance or limits to automatically generated dimension text. You are either prompted for the tolerance or specify it when you create the dimension. The Automatic Tolerance/Limits modal only affects dimensions generated after you set it.

These are the menu choices:

```
AUTOMATIC TOLERANCE/LIMITS
1.OFF
2.ENTER PRESET TOLERANCE
3.SET WHEN USED
```

Enter:

- 1 To not add tolerance or limits to automatically generated dimension text. The system displays the following message:

```
AUTOMATIC TOLERANCING - OFF
```

- 2 To enter the tolerance you want to use.

TOLERANCING METHOD	Enter:
1.TOLERANCE	1 To automatically add the tolerance to the dimension.
2.LIMITS	2 To automatically create limits from the dimension.
3.SET WHEN USED	3 To have the system prompt you for the tolerancing method when you create the dimension.

According to your selection, the system confirms your choice with one of the following messages:

```
TOLERANCING METHOD: TOLERANCE
LIMITS
SET WHEN USED
```

Next, the system prompts for the desired tolerance:

```
TOLERANCE
1.UPPER = .nnn
2.LOWER = -.nnn
```

Enter the values for the upper and lower tolerance to be automatically added to dimensions at creation. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

After setting this modal, the system adds the tolerance or limits you specified to the first display of dimensions you create.

- 3 To enter the tolerance when you create the dimension.

TOLERANCING METHOD

- 1. TOLERANCE
- 2. LIMITS
- 3. SET WHEN USED

Enter:

- 1 To automatically add the tolerance to the dimension.
- 2 To automatically create limits from the dimension.
- 3 To have the system prompt you for the tolerancing method when you create the dimension.

According to your selection, the system confirms your choice with one of the following messages:

TOLERANCING METHOD: TOLERANCE
LIMITS
SET WHEN USED

When you create dimensions after setting this modal, the system prompts you for the tolerance or limits you want to specify. If you do not want to have a tolerance added to a dimension, enter zero for both the upper and lower values.

16.1.4.6 Dimension Text Orientation

These modals allow you to control the angle and placement of dimension text. The angle is either horizontal to the work plane (ANSI standard) or parallel to the dimension line. The placement is above or below a solid line, or in the middle of a broken line.

The menu for this modal is:

DIMENSION TEXT ORIENTATION
1.DIMENSION TEXT ANGLE
2.PLACEMENT

The following paragraphs describe these choices and their resulting prompts and actions.

16.1.4.6.1 Dimension Text Angle

With this modal, you can control the angle of the dimension text with respect to the dimension line. Horizontal means horizontal on the current workplane. This corresponds to ANSI standard format and is the system default. You can choose to have only linear dimensions parallel to the dimension line. This leaves text on angular, radius, diameter, thickness, taper, chamfer, arc length, and rectangular coordinate dimensions in the horizontal orientation. Existing dimensions are not affected.

DIMENSION TEXT ANGLE

- 1.HORIZONTAL
- 2.PARALLEL-LINEAR DIMENSIONS ONLY

Enter:

- 1 To have all text horizontal in the workplane. This is the system default.
- 2 To draw text parallel to the dimension line for horizontal, vertical, and parallel dimensions only.

16.1.4.6.2 Dimension Text Placement

With this modal, you can control the placement of dimension text in relation to the dimension line. If the text is placed above or below the dimension line, the line is solid, otherwise it is broken. The system default is middle (ANSI standard). This modal does not affect existing dimensions.

DIMENSION TEXT PLACEMENT

- 1.MIDDLE
- 2.ABOVE
- 3.BELOW

Enter:

- 1 To write text in the middle of a broken dimension line. This is the system default.
- 2 To write text above a solid dimension line.
- 3 To write text below a solid dimension line.

16.1.4.7 Foot Symbol on Value

This modal controls display of the foot symbol. Values greater than or equal to the setting are displayed with the foot symbol. Values less than the setting are displayed in inches. This symbol is available only when the units of measure are feet/inches.

1.VAL >...FT = n.nnnn

Enter the number (in feet) for which the system will display equal or greater values with the foot symbol.

For example, if this modal is set to 2 ft., 15 inches is displayed as 15". If this modal is set to 0 ft., 15 inches is displayed as 1'3".

When you enter ICEM DDN, this modal is set to 0 ft. When you change the modal, existing dimensions are not affected.

16.1.5 Text Angle Control

With this modal, you can control the angle at which the text of the note is written. This modal affects the operation of 16.10 NOTE and 16.11 LABEL.

TEXT ANGLE CONTROL

- 1.HORIZONTAL
- 2.ENTER ANGLE
- 3.PARALLEL TO LINE/ARC
- 4.TOTAL ANGLE CONTROL

Enter:

- 1 To select no text angle control (to write all text horizontally).
- 2 To write notes or labels at the angle you specify.
- 3 To have the option of writing a note parallel to a line or arc.
- 4 To write notes at the angle you specify or parallel to a line or arc.

16.1.6 Drafting Scale Factor

With this modal, you can set the scale factor used in generating dimensions, labels, and notes. The following factors are affected:

- Character size
- Arrowhead size of all dimensions and labels
- Distance from the text of a dimension to the dimension line
- Distance the extension line is offset from a reference point, line, and so on.
- Distance the extension line extends past the dimension line
- Spacing and dash size for centerlines
- Preset balloon and datum target symbol radii

This operation affects all newly created dimensions.

DRAFTING SCALE OR RATIO

1.SCALE = n.nnnn
2.RATIO = n.nnnn

Enter the drafting scale the system will use to produce the drawing. Upon entry to the system, this value is preset to 1. When the drafting scale is halved, the other affected drafting modals are doubled. When the drafting scale is doubled, the other affected drafting modals are halved, and so on.

or

Enter a ratio to the current drafting scale, which proportionately affects the drafting scale factor.

16.1.7 Arrowhead Length

Use this modal to set the length of arrowheads and dimension origin circles. Arrowhead width is one-third the length. Only the dimensions and arrowheads you create after changing this modal are affected.

ARROW LENGTH = n.nnnn

Enter the arrowhead length. This value is preset to 3.81 mm (0.15 in).

16.1.8 Extension/Dimension Line Control

With this modal, you can control the generation of extension and dimension lines. An extension line extends from an entity and is the line to which a dimension is measured. A dimension line is an arrow from the text to the extension line indicating the dimension being measured. The tip of the dimension arrow touches the extension line. You can suppress either or both of the extension lines associated with a dimension. (This means they will not be drawn on the screen.) You can suppress dimension arrows individually, with just the associated extension line also suppressed, or with both extension lines suppressed.

EXTENSION/DIMENSION LINE CONTROL

- 1.NO SUPPRESSION
- 2.FIRST EXTENSION
- 3.SECOND EXTENSION
- 4.BOTH EXTENSION
- 5.FIRST DIMENSION AND EXTENSION
- 6.SECOND DIMENSION AND EXTENSION
- 7.BOTH EXTENSION AND FIRST
DIMENSION
- 8.BOTH EXTENSION AND SECOND
DIMENSION

Enter:

- 1 To generate dimension and extension lines to both entities.
- 2 To suppress the extension line that extends from the first entity selected.
- 3 To suppress the extension line that extends from the second entity selected.
- 4 To suppress both extension lines.
- 5 To suppress the dimension arrow and associated extension line for the first entity selected.
- 6 To suppress the dimension arrow and associated extension line for the second entity selected.
- 7 To suppress both extension lines and the dimension arrow for the first entity selected.
- 8 To suppress both extension lines and the dimension arrow for the second entity selected.

16.1.9 Arrow/Leader Control

With this set of modals, you can control the placement of arrows in a dimension or the placement of label leaders.

The menu for this section is:

ARROW/LEADER CONTROL

- 1.ARROW PLACEMENT
- 2.LABEL LEADER PLACEMENT

The following pages describe these choices and their resulting prompts.

16.1.9.1 Arrow Placement

With this modal, you can determine the placement of the arrows generated by the dimension functions in 16.2 DIMENSION. Changing this modal does not affect existing dimensions.

ARROW PLACEMENT

1. AUTOMATIC
2. ARROWS INSIDE
3. ARROWS OUTSIDE

Enter:

- 1 To allow the system to automatically determine the placement of arrows according to the placement of text and the available space.
- 2 To generate dimensions with the arrows inside (between) the entities or with extension lines to which the dimensions extend.
- 3 To generate dimensions with the arrows outside the entities or with extension lines to which the dimensions extend.

Figures 1-4 and 1-5 are examples of two types of dimensions.

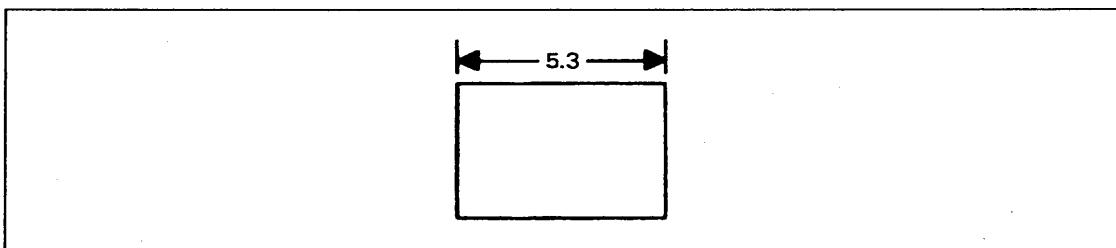


Figure 1-4. Automatic Placement and Placement Inside Entities (1.AUTO or 2.ARROWS IN)

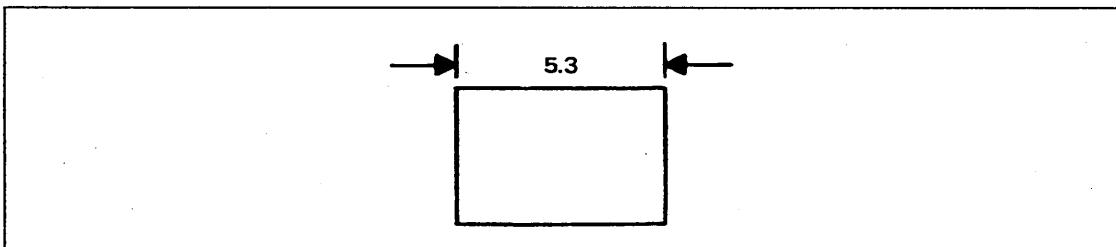


Figure 1-5. Placement Outside Entities (3.ARROWS OUT)

16.1.9.2 Label Leader Placement

With this modal, you can determine the placement of the label leader with respect to the label text (refer to 16.11 LABEL). Changing this modal does not affect existing labels.

LABEL LEADER PLACEMENT

- 1.LEADER TO FIRST TEXT LINE
- 2.LEADER TO MIDDLE TEXT LINE

Enter:

- 1 To start the label leader line from the first text line (refer to 16.11 LABEL).
- 2 To start the label leader line from the middle text line (refer to 16.11 LABEL).

16.1.10 Arrow Alignment

With this modal, you can create vertical or horizontal dimensions aligned with previously created dimensions of the same type (figure 1-6). By default, the arrow alignment is off.

ARROW ALIGNMENT

- 1.ON
- 2.OFF

Enter:

- 1 To have the system request an alignment dimension when defining horizontal or vertical dimensions and then to align the dimension appropriately.
- 2 To avoid the system request for an alignment dimension.

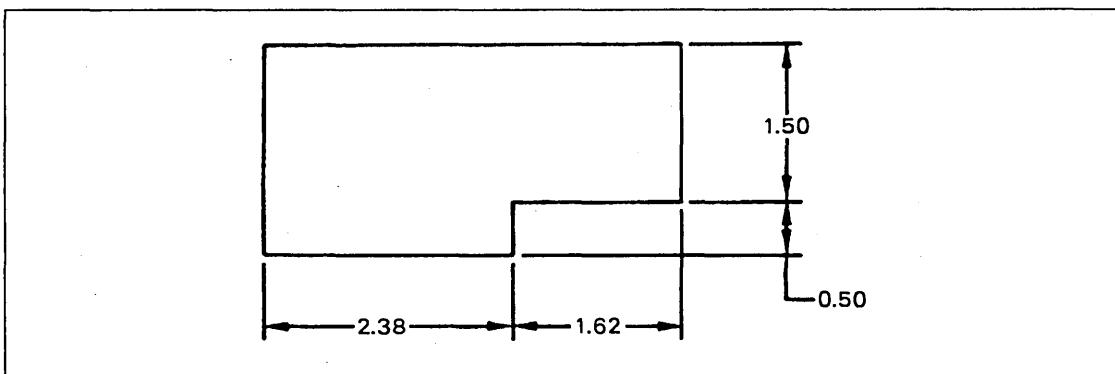


Figure 1-6. Aligned Arrowheads

16.1.11 Dimension Offset Distances

With this modal, you can set the following distances in the generation of dimensions (see figure 1-7):

- The distance from the text to the dimension lines.
- The distance from an extension line to the entity from which it extends.
- The distance that an extension line extends beyond a dimension line.

When you select this modal, the system displays:

DIMENSION OFFSET DISTANCES

1.TEXT-DIMENSION = n.nnnn
 2.EXTENSION-POINT = n.nnnn
 3.EXTENSION-DIMENSION = n.nnnn

Enter the distance from the text to the dimension line. When you enter the system, this value is preset to 2.54 mm (0.10 in).

Enter the distance from the extension line to the entity. This value is preset to 1.59 mm (0.0625 in).

Enter the distance that the extension line extends beyond the dimension line. This value is preset to 3.18 mm (0.125 in).

Altering the dimension offset distances only affects subsequently generated dimensions.

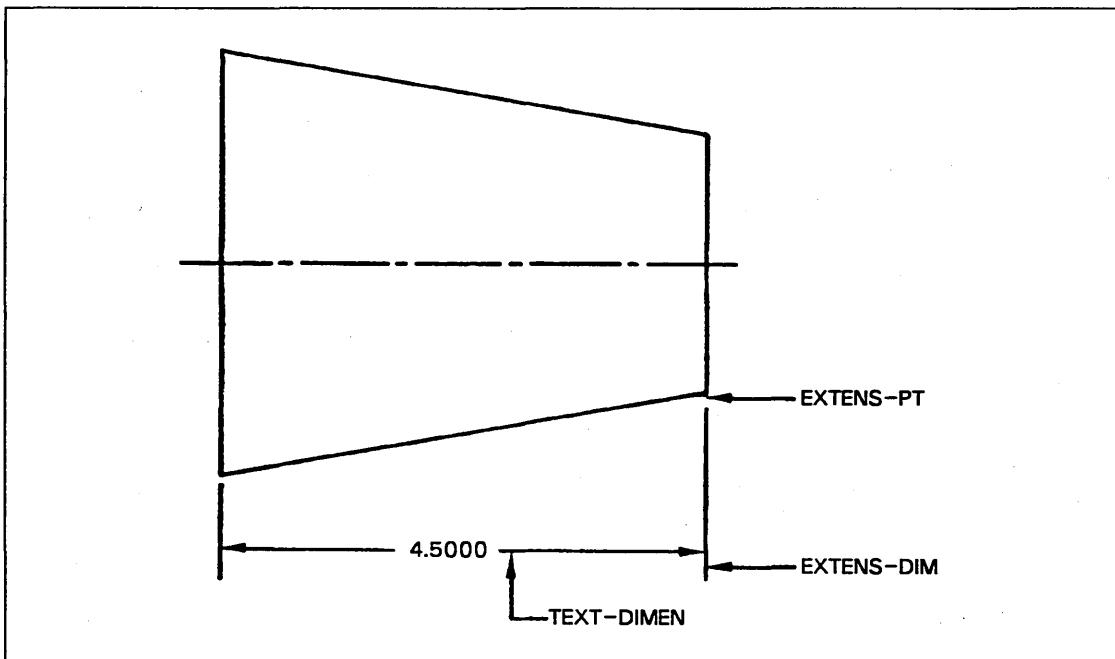


Figure 1-7. Dimension Offset Distances

16.1.12 Dimension Origin

This modal allows you to control which side (if any) of a dimension will specify a dimension origin. A dimension origin is graphically represented as a circle (instead of an arrowhead) at the end of a dimension line. The diameter of the dimension origin circle is determined by the arrowhead length (16.1.7). This modal applies to horizontal, vertical, parallel, and thickness dimensions. The system default dimension origin is 3.NONE.

DIMENSION ORIGIN

- 1.FIRST SIDE
- 2.SECOND SIDE
- 3.NONE

Enter:

- 1 To specify the first side selected as the dimension origin side.
- 2 To specify the second side selected as the dimension origin side.
- 3 To specify that neither side of a dimension will be the dimension origin. This is the default dimension origin.

16.1.13 Automatic Tail Location

With this modal, you can control the entry of the tail location while creating or modifying the origin of drafting entities with tails.

AUTOMATIC TAIL LOCATION

- 1.ON
- 2.OFF

Enter:

- 1 To automatically place the tail and eliminate prompts for tail location. With this modal on, the system chooses the most efficient tail location for each new drafting entity with a tail.
- 2 To request a tail location when needed.

NOTE

The parallel and thickness dimension tails are automatically defined when the text is not completely outside the extension lines.

16.1.14 Isometric Drafting

With this modal, you can choose whether or not to use isometric drafting. Isometric drafting allows you to create drafting entities using the independent workspace feature of ICEM Design/Drafting. This permits you to create drafting entities in a work plane that is not aligned with the view plane. Thus, you can create drafting entities while in a three-dimensional, isometric view of the model. Isometric drafting entities can be displayed in either their view of definition or in all views.

ISOMETRIC DRAFTING

- 1.ON--DISPLAY IN VIEW OF DEF ONLY
- 2.ON--DISPLAY IN ALL VIEWS
- 3.OFF

Enter:

- 1 To activate isometric drafting and restrict the display of entities to their view of definition only. Selective view blanking is turned on. The drafting entities generated are automatically selectively view blanked in all but the current work view.
- 2 To activate isometric drafting and display entities in all views. Selective view blanking is turned off.
- 3 To deactivate isometric drafting. This is the system default. Selective view blanking is turned off.

16.1.15 Drafting Standards

With this modal, you can change the ANSI drafting standards you are currently using to define new entities from 1982 standards to 1973 standards, or vice versa. This modal is first set when you enter a new part and drafting entities already defined do not change when you alter the modal.

If you are currently using 1973 ANSI drafting standards, the system displays:

SWITCH TO 1982 ANSI DRAFTING STANDARD?

Enter:

- Y To switch to 1982 ANSI drafting standards.
- N To continue using 1973 ANSI drafting standards.

If you are currently using 1982 ANSI drafting standards, the system displays:

SWITCH TO 1973 ANSI DRAFTING STANDARD?

Enter:

- Y To switch to 1973 ANSI drafting standards.
- N To continue using 1982 ANSI drafting standards.

16.1.16 Display Drafting Modals

Use this modal to display the current settings of all the drafting modals, as shown in figure 1-8. Enter] to view the second set of modals and] again to view the third set.

There are no system prompts or actions for this modal. After viewing all the modals, enter] or [to return to 16.1 DRAFTING MODALS.

CURRENT DRAFTING MODALS	
1.CHARACTER SIZE:	n.nnnn
2.SPACING RATIO:	n.nnnn
3.ASPECT RATIO:	n.nnnn
4.DOWNSPACE RATIO:	n.nnnn
5.TOLERANCE RATIO:	n.nnnn
6.CHARACTER SET:	STANDARD
7.CHARACTER SLANT:	OFF
8.CHARACTER OUTPUT:	FINE
9.SPECIAL SET PREFIX:	\
10.TEXT JUSTIFICATION:	LEFT
11.TEXT ORIGIN METHOD:	SCREEN POS
12.DECIMAL PLACES:	n
13.SECONDARY DECIMAL PLACES:	n
14.FRACTIONS:	OFF
15.AUTOMATIC DIMENSION TEXT:	ON
16.LINEAR DIMENSION SYMBOL:	NONE
17.DIMENSIONING UNITS:	STANDARD
18.DUAL DIMENSIONING METHOD:	BRCKT+POS
CURRENT DRAFTING MODALS	
1.TEXT ANGLE:	HORIZONTAL
2.DRAFTING SCALE:	n.nnnn
3.ARROWHEAD LENGTH:	n.nnnn
4.TEXT-DIMENSION:	n.nnnn
5.EXTENSION-POINT:	n.nnnn
6.EXTENSION-DIMENSION:	n.nnnn
7.EXTENSION LINE:	NO SUPPRESSION
8.DIMENSION LINE:	NO SUPPRESSION
9.ARROW PLACEMENT:	AUTOMATIC
10.LEADER PLACEMENT:	FIRST
11.ARROW ALIGNMENT:	OFF
12.AUTO TAIL LOCATION:	ON
13.DIMENSION ORIGIN:	NONE
14.DRAFTING STANDARD:	ANSI 1982
15.FEET/INCHES ROUNDING:	1/64 INCH
16.FOOT SYMBOL ON VALUE:	> n FT

Figure 1-8. Drafting Modals Display

(Continued)

(Continued)

CURRENT DRAFTING MODALS	
1. SPHERICAL DIMENSIONS:	OFF
2. REPETITIVE DIM TEXT:	OFF
3. AUTO TOL/LIM(METHOD):	NO PRESET(SET)
4. ISOMETRIC DRAFTING:	OFF
5. DIM TEXT ANGLE:	HORIZONTAL
6. DIM TEXT PLACEMENT:	MIDDLE

Figure 1-8. Drafting Modals Display

16.2 Dimension

With this section, you can generate dimensions. The menu for this option is:

DIMENSION

1. HORIZONTAL
2. VERTICAL
3. PARALLEL
4. THICKNESS
5. ANGULAR
6. RADIUS
7. DIAMETER
8. TAPER
9. CHAMFER
10. ARC LENGTH
11. RECTANGULAR COORDINATE

The following pages describe these choices and their resulting prompts.

16.2.1 Horizontal

With this choice, you can produce a horizontal dimension between two entities. The dimension includes extension lines, arrowheads, horizontal dimension lines, and text. You enter the text, or the system automatically calculates and inserts it, depending on the setting of 16.1.4.1 AUTOMATIC DIMENSION TEXT.

INDICATE ENTITY

Use the graphics cursor to select the first entity. Place the graphics cursor near the end from which the extension line is to be drawn. Allowable entities include points, lines, arcs or circles, conics, splines (two- and three-dimensional), composite curves, point sets, machine curves, and centerlines.

INDICATE ENTITY

Use the graphics cursor to select the second entity.

If 16.1.10 ARROW ALIGNMENT IS OFF, the dimension is created if 16.1.10 ARROW ALIGNMENT is on, the system displays:

INDICATE DIMENSION

Use the graphics cursor to select the existing horizontal dimension with which the new dimension is to be aligned. Enter] to use the previously created or modified horizontal dimension as the alignment dimension. Enter [to create the new dimension without alignment.

If you indicate the wrong type of dimension, the system displays:

THE ALIGNMENT DIMENSION MUST HAVE SIMILAR ORIENTATION.

Enter] or [to continue.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3.SET WHEN USED, the system displays the following prompt.

TOLERANCING METHOD

- 1.TOLERANCE
- 2.LIMITS

Enter:

- 1 To automatically add tolerance to the dimension.
- 2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt:

TOLERANCE

- 1.UPPER = .nnn
- 2.LOWER = -.nnn

Enter the values for the upper and lower tolerance to be automatically added to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

16.2.1 Horizontal

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features.

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

INDICATE TEXT ORIGIN

Indicate the location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the distance between the two entities selected and writes that distance in the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you enter the text for the dimension when you receive the following prompt.

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Either enter] or RETURN twice to end text entry. The system allows the entry of at least 100 uppercase or lowercase characters, minus the number of text lines.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off and 16.1.4.4 DIMENSIONING UNITS is set to dual dimensioning, the system requires secondary text.

ENTER SECONDARY TEXT

Enter the text to be written below the standard text. Enter] for the system to convert the standard text.

If this is not possible, the system redisplays the prompt.

16.2.2 Vertical

With this choice, you can produce a vertical dimension between two entities. The dimension includes extension lines, arrowheads, vertical dimension lines, and text. You enter the text, or the system automatically calculates and inserts it, depending on the setting of 16.1.4.1 AUTOMATIC DIMENSION TEXT.

INDICATE ENTITY

Use the graphics cursor to select the first entity. Place the graphics cursor near the end from which the extension line is to be drawn. Allowable entities include points, lines, arcs, circles, conics, splines (two- or three-dimensional), composite curves, point-sets, machine curves, and centerlines.

INDICATE ENTITY

Use the graphics cursor to select the second entity.

If 16.1.10 ARROW ALIGNMENT is off, the dimension is created. If 16.1.10 ARROW ALIGNMENT is on, the system displays:

INDICATE DIMENSION

Use the graphics cursor to select the existing vertical dimension with which the new dimension is to be aligned. Enter] to use the previously created or modified vertical dimension as the alignment dimension. Enter [to create the new dimension without alignment.

If you indicate the wrong type of dimension, the system displays:

THE ALIGNMENT DIMENSION MUST HAVE SIMILAR ORIENTATION.

Enter] or [to continue.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3.SET WHEN USED, the system displays the following prompt:

TOLERANCING METHOD

- 1.TOLERANCE
- 2.LIMITS

Enter:

- 1 To automatically add tolerance to the dimension.
- 2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt.

TOLERANCE

- 1.UPPER = .nnn
- 2.LOWER = -.nnn

Enter the values for the upper and lower tolerance to be automatically added to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

16.2.2 Vertical

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features.

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

INDICATE TEXT ORIGIN

Indicate the location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the distance between the two entities selected and writes that distance in the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you enter the text for the dimension.

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Either enter] or RETURN twice to terminate entry of text. The system allows the entry of at least 100 uppercase or lowercase characters, minus the number of text lines.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off and 16.1.4.4 DIMENSIONING UNITS is set to dual dimensioning, the system requires secondary text.

ENTER SECONDARY TEXT

Enter the text to be written below the standard text.

Enter] for the system to convert the standard text. If this is not possible, the system redisplays the prompt.

If 16.1.13 AUTOMATIC TAIL LOCATION is off and the dimension has a tail on it, the system displays:

TAIL LOCATION
1.START OF TEXT
2.END OF TEXT
3.AUTOMATIC

Enter:

- 1 To place the tail to the left of the text.
- 2 To place the tail to the right of the text.
- 3 To place the tail automatically.

16.2.3 Parallel

With this choice, you can produce a parallel dimension between two entities. The dimension includes extension lines, arrowheads, dimension lines, and text. If the entities selected are two parallel lines, the dimension lines produced are perpendicular to the two parallel lines and the distance between the lines is measured perpendicularly. You enter the text, or the system automatically calculates and inserts it, depending on the setting of 16.1.4.1 AUTOMATIC DIMENSION TEXT.

NORMAL TO PARALLEL LINES?

Enter:

- Y To indicate that the measurement is the normal distance from one line to the end of another entity. Extension lines are parallel to the first line chosen, and the distance measured is perpendicular to the first line.
- N To indicate that the measurement is the two-dimensional distance between the ends of two entities.

If you select Y, the system displays:

INDICATE ENTITY

Use the graphics cursor to select a line or center line from which a perpendicular dimension is to be made. Place the graphics cursor near the end from which the extension line is to be drawn.

INDICATE ENTITY

Use the graphics cursor to select the end of the entity to which the dimension is to be drawn. Allowable entities include points, lines, arcs or circles, conics, splines (two- or three-dimensional), composite curves, point sets, machine curves, and centerlines.

If you select N, the system displays:

INDICATE ENTITY

Use the graphics cursor to select the first entity. Place the graphics cursor near the end to be used for the dimension. Allowable entities include points, lines, arcs or circles, conics, splines (two- or three-dimensional), composite curves, point sets, machine curves, and centerlines.

INDICATE ENTITY

Use the graphics cursor to select the second entity.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3.SET WHEN USED, the system displays the following prompt:

TOLERANCING METHOD

1.TOLERANCE
2.LIMITS

Enter

- 1 To automatically add tolerance to the dimension.
- 2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt:

TOLERANCE

1.UPPER = .nnn
2.LOWER = -.nnn

Enter the values for the upper and lower tolerance to be automatically added to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

INDICATE TEXT ORIGIN

Indicate the location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the distance between the two entities selected and writes that distance in the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you enter the text for the dimension.

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Either enter] or RETURN twice to terminate entry of text. The system allows the entry of at least 100 uppercase or lowercase characters, minus the number of text lines.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off and 16.1.4.4 DIMENSIONING UNITS is set to dual dimensioning, the system requires secondary text.

ENTER SECONDARY TEXT

Enter the text to be written below the standard text. Enter] for the system to convert the standard text. If this is not possible, the system redisplays the prompt.

If 16.1.13 AUTOMATIC TAIL LOCATION is off and the dimension has a tail on it, the system displays:

TAIL LOCATION

- 1.START OF TEXT
- 2.END OF TEXT
- 3.AUTOMATIC

Enter:

- 1 To place the tail to the left of the text.
- 2 To place the tail to the right of the text.
- 3 To place the tail automatically.

16.2.4 Thickness

With this choice, you can produce a thickness dimension between two curves. The dimension measures the distance between the two curves from a point on the first curve, along an imaginary line that extends from that position and is normal to the second curve, to the position at which this imaginary line intersects the second curve.

INDICATE 2-D CURVE

Use the graphics cursor to select the first curve. The placement of the graphics cursor along the curve is significant; the measurement is made from the point on the curve nearest this position.

INDICATE 2-D CURVE

Use the graphics cursor to select the other curve.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3. SET WHEN USED, the system displays the following prompt:

TOLERANCING METHOD

Enter:

1.TOLERANCE

1 To automatically add tolerance to the dimension.

2.LIMITS

2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt:

TOLERANCE

Enter the values for the upper and lower tolerance to be automatically added to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

INDICATE TEXT ORIGIN

Indicate the location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD. The arrow placement and text origin are constrained by the first curve pick location. The system automatically alters the given origin to the nearest possible location.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the distance between the two entities selected and writes that distance in the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you must enter the text for the dimension.

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Either enter] or RETURN twice to terminate entry of text. The system allows the entry of at least 100 uppercase or lowercase characters, minus the number of text lines.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off and 16.1.4.4 DIMENSIONING UNITS is set to dual dimensioning, the system requires secondary text.

ENTER SECONDARY TEXT

Enter the text to be written below the standard text. Enter] for the system to convert the standard text. If this is not possible, the system redisplays the prompt.

If 16.1.13 AUTOMATIC TAIL LOCATION is off and the dimension has a tail on it, the system displays:

TAIL LOCATION
1.START OF TEXT
2.END OF TEXT
3.AUTOMATIC

Enter:

- 1 To place the tail to the left of the text.
- 2 To place the tail to the right of the text.
- 3 To place the tail automatically.

The thickness dimension is drawn with the arrow placement determined by modal 16.1.9.1 ARROW PLACEMENT.

16.2.5 Angular

With this choice, you can produce a dimension that gives the angle between two lines.

INDICATE ENTITY

Use the graphics cursor to select the first line or centerline.

INDICATE ENTITY

Use the graphics cursor to select the second line or centerline.

In selecting these two lines, the following conditions affect this operation:

- The order in which you select the lines is significant because the angle is measured from the first line to the second line in a counterclockwise direction.
- The end of the line you select is significant because the angle is measured from the line (extended if necessary) on the side of the vertex corresponding to the end of the line chosen.

Figure 1-9 shows examples of how to control this operation through the selection order and the choice of the end selected.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3. SET WHEN USED, the system displays the following prompt:

TOLERANCING METHOD

1.TOLERANCE

2.LIMITS

Enter:

- 1 To automatically add tolerance to the dimension.
- 2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt:

TOLERANCE

1.UPPER = .nnn

2.LOWER = -.nnn

Enter the values for the upper and lower tolerance to be automatically added to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

INDICATE TEXT ORIGIN

Indicate the location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the angle between the two entities selected and writes that angle in the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you must enter the text for the dimension when you receive the following prompt:

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Enter either] or RETURN twice to terminate entry of text. The system allows the entry of at least 100 uppercase or lowercase characters, minus the number of text lines.

If 16.1.13 AUTOMATIC TAIL LOCATION is off and the dimension has a tail on it, the system displays:

TAIL LOCATION

- 1.START OF TEXT
- 2.END OF TEXT
- 3.AUTOMATIC

Enter:

- 1 To place the tail to the left of the text.
- 2 To place the tail to the right of the text.
- 3 To place the tail automatically.

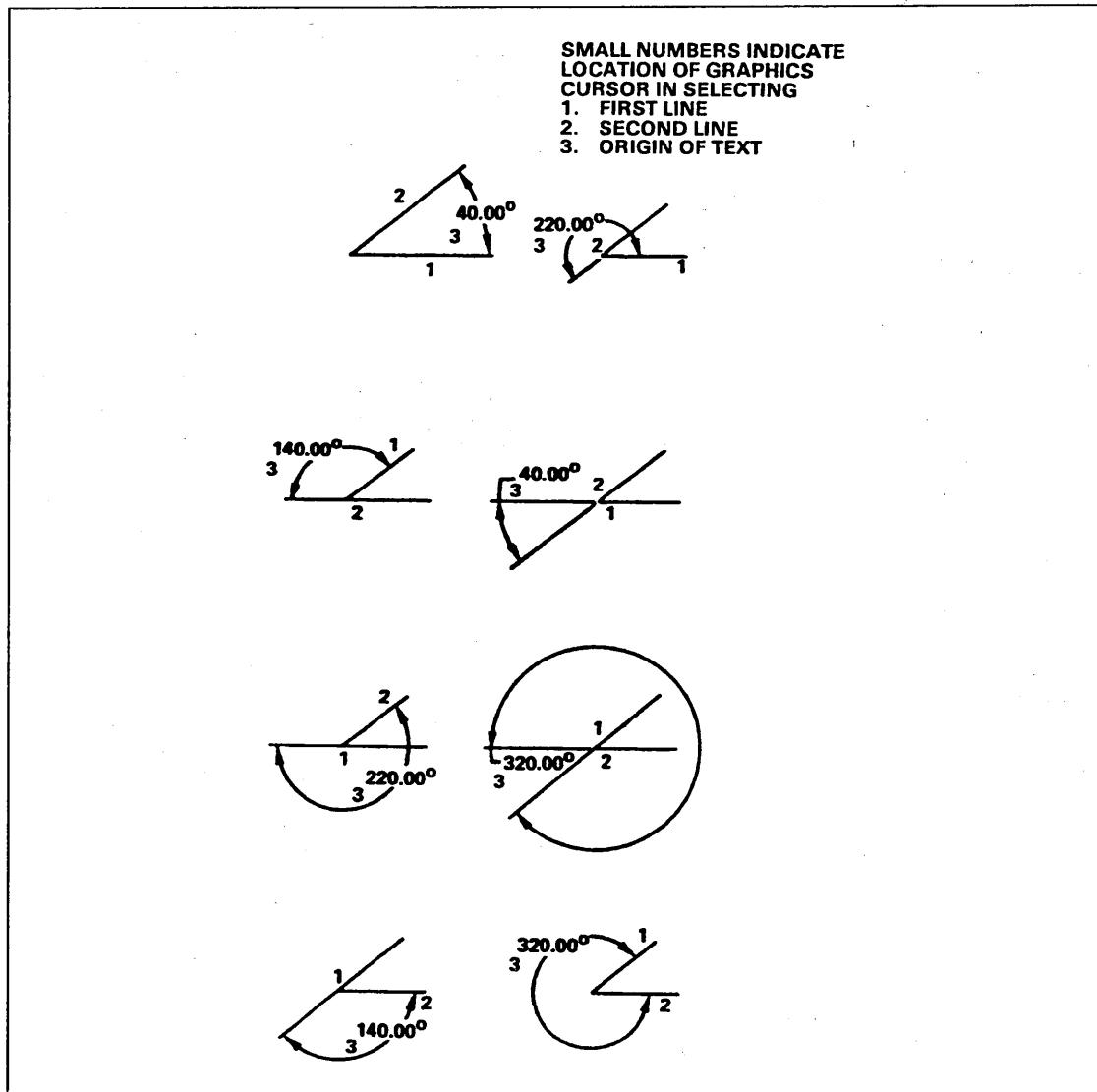


Figure 1-9. Angular Dimension Operation

16.2.6 Radius

With this choice, you can draw the dimension of the radius of an arc or a circle.

INDICATE ARC

Use the graphics cursor to select the arc or circle whose radius you want to dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3. SET WHEN USED, the system displays the following prompt:

TOLERANCING METHOD

- 1. TOLERANCE
- 2. LIMITS

Enter:

- 1 To automatically add tolerance to the dimension.
- 2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt:

TOLERANCE

- 1. UPPER = .nnn
- 2. LOWER = -.nnn

Enter the values for the upper and lower tolerance to be automatically added to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

INDICATE TEXT ORIGIN

Indicate the location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the radius of the arc or circle and writes that radius in the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you must enter the text for the dimension when you receive the following prompt:

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Enter either] or RETURN twice to terminate entry of text. The system allows the entry of at least 100 uppercase or lowercase characters, minus the number of text lines.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off and 16.1.4.4 DIMENSIONING UNITS is set to dual dimensioning, the system requires secondary text.

ENTER SECONDARY TEXT

Enter the text to be written below the standard text. Enter] for the system to convert the standard text. If this is not possible, the system redisplays the prompt.

When you are using 1973 ANSI standards, the dimension is drawn with an R following the text to indicate a radius dimension. When you are using 1982 ANSI standards, an R precedes the radius.

If 16.1.13 AUTOMATIC TAIL LOCATION is off and the dimension has a tail on it, the system displays:

TAIL LOCATION

- 1. START OF TEXT
- 2. END OF TEXT
- 3. AUTOMATIC

Enter:

- 1 To place the tail to the left of the text.
- 2 To place the tail to the right of the text.
- 3 To place the tail automatically.

16.2.7 Diameter

Use this menu choice to draw the dimension of the diameter of a circle.

INDICATE ENTITY

Use the graphics cursor to select the circle or bolt circle centerline whose diameter you want to dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3. SET WHEN USED, the system displays the following prompt:

TOLERANCING METHOD

1.TOLERANCE
2.LIMITS

Enter:

- 1 To automatically add tolerance to the dimension.
- 2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt:

TOLERANCE

1.UPPER = .nnn
2.LOWER = -.nnn

Enter the values for the upper and lower tolerance to be automatically added to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

INDICATE TEXT ORIGIN

Indicate the location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the diameter of the arc or circle and writes that diameter in the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you must enter the text for the dimension when you receive the following prompt:

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Enter either] or RETURN twice to terminate entry of text. The system allows the entry of at least 100 uppercase or lowercase characters, minus the number of text lines.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off and 16.1.4.4 DIMENSIONING UNITS is set to dual dimensioning, the system requires secondary text.

ENTER SECONDARY TEXT

Enter the text to be written below the standard text. Enter] for the system to convert the standard text. If this is not possible, the system redisplays the prompt.

If 16.1.13 AUTOMATIC TAIL LOCATION is off and the dimension has a tail on it, the system displays:

TAIL LOCATION

- 1. START OF TEXT
- 2. END OF TEXT
- 3. AUTOMATIC

Enter:

- 1 To place the tail to the left of the text.
- 2 To place the tail to the right of the text.
- 3 To place the tail automatically.

The system draws the diameter dimension for the circle.

16.2.8 Taper

With this choice, you can draw a slope or taper dimension to two lines.

TAPER

- 1.CONICAL
- 2.FLAT

INDICATE LINE

Enter:

- 1 To generate a conical taper dimension.
- 2 To generate a flat taper dimension.

INDICATE LINE

With the graphics cursor, select a line opposite the side where the taper dimension will be placed.

With the graphics cursor, select the line to which the dimension arrow will point.

Figure 1-10 shows the two taper dimensions.

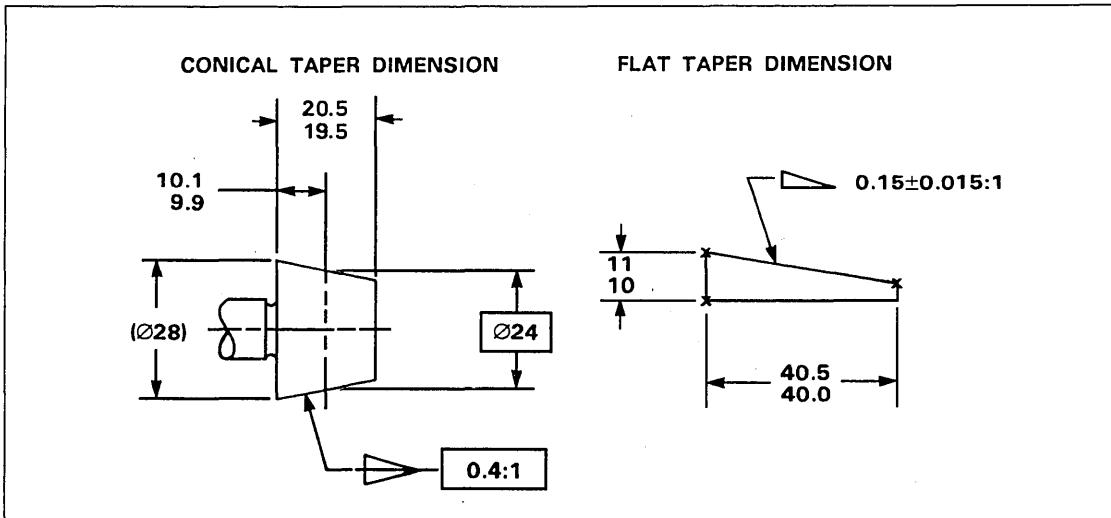


Figure 1-10. Conical and Flat Taper Dimensions

The leader will proceed from the middle of the first line of text to the second selected line. Indicate where on the second line the arrow is to point.

LEADER LINE METHOD

- 1.SELECTED POSITION
- 2.ENTER SLOPE

Enter:

- 1 To draw an arrow that points to the position you selected on the second line.
- 2 To enter the slope of the leader with this prompt:

SLOPE = n.nnnn

Enter the leader angle. The angle is measured in a counterclockwise direction.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3. SET WHEN USED, the system displays the following prompt:

TOLERANCING METHOD

- 1.TOLERANCE
2.LIMITS

Enter:

- 1 To automatically add tolerance to the dimension.
- 2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt:

TOLERANCE

- 1.UPPER = .nnn
2.LOWER = -.nnn

Enter the values for the upper and lower tolerance to be automatically added to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

The system requests the starting location for the text.

INDICATE TEXT ORIGIN

Indicate the starting location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the slope value and generates the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you must enter the text for the dimension when you receive the following prompt:

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end the line. Enter] or RETURN twice to terminate the entry of text. The system allows the entry of at least 100 upper or lower case characters, minus the number of text lines.

If 16.1.13 AUTOMATIC TAIL LOCATION is off, the system displays:

TAIL LOCATION

- 1.START OF TEXT
- 2.END OF TEXT
- 3.AUTOMATIC

Enter:

- 1 To place the tail to the left of the text.
- 2 To place the tail to the right of the text.
- 3 To place the tail automatically.

16.2.9 Chamfer

With this choice, you can draw a 45° chamfer dimension. This is a measurement of the longitudinal direction of a chamfer. Refer to figure 1-11 for an example.

INDICATE CHAMFER

Use the graphics cursor to select the line from which to calculate the chamfer dimension. Select the line at the position where the leader line points.

The leader proceeds from the chamfer you selected.

The vertical position of the leader line tail is determined by 16.1.9.2 LEADER PLACEMENT. Indicate where on the entity the arrow is to point.

LEADER LINE METHOD

- 1.SELECTED POSITION
- 2.ENTER SLOPE

Enter:

- 1 To draw an arrow that points to the position you selected on the chamfer.
- 2 To enter the slope of the leader with this prompt:

SLOPE = n.nnnn

Enter the leader angle. The angle is measured in a counterclockwise direction.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3.SET WHEN USED, the system displays the following prompt:

TOLERANCING METHOD

- 1.TOLERANCE
- 2.LIMITS

Enter:

- 1 To automatically add tolerance to the dimension.
- 2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt:

TOLERANCE

- 1.UPPER = .nnn
- 2.LOWER = -.nnn

Enter the values for the upper and lower tolerance to be added automatically to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

The system requests the starting location for the text.

INDICATE TEXT ORIGIN

Indicate the starting location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the slope value and generates the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you must enter the text for the dimension when you receive the following prompt:

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end the line. Enter] or RETURN twice to terminate the entry of text. The system allows the entry of at least 100 uppercase or lowercase characters, minus the number of text lines.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off and 16.1.4.4 DIMENSIONING UNITS is set to dual dimensioning, the system requires secondary text.

ENTER SECONDARY TEXT

Enter the text to be written below the standard text. Enter] for the system to convert the standard text. If this is not possible, the system redisplays the prompt.

If 16.1.13 AUTOMATIC TAIL LOCATION is off, the system displays:

TAIL LOCATION

- 1.START OF TEXT
- 2.END OF TEXT
- 3.AUTOMATIC

Enter:

- 1 To place the tail to the left of the text.
- 2 To place the tail to the right of the text.
- 3 To place the tail automatically.

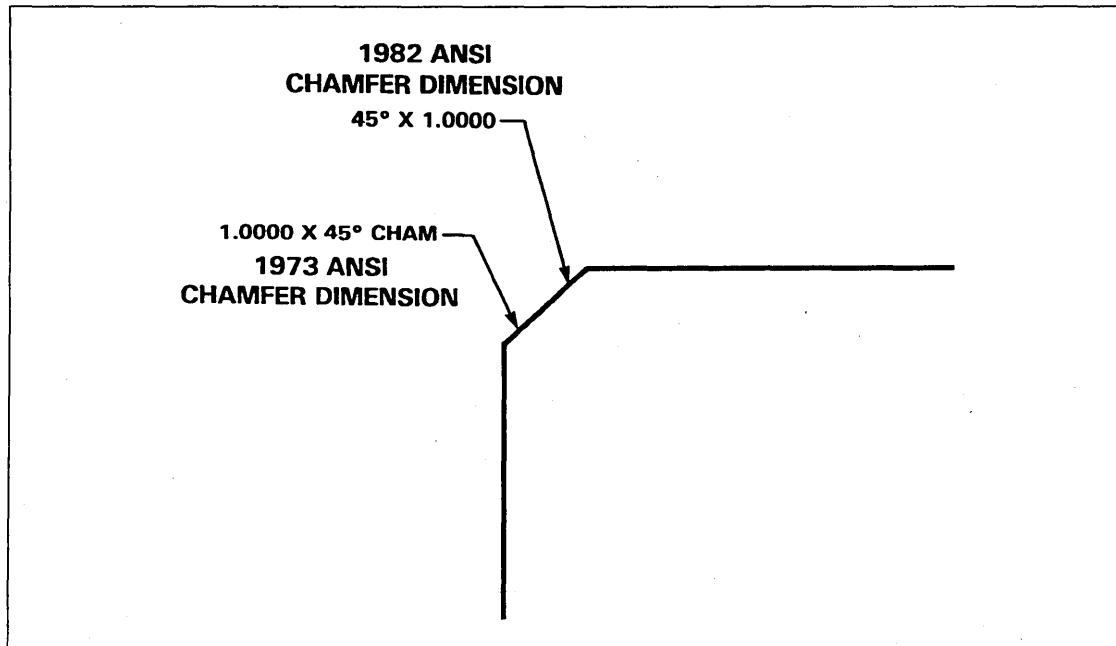


Figure 1-11. Chamfer Dimensions

16.2.10 Arc Length

With this choice, you can produce an arc length dimension. This is a measurement of length on the circumference of an arc. Figure 1-12 shows examples of arc length dimensions.

INDICATE ARC

Use the graphics cursor to select the arc whose circumference you want to dimension. The length of the circumference is measured from one endpoint of the arc to the other.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3.SET WHEN USED, the system displays the following prompt:

TOLERANCING METHOD

- 1. TOLERANCE
- 2. LIMITS

Enter:

- 1 To automatically add tolerance to the dimension.
- 2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt:

TOLERANCE

- 1. UPPER = .nnn
- 2. LOWER = -.nnn

Enter the values for the upper and lower tolerance to be added automatically to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

The system requests the starting location for the text.

INDICATE TEXT ORIGIN

Indicate the starting location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the length and generates the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you enter the text for the dimension when you receive the following prompt:

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Either enter] or RETURN twice to end text entry. The system allows the entry of at least 100 uppercase or lowercase characters, minus the number of text lines.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off and 16.1.4.4 DIMENSIONING UNITS is set to dual dimensioning, the system requires secondary text.

ENTER SECONDARY TEXT

Enter the text to be written below the standard text. Enter] for the system to convert the standard text. If this is not possible, the system redisplays the prompt.

If 16.1.13 AUTOMATIC TAIL LOCATION is off and the dimension has a tail on it, the system displays:

TAIL LOCATION

- 1.START OF TEXT
- 2.END OF TEXT
- 3.AUTOMATIC

Enter:

- 1 To place the tail to the left of the text.
- 2 To place the tail to the right of the text.
- 3 To place the tail automatically.

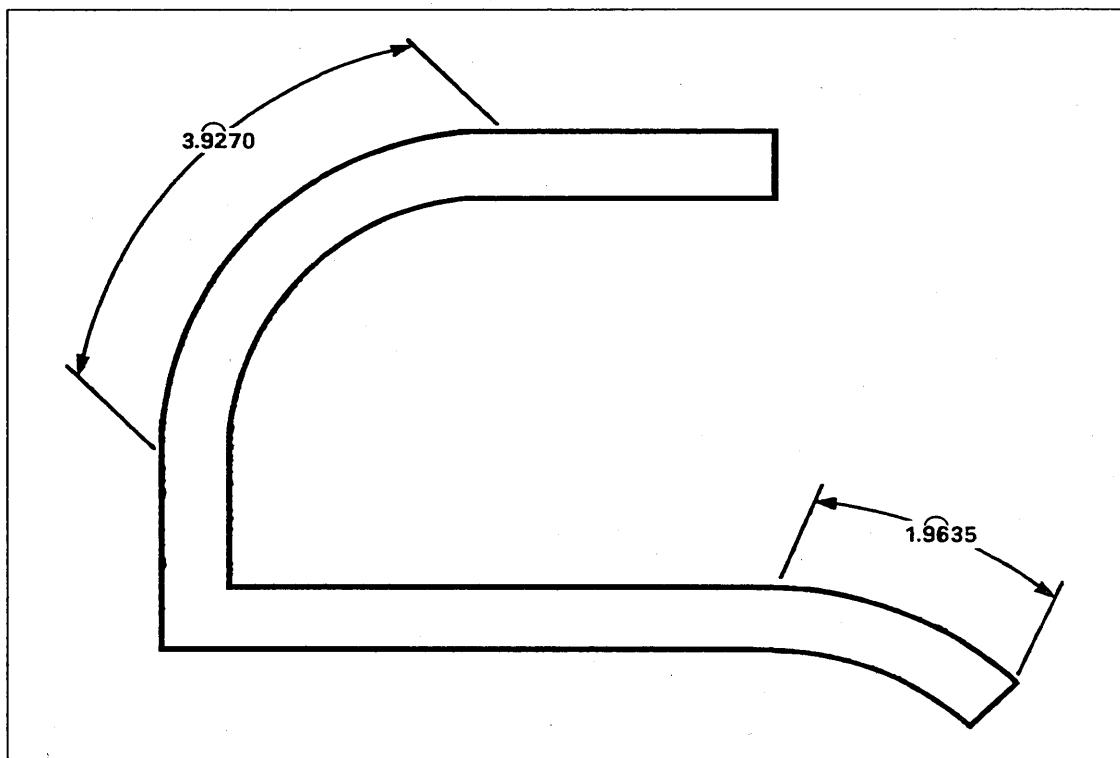


Figure 1-12. Arc Length Dimensions

16.2.11 Rectangular Coordinate

With this choice, you can draw a rectangular coordinate dimension specifying the distance between an origin and a selected entity in either a horizontal or vertical direction. Figure 1-13 shows an example of rectangular coordinate dimensions. The menu for this section is:

```
RECTANGULAR COORDINATE
1.DEFINE ORIGIN
2.HORIZONTAL
3.VERTICAL
```

If you choose 1.DEFINE ORIGIN, the system asks you to indicate the origin point (or datum) from which subsequent rectangular ordinate dimensions are measured. You must define an origin before you can choose 2.HORIZONTAL or 3.VERTICAL.

INDICATE POINT

Use the graphics cursor to select the point to be used as the origin.

The system confirms your choice with the following message:

ORIGIN POINT DEFINED

If you choose 2.HORIZONTAL or 3.VERTICAL, the system makes sure you defined an origin point. If you did not, you receive the following message:

DIMENSION ORIGIN HAS NOT BEEN DEFINED

You return to the Rectangular Coordinate menu in order to define an origin.

If you defined an origin, the system asks for the entity to which you want to dimension.

INDICATE ENTITY

Use the graphics cursor to indicate the entity to which to dimension. Place the cursor near the end of the curve from which the extension line is to be drawn. Allowable entities include points, lines, arcs, conics, splines, composite curves, point sets, machine curves, and centerlines.

If 16.1.10 ARROW ALIGNMENT is on, the system displays:

INDICATE DIMENSION

Use the graphics cursor to select the existing dimension you want to align with the new dimension. Enter] to use the previously created or modified horizontal dimension as the alignment dimension. Enter [to create the new dimension without alignment.

If you indicate the wrong type of dimension, the system displays:

THE ALIGNMENT DIMENSION MUST HAVE SIMILAR ORIENTATION.

Enter] or [to continue.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.2 ENTER PRESET TOLERANCE or 16.1.4.5.3 SET WHEN USED with tolerancing method 3.SET WHEN USED, the system displays the following prompt:

TOLERANCING METHOD
1.TOLERANCE
2.LIMITS

Enter:

- 1 To automatically add tolerance to the dimension.
- 2 To automatically create limits from the dimension.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and you choose 16.1.4.5.3 SET WHEN USED, the system displays the following prompt:

TOLERANCE
1.UPPER = .nnn
2.LOWER = -.nnn

Enter the values for the upper and lower tolerance to be automatically added to the dimension. Refer to section 16.13.4 ADD/CHANGE TOLERANCE OR LIMITS for a description of tolerance values and some examples.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on and 16.1.4.3.3 REPETITIVE FEATURES is on, the system prompts for the number of features:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

INDICATE TEXT ORIGIN

Indicate the location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is on, the system calculates the distance between the two entities selected and writes that distance in the dimension. If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off, you enter the text for the dimension when you receive the following prompt:

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Either enter] or RETURN twice to end text entry. The system allows the entry of at least 100 uppercase or lowercase characters, minus the number of text lines.

If 16.1.4.1 AUTOMATIC DIMENSION TEXT is off and 16.1.4.4 DIMENSIONING UNITS is set to dual dimensioning, the system requires secondary text.

ENTER SECONDARY TEXT

Enter the text to be written below the standard text. Enter] for the system to convert the standard text. If this is not possible, the system redisplays the prompt.

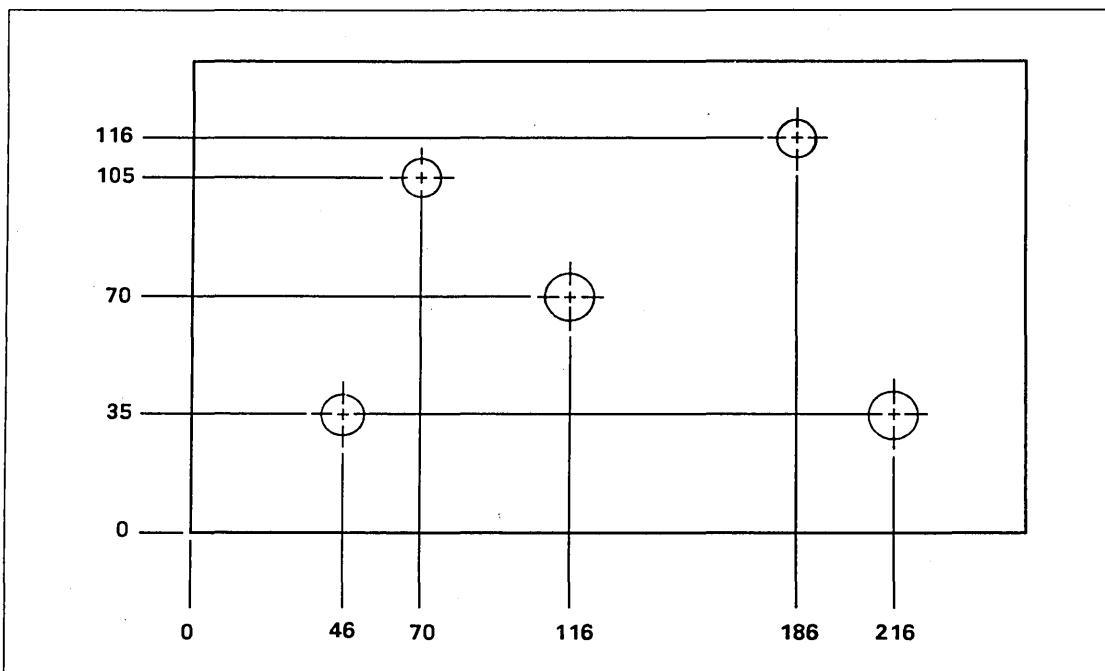


Figure 1-13. Rectangular Coordinate Dimensions

16.3 Section Lining

With this choice, you can produce section lining within specified boundaries. You define the appearance of the section lining with modal settings. You enter the angle of the lines, the distance between them, and the tolerance between the end of the lines and the boundaries. You can also specify areas (islands) within the main boundary that are not to be section lined.

First, the following menu is displayed:

```
SECTION LINING
1.MODALS
2.DEFINITION
```

Enter:

- 1 To set the section lining modals. The current values are displayed with the menu items.

SECTION LINING MODALS	
1.MATERIAL TYPE	IRON
2.VISIBILITY	VIEW OF DEFINITION
3.ALIGNMENT	OFF

Enter:

- 1 To select the material type used for section lining. Figure 1-14 shows a sample of each type.

SECTION LINING MATERIAL TYPE	
1.IRON	
2.STEEL	
3.BRASS/COPPER	
4.RUBBER/PLASTIC	
5.REFRACTORY	
6.GLASS/SLATE	
7.LEAD	
8.ALUMINUM/MAGNESIUM	

Select the type of section lining you want to generate.

- 2 To display the section lining in either the view of definition or in all views. Only section lining generated after setting this modal is affected.

SECTION LINING VISIBILITY	
1.VIEW OF DEFINITION	
2.ALL VIEWS	

Enter:

- 1 To generate section lining visible only in its view of definition. This is the default display.
- 2 To generate section lining visible in all views.

- 3 To control the alignment of new section lining to existing section lining.

SECTION LINING ALIGNMENT

- 1.ON
2.OFF

Enter:

- 1 To activate section lining alignment.
2 To deactivate section lining alignment. This is the default setting.

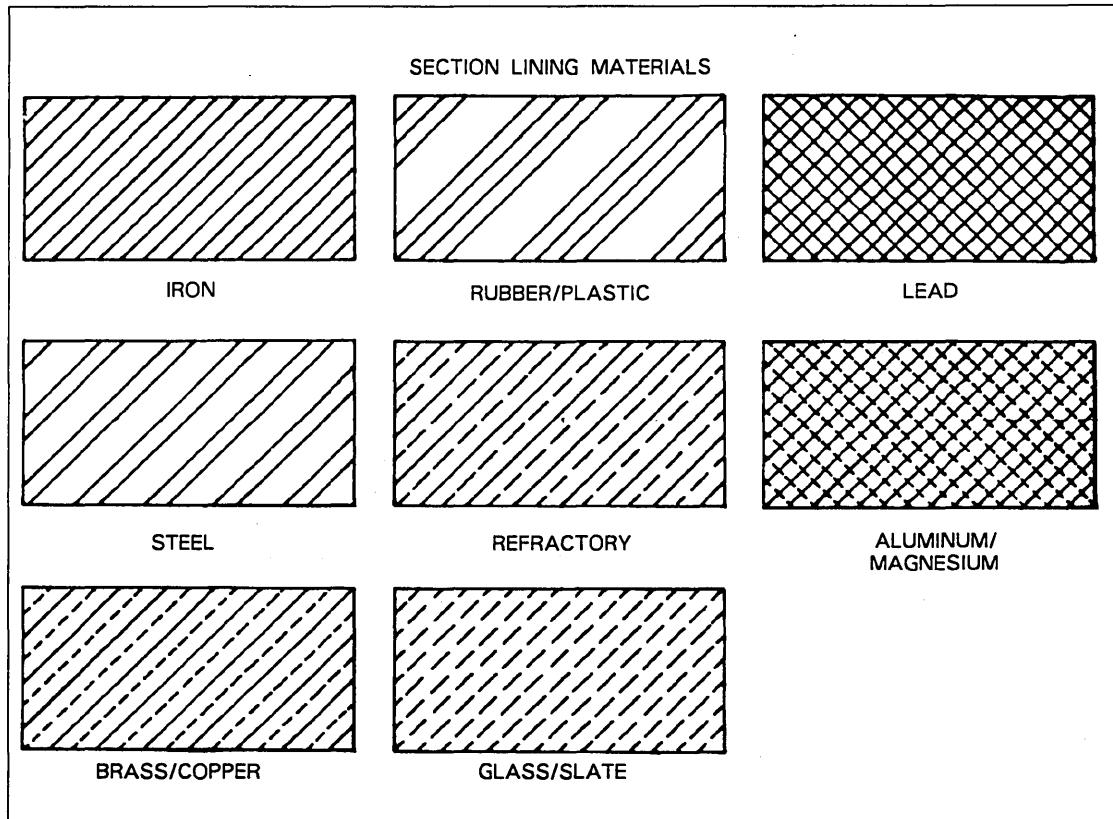


Figure 1-14. Section Lining Material Types

- 2 To proceed with the section lining operation.

If the section lining alignment modal is on, you can select existing section lining to match its angle and spacing. The alignment factor determines the offset used. A factor of 1.0 gives full alignment, that is, the new section lines are in line with the existing section lines. A factor of 0.5 gives half-alignment; the new section lines are halfway between the existing section lines. Figure 1-15 shows examples of alignment.

INDICATE SECTION LINING FOR ALIGNMENT

Select the section lining entity to which you want to align the new section lines.

ALIGNMENT = n.nnnn

Enter an alignment factor from 0.0 to 1.0.

Enter] to bypass alignment.

If you enter a value outside the range of 0.0 to 1.0, the system displays the following message:

VALUE IS OUTSIDE RANGE (0.0 - 1.0)

Continue with the next set of prompts.

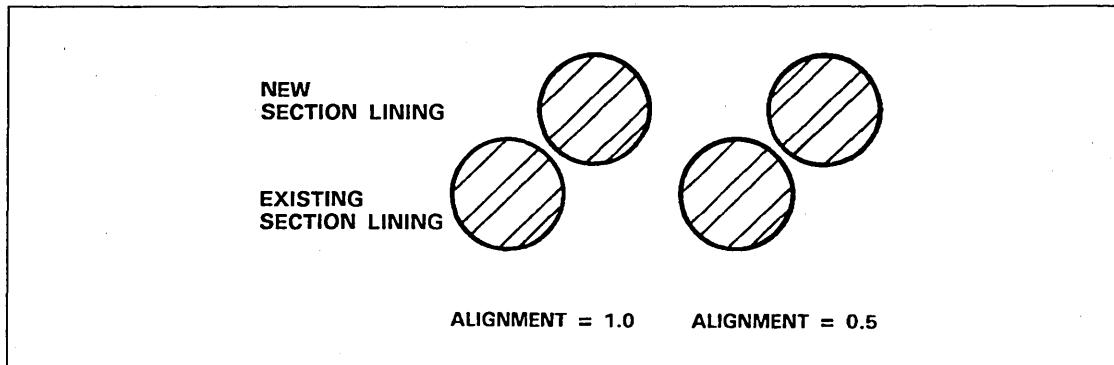


Figure 1-15. Section Lining Alignment Examples

Specify (or accept) values for the section lines. (These prompts appear right after menu selection 2.DEFINITION if the section lining alignment modal is off.)

1. ANGLE = n.nnnn

Enter the angle between the section lines and the positive x transform axis.

2. DISTANCE = n.nnnn

Enter the normal distance between the section lines. The preset value is 6.35 mm (0.25 in).

3. TOLERANCE = n.nnnn

Specify the maximum distance the section lines may be short of or extend beyond a boundary line. The preset value is 0.25 mm (0.01 in).

Specify the main boundary of the section lining:

SECTION LINING MAIN BOUNDARY
INDICATE ENTITY

Use the graphics cursor to screen select the curves defining the outside boundary of the area to be section lined. The allowable entity types for the boundary are lines, arcs, conics, 2-D splines, 3-D splines, machine curves, and Bezier curves.

The INDICATE ENTITY prompt repeats for continued boundary selection until you enter] or CTRL-E. Enter] if you have selected all entities and proceed to the SECTION LINING ISLANDS prompt.

If you have not selected all entities, you can choose additional boundary entities with the Entity Selection menu. Enter CTRL-E to receive the menu:

```
CONSTRAINT SET = NONE
ENTITY SELECTION
1. SCREEN SELECT
2. CHAIN
:
```

Enter] at the Entity Selection menu when you finish defining the main boundary. (This is explained in more detail after the prompts.)

Select the areas (islands) to be omitted from section lining:

```
SECTION LINING ISLANDS
SELECT NEXT ISLAND
INDICATE ENTITY
```

Use the graphics cursor to screen select the curves defining areas (islands) to be omitted from section lining. The allowable entity types for island boundaries are lines, arcs, conics, 2-D splines, 3-D splines, machine curves, and Bezier curves.

The INDICATE ENTITY prompt repeats for continued selection of single island boundaries until you enter] or CTRL-E. Enter] if you have selected all entities bounding a single island.

If you have not selected all entities, you can choose additional island entities with the Entity Selection menu. Enter CTRL-E to receive the menu:

```
CONSTRAINT SET = NONE
ENTITY SELECTION
1. SCREEN SELECT
2. CHAIN
:
```

Enter] at the Entity Selection menu when you finish defining each island (More explanation follows.)

When you have completed all islands (or if you do not want to specify islands) enter] at the SELECT NEXT ISLAND/INDICATE ENTITY prompt. The system draws the section lining.

You can select boundary entities (main or island) using a combination of screen and chain select from the Entity Selection menu. For example, you can screen select two arcs, chain select three lines, and complete the boundary by screen selecting a spline. Complete the boundary definition by entering] at the INDICATE ENTITY prompt or the Entity Selection menu.

It is best to select boundary entities near intersection points so no ambiguity arises when two entities have more than one point of intersection. When you select the first entity, place the graphics cursor near the intersection of the first and second entities. Then select the second entity near the intersection of the second and third entities. Select all entities in this fashion. Select the last entity near its intersection with the first entity.

Section lining is most effective when done within a closed boundary. However, the system does allow reasonable boundary gaps. If you specify an open boundary that is too large, the system places an asterisk (*) at the first gap where the two entities should intersect. You receive the following prompt:

SECTION LINING BOUNDARY IS OPEN AT *
CANNOT CONTINUE, ACKNOWLEDGE

If an island contains the main boundary, the system treats the island as a main boundary and the main boundary as an island. If you select an island within an island within the main boundary, the system section lines the innermost island. Figure 1-16 shows the results.

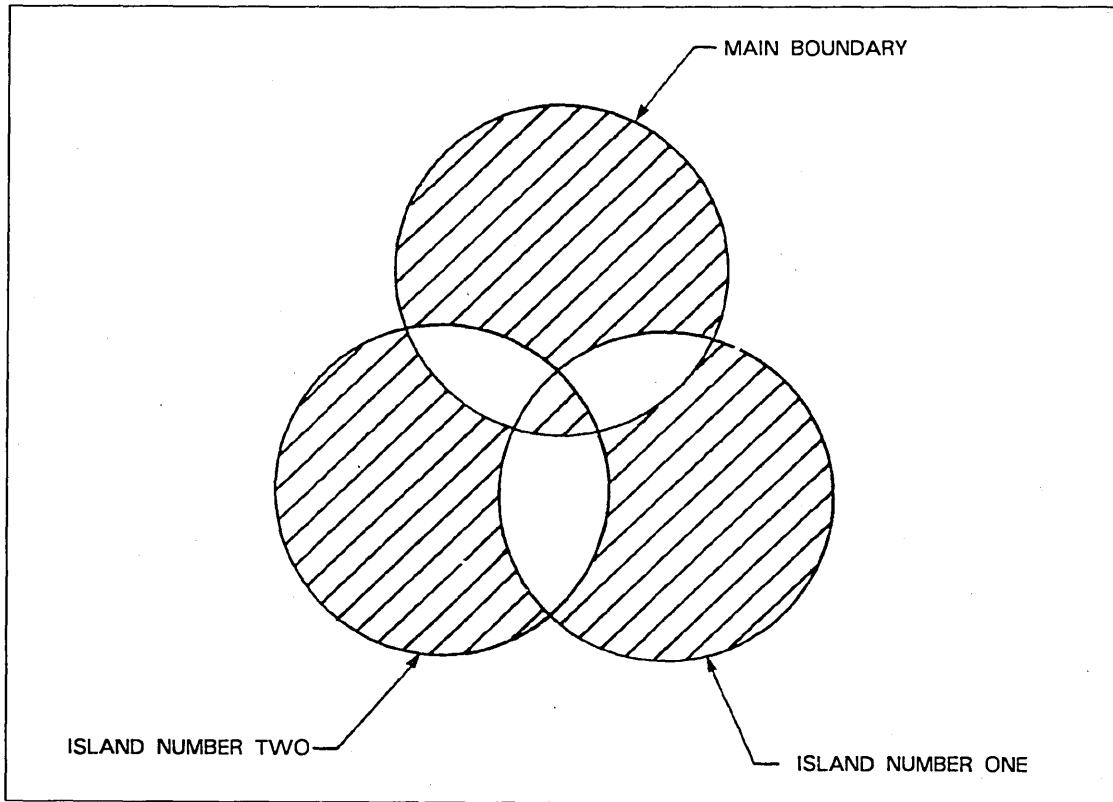


Figure 1-16. Section Lining Islands

16.4 Balloon

With this choice, you can draw a balloon with an arrow pointing to an entity in the current drawing. The system displays the drawing and part numbers inside the balloon.

INDICATE 2-D CURVE

Use the graphics cursor to select a point or two-dimensional curve in the current drawing. The arrow from the balloon touches this entity at the location selected.

INDICATE BALLOON CENTER

Use the graphics cursor to select a position for the center of the balloon.

RADIUS = n.nnnn

Enter the radius of the balloon.

ENTER DETAIL NUMBER

Enter the detail number (fewer than nine characters) and the sheet number (fewer than nine characters) to be written in the balloon. If only one number is desired, enter] for the sheet number.

ENTER SHEET NUMBER

16.5 Datum Target

Selection of this option allows you to define a datum target symbol as shown in figure 1-17. The symbol is composed of arcs, point sets, section lining, and notes all joined into one group entity. You can select this option only when using 1982 ANSI drafting standards. Otherwise this menu item is blanked.

The datum target may be specified by a point, line, circular area, or existing entity.

TARGET
 1.POINT WITHOUT AREA
 2.POINT WITH AREA
 3.LINE
 4.CIRCLE
 5.EXISTING ENTITY

Enter:

- 1 To draw an X at a specified point. You may represent a datum target point or a datum target line on edge with this symbol.
- 2 To draw an X at a specified point. You may represent a small datum target area with this symbol.
- 3 To draw a phantom line through the two points and extend the current extension distance beyond the points. You may represent a datum target line in direct view or a target area on edge with this symbol.
- 4 To draw a section lined area bounded by a phantom circle. You may represent a circular datum area with this symbol.
- 5 To draw the datum target symbol leader to an arbitrary existing entity.

The symbol diameter, normally three-and-one-half-character heights, may be varied no matter which method of specifying the target you choose. The pen number used in drawing the datum target may be set differently from the current pen for point, line, or arc target. However, the target symbol (circle, notes, and leader line) is always defined using the current pen number.

TARGET SYMBOL DATA
 1.RADIUS = n.nnnn
 2.TARGET PEN = n

Enter the symbol radius and the target pen number. (The system requests the target pen number only when you previously requested menu choice 1.POINT WITHOUT AREA, 2.POINT WITH AREA, 3.LINE, or 4.CIRCLE.)

The following prompts vary, depending on the target.

If you enter 1.POINT WITHOUT AREA as the target, the system prompts for the point at which to place the target.

INDICATE POINT

Indicate the point at which the system will place the X.

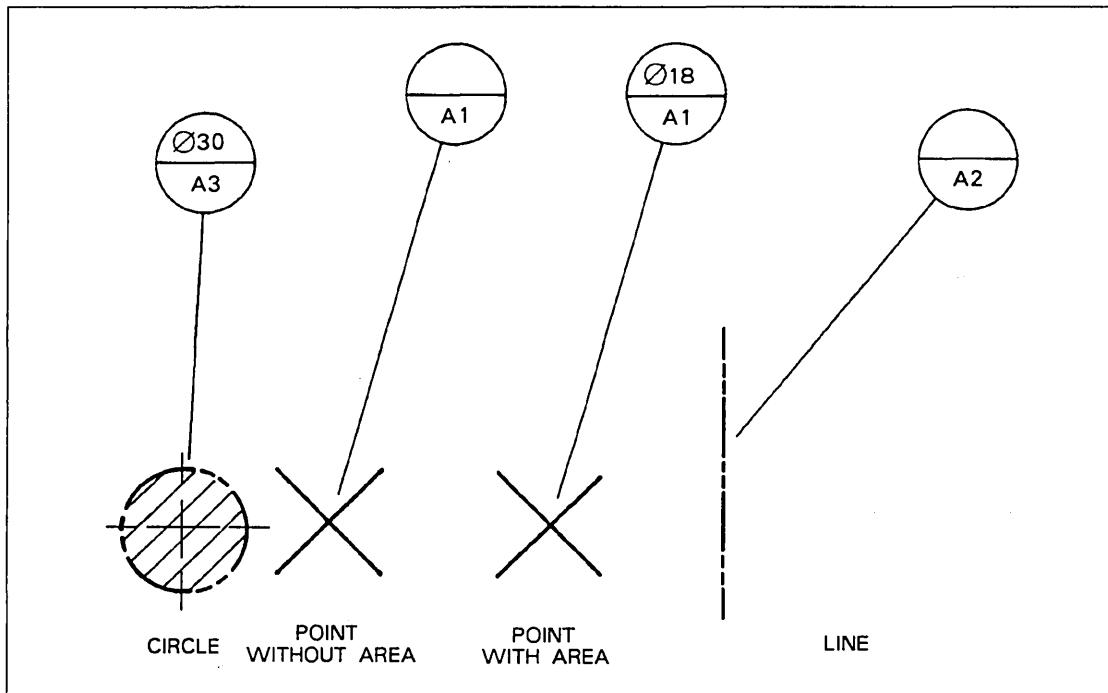


Figure 1-17. Datum Target Symbols

If you enter 2.POINT WITH AREA as the target, the system prompts you to indicate where to place the target.

INDICATE POINT

Indicate the point where the system will place the X.

DIAMETER = n.nnnn

Enter the diameter of the represented area to be written in the note above the line in the target symbol.

If you enter 3.LINE as the target, the system prompts you for the points through which to draw the line.

**INDICATE POINT
INDICATE POINT**

Indicate the first and second points through which the line is to be drawn.

A point set is drawn in the phantom font through the two points and extending the extension dimension distance beyond the points. The system then prompts you to indicate the position on the line where the target leader is to point.

INDICATE POSITION

Indicate the position on the phantom point set to which the target leader is to point.

If you enter 4.CIRCLE as the target, the system requests the necessary data to define the section lined phantom circle.

16.5 Datum Target

INDICATE POINT

Select the target circle center point.

DIAMETER = n.nnnn

Enter the diameter of the target circle. This value is written in the upper half of the symbol circle.

SECTION DATA

1.ANGLE = n.nnnn

2.DISTANCE = n.nnnn

Enter the angle of the section lining and the distance between section lines.

If you enter 5.EXISTING ENTITY as the target, the system requests an entity.

INDICATE ENTITY

Select a point or curve at the position to which the leader is to point.

After the above prompts are satisfied, the system requests a target reference number to be placed in the lower half of the datum symbol circle.

ENTER TARGET IDENTIFIER

Enter a letter identifying the datum followed by a target number.

INDICATE POSITION

Use the current text origin mode to specify the target circle center.

16.6 Datum Feature

With this choice, you can display a datum feature symbol and associated text within a feature frame on the drawing (see figure 1-18). The system asks whether the frame is to have a leader to an entity and how to draw that leader.

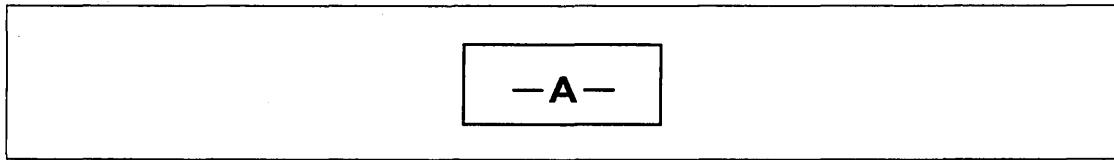


Figure 1-18. Datum Feature Symbol

LEADER DESIRED?

Enter:

Y To produce a connecting leader.

N To not produce a connecting leader. The system proceeds to the ENTER TEXT prompt.

If you select Y, the system displays:

INDICATE ENTITY

Use the graphics cursor to select the datum reference entity. Select a point, line, arc, conic, spline, or dimension. Select curves near the entity end to which the connection is desired. Dimensions are connected to the arrowhead nearest the frame.

Next, the system prompts you to indicate where the connection line should join the feature frame.

CONNECTION TO FEATURE FRAME

- 1.LEFT SIDE
- 2.RIGHT SIDE
- 3.LOWER LEFT CORNER
- 4.LOWER RIGHT CORNER
- 5.UPPER LEFT CORNER
- 6.UPPER RIGHT CORNER

Enter:

- 1 To select the midpoint of the left edge of the frame.
- 2 To select the midpoint of the right edge of the frame.
- 3 To select the lower left corner of the frame.
- 4 To select the lower right corner of the frame.
- 5 To select the upper left corner of the frame.
- 6 To select the upper right corner of the frame.

Now specify the type of connection leader line to be drawn.

CONNECTION METHOD

- 1.JOG
- 2.DIRECT
- 3.MODIFY ORIGIN TO MAKE DIRECT

Enter:

- 1 To produce a horizontal line two times the text dimension distance, followed by a line directly to the datum.
- 2 To produce a straight line from the datum to the feature frame.
- 3 To modify the position of the frame so that a horizontal or vertical line can be drawn.

ARROWHEAD DESIRED?

Enter:

Y To produce an arrowhead at the end of the connection line.

N To omit the arrowhead.

ENTER TEXT

Enter the datum feature identifying character(s). Up to two characters are permitted.

Figure 1-19 shows examples of the connection methods with datum feature symbols.

When the system displays the following menu, select a method to indicate the position of the lower left corner of the datum feature symbol.

ORIGIN METHOD

- 1.SCREEN POSITION
- 2.ENTER COORDINATES
- 3.EXISTING POINT
- 4.BELOW EXISTING TEXT
- 5.ABOVE EXISTING TEXT

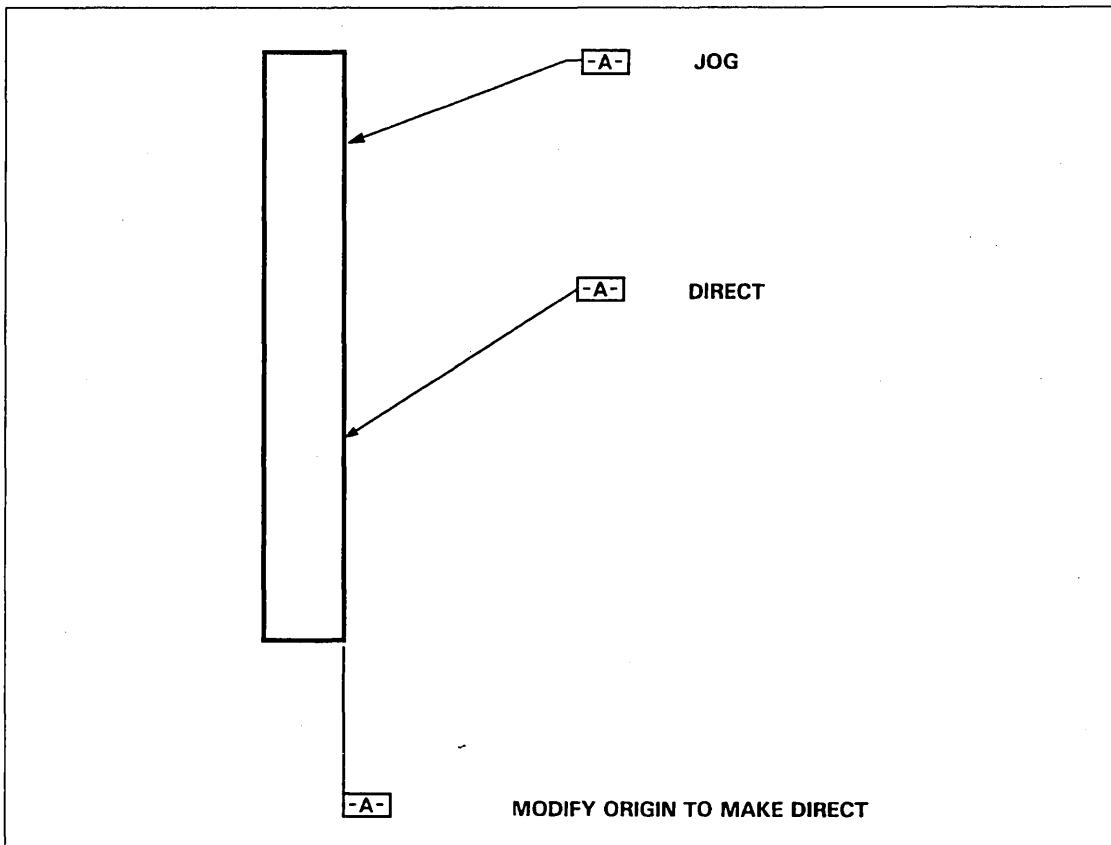


Figure 1-19. Connection Methods with Datum Feature Symbols

Enter:

- 1 To use the graphics cursor to select a screen position.

INDICATE POSITION

Use the graphics cursor to select the location of the lower left corner of the feature frame.

- 2 To enter the transform coordinates of a position.

FRAME COORDINATES

1. ORIGIN XT = n.nnnn
2. YT = n.nnnn

Enter the transform coordinates for the lower left corner of the feature frame.

- 3 To use the graphics cursor to select an existing point as the position.

INDICATE POINT

Use the graphics cursor to select a point as the location of the lower left corner of the feature frame.

- 4 To place the datum feature symbol directly below an existing note, label, dimension, or feature frame.

INDICATE DRAFTING ENTITY

Use the graphics cursor to select an existing drafting entity.

- 5 To place the datum feature symbol directly above an existing note, label, dimension, or feature frame.

INDICATE DRAFTING ENTITY

Use the graphics cursor to select an existing drafting entity.

16.7 Geometric Tolerance Frames

With this menu, you can display geometric tolerance or composite geometric tolerance symbols and associated text within a feature frame on a drawing.

GEOMETRIC TOLERANCE FRAMES

- 1.GEOMETRIC TOLERANCE**
- 2.COMPOSITE GEOMETRIC TOLERANCE**

The following pages describe these menu choices and their resulting prompts.

16.7.1 Geometric Tolerance

The system first asks whether to draw a leader connecting the geometric tolerance to the entity.

LEADER DESIRED?

Enter:

- Y To produce a connecting leader to an entity.
- N To not produce a connecting leader to an entity. The system proceeds to the GEOMETRIC CHARACTERISTIC prompt.

If you select Y, the system asks how to draw the leader:

INDICATE DATUM REFERENCE ENTITY

Use the graphics cursor to select the datum reference entity. Select a point, line, arc, conic, spline, or dimension. Indicate curves near the end of the entity where you want the connection. The dimensions will be connected to the arrowhead nearest the feature frame.

Indicate where you want the connection line to join the feature frame.

CONNECTION TO FEATURE FRAME

- 1.LEFT SIDE
- 2.RIGHT SIDE
- 3.LOWER LEFT CORNER
- 4.LOWER RIGHT CORNER
- 5.UPPER LEFT CORNER
- 6.UPPER RIGHT CORNER

Enter:

- 1 To select the midpoint of the left edge of the frame.
- 2 To select the midpoint of the right edge of the frame.
- 3 To select the lower left corner of the frame.
- 4 To select the lower right corner of the frame.
- 5 To select the upper left corner of the frame.
- 6 To select the upper right corner of the frame.

Specify the type of connection line to be drawn.

CONNECTION METHOD

- 1.JOG
- 2.DIRECT
- 3.MODIFY ORIGIN TO MAKE DIRECT

Enter:

- 1 To produce a horizontal line that is two times the text dimension distance, followed by a line directly to the datum.
- 2 To produce a straight line from the datum to the feature frame.
- 3 To modify the position of the frame in order to draw a horizontal or vertical line.

ARROWHEAD DESIRED?

Enter:

- Y** To produce an arrowhead at the end of the connection line.

- N** To omit the arrowhead.

Figure 1-20 shows examples of the connection methods with geometric tolerance symbols.

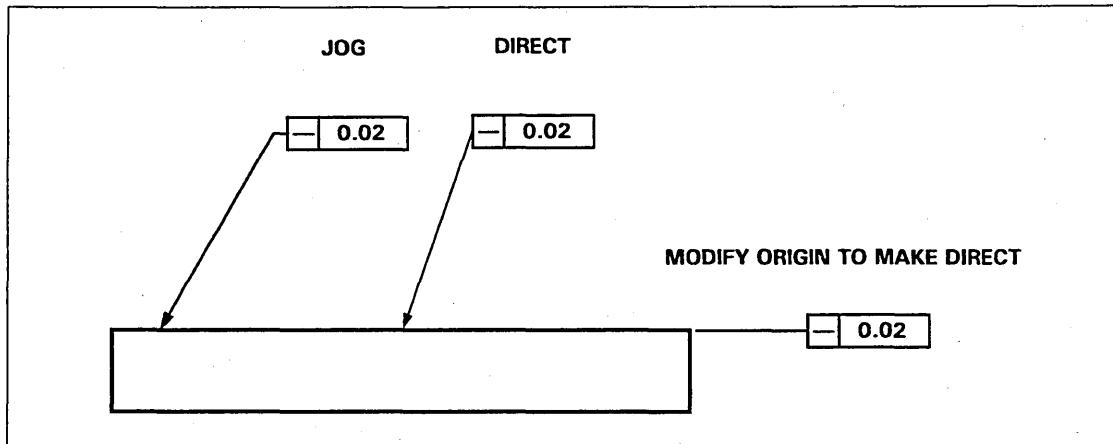


Figure 1-20. Connection Methods with Geometric Tolerance Symbols

Now select the geometric characteristic symbol.

- GEOMETRIC CHARACTERISTIC**
1. STRAIGHTNESS
 2. FLATNESS
 3. CIRCULARITY
 4. CYLINDRICITY
 5. PROFILE OF A LINE
 6. PROFILE OF A SURFACE
 7. ANGULARITY
 8. PERPENDICULARITY
 9. PARALLELISM
 10. POSITION
 11. CONCENTRICITY
 12. CIRCULAR RUNOUT
 13. TOTAL RUNOUT
 14. SYMMETRY

Enter the number of the desired geometric characteristic. The symbol associated with the characteristic selected is drawn in the leftmost section of the frame. Enter] to omit the symbol and proceed to the ENTER DATUM REFERENCE prompt. Enter [to return to the LEADER DESIRED? prompt.

Note that the geometric characteristic symbol for symmetry is not allowed in 1982 ANSI drafting standards. In such a case, menu item number 14 is blanked.

If you enter 6.PROFILE OF A SURFACE for the geometric characteristic and previously chose 1.JOG from the Connection Method menu in a part using 1982 ANSI drafting standards, the system asks whether to attach a symbol signifying that the profile applies to surfaces all around the part.

PROFILE ALL AROUND?

Enter:

Y Draw a circle at the junction of the leader from the feature frame.

N To not display the circle.

The system now allows you to enter the tolerance and datum references for this frame. For 1973 ANSI drafting standards, the order of the prompts for the tolerance and datum references is reversed. The system requests the datum references first, then the tolerance.

ENTER TOLERANCE

Enter the tolerance. This entry can contain up to 20 characters, including the decimal point. Use \O for the diameter symbol. (See 16.1.1.5 SPECIAL SET PREFIX for more information.)

- TOLERANCE MODIFIER**
1. MMC
 2. LMC
 3. RFS
 4. PROJECTED TOLERANCE ZONE

Enter:

- 1 To select maximum material condition (MMC).
 - 2 To select least material condition (LMC).
 - 3 To select regardless of feature size (RFS).
 - 4 To select projected tolerance zone.
-]
To omit the symbol and proceed to the datum references.

ENTER DATUM REFERENCE

Enter a datum reference. Each reference can consist of 1 to 20 alphanumeric characters. Up to five datum references can be included in a single frame. Enter] to omit the reference and draw the feature frame.

DATUM MODIFIER

1. MMC
2. LMC
3. RFS
4. PROJECTED TOLERANCE ZONE

Enter the number of the desired datum modifier. Enter] to omit the symbol following the datum. These selections are the same as for the Tolerance Modifier menu.

Select the method to indicate the position of the lower left corner of the feature frame.

ORIGIN METHOD

1. SCREEN POSITION
2. ENTER COORDINATES
3. EXISTING POINT
4. BELOW EXISTING TEXT
5. ABOVE EXISTING TEXT

Enter:

- 1 To use the graphics cursor to select a screen position.

INDICATE POSITION

Use the graphics cursor to select the location of the lower left corner of the feature frame.

- 2 To enter the transform coordinates of a position.

FRAME COORDINATES

1. ORIGIN XT = n.nnnn
2. YT = n.nnnn

Enter the transform coordinates for the lower left corner of the feature frame.

- 3 To use the graphics cursor to select an existing point as the position.

INDICATE POINT

Use the graphics cursor to select a point as the location of the lower left corner of the feature frame.

- 4 To place the feature frame directly below an existing note, label, dimension, or feature frame.

INDICATE DRAFTING ENTITY

Use the graphics cursor to select an existing drafting entity.

- 5 To place the feature frame directly above an existing note, label, dimension, or feature frame.

INDICATE DRAFTING ENTITY

Use the graphics cursor to select an existing drafting entity.

The system displays the complete geometric tolerance symbol. See figure 1-21 for the symbols.

	.015	STRAIGHTNESS			
	.002	FLATNESS			
	.030	CIRCULARITY			
	.005	CYLINDRICITY			
	.006	PROFILE OF A LINE			
	.06	A	B	PROFILE OF A SURFACE	
	.002	A	ANGULARITY		
	.08	(M)	A	PERPENDICULARITY	
	.03	A	PARALLELISM		
	.01	(M)	B	(M)	POSITION
	.004	A	B	CONCENTRICITY	
	.005	A	B	CIRCULAR RUNOUT	
	A	.02	TOTAL RUNOUT (ANSI'73)		
	.02	A	TOTAL RUNOUT (ANSI'82)		
	A	.005	SYMMETRY (ANSI'73)		

Figure 1-21. Geometric Tolerance Symbols

16.7.2 Composite Geometric Tolerance

The system first asks whether to draw a leader connecting the geometric tolerance to the entity.

LEADER DESIRED?

Enter:

- Y** To produce a connecting leader to an entity.
- N** To not produce a connecting leader to an entity. The system proceeds to the **GEOMETRIC CHARACTERISTIC** prompt.

If you select Y, the system asks how to draw the leader:

INDICATE DATUM REFERENCE ENTITY

Use the graphics cursor to select the datum reference entity. Select a point, line, arc, conic, spline, or dimension. Indicate curves near the end of the entity where you want the connection. The dimensions will be connected to the arrowhead nearest the feature frame.

Indicate where you want the connection line to join the feature frame.

CONNECTION TO FEATURE FRAME

- 1.LEFT SIDE
- 2.RIGHT SIDE
- 3.LOWER LEFT CORNER
- 4.LOWER RIGHT CORNER
- 5.UPPER LEFT CORNER
- 6.UPPER RIGHT CORNER

Enter:

- 1 To select the midpoint of the left edge of the frame.
- 2 To select the lower right corner of the top frame.
- 3 To select the lower left corner of the frame.
- 4 To select the lower right corner of the bottom frame.
- 5 To select the upper left corner of the top frame.
- 6 To select the upper right corner of the top frame.

Specify the type of connection line to be drawn.

CONNECTION METHOD

1. JOG
2. DIRECT
3. MODIFY ORIGIN TO MAKE DIRECT

Enter:

- 1 To produce a horizontal line that is two times the text dimension distance, followed by a line directly to the datum.
- 2 To produce a straight line from the datum to the feature frame.
- 3 To modify the position of the frame in order to draw a horizontal or vertical line.

ARROWHEAD DESIRED?

Enter:

Y To produce an arrowhead at the end of the connection line.

N To omit the arrowhead.

Figure 1-22 shows examples of the connection methods with composite geometric tolerance symbols.

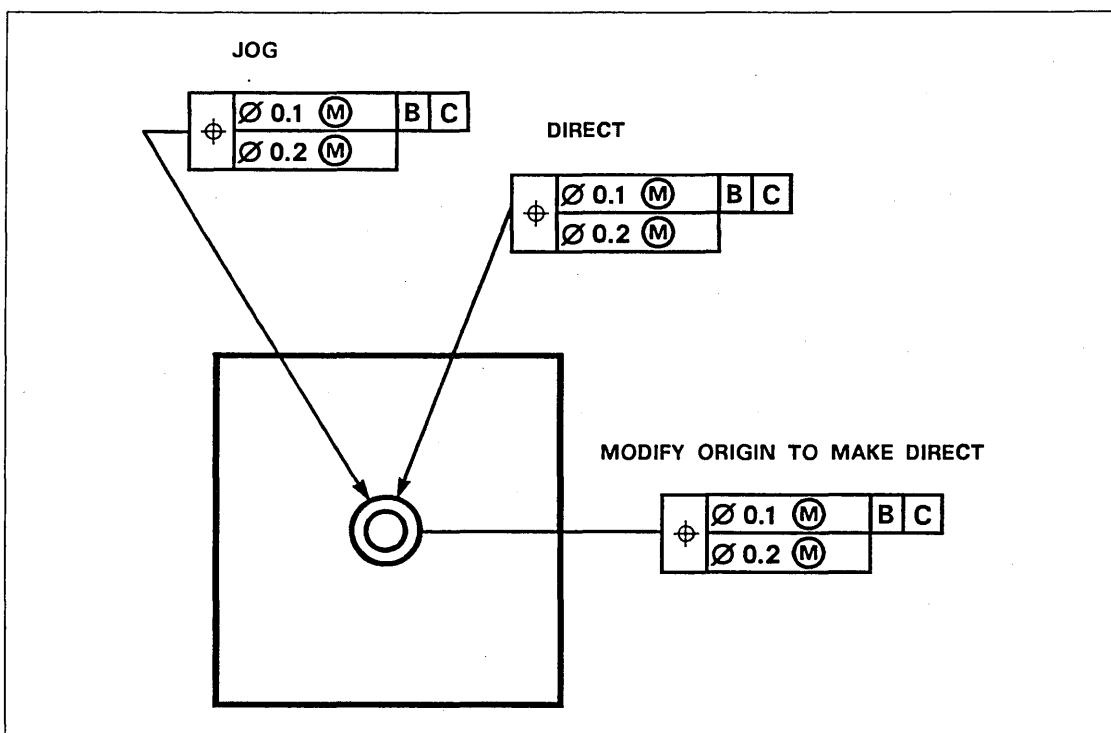


Figure 1-22. Connection Methods with Composite Geometric Tolerance Symbols

Now select the geometric characteristic and symbols.

GEOMETRIC CHARACTERISTIC

- 1. STRAIGHTNESS
- 2. FLATNESS
- 3. CIRCULARITY
- 4. CYLINDRICITY
- 5. PROFILE OF A LINE
- 6. PROFILE OF A SURFACE
- 7. ANGULARITY
- 8. PERPENDICULARITY
- 9. PARALLELISM
- 10. POSITION
- 11. CONCENTRICITY
- 12. CIRCULAR RUNOUT
- 13. TOTAL RUNOUT
- 14. SYMMETRY

Enter the number of the desired geometric characteristic. The symbol associated with the characteristic selected is drawn in the leftmost section of the frame. Enter] to return to the Geometric Tolerance Frames menu. Enter [to return to the LEADER DESIRED? prompt.

Note that the geometric characteristic symbol for symmetry is not allowed in 1982 ANSI drafting standards. In such a case, menu item number 14 is blanked.

If you enter 6.PROFILE OF A SURFACE for the geometric characteristic and previously chose 1.JOG from the Connection Method menu in a part using 1982 ANSI drafting standards, the system asks whether to attach a symbol signifying that the profile applies to surfaces all around the part.

PROFILE ALL AROUND?

Enter:

Y Draw a circle at the junction of the leader from the feature frame.

N To not display the circle.

The system now allows you to enter the tolerance and datum references for the top frame, then the tolerance and datum references for the bottom frame. For 1973 ANSI drafting standards, the order of the prompts for the tolerance and datum references is reversed. The system requests the datum references and tolerance for the top frame first, then the datum references and tolerance for the bottom frame.

ENTER TOLERANCE FOR TOP FRAME

Enter the tolerance for the top frame. This entry can contain up to 20 characters, including the decimal point. Use \0 for the diameter symbol.

TOLERANCE MODIFIER
 1.MMC
 2.LMC
 3.RFS
 4.PROJECTED TOLERANCE ZONE

ENTER DATUM REFERENCE FOR
 TOP FRAME

DATUM MODIFIER
 1.MMC
 2.LMC
 3.RFS
 4.PROJECTED TOLERANCE ZONE

ENTER TOLERANCE FOR BOTTOM FRAME

TOLERANCE MODIFIER
 1.MMC
 2.LMC
 3.RFS
 4.PROJECTED TOLERANCE ZONE

Enter:

- 1 To select maximum material condition (MMC).
 - 2 To select least material condition (LMC).
 - 3 To select regardless of feature size (RFS).
 - 4 To select projected tolerance zone.
-]
 To omit the symbol and proceed to the datum references.

Enter a datum reference for the top frame. Each reference can consist of 1 to 20 alphanumeric characters. Up to five datum references can be included in a single frame. Enter] to omit the reference and proceed to the ENTER TOLERANCE FOR BOTTOM FRAME prompt. If you enter] to both the ENTER TOLERANCE FOR TOP FRAME and ENTER DATUM REFERENCE FOR TOP FRAME prompts, the system returns to the Geometric Tolerance Frames menu.

Enter the number of the desired datum modifier. Enter] to omit the symbol following the datum. These selections are the same as for the Tolerance Modifier menu.

Enter the tolerance for the bottom frame. This entry can contain up to 20 characters, including the decimal point. Use \O for the diameter symbol.

Enter the number of the desired tolerance modifier. Enter] to omit the symbol following the tolerance. These selections are the same as for the Tolerance Modifier menu.

**ENTER DATUM REFERENCE FOR
BOTTOM FRAME**

Enter a datum reference for the bottom frame. Each reference can consist of 1 to 20 alphanumeric characters. Up to five datum references can be included in a single frame. Enter] to omit the reference and draw the feature frame. If you enter] to both the ENTER TOLERANCE FOR BOTTOM FRAME and ENTER DATUM REFERENCE FOR BOTTOM FRAME prompts, the system returns to the Geometric Tolerance Frames menu.

DATUM MODIFIER

- 1.MMC
- 2.LMC
- 3.RFS
- 4.PROJECTED TOLERANCE ZONE

Enter the number of the desired datum modifier. Enter] to omit the symbol following the datum. These selections are the same as for the Tolerance Modifier menu.

Select the method to indicate the position of the lower left corner of the feature frame.

ORIGIN METHOD

- 1.SCREEN POSITION
- 2.ENTER COORDINATES
- 3.EXISTING POINT
- 4.BELOW EXISTING TEXT
- 5.ABOVE EXISTING TEXT

Enter:

- 1 To use the graphics cursor to select a screen position.

INDICATE POSITION

Use the graphics cursor to select the location of the lower left corner of the feature frame.

- 2 To enter the transform coordinates of a position.

FRAME COORDINATES

- 1.ORIGIN XT = n.nnnn
- 2. YT = n.nnnn

Enter the transform coordinates for the lower left corner of the feature frame.

- 3 To use the graphics cursor to select an existing point as the position.

INDICATE POINT

Use the graphics cursor to select a point as the location of the lower left corner of the feature frame.

- 4 To place the feature frame directly below an existing note, label, dimension, or feature frame.

INDICATE DRAFTING ENTITY

Use the graphics cursor to select an existing drafting entity.

- 5 To place the feature frame directly above an existing note, label, dimension, or feature frame.

INDICATE DRAFTING ENTITY

Use the graphics cursor to select an existing drafting entity.

The system displays the complete composite geometric tolerance symbol. See figure 1-23 for an example of a composite geometric tolerance.

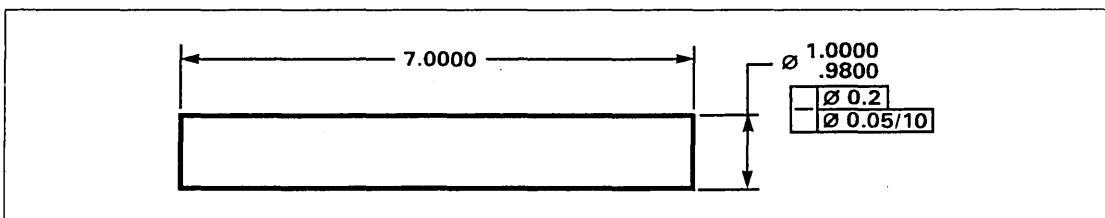


Figure 1-23. Composite Geometric Tolerance

16.8 Surface Texture-Basic Symbol

With this choice, you can display the surface texture-basic symbol either displaced from an entity or entities and joined to the entities by arrows, or you can display it attached directly to the entity. Refer to figure 1-24.

The system displays the roughness average (R_a) in the surface texture symbol.

The entire surface texture symbol can be deleted by using 3.2 FROM TYPE and by selecting arrays and groups as the entity type.

The menu for this section is:

```
SURFACE TEXTURE-BASIC SYMBOL
1.SYMBOL ATTACHED TO ARROW
2.SYMBOL ATTACHED TO ENTITY
```

These menu choices are described here and on the following pages.

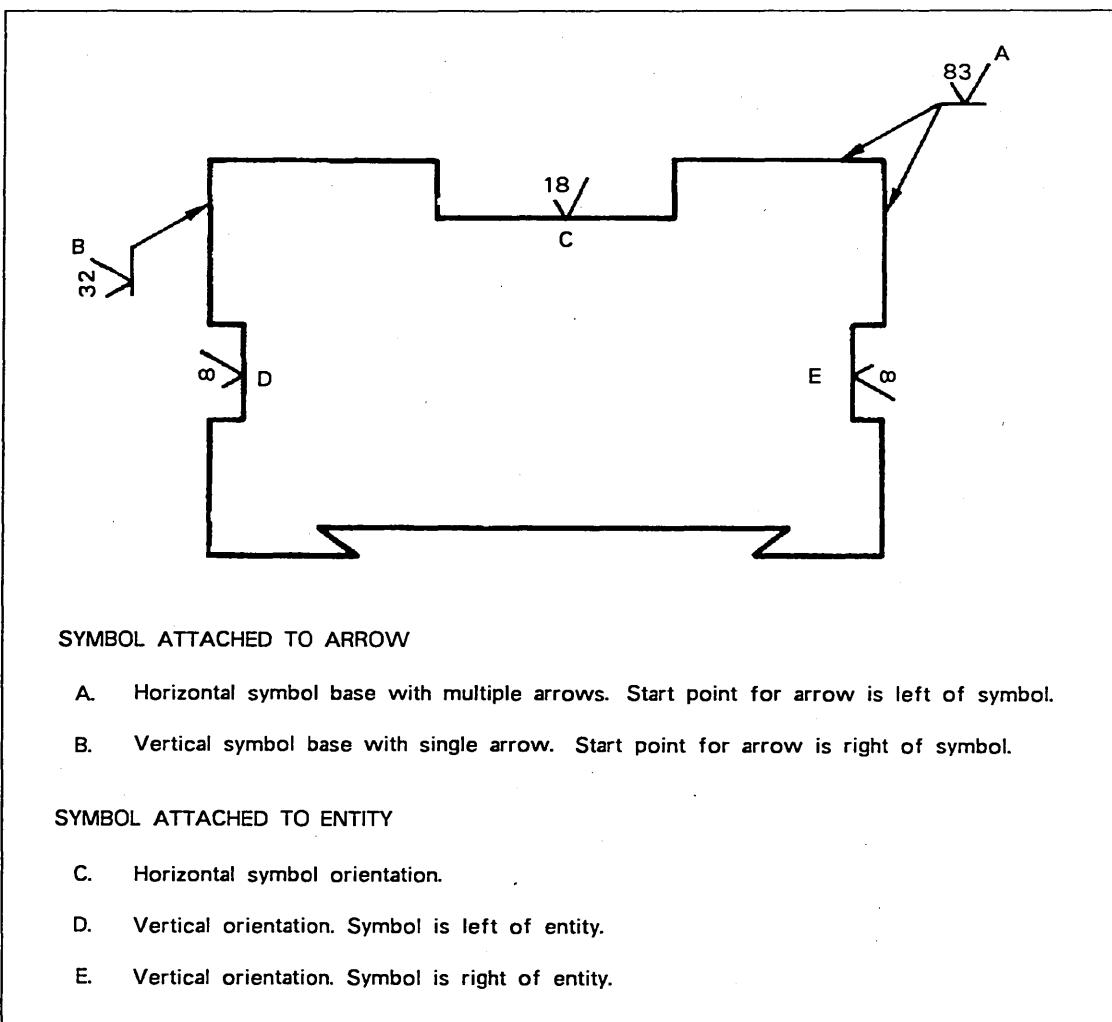


Figure 1-24. Surface Texture Symbols

16.8.1 Symbol Attached to Arrow

With this choice, you can display a surface texture symbol and its base displaced from an entity or entities and joined to the entities with one or more arrows.

ROUGHNESS = n.nnnn

Enter the value of Ra using micrometers for SI units or microinches for U.S. customary units. When using SI units, the system displays the number of decimal places specified by 16.1.4.2 DECIMAL/FRACTIONS. When using U.S. customary units, Ra is displayed as a whole number without a decimal point. Entering 0 produces a symbol with no text. The default value is 1.60 μm (63 μin).

INDICATE 2-D CURVE

Use the graphics cursor to indicate the approximate position of the arrow endpoint on the curve. Up to 10 positions can be selected. Enter] to terminate selection of the positions.

Select the symbol base orientation.

SYMBOL BASE
1.HORIZONTAL
2.VERTICAL

Enter:

- 1 To display the surface texture symbol above a horizontal base.
- 2 To display the surface texture symbol to the left of a vertical base.

Select the relative location of the arrow's starting point.

STARTING POINT FOR ARROW
1.LEFT OF SYMBOL
2.RIGHT OF SYMBOL

Enter:

- 1 To start the arrow to the left of a horizontal symbol base, or at the bottom of a vertical symbol base.
- 2 To start the arrow to the right of a horizontal symbol base or at the top of a vertical symbol base.

INDICATE POSITION

Use the graphics cursor to indicate where the arrow or arrows attach to the symbol base. This determines the exact placement of the symbol.

The system displays the symbol, symbol base, Ra value, and arrows.

16.8.2 Symbol Attached to Entity

With this choice, you can display a symbol directly attached to an entity.

ROUGHNESS = n.nnnn

Enter the value of Ra using micrometers for SI units or microinches for U.S. customary units. When using SI units, the system displays the number of decimal places specified by 16.1.4.2 DECIMAL/FRACTIONS. When using U.S. customary units, Ra is displayed as a whole number without a decimal point. Entering 0 produces a symbol with no text. The default value is 1.60 μm (63 μin).

SYMBOL ORIENTATION

- 1.HORIZONTAL
- 2.VERTICAL-LEFT OF ENTITY
- 3.VERTICAL-RIGHT OF ENTITY

Enter:

- 1 To display the symbol horizontally above the entity.
- 2 To display the symbol vertically to the left of the entity.
- 3 To display the symbol vertically to the right of the entity.

INDICATE 2-D CURVE

Use the graphics cursor to select the approximate position of the symbol on the entity.

The system displays the symbol and the Ra value.

16.9 Arrow

With this choice, you can select one of two types of arrows: an arrowhead on a curve, or cross-section arrows. The menu for this section is:

ARROW TYPE

- 1.ARROWHEAD ON CURVE
- 2.SECTION ARROWS

The following sections describe the choices in this menu.

16.9.1 Arrowhead on Curve

With this choice, you can draw an arrowhead at any position along an existing curve. The system treats arrowheads created by this operation as triangles and therefore they can be used in the same way as triangles.

ARROWHEAD PLACEMENT

1. AT CURVE END
2. AT SCREEN POSITION ON CURVE

Enter:

- 1 To place the arrowhead at the indicated curve end.

INDICATE CURVE

Use the graphics cursor to select a line, arc, conic, spline, composite curve, point set, or three-dimensional spline. An arrowhead is drawn at the end nearest the graphics cursor.

- 2 To place the arrowhead along a curve at the indicated position.

INDICATE 2-D CURVE

Use the graphics cursor to select a curve at the position where you want the arrowhead tip. You can select a line, arc, conic, or spline.

INDICATE DIRECTION OF ARROW

Use the graphics cursor to select the direction along the curve that the arrowhead will point. An arrowhead is then displayed along the curve.

The size of arrowheads created by this operation is controlled by the value of 16.1.7 ARROWHEAD LENGTH. The sizes of arrowheads you define under this operation are not dynamically adjusted when 16.1.7 ARROWHEAD LENGTH is changed.

16.9.2 Section Arrows

With this choice, you can create cross-section arrows for declaring details or cutouts for drafting.

SECTION ARROW DATA

1.THICKNESS = n.nnnn

Enter the thickness of the section arrow main body. The default is .5 times the length set in 16.1.7 ARROWHEAD LENGTH.

2.ARROW LENG = n.nnnn

Enter the length of the section arrowhead. The default value is 5 times the length set in 16.1.7 ARROWHEAD LENGTH.

3.ARROW WID = n.nnnn

Enter the width of the section arrowhead. The default is 1.66 times the length set in 16.1.7 ARROWHEAD LENGTH.

4.MIN NECK = n.nnnn

Enter the minimum distance between an arrowhead and the first jog of the main body. The default value is 1.5 times the length set in 16.1.7 ARROWHEAD LENGTH.

5.LEVEL = nn

Enter the level on which to define the section arrows. The default is the current level.

Refer to figure 1-25 for an illustration of section arrow dimensions. Enter] to continue.

ARROWHEAD POSITION

- 1.AT START
- 2.AT END
- 3.AT BOTH ENDS
- 4.NO ARROWHEADS

Enter:

- 1 To place the arrowhead at the start.
- 2 To place the arrowhead at the end.
- 3 To place the arrowhead at both ends.
- 4 To draw thick bars without arrowheads.

INDICATE POSITION

Use the graphics cursor to select the screen position where the section arrow starts.

INDICATE POSITION

Use the graphics cursor to select the next section arrow jog position. The system specifies the positions in the middle of the section arrow joints.

After you specify jog positions, entering] results in the definition and display of the section arrow. A section arrow is defined as a point set curve (similar to the way that N-gon entities are defined). You can section line this arrow in a separate process (refer to 16.3 SECTION LINING).

If 1.2.1 CONTINUE OPERATION MODE is on, the system redisplays the INDICATE POSITION prompt in anticipation of the start of another section arrow. Entering] terminates this function and returns you to the next higher menu. If 1.2.1 CONTINUE OPERATION MODE is off, the system returns to the next higher menu.

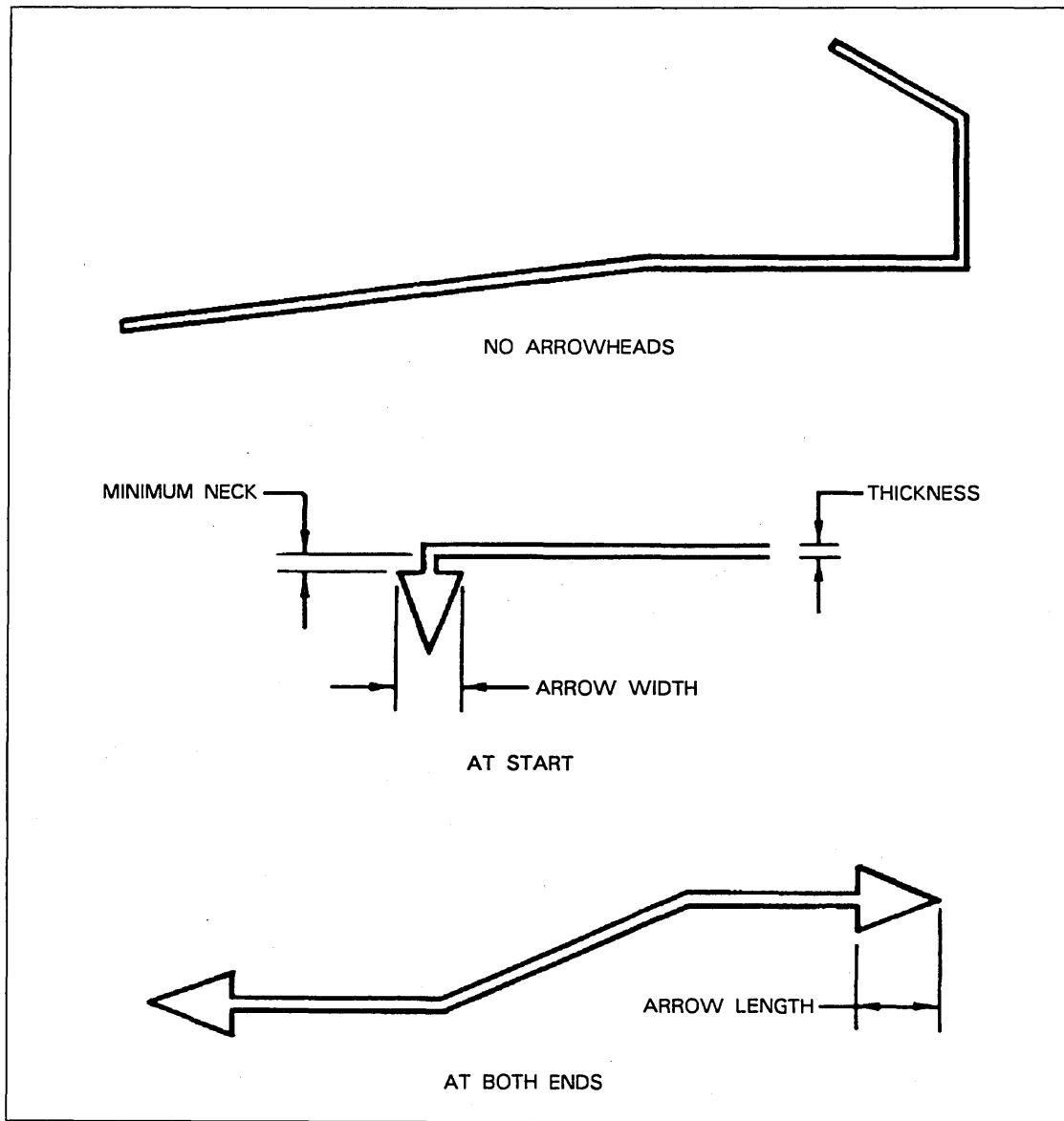


Figure 1-25. Section Arrow Dimensions

16.10 Note

With this choice, you can write text on the drawing. Select the location of the note and, optionally, either the angle at which the text is to be written or the line or arc that the text is to follow. 16.1.1.1 CHARACTER SIZE sets the size of the characters.

You then indicate the location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.5 TEXT ANGLE CONTROL is set to 2.ENTER ANGLE, the system displays:

TEXT ANGLE = n.nnnn

Enter the angle at which to display the text.

If 16.1.5 TEXT ANGLE CONTROL is set to 3.PARALLEL TO LINE/ARC, the system displays:

PARALLEL TO LINE OR ARC?

Enter:

Y To write a note parallel to a line or arc.

N To write a note that is not parallel to a line or arc.

If you answer Y, the system asks you to indicate the curve to which the note is to be parallel.

INDICATE 2-D CURVE

Select the line or arc to which the note is to be parallel.

If you select a line, the system requests the text direction of the note.

TEXT DIRECTION

Enter:

1.POSITIVE X
2.NEGATIVE X

1 To direct the text along the positive x-axis (horizontally).

2 To direct the text along the negative x-axis (horizontally and upside down).

For vertical lines, choosing 1.POSITIVE X causes the text to be displayed along the positive y-axis (vertically).

If you select an arc, the system requests the text rotation direction for the note. Notes parallel to an arc must be only one line long.

TEXT ROTATION

Enter:

1.CLOCKWISE
2.COUNTERCLOCKWISE

1 To rotate the text in a clockwise direction.

2 To rotate the text in a counterclockwise direction.

If 16.1.5 TEXT ANGLE CONTROL is set to 4.TOTAL ANGLE CONTROL, the system displays:

PARALLEL TO LINE OR ARC?

Enter:

Y To write a note parallel to a line or arc.

N To enter a new angle.

If you select Y, the system asks you to indicate the curve to which the note is to be parallel.

INDICATE 2-D CURVE

Select the line or arc to which the note is to be parallel.

If you select a line, the system asks for the text direction of the note.

TEXT DIRECTION

- 1.POSITIVE X
- 2.NEGATIVE X

Enter:

1 To direct the text along the positive x-axis (horizontally).

2 To direct the text along the negative x-axis (horizontally and upside down).

For vertical lines, choosing 1.POSITIVE X causes the text to be displayed along the positive y-axis (vertically).

If you select an arc, the system asks for the text rotation of the note. Notes parallel to an arc must be only one line long.

TEXT ROTATION

- 1.CLOCKWISE
- 2.COUNTERCLOCKWISE

Enter:

1 To rotate the text in a clockwise direction.

2 To rotate the text in a counterclockwise direction.

If you select N in response to the PARALLEL TO LINE OR ARC? prompt, the system displays:

TEXT ANGLE = n.nnnn

Enter the angle at which the text is to be displayed.

After all other prompts are satisfied, or if 16.1.5 TEXT ANGLE CONTROL is set to 1. HORIZONTAL (the default setting), the system displays:

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Enter either] or RETURN twice to terminate entry of text. The system allows the entry of at least 500 uppercase or lowercase characters, minus twice the number of text lines.

To leave a blank line within a note, type one or more blank spaces, followed by a carriage return.

16.11 Label

With this choice, you can write a label on the drawing that includes text and an arrow from the text (called a leader) to an entity on the drawing. The leader includes a tail, which is a short horizontal bar extending from the beginning or end of the text, depending on the setting of 16.1.13 AUTOMATIC TAIL LOCATION.

INDICATE CURVE

Select the entity to which the arrow points.

If 16.1.13 AUTOMATIC TAIL LOCATION is off and the dimension has a tail on it, the system displays:

TAIL LOCATION

- 1.START OF TEXT
- 2.END OF TEXT
- 3.AUTOMATIC

Enter:

- 1 To place the tail to the left of the text.
- 2 To place the tail to the right of the text.
- 3 To place the tail automatically.

The vertical position of the leader line tail is determined by 16.1.9.2 LEADER PLACEMENT. The leader touches the first text line by default. Indicate where on the entity the arrow is to point.

LEADER LINE METHOD

- 1.SELECTED POSITION
- 2.ENTER SLOPE

Enter:

- 1 To draw an arrow that points to the position on the entity where you selected the entity.
- 2 To enter the slope (angle) of the arrow.

SLOPE =

Enter the angle. The angle is measured in a counterclockwise direction from the positive x transform axis.

INDICATE TEXT ORIGIN

Indicate the location of the text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD.

If 16.1.5 TEXT ANGLE CONTROL is set to 2.ENTER ANGLE or 4.TOTAL ANGLE CONTROL, the system displays:

TEXT ANGLE = n.nnnn

Enter the angle at which the system is to display the text.

After all other prompts are satisfied, or if you set 16.1.5 TEXT ANGLE CONTROL to 1.NONE (the default setting), the system displays:

ENTER TEXT

Enter the text to be written in the dimension. Enter RETURN once to end a line. Enter either] or RETURN twice to terminate entry of text. The system allows the entry of at least 500 uppercase or lowercase characters, minus twice the number of text lines.

16.12 Centerline

With this choice, you can draw a centerline between points, circles, or bolt circles (a circular array of points or circles). Select the type of centerline:

DEFINITION TYPE

- 1.EXISTING POINTS
- 2.EXISTING CIRCLE(S)
- 3.BOLT CIRCLE

Enter:

- 1 To draw a centerline between points.

INDICATE POINTS

Use the graphics cursor to select up to 12 existing points through which the centerline can be drawn. If more than two points are selected, all the points must be collinear.

If you select two points, the centerline constructed by the system begins at one point, ends at the other, and has a short dash at its midpoint. If you select more than two points, the centerline begins and ends at the outermost points and has a short dash through each of its inner points.

- 2 To draw a centerline between circles.

INDICATE CIRCLES

Use the graphics cursor to select up to six collinear circles through whose centers the centerline will be drawn. You must select the circles sequentially, from one end to the other, and they must not have coincident centers.

If you select a single circle, horizontal and vertical centerlines are drawn through the center. If you select more than one circle, one common centerline is drawn through all the centers, and separate centerlines, perpendicular to the common centerline, are drawn through each center.

- 3 To draw a centerline through a bolt circle.

INDICATE CIRCULAR ARRAY

Use the graphics cursor to select a circular array of points or circles.

A circular centerline is drawn through all the points or centers, and a separate centerline normal to the circular centerline is drawn through each center.

16.13 Modify Drafting Entity

With this choice, you can modify various drafting entities without redefining them. Select the type of modification. The menu choices are:

- DRAFTING ENTITY MODIFICATION TYPE
- 1. NEW TEXT ORIGIN
 - 2. BASIC
 - 3. REFERENCE
 - 4. ADD/CHANGE TOLERANCE OR LIMITS
 - 5. NEW DISPLAY VALUES
 - 6. MODIFY TEXT
 - 7. MODIFY SLANT STATUS
 - 8. MODIFY ANGLE
 - 9. DELETE TOLERANCE OR LIMITS
 - 10. EXTENSION LINE STATUS
 - 11. DIMENSION LINE STATUS
 - 12. CHARACTER SET
 - 13. BALLOON MODIFICATION
 - 14. DATUM TARGET MODIFICATION
 - 15. FEATURE FRAME MODIFICATION
 - 16. DIMENSION ORIGIN MODIFICATION
 - 17. MODIFY DECIMAL PLACES
 - 18. DIMENSION NOT TO SCALE

These menu choices are explained in the following pages.

16.13.1 New Text Origin

With this choice, you can change the origin of text written in a label, note, or dimension.

INDICATE DRAFTING ENTITY

Select the text you want to move.

If 16.1.10 ARROW ALIGNMENT is on and you selected a vertical or horizontal dimension, the system displays:

INDICATE DIMENSION

Use the graphics cursor to select the existing horizontal or vertical dimension with which the new dimension is to be aligned. Enter] to use the previously created or modified horizontal dimension as the alignment dimension. Enter [to create the new dimension without alignment.

If you indicate the wrong type of dimension, the system displays:

THE ALIGNMENT DIMENSION MUST HAVE SIMILAR ORIENTATION.

Enter] or [to continue.

INDICATE TEXT ORIGIN

Indicate the location of the lower left corner of the first character of text to be written. This procedure is described in 16.1.3 TEXT ORIGIN METHOD. If 1.14.3 ENTITY DRAGGING is on, move the entity around the screen. If ENTITY DRAGGING is off, use the cursor to move the entity. (Entity dragging is available only on certain terminals. See the ICEM Design/Drafting Introduction and System Controls manual for further explanation.)

If 16.1.13 AUTOMATIC TAIL LOCATION is off and the dimension has a tail on it, the system displays:

TAIL LOCATION

- 1.START OF TEXT
- 2.END OF TEXT
- 3.AUTOMATIC

Enter:

- 1 To place the new tail to the left of the text.
- 2 To place the new tail to the right of the text.
- 3 To place the tail automatically.

For thickness dimensions, the text origin you specified does not affect the location of the dimension line; instead, the text origin location is modified to the closest possible location.

16.13.2 Basic

With this choice, you can add or delete a rectangle around the text in a label, note, or dimension.

INDICATE DRAFTING ENTITY

Select the label, note, or dimension text you want to modify.

If you select a note that is parallel to an arc, the system responds:

BASIC BOX NOT ALLOWED FOR NOTES PARALLEL TO ARCS

and returns to the INDICATE DRAFTING ENTITY prompt.

If there is a rectangle around the text in the label, note, or dimension, the system deletes the rectangle.

If there is not a rectangle around the text, a rectangle is drawn.

16.13.3 Reference

With this choice, you can add or delete parentheses around the text in a dimension.

INDICATE DIMENSION

Select the dimension text you want to modify.

If parentheses exist around the text in a dimension, the system deletes them.

If parentheses do not exist around the text, the system adds them. The system also trims extension lines automatically.

16.13.4 Add/Change Tolerance or Limits

With this choice, you can add a tolerance to dimension text or change an existing tolerance.

First, indicate whether you desire tolerances or limits.

TOLERANCING METHOD

- 1. **TOLERANCE**
- 2. **LIMITS**

Enter:

- 1 To write tolerances as described previously.
- 2 To write limits in full character size. The upper limit is the dimension and the positive value. The lower limit is the remainder after subtracting the entered negative value from the dimension.

The system then requests the desired tolerance:

TOLERANCE

- 1. **UPPER = .nnn**
- 2. **LOWER = -.nnn**

Enter the values you desire for the upper and lower tolerance.

The previously entered data are the default values for this entry. If you enter both plus and minus tolerances, the system writes them one above the other, immediately following the text in the dimension. If the tolerances you enter are opposite in sign and equal in value, the displayed tolerance is written on one line with full-size characters and a plus-or-minus symbol.

If you are dimensioning in inches, the number of decimal places used in the tolerance is the same as the number used in the main dimension selected according to ANSI standards.

If you are dimensioning in millimeters, the number of decimal places used in the tolerance is determined by the current drafting decimal places. Significant digits are retained out to the number of decimal places in all cases. Trailing zeros are kept only if needed to force both the upper and lower tolerances to have the same number of decimal places.

Figure 1-26 gives examples of dimensions containing tolerances. The height of the tolerance characters is determined by the tolerance ratio. (See sections 16.1.1.4 DISPLAY RATIOS and 16.13.5 NEW DISPLAY VALUES to set the tolerance ratio.)

INDICATE DIMENSION

Select the dimension you want to modify.

The system modifies the dimension by adding or changing the tolerance or limit.

If the dimension is user-entered, you cannot modify it. You receive the following message:

CANNOT ADD TOLERANCE/LIMITS TO USER-ENTERED DIMENSIONS

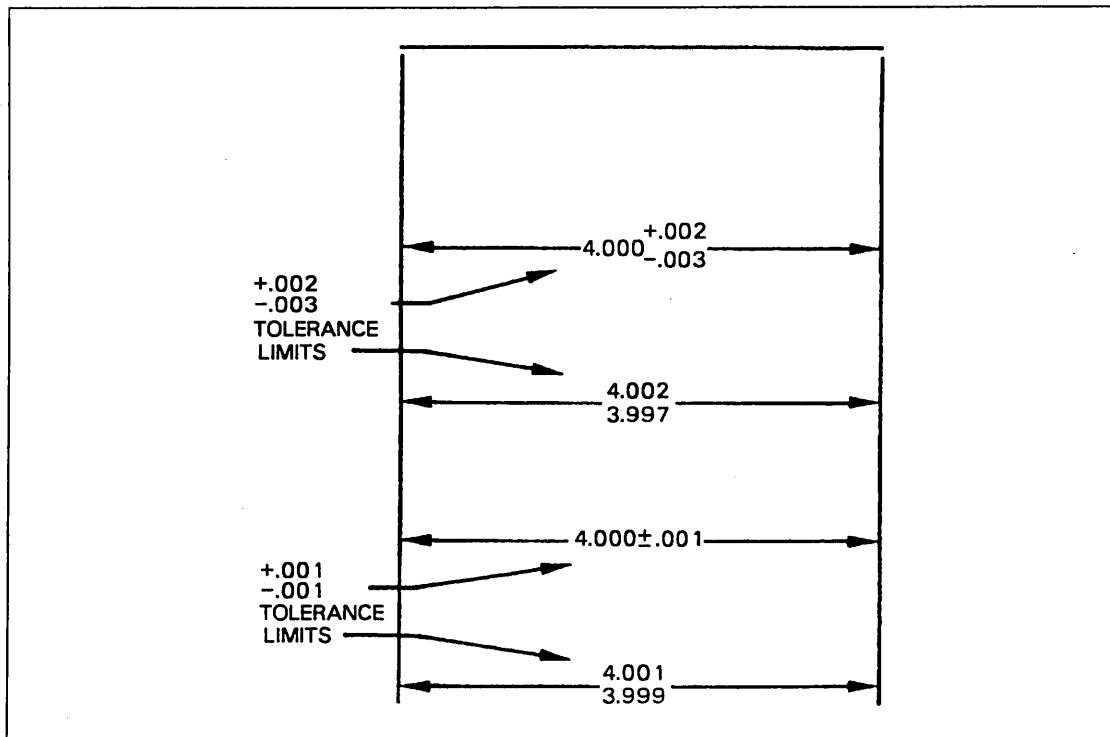


Figure 1-26. Tolerances and Limits

16.13.5 New Display Values

With this choice, you can change the display values of text, dimension lines, and extension lines.

The menu choices are:

- NEW DISPLAY VALUES
 - 1. SINGLE ENTITY ALL VALUES
 - 2. MULTIPLE ENTITIES ONE VALUE
 - 3. MULTIPLE ENTITIES MULTIPLE VALUES

16.13.5.1 Single Entity All Values

The system displays the current text, dimension line, and extension line values applicable to this drafting entity and requests the desired values.

INDICATE DRAFTING ENTITY

Select the text you want to modify.

The display values that can be modified depend on the entity type. Refer to table 1-2 for the applicable values. X indicates that the entity is always present; S indicates that the entity is sometimes present.

Table 1-2. Display Values

Display Values	Entity Type¹						
	32	33	34	35	36	37	40
CHARACTER SIZE = n.nnnn	X	X	X	X	X	X	X
DRAFTING SCALE = n.nnnn	X	X	X	X	X	X	X
SPACING RATIO = n.nnnn	X	X	X	X	X	X	X
ASPECT RATIO = n.nnnn	X	X	X	X	X	X	X
DOWNSPACE RATIO = n.nnnn	X	X	X	X	X	X	X
TOLERANCE RATIO = n.nnnn		X	X	X	X	X	X
ARROWHEAD LENGTH = n.nnnn	X	X	X	X	X		S
TEXT-DIMENSION = n.nnnn (text dimension distance)	X	X	X	X	X		
EXTENSION-POINT = n.nnnn (extension offset distance)		X		X		X	S
EXTENSION-DIMENSION = n.nnnn (extension dimension distance)		X		X		X	

1. Refer to table 1-1, Code Numbers of Entity Types, in the ICEM Design/Drafting Data Management manual for a description of entity type numbers.

After you select the drafting entity, the system displays:

1.CHARACTER SIZE	= n.nnnn	Enter the new character size.
2.DRAFTING SCALE	= n.nnnn	Enter the new drafting scale factor. This factor affects the character size, arrowhead size, text dimension distance, extension offset distance, and extension dimension distance.
3.SPACING RATIO	= n.nnnn	Enter the new character spacing ratio.
4.ASPECT RATIO	= n.nnnn	Enter the new character width-to-height ratio.
5.DOWNSPACE RATIO	= n.nnnn	Enter the new character downspace ratio.
6.TOLERANCE RATIO	= n.nnnn	Enter the new tolerance character to standard character size ratio.
7.ARROWHEAD LENGTH	= n.nnnn	Enter the new arrowhead length.
8.TEXT-DIMENSION	= n.nnnn	Enter the new text to dimension line distance.
9.EXTENSION-POINT	= n.nnnn	Enter the new entity to extension line distance.
10.EXTENSION-DIMENSION	= n.nnnn	Enter the new distance the extension line extends beyond the dimension line.

The system redefines the entity and displays it with the new values.

2

16.13.5.2 Multiple Entities One Value

The system allows you to select one or more entities for which you want to alter the same display value.

First, select the display value you want to alter.

DISPLAY VALUE

- 1. CHARACTER SIZE
- 2. DRAFTING SCALE
- 3. SPACING RATIO
- 4. ASPECT RATIO
- 5. DOWNSPACE RATIO
- 6. TOLERANCE RATIO
- 7. ARROWHEAD LENGTH
- 8. TEXT-DIMENSION
- 9. EXTENSION-POINT
- 10. EXTENSION-DIMENSION

Enter the number corresponding to the display value you want to change.



The system prompts you to enter the display value.

xxxxxxxxxx = n.nnnn

Enter the new display value to be used in the selected entities. The current modal value is the default value. (The x's represent the display value you chose above.)

Then the system requests that you select the entities you want to modify.

INDICATE DRAFTING ENTITY

Use the graphics cursor to screen select the entities you want to modify.

If you want to use another method of selection, enter E or CTRL-E to receive the following menu:

ENTITY SELECTION

- 1. SCREEN SELECT
- 2. CHAIN
- 3. REGION IN
- ⋮
- 4. All

Use the Entity Selection menu to select the entities you want to modify. For more information, refer to the ICEM Design/Drafting Introduction and System Controls manual.

The system defines all selected entities with the new display value.

16.13.5.3 Multiple Entities Multiple Values

The system allows you to select one or more entities for which you want to alter one or more display values.

First, select the display value(s) you want to alter.

- DISPLAY VALUE**
1. CHARACTER SIZE
 2. DRAFTING SCALE
 3. SPACING RATIO
 4. ASPECT RATIO
 5. DOWNSPACE RATIO
 6. TOLERANCE RATIO
 7. ARROWHEAD LENGTH
 8. TEXT-DIMENSION
 9. EXTENSION-POINT
 10. EXTENSION-DIMENSION

Enter the number corresponding to the display value you want to change.

The system prompts you to enter the display value.

xxxxxxxxxx = n.nnnn

Enter the new display value to be used in the selected entities. The current modal value is the default value. (The x's represent the display value you chose above.)

The system returns to the Display Value menu where you can pick another display value to modify. When you have modified all the values you want, enter] at the Display Value menu.

Then the system requests that you select the entities you want to modify.

INDICATE DRAFTING ENTITY

Use the graphics cursor to screen select the entities you want to modify.

If you want to use another method of selection, enter E or CTRL-E to receive the following menu.

- ENTITY SELECTION**
1. SCREEN SELECT
 2. CHAIN
 3. REGION IN
 - :

Use the Entity Selection menu to select the entities you want to modify. For more information, refer to the ICEM Design/Drafting Introduction and System Controls manual.

The system defines all selected entities with the new display values.

16.13.6 Modify Text

With this choice, you can change the text in a note, dimension, or label.

INDICATE DRAFTING ENTITY

Select the text you want to modify.

Select the type of change from the menu displayed:

TEXT MODIFICATION TYPE

- 1.DELETE LINE
- 2.INSERT LINE
- 3.REPLACE STRING
- 4.ADD PREFIX TEXT (1982 ANSI) OR ADD SUFFIX TEXT (1973 ANSI)
- 5.REMOVE PREFIX TEXT (1982 ANSI) OR REMOVE SUFFIX TEXT (1973 ANSI)

Enter:

- 1 To delete an entire line of text. The text becomes user-entered, even if it was previously automatic.

ENTER LINE NUMBER

Enter the number of the line you want to delete.

- 2 To insert a new line of text. The text becomes user-entered, even if it was previously automatic.

ENTER LINE NUMBER

Enter the number of the line after which you want to insert new text.
Enter 0 to insert a new first line.

ENTER NEW STRING

Enter the new string.

- 3 To replace a part of a line. You can use this menu selection to add any symbol to dimension text. The text becomes user-entered, even if it was previously automatic.

ENTER LINE NUMBER

Enter the number of the line where you want to replace a string of characters.

ENTER OLD STRING

Enter the string you want to replace.
The system searches the line selected for this string of characters and deletes it.

ENTER NEW STRING

Enter the new string. The system replaces the deleted string with the new string.

- 4 To add certain special symbols to the automatic dimension text. Adding these symbols does not affect the existing automatic status of the text.

ADD PREFIX TEXT (1982 ANSI) or
 ADD SUFFIX TEXT (1973 ANSI)
 1.DIAMETER SYMBOL
 2.RADIUS SYMBOL
 3.SPHERICAL DIAMETER SYMBOL
 4.SPHERICAL RADIUS SYMBOL
 5.SQUARE SYMBOL
 6.REPETITIVE FEATURES TEXT

Enter:

- 1 To add a diameter symbol to the beginning or end of the text (depending on the drafting standard in effect).
- 2 To add a radius symbol to the beginning or end of the text.
- 3 To add a spherical diameter symbol to the beginning or end of the text.
- 4 To add a spherical radius symbol to the beginning or end of the text.
- 5 To add a square symbol to the beginning of the text (1982 ANSI only).
- 6 To add repetitive features text to the beginning of the text (1982 ANSI only).

If the dimension text is user-entered, the system displays:

CANNOT ADD SYMBOL TO KEY-IN TEXT

If you select any of the symbols represented by menu choices 1-5 above and one of those symbols is already present in the text, the system displays:

SYMBOL INCOMPATIBLE WITH EXISTING TEXT

If you choose a symbol which does not make sense for a particular dimension (e.g., the square symbol for a diameter dimension), the system displays:

SYMBOL INCOMPATIBLE WITH THIS DIMENSION

Enter] or [to continue in each case.

You may use 3.REPLACE STRING from the Modify Text menu to add any symbol to dimension text; however, you must key in the text.

If you choose 6.REPETITIVE FEATURES, the system displays:

NO. TIMES = 1

Enter the number of times the feature occurs. If you accept the default value of 1, the repetitive feature text is not added.

If you enter 0 or a negative number, the system displays:

NUMBER MUST BE A POSITIVE INTEGER

Enter] or [to continue.

- 5 To remove certain special symbols from the automatic dimension text. Removing these symbols does not affect the existing automatic status of the dimension text.

REMOVE PREFIX TEXT (1982 ANSI) or
REMOVE SUFFIX TEXT (1973 ANSI)
1.DIAMETER SYMBOL
2.RADIUS SYMBOL
3.SPHERICAL DIAMETER SYMBOL
4.SPHERICAL RADIUS SYMBOL
5.SQUARE SYMBOL
6.REPETITIVE FEATURES TEXT

Enter:

- 1 To remove the diameter symbol from the text.
- 2 To remove the radius symbol from the text.
- 3 To remove the spherical diameter symbol from the text.
- 4 To remove the spherical radius symbol from the text.
- 5 To remove the square symbol from the text.
- 6 To remove repetitive features text from the dimension text.

If the dimension text is user-entered, the system displays:

CANNOT REMOVE SYMBOL FROM KEY-IN TEXT

If the symbol you choose for removal is not contained in the text, the system displays:

SYMBOL NOT FOUND IN TEXT

Enter] or [to continue in both cases.

16.13.7 Modify Slant Status

With this choice, you can change the slant status of the characters in a note, label, or dimension.

INDICATE DRAFTING ENTITY

Select the text you want to modify.

The text selected is rewritten with a slant if it is normal or rewritten normally if it is slanted.

16.13.8 Modify Angle

With this choice, you can change the angle at which the note text is written.

INDICATE DRAFTING ENTITY

Select the text you want to modify.

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.

16.13.9 Delete Tolerance or Limits

With this choice, you can delete an existing tolerance or limit in a dimension.

INDICATE DRAFTING ENTITY

Select the text you want to modify.

The system deletes the tolerance or limit information and redisplays the selected entity.

16.13.10 Extension Line Status

With this choice, you can change the display status of an extension line.

INDICATE DIMENSION

Use the graphics cursor to select a horizontal, vertical, parallel, or angular dimension.

INDICATE SIDE

Use the graphics cursor to select the extension line on one side of the dimension.

The system turns off the display of the selected extension line if it is currently on or turns on the display if it is currently off. The system redisplays the modified dimension.

16.13.11 Dimension Line Status

With this choice, you can change the display status of a dimension line.

INDICATE DIMENSION

Use the graphics cursor to select a horizontal, vertical, parallel, or angular dimension.

INDICATE SIDE

Use the graphics cursor to select the dimension line on one side of the dimension text.

The system turns off the display of the dimension line and associated arrowhead if it is currently on, or turns on the display if it is currently off. The system redisplays the modified dimension.

16.13.12 Character Set

With this choice, you can change the character set used in the display of a drafting entity.

INDICATE DRAFTING ENTITY

Select the text you want to modify.

If you use a character set that you created, the system displays:

CHARACTER SET NAME cccc

(The letters cccc represent the 4-character set name.)

The system prompts you for the character set you want to use.

NEW CHARACTER SET

- 1. STANDARD
- 2. USER-GENERATED

Enter:

- 1 To use the standard character set to display the characters of the selected text. The system redisplays the entity with standard characters.
- 2 To use a user-generated character set to display the characters of the selected text.

If you select 2.USER-GENERATED, the system prompts you for the character set.

ENTER CHARACTER SET NAME

Enter the name of the user-generated character set.

If the system finds the character set, the system redisplays the drafting entity with the requested character set. If the character set is not found, the system displays:

CHARACTER SET cccc NOT FOUND

(The letters cccc represent the name of the character set.)

The system then returns to the preceding prompt.

16.13.13 Balloon Modification

With this choice, you can modify a balloon (created with 16.4 BALLOON) by changing its origin, changing its radius, replacing its text, adding or removing an adjacent balloon circle, or moving its arrow leader.

BALLOON MODIFICATION

- 1.MODIFY ORIGIN
- 2.CHANGE RADIUS
- 3.REPLACE TEXT
- 4.ADD ADJACENT BALLOON
- 5.REMOVE ADJACENT BALLOON
- 6.CHANGE REFERENCE POINT

Enter:

- 1 To move the center position of the balloon circle by responding to the following prompts:

INDICATE BALLOON

Select the balloon you want to modify.

INDICATE NEW SYMBOL
CENTER

Use the graphics cursor to select the new center for the balloon circle. If the selected balloon has more than one symbol circle, the position you indicate becomes the new center of the first symbol created.

- 2 To change the radius of an existing balloon. The radius change affects all circles included in the selected balloon.

INDICATE BALLOON

Select the balloon you want to modify.

RADIUS = n.nnnn

Enter the desired radius of the balloon.

- 3 To replace the text in the balloon.

INDICATE BALLOON

Select the balloon you want to modify.

If the balloon you select has more than one symbol circle, the system asks which circle's text you want to change:

INDICATE BALLOON CIRCLE

Select the circle within the balloon whose text you want to replace.

ENTER DETAIL NUMBER
ENTER SHEET NUMBER

Enter the detail number (up to nine characters) and the sheet number (up to nine characters) that are to be written in the balloon. If you want only one number, enter] for the sheet number.

4 To add an adjacent balloon circle to an existing balloon.

INDICATE BALLOON

Select an existing balloon to which you want to add an adjacent balloon circle.

ADJACENT SYMBOL PLACEMENT

- 1.TO THE RIGHT
- 2.TO THE LEFT
- 3.ABOVE
- 4.BELOW

Enter the number corresponding to the side of the existing single-circle balloon where the adjacent circle is to be added.

The radius of the added symbol is the same as the original circle.

If the selected balloon already has more than one symbol circle, the added symbol is placed adjacent to the last added circle in the same direction. In this case, the ADJACENT SYMBOL PLACEMENT prompt does not appear. The system proceeds with the following prompts:

ENTER DETAIL NUMBER

Enter the detail number (up to nine characters) and the sheet number (up to nine characters) to be written in the balloon. If only one number is desired, enter] for the sheet number.

ENTER SHEET NUMBER

5 To remove a symbol circle from a multiple-circled balloon.

INDICATE BALLOON

Select the balloon you want to modify.

If the balloon selected has only one symbol circle, the system responds with

CANNOT REMOVE SINGLE SYMBOL

and you return to the Balloon Modification menu.

If the balloon has more than one symbol circle, you receive the following prompt:

INDICATE SYMBOL CIRCLE

Select the symbol circle you want to remove.

The system removes the indicated symbol circle and then repositions other circles in the balloon if necessary to maintain a continuous string of symbol circles.

6 To move the arrow leader so it points to a different position.

INDICATE ENTITY

Select an entity near the position where you want the arrow leader to point.

16.13.14 Datum Target Modification

With this menu choice, you can modify a datum target symbol (created with 16.5 DATUM TARGET) by changing its origin, changing its radius, replacing its target number, or adding or removing an adjacent target symbol.

DATUM TARGET MODIFICATION
 1.MODIFY ORIGIN
 2.CHANGE RADIUS
 3.REPLACE TARGET NUMBER
 4.ADD ADJACENT SYMBOL
 5.REMOVE ADJACENT SYMBOL

Enter:

- 1 To move the center position of the datum target symbol circle.

INDICATE DATUM TARGET SYMBOL

Select the datum target symbol you want to modify.

INDICATE NEW SYMBOL CENTER

Use the graphics cursor to select the new center for the datum target symbol. If the datum target symbol you select has more than one symbol circle, the position you indicate becomes the new center of the first symbol created.

- 2 To change the radius of an existing datum target symbol. The radius change affects all circles included in the selected datum target symbol.

INDICATE DATUM TARGET SYMBOL

Select the datum target symbol you want to modify.

RADIUS = n.nnnn

Enter the desired radius of the datum target symbol.

- 3 To replace the target number of the datum target symbol.

INDICATE DATUM TARGET SYMBOL

Select the datum target symbol you want to modify.

If the selected datum target symbol has more than one symbol circle, the system asks which circle's target number you want to change:

INDICATE SYMBOL CIRCLE

Select the circle within the datum target symbol whose target number you want to replace.

ENTER TARGET IDENTIFIER

Enter a letter identifying the datum, followed by a target number.

4 To add an adjacent datum target symbol circle to an existing datum target symbol.

INDICATE DATUM TARGET SYMBOL

Select the datum target symbol you want to modify.

ADJACENT SYMBOL PLACEMENT

- 1. TO THE RIGHT
- 2. TO THE LEFT
- 3. ABOVE
- 4. BELOW

Enter the number corresponding to the side of the existing single-circle symbol where the adjacent symbol will be added.

The radius of the added symbol is the same as the original circle.

If the selected datum target symbol already has more than one symbol circle, the added symbol is placed adjacent to the last added circle in the same direction. In this case, the ADJACENT SYMBOL PLACEMENT prompt does not appear.

ENTER TARGET IDENTIFIER

Enter a letter identifying the datum, followed by a target number.

5 To remove a symbol circle from a multiple-circled datum target symbol.

INDICATE DATUM TARGET SYMBOL

Select the datum target symbol you want to modify.

If the datum target symbol you select has only one symbol circle, the system responds with:

CANNOT REMOVE SINGLE SYMBOL

and returns you to the Datum Target Modification menu.

INDICATE SYMBOL CIRCLE

Select the symbol circle you want to remove.

The system removes the indicated symbol circle and then repositions other circles in the datum target symbol if necessary to maintain a continuous string of symbol circles.

16.13.15 Feature Frame Modification

With this choice, you can replace, add, or remove a single frame section from a geometric tolerance (16.7.1), composite geometric tolerance (16.7.2), or datum feature (16.6) symbol. A frame section is the portion of the symbol enclosed by two consecutive vertical bars.

FEATURE FRAME MODIFICATION
 1.REPLACE FRAME SECTION
 2.ADD FRAME SECTION
 3.REMOVE FRAME SECTION

Enter:

- 1 To replace an existing frame section.

INDICATE FEATURE FRAME

Select the datum feature, geometric tolerance, or composite geometric tolerance symbol you want to modify.

If the selected feature frame is a datum feature symbol, the system responds:

ENTER NEW TEXT

Enter the text to replace the existing text within the datum feature symbol.

If the selected feature frame is a geometric tolerance or composite geometric tolerance symbol with more than one frame section, the system responds:

INDICATE FRAME SECTION

Select the individual frame section you want to replace within the geometric tolerance or composite geometric tolerance symbol.

If the indicated frame section contains a geometric characteristic symbol, the system responds:

GEOMETRIC CHARACTERISTIC

- 1.Straightness
- 2.Flatness
- 3.Circularity
- 4.Cylindricity
- 5.Profile of a Line
- 6.Profile of a Surface
- 7.Angularity
- 8.Perpendicularity
- 9.Parallelism
- 10.Position
- 11.Concentricity
- 12.Circular Runout
- 13.Total Runout
- 14.Symmetry

Enter the number of the symbol to replace the existing symbol.

Note that the geometric characteristic symbol for symmetry is not allowed in 1982 ANSI drafting standards. In such a case, menu item number 14 is blanked.

If the indicated frame section contains a datum, the system responds:

ENTER NEW DATUM REFERENCE

Enter a datum reference consisting of 1 to 20 alphanumeric characters.

DATUM MODIFIER

1.MMC

2.LMC

3.RFS

4.PROJECTED TOLERANCE ZONE

Choose the number of the desired datum modifier. Enter:

- 1 To select maximum material condition.
- 2 To select least material condition.
- 3 To select regardless of feature size.
- 4 To select projected tolerance zone.
-] To omit the symbol.

If the indicated frame section contains a tolerance, the system responds:

ENTER NEW TOLERANCE

Enter a 1- to 20-character tolerance. Use \O for the diameter symbol. (See section 16.1.1.5 SPECIAL SET PREFIX for information about other symbols.)

TOLERANCE MODIFIER

1.MMC

2.LMC

3.RFS

4.PROJECTED TOLERANCE ZONE

Choose the number of the desired tolerance modifier. Enter:

- 1 To select maximum material condition (MMC).
- 2 To select least material condition (LMC).
- 3 To select regardless of feature size (RFS).
- 4 To select projected tolerance zone.
-] To omit the symbol.

- 2 To add a datum frame section to an existing feature frame. The system places the added frame section at the end of the datum frame section(s) that are currently displayed in the entity.

INDICATE FEATURE FRAME

Select the geometric tolerance or composite geometric tolerance symbol you want to modify.

If the feature frame you selected is a datum feature symbol, the system responds:

CANNOT ADD TO A DATUM FEATURE

and returns you to the Feature Frame Modification menu.

If the feature frame you selected is a composite geometric tolerance symbol, the system asks whether you want to add the datum to the top or bottom frames.

COMPOSITE GEOMETRIC TOLERANCE

- 1.ADD TO TOP
2.ADD TO BOTTOM

Enter:

- 1 To add a datum to the top frame sections.
- 2 To add a datum to the bottom frame sections.

If the feature frame you selected has six datum and tolerance frames, the system responds:

CANNOT EXCEED 6 DATUM AND TOLERANCE FRAME SECTIONS

and returns you to the Feature Frame Modification menu.

ENTER DATUM REFERENCE

Enter a datum reference consisting of 1 to 20 alphanumeric characters.

DATUM MODIFIER

- 1.MMC
2.LMC
3.RFS
4.PROJECTED TOLERANCE ZONE

Choose the number of the desired datum modifier. Enter:

- 1 To select maximum material condition.
- 2 To select least material condition.
- 3 To select regardless of feature size.
- 4 To select projected tolerance zone.
-] To omit the symbol.

3 To remove a single frame section.

INDICATE FEATURE FRAME

Select the geometric tolerance or composite geometric tolerance symbol containing the frame you want to remove.

Enter [to return to the Feature Frame Modification menu.

Enter] to return to the Modify Drafting Entity menu.

If the selected feature frame is a datum feature symbol or a geometric tolerance symbol with only one frame section, the system responds:

CANNOT REMOVE SINGLE FRAME SECTION

and returns you to the Feature Frame Modification menu.

If you wish to delete this entity, use ICEM Design/Drafting menu choice 3.DELETE.

INDICATE FRAME SECTION
TO BE REMOVED

Select the frame section you want to remove from the geometric tolerance or composite geometric tolerance symbol.

The system redisplays the geometric tolerance or composite geometric tolerance symbol with the indicated frame section removed.

If you try to remove a top or bottom frame of a composite geometric tolerance symbol that contains a single top or bottom frame, the system responds:

CANNOT REMOVE SINGLE TOP
OR BOTTOM FRAME SECTION

If you try to remove the geometric characteristic symbol of a composite geometric tolerance, the system responds:

CANNOT REMOVE CHARACTERISTIC SYMBOL
OF A COMPOSITE GEOMETRIC TOLERANCE

In both cases you return to the Feature Frame Modification menu.

16.13.16 Dimension Origin Modification

With this selection you can add, change the side of, or remove a dimension origin from an existing dimension. The dimension origin, graphically represented as a circle (instead of an arrowhead) at the end of a dimension line, can be added or removed from an existing horizontal, vertical, parallel, or thickness dimension.

MODIFY DIMENSION ORIGIN

- 1.ADD
- 2.SWITCH SIDE
- 3.DELETE

Enter:

- 1 To add a dimension origin to the side of the dimension by your response to the following prompts:

INDICATE DIMENSION

Use the graphics cursor to select a dimension.

INDICATE SIDE

Indicate the side of the dimension to which you want to add the origin. Use the graphics cursor to indicate that side of the arrowhead.

The system defines dimension origin on the indicated side and then redisplays the dimension.

If a dimension origin already exists on the selected side, the system responds:

DIMENSION ORIGIN ALREADY EXISTS, CANNOT ADD

and returns to the MODIFY DIMENSION ORIGIN prompt without requesting a side.

- 2 To switch an existing dimension origin from one side of the dimension to the other.

INDICATE DIMENSION

Use the graphics cursor to select a dimension.

The system switches the dimension origin to the opposite side and redisplays the dimension.

If there is not an existing dimension origin, there is no change to the dimension.

- 3 To delete a dimension origin.

INDICATE DIMENSION

Use the graphics cursor to select a dimension.

The system deletes the origin and redisplays the dimension.

If there is not an existing dimension origin, there is no change to the dimension.

16.13.17 Modify Decimal Places

With this choice, you can change the number of decimal places displayed on a dimension or multiple dimensions.

If 16.1.18 DUAL DIMENSIONING is on, the system displays:

DECIMAL PLACES

- 1.PRIMARY = n
- 2.SECONDARY = n

Enter the number of decimal places to be used in the selected entities. The primary value affects the upper text in a dual dimension or the only text in a nondual dimension. The secondary value affects the lower text in a dual dimension but has no effect on a nondual dimension.

If 16.1.18 DUAL DIMENSIONING is off, the system displays:

DECIMAL PLACES = n

Enter the number of decimal places to be used in the selected entities.

When you enter the number of decimal places, the system asks which entities you want to modify.

INDICATE ENTITY

Use the graphics cursor to screen select the entities you want to modify.

If you want to use another method of selection, enter E or CTRL-E to receive the following menu:

ENTITY SELECTION

- 1.SCREEN SELECT
- 2.CHAIN
- 3.REGION IN
- :

Use the Entity Selection menu to select the entities. For more information, refer to the ICEM Design/Drafting Introduction and System Controls manual.

If you are dimensioning in millimeters, this modification has no effect (unless the selected dimension has significant digits to the specified number of places). The system does not display trailing zeros for metric units, and tolerances and limits are not affected.

If you are dimensioning in inches, the decimal places have an effect on both the main dimension and the tolerance to ensure that the dimension conforms to standards.

If the dimension is user-entered, the following error message appears to inform you that one or more of the selected dimensions could not be updated:

**DECIMAL PLACES NOT MODIFIED
FOR USER-ENTERED DIMENSION(S)**

16.13.18 Dimension Not to Scale

With this choice, you can add or delete a line under a dimension. A line drawn under a dimension indicates the dimension is not to scale.

INDICATE DIMENSION

Use the graphics cursor to select a dimension.

If the dimension selected is a label, the system responds:

LABELS ARE NOT SELECTABLE

and returns to the INDICATE DIMENSION prompt.

If there is a line under the dimension, the system deletes the line.

If there is not a line under the dimension, the system draws a line.

16.14 Detail Magnify

With this choice, you can produce a magnified drawing of an area in a circle. The entities you can magnify are points, lines, arcs, conics, splines, point-sets, rectangular arrays, circular arrays, centerlines, and section lining. Automatic dimensioning performed on magnified entities reflects the size of the original, unmagnified entities.

Indicate whether the area to be magnified is within an existing circle or whether you must construct a circle.

EXISTING CIRCLE FOR REGION

Enter:

TO MAGNIFY?

Y To use an existing circle.

N To create a circle around the area you want to magnify.

If you select Y, the system asks you to indicate the existing circle.

INDICATE ARC OR CIRCLE

Use the graphics cursor to select an existing circle.

If you select N, the system asks you to indicate positions for the center and edge of an imaginary circle enclosing the area to be magnified.

INDICATE CENTER

Use the graphics cursor to select the center position of the circle to be created.

INDICATE EDGE

Use the graphics cursor to select a position on the edge of the circle to be created.

Now indicate the location and size (optional) of the circle in which the magnified reproduction is to be drawn. Select the method to enter the location and size of the circle.

REGION FOR MAGNIFICATION DISPLAY

1. SELECT EXISTING CIRCLE
2. INDICATE CENTER AND EDGE POSITIONS
3. INDICATE CENTER AND ENTER MAG

Enter:

- 1 To select an existing circle.

INDICATE ARC OR CIRCLE

Use the graphics cursor to select an existing circle.

SELECTION OK?

Enter:

- Y To indicate the selected circle is correct.

- N To indicate the selected circle is incorrect. The system repeats the Region for Magnification Display menu so you can select a different circle.

- 2 To create a new circle by indicating the center and edge positions.

The system asks you about a border for the region, and asks you to indicate the center and size of the circle.

DO YOU WANT THE REGION BORDERED?

Enter:

- Y To draw a border around the magnified drawing.

- N To not draw a border.

INDICATE CENTER

Use the graphics cursor to select the center position of the circle.

INDICATE EDGE

Specify the size of the circle by using the graphics cursor to select a position on the edge.

SELECTION OK?

Enter:

- Y To indicate that the region for magnification is satisfactory.

- N To return to the Region for Magnification Display menu.

- 3 To create a new circle by indicating both the center position and the magnification size (which determines the size of the circle).

The system asks you whether to border the drawing, the location of the center, and the magnification factor. If you ask for a border, the system creates a circle large enough to contain the magnified drawing.

DO YOU WANT THE REGION
BORDERED?

Enter:

Y To draw a border around the magnified drawing.

N To not draw a border.

INDICATE CENTER

Use the graphics cursor to select the center position of the circle.

MAGNIFICATION = n.nn

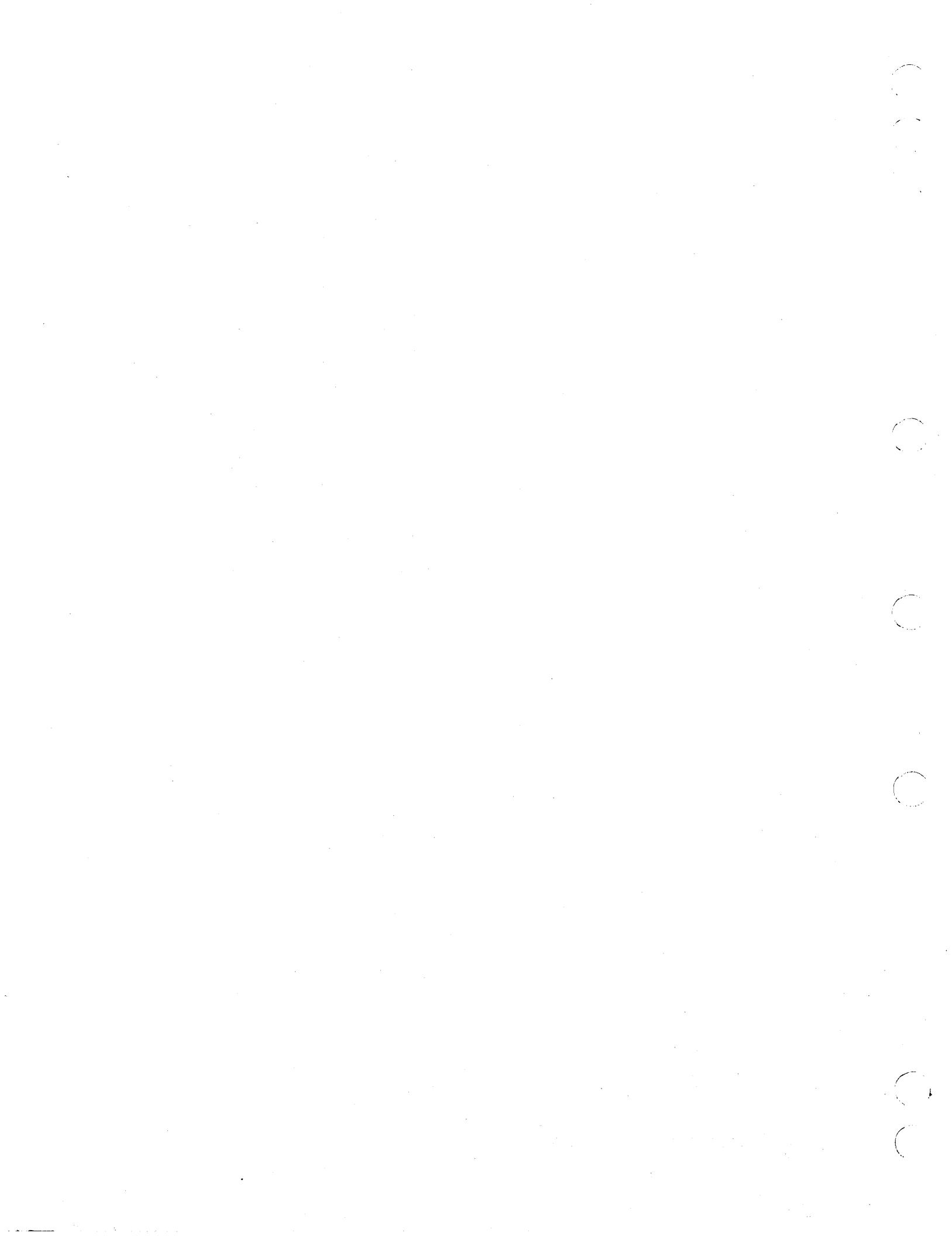
Enter the magnification factor desired. The size of the new drawing is the size of the old drawing magnified by this factor. For example, a magnification factor of 1 causes no magnification or reduction; a factor of 2 causes a drawing twice as large as the original, and so on.

SELECTION OK?

Enter:

Y To indicate that the region for magnification is satisfactory.

N To return to the Region for Magnification Display menu and specify a new circle.



Menu 18: Analysis

2

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Menu 18: Analysis

2

Using Menu 18

You can use this menu to calculate and display the following information about entities in the current part:

- Properties of a two-dimensional spline, including slope and curvature
- Area and perimeter of a two-dimensional, closed figure
- Two-dimensional analysis
- Three-dimensional analysis

The Analysis menu is:

```

ANALYSIS
1.SPLINE ANALYSIS
2.ANALYTIC AREA/PERIMETER
3.2-D SECTION ANALYSIS
4.3-D ANALYSIS

```

The analysis functions give you the following capabilities:

Menu Title	Description
18.1 SPLINE ANALYSIS	Calculates and displays properties of a two-dimensional spline.
18.2 ANALYTIC AREA/PERIMETER	Calculates and displays the area and perimeter of a single closed planar figure.
18.3 2-D SECTION ANALYSIS	Performs analysis on any two-dimensional figure including perimeter, area, center of gravity, axial moments of area, axial moments of inertia, principal moments of inertia, principal axes angles, products of inertia, axial radii of gyration, polar moment of inertia, and polar radius of gyration.
18.4 3-D ANALYSIS	Calculates specific analytical properties of certain three-dimensional figures including surface area, volume, weight, first moment and center of mass, moment of inertia, radius of gyration, spherical moment of inertia, and spherical radius of gyration.

18.1 Spline Analysis

With this choice, you can analyze a two-dimensional spline by displaying various graphs and charts.

The spline being analyzed here is parametrically defined by chord length. For example, if t is the parameter, then a value of t corresponds to a position along the spline. This relationship is denoted by $s(t)$, meaning that the spline is defined and dependent on the parameter t . Figure 2-1 illustrates the parametrization by cumulative chord length.

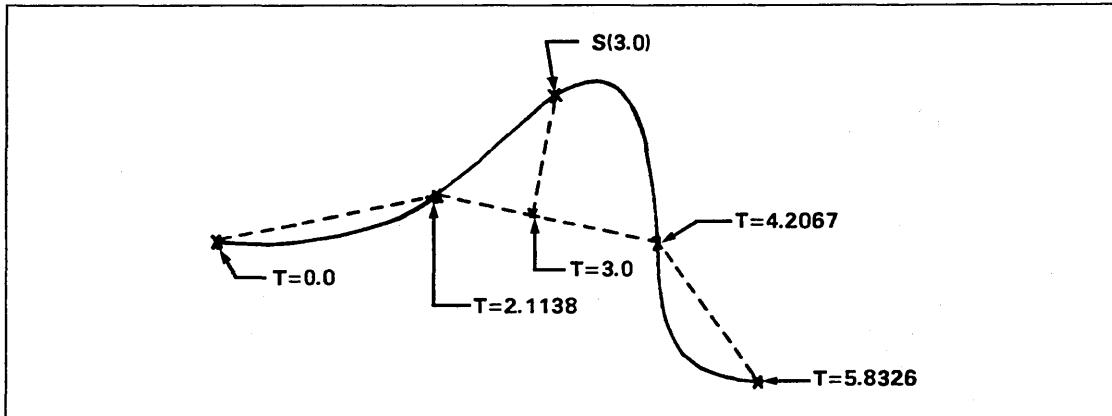


Figure 2-1. Spline with Chord Length Parametrization

Throughout the discussion of the analysis options, the spline in figure 2-2 is used.

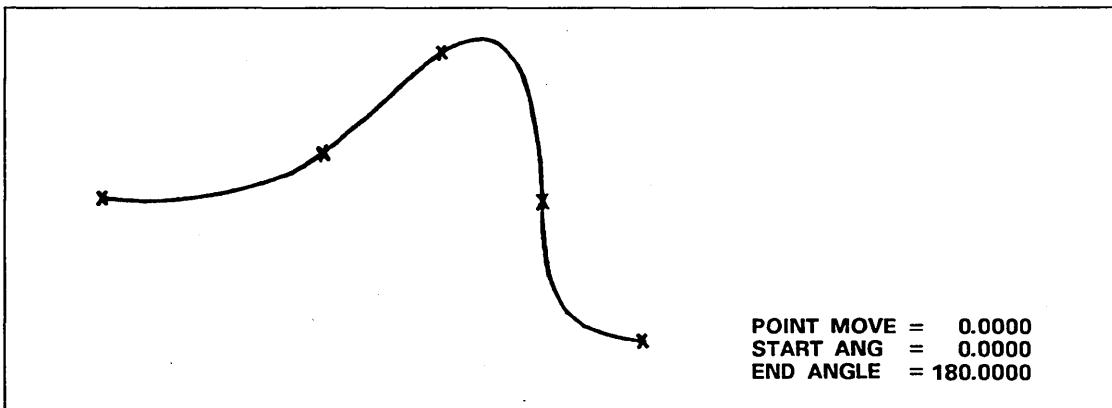


Figure 2-2. Spline Used for Example Analysis

The choices available in menu 18.1 include the following five graphs (menu options 1-5) and five charts (menu option 6).

- slope
- curvature
- radius of curvature
- x transform coordinate
- y transform coordinate

All five graphs describe the properties of the spline as they vary by parameter value. Consequently, the horizontal axis of the graphs is the cumulative chord length parameter.

Entries in the charts are listed by spline points or segments. The spline points are the positions used to define the spline. The segments are the pieces of the curve between the spline points. The segment NUM starts at spline point NUM and ends at spline point NUM + 1. The quantities tabulated in the charts are shown in figures 2-3 and 2-4.

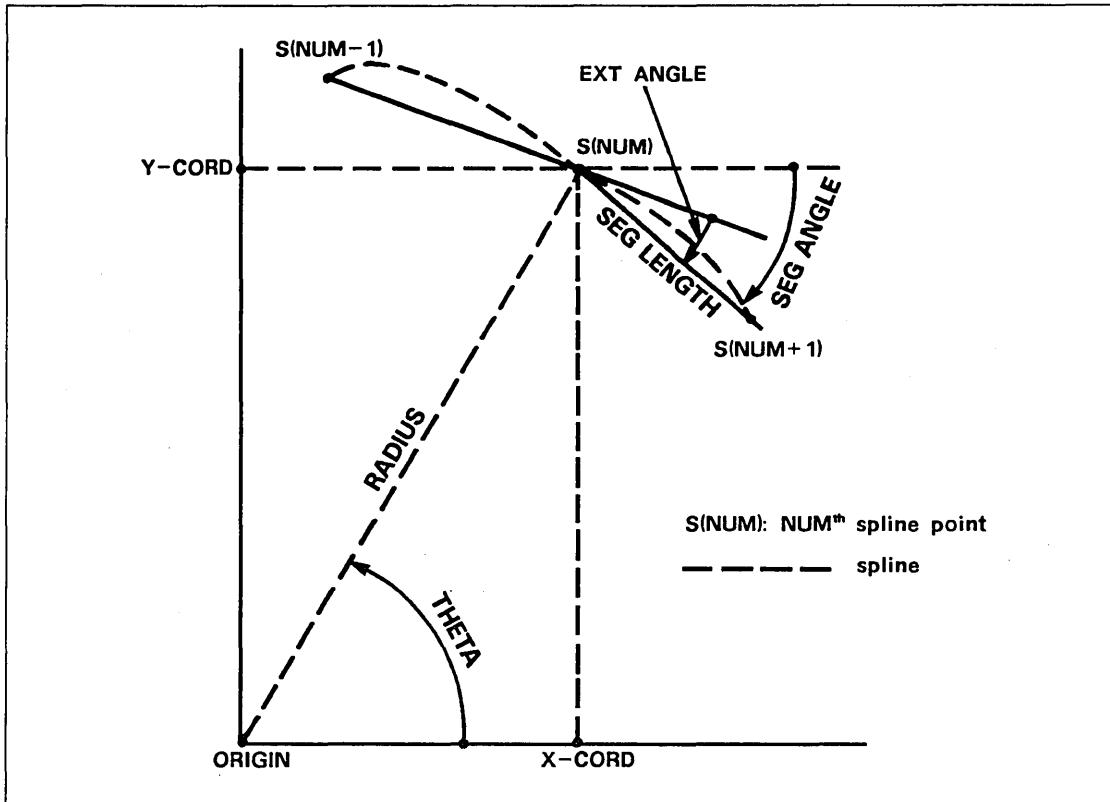


Figure 2-3. Quantities in Charts 1 and 2

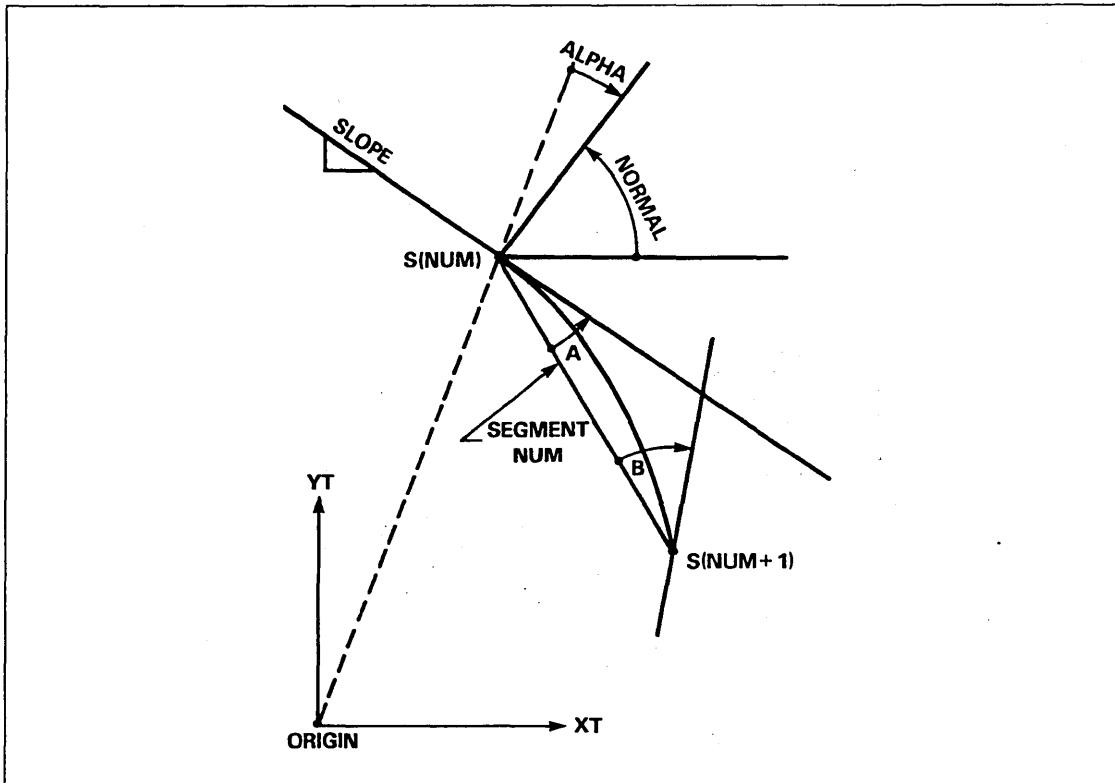


Figure 2-4. Quantities in Charts 2 and 3

INDICATE 2-D SPLINE

Use the graphics cursor to select a spline.

Choose the type of information you want to display.

SELECT ANALYSIS
 1.SLOPE
 2.CURVATURE
 3.RAD OF CURV
 4.XT
 5.YT
 6.EXTENDED ANALYSIS

Enter:

- 1 To display a graph showing the slope of the spline. The current display is replaced by a plot of the spline's parametric range along the horizontal axis of the graph versus the spline's slope along the vertical axis of the graph. The slope is the rise/run at a specific position (parameter) along the spline. When the run is 0 (that is, the spline is at 90° or 270°), the slope is considered infinite. In this case, the plot is broken by a vertical dashed line to represent the infinite slope. (Refer to figures 2-5 and 2-7.)

- 2 To display a graph of the curvature along the spline. The current display is replaced by a plot of the spline's parametric range along the horizontal axis of the graph versus its curvature along the vertical axis of the graph. The curvature is a measure of the change in direction of the spline. The curvature plot reveals any unexpected irregularities, such as inflections, roughness, and bumps. A spline with a generally small curvature (for example, less than 0.1000) is a very smooth curve. The curvature is the inverse of the radius of curvature. (Refer to figure 2-6.)
- 3 To display a graph of the radii of curvature along the spline. The current display is replaced by a plot of the spline's parametric range along the horizontal axis of the graph versus the spline's radius of curvature along the vertical axis of the graph. The radius of curvature at a point on the spline is the radius of the circular arc that has the same curvature as the spline at that point. The radius of curvature is always positive and is infinite at any inflection point on a spline. It is also infinite along any flat portion of a spline. The radius of curvature is the inverse of the curvature. (Refer to figure 2-7.)
- 4 To display a graph of the xt coordinates. The current display is replaced by a plot of the spline's parametric range along the horizontal axis of the graph versus the spline's xt range along the vertical axis of the graph. (Refer to figure 2-8.)
- 5 To display a graph of the yt coordinates. The current display is replaced by a plot of the spline's parametric range along the horizontal axis of the graph versus the spline's yt range along the vertical axis of the graph. (Refer to figure 2-9.)
- 6 To display a list of the mathematical properties of the spline.

1.TOLERANCE FACTOR = n.nnnn

Enter the chord height error tolerance for the spline analysis.

OUTPUT DEVICE

1.CRT

2.LINE PRINTER

Specify whether the results of the analysis are to be sent to the terminal screen (displays four digits after the decimal) or to a local file, TAPE 4, that includes carriage control characters for output to a line printer (displays seven digits after the decimal). When you select 1.CRT, five charts are displayed in succession on the terminal screen.

18.1 Spline Analysis

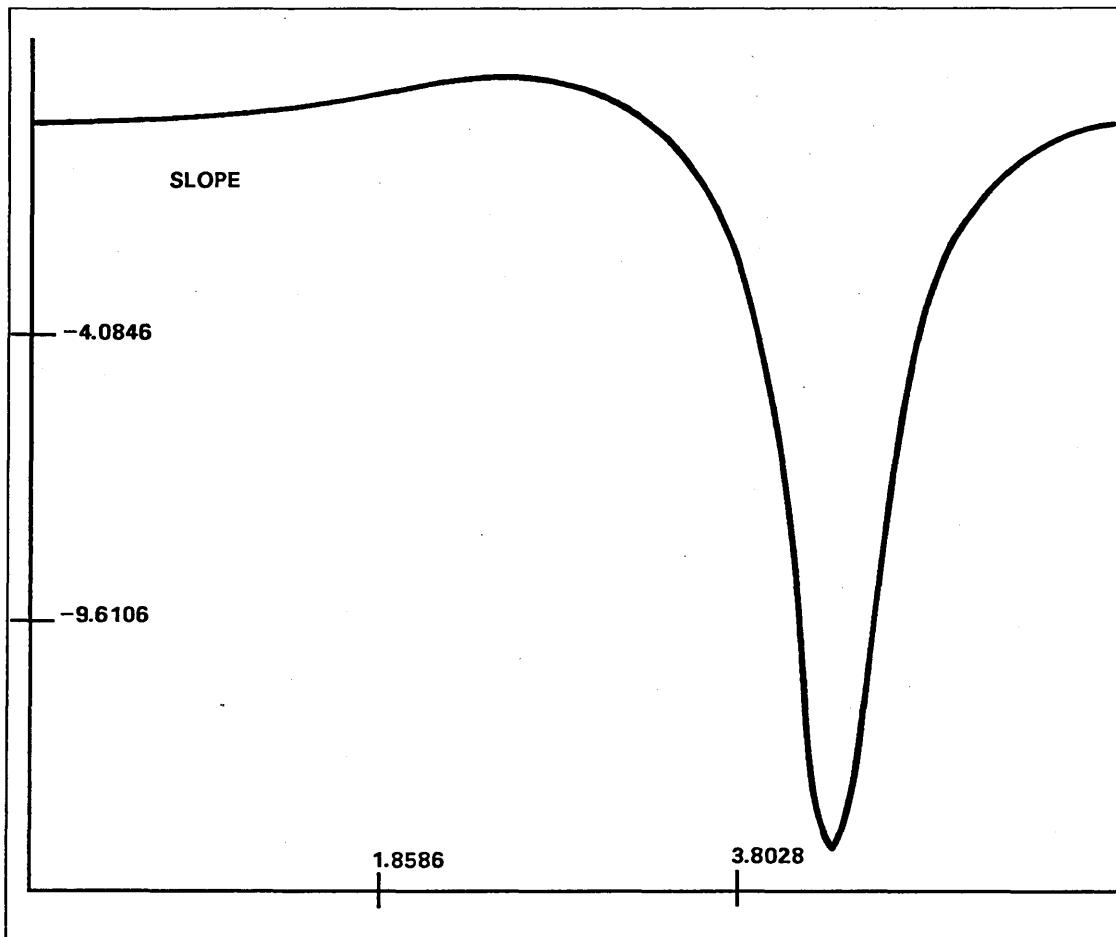


Figure 2-5. Slope of the Spline

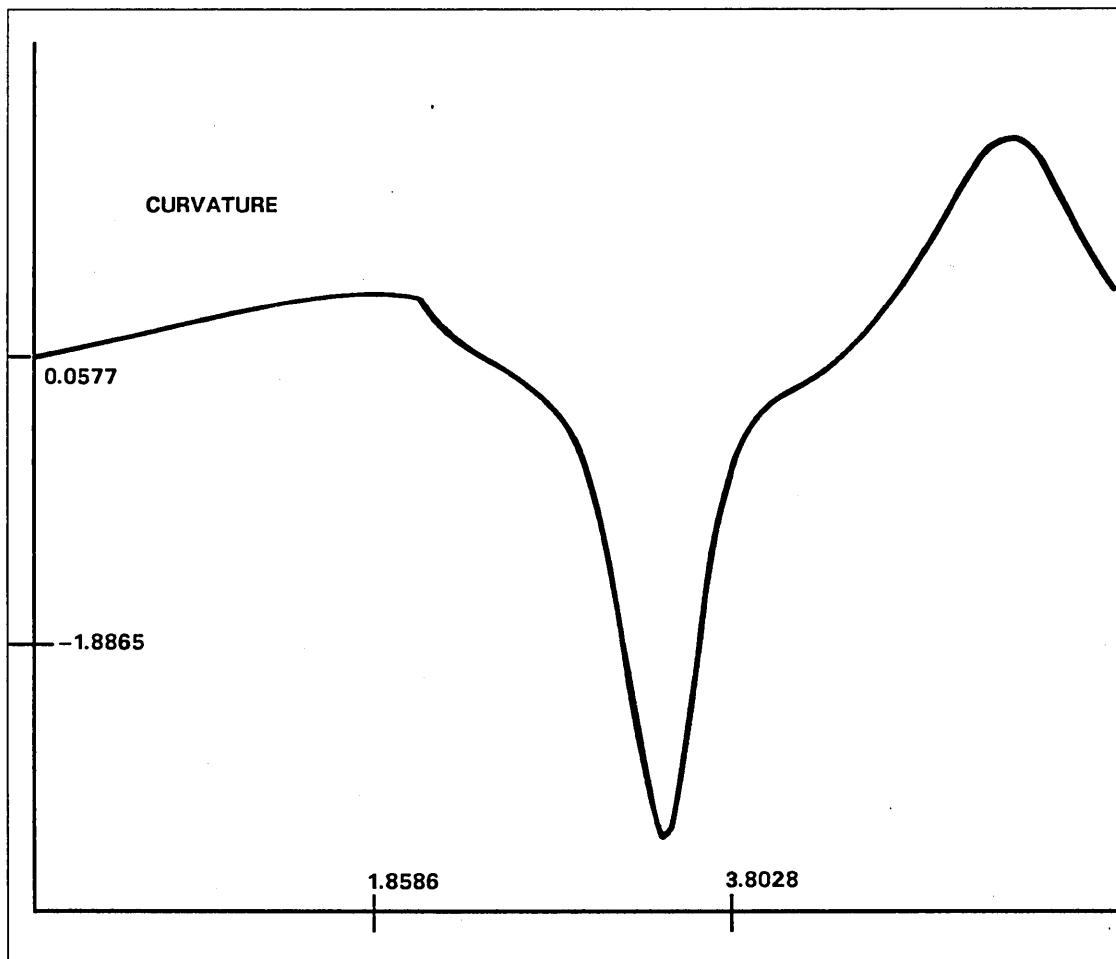


Figure 2-6. Curvature of the Spline

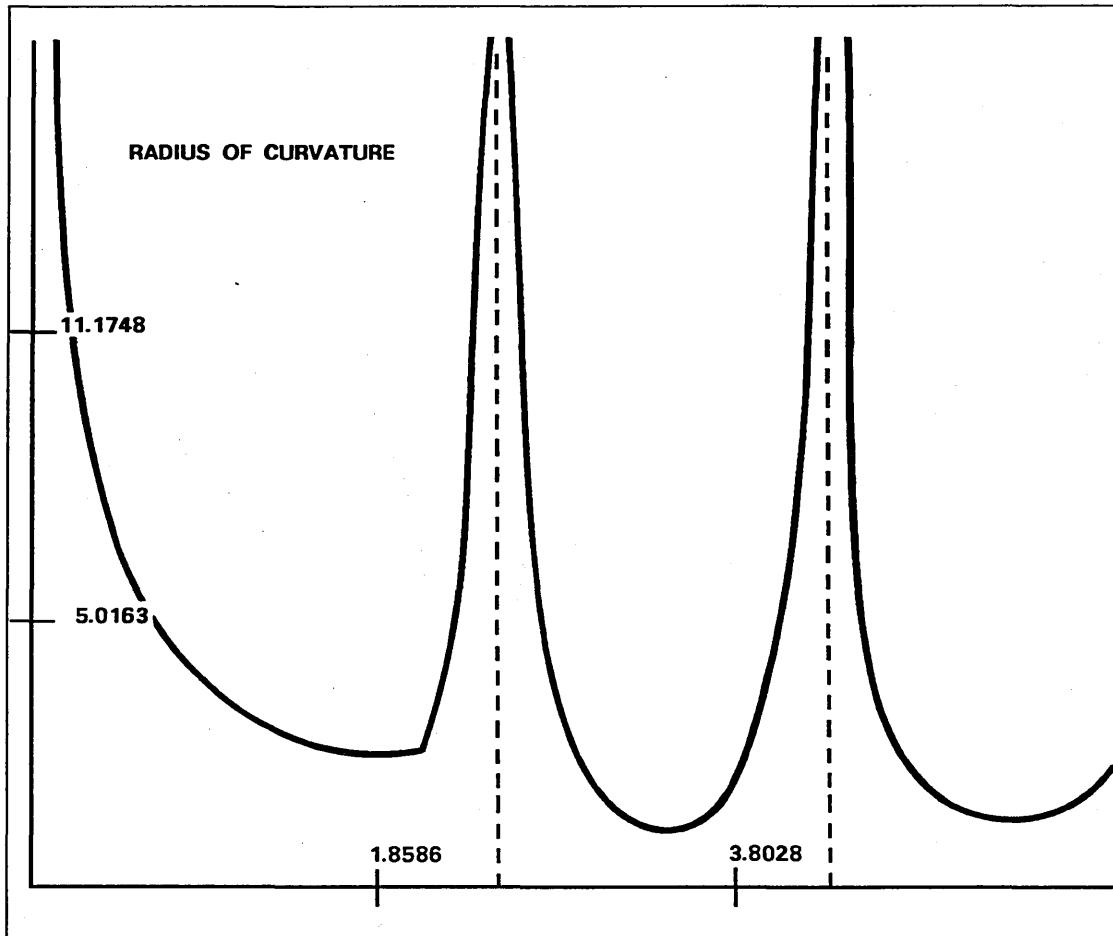


Figure 2-7. Radius of Curvature of the Spline

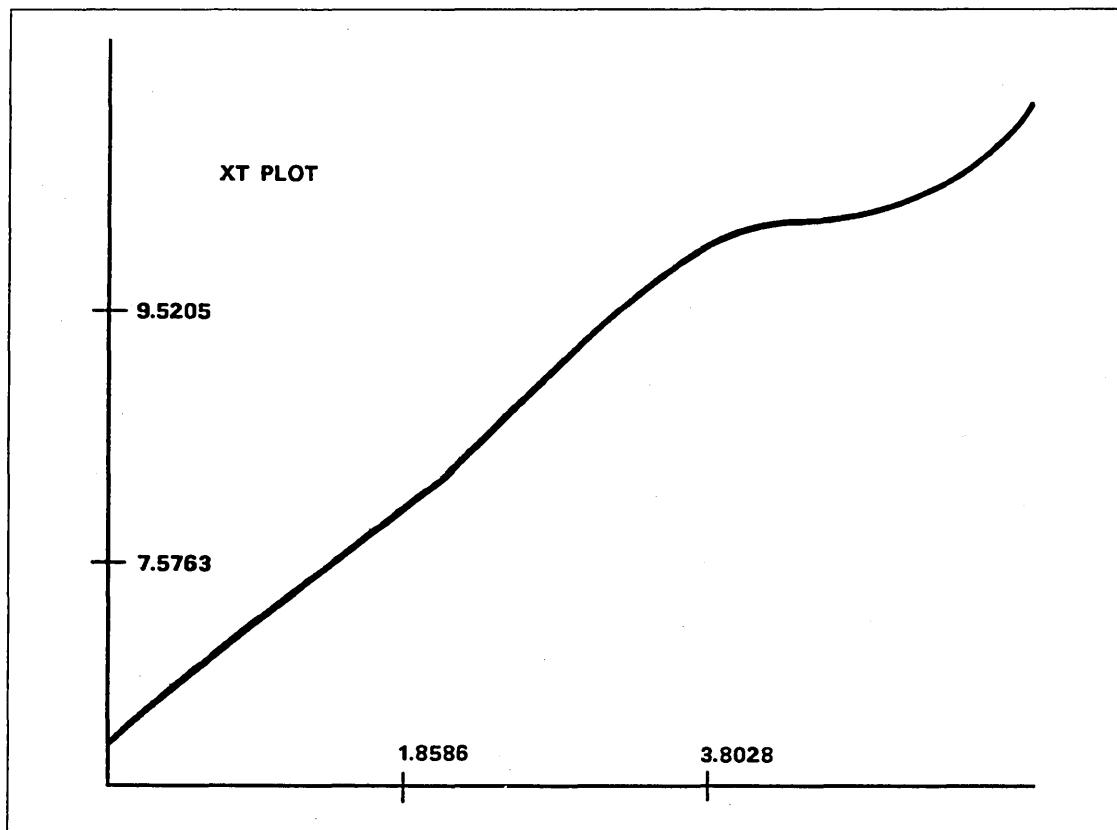


Figure 2-8. X Transform Range of the Spline

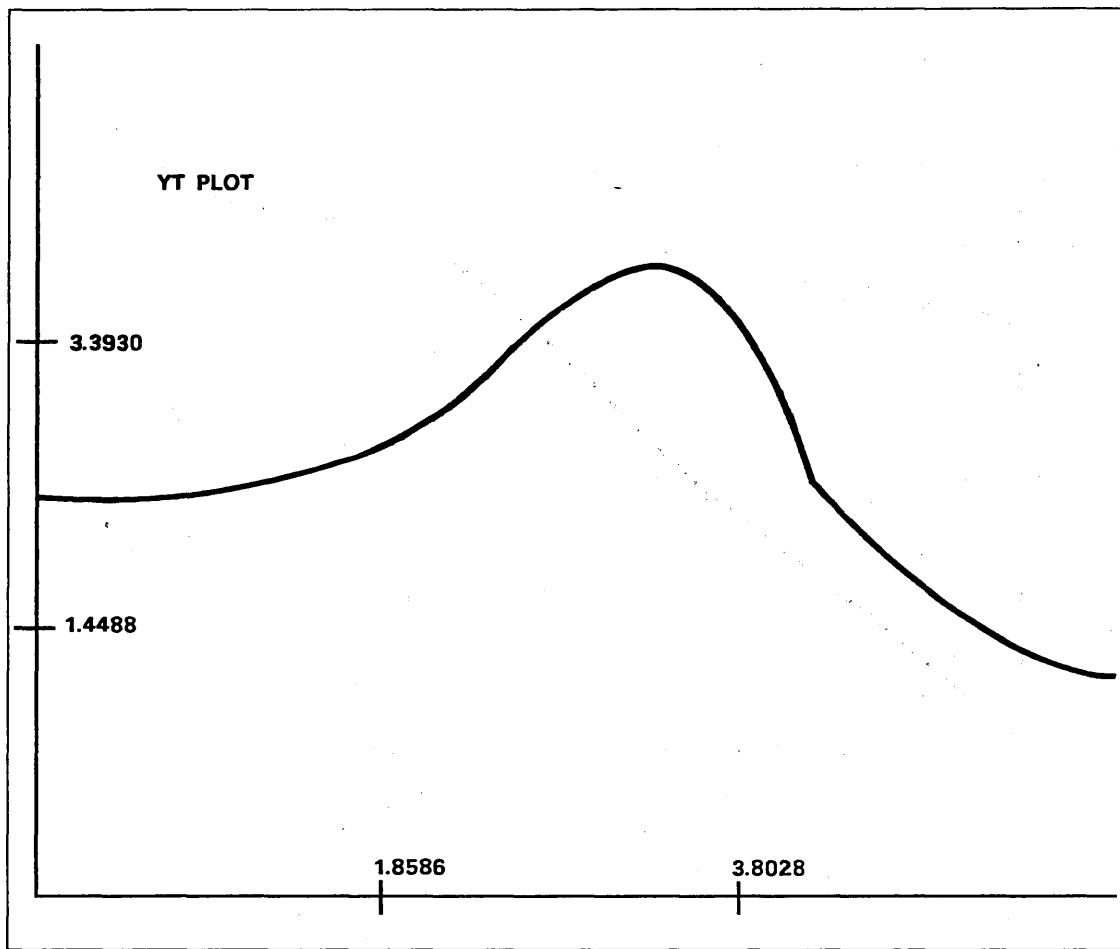


Figure 2-9. Y Transform Range of the Spline

If you select 1.CRT, the system displays:

CONTINUE?

Enter:

Y To display the next chart.

N To end spline analysis.

This prompt appears after each of the five charts is displayed.

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:

XSCL = YSCL?

Enter:

Y To leave the graph unchanged.

N To stretch the graph in the x transform direction and redisplay it to fill the screen.

Figures 2-10 through 2-14 show examples of each of the five charts, in the order displayed. A value of 99999.9999 for any chart entry indicates the measured quantity for that item is greater than or equal to 99999.9999. This may occur for spline characteristics that can have an infinite value, such as slope.

NUM	THETA	RADIUS	X-CORD	Y-CORD
1	20.9297	6.5386	6.1071	2.3357
2	19.0640	8.6381	8.1643	2.8214
3	13.4033	10.5076	10.2214	2.4357
4	5.7226	11.2347	11.1786	1.1214

NUM Point on the spline being analyzed.

THETA Angle from the positive xt-axis of the location of the point with reference to the xt=0, yt=0 origin.

RADIUS Point's distance from the xt=0, yt=0 origin. THETA and RADIUS compose the polar coordinates of the point.

X-CORD Position of the point on the xt-axis.

Y-CORD Position of the point on the yt-axis.

(Figure 2-3 illustrates the quantities given in this chart.)

Figure 2-10. Example and Description of Chart 1

NUM	SEG LENGTH	SEG ANGLE	EXT ANGLE	SLOPE	NORMAL
1	2.1138	13.2841	---	- 0.0000	90.0000
2	2.0929	-10.6195	-23.9036	0.7464	126.7372
3	1.6259	-53.9343	-43.3148	-12.9739	4.4075
4	---	---	---	- 0.0000	90.0000
LENGTH = 5.8326					
NUM	Point on the spline being analyzed.				
SEG LENGTH	Chord length from the current point to the next point on the spline. This is the linear distance, not the curve length along the segment. The SEG LENGTH for the last spline point has no value because there are no subsequent spline points to which to measure.				
SEG ANGLE	Angle (in degrees) that the chord of the spline makes with the positive xt-axis. SEG ANGLE has no value for the last spline point.				
EXT ANGLE	Change in the external angle that the current chord makes with reference to the previous chord segment. EXT ANGLE has no value for the first spline point since there is no previous segment, and it has no value for the last spline point because there is no segment starting at that point. The angle between any three consecutive spline definition points must not be less than 90°. Thus, the external angle value should be between -90° and 90°.				
SLOPE	Tangent of the curve at the current spline point.				
NORMAL	Angle (in degrees) of the normal vector from the current spline point. The normal vector is not directly associated with the current chord angle; rather, it is a function of the tangent vector of the spline at the current point.				
LENGTH	Sum of the segment lengths along the spline. This is the upper parameter of the spline.				
(Figures 2-3 and 2-4 illustrate the quantities given in this chart.)					

Figure 2-11. Example and Description of Chart 2

NUM	ALPHA	TAN ANGLE A	TAN ANGLE B
1	69.0703	-0.2361	0.4338
2	-72.3268	1.0858	-3.7250
3	- 8.9957	-0.6166	1.3731
4	84.2714	---	---

NUM Point on the spline being analyzed.

ALPHA Angle (in degrees) between the polar angle of the start of the chord with respect to the xt,yt origin and the normal of the curve at the current spline point.

TAN ANGLE A Tangent of the angle between the chord starting at point NUM and the tangent vector at the start of the segment.

TAN ANGLE B Tangent of the angle between the chord starting at point NUM and the tangent vector at the end of the segment.

(Figure 2-4 illustrates the quantities given in this chart.)

Figure 2-12. Example and Description of Chart 3

NUM	CURV A	CURV B	DELTA CURV	ENERGY
1	0.0335	0.4814	---	18.3722
2	0.4615	-0.1060	0.0001	864.6981
3	-0.1061	0.5345	-0.0001	191.8355
TOTAL ENERGY = 1074.9058				
NUM	Segment of the spline being analyzed. The segment starts at spline point NUM and goes to spline point NUM + 1.			
CURV A	Curvature at the start of the current segment.			
CURV B	Curvature at the end of the current segment.			
DELTA CURV	Change in the curvature between the end of the previous segment and the start of the current segment. DELTA CURV has no value for the first segment.			
ENERGY	Measure of the roughness of a spline. It is normally desirable to minimize the change in curvature of a curve fit through points. The strain energy is measured as the integral of the curvature squared along a spline segment.			
TOTAL ENERGY	Sum of the strain energy for each segment of the spline.			

Figure 2-13. Example and Description of Chart 4

SEG	PT	CURVE POINTS		SEG	PT	CURVE POINTS	
		XT	YT			XT	YT
1	1	6.1071	2.3357	2	36	9.6529	3.9071
1	2	6.2213	2.3360	2	37	9.6885	3.9012
1	3	6.3354	2.3370	2	38	9.7233	3.8912
1	4	6.4492	2.3393	2	39	9.7573	3.8769
1	5	6.5626	2.3430	2	40	9.7905	3.8582
1	6	6.6756	2.3486	2	41	9.8229	3.8348
1	7	6.7881	2.3564	2	42	9.8543	3.8067
1	8	6.8999	2.3668	2	43	9.8848	3.7736
1	9	7.0111	2.3801	2	44	9.9144	3.7355
1	10	7.1215	2.3966	2	45	9.9430	3.6920
1	11	7.2310	2.4169	2	46	9.9705	3.6431
1	12	7.3396	2.4411	2	47	9.9970	3.5886
1	13	7.4472	2.4697	2	48	10.0224	3.5283
1	14	7.5536	2.5029	2	49	10.0467	3.4620
1	15	7.6589	2.5413	2	50	10.0698	3.3896
1	16	7.7629	2.5850	2	51	10.0918	3.3110
1	17	7.8655	2.6346	2	52	10.1126	3.2259
1	18	7.9666	2.6902	2	53	10.1321	3.1342
1	19	8.0663	2.7524	2	54	10.1503	3.0357
2	1	8.1643	2.8214	2	55	10.1673	2.9303
2	2	8.2072	2.8542	2	56	10.1829	2.8177
2	3	8.2503	2.8887	2	57	10.1971	2.6979
2	4	8.2938	2.9248	2	58	10.2100	2.5706
2	5	8.3376	2.9623	3	1	10.2214	2.4357
2	6	8.3815	3.0010	3	2	10.2261	2.3721
2	7	8.4257	3.0408	3	3	10.2307	2.3084
2	8	8.4701	3.0814	3	4	10.2352	2.2446
2	9	8.5145	3.1228	3	5	10.2401	2.1811
2	10	8.5591	3.1648	3	6	10.2454	2.1179
2	11	8.6038	3.2072	3	7	10.2514	2.0552
2	12	8.6485	3.2498	3	8	10.2583	1.9932
2	13	8.6932	3.2924	3	9	10.2663	1.9320
2	14	8.7379	3.3350	3	10	10.2757	1.8718
2	15	8.7825	3.3773	3	11	10.2867	1.8128
2	16	8.8271	3.4191	3	12	10.2995	1.7551
2	17	8.8715	3.4604	3	13	10.3143	1.6988
2	18	8.9158	3.5009	3	14	10.3314	1.6442
2	19	8.9600	3.5404	3	15	10.3509	1.5914
2	20	9.0039	3.5788	3	16	10.3731	1.5405
2	21	9.0476	3.6160	3	17	10.3982	1.4917
2	22	9.0910	3.6517	3	18	10.4265	1.4452
2	23	9.1341	3.6858	3	19	10.4581	1.4012
2	24	9.1769	3.7182	3	20	10.4932	1.3598
2	25	9.2193	3.7486	3	21	10.5322	1.3211
2	26	9.2613	3.7768	3	22	10.5751	1.2853
2	27	9.3029	3.8029	3	23	10.6223	1.2526
2	28	9.3440	3.8264	3	24	10.6739	1.2232

Figure 2-14. Example and Description of Chart 5

(Continued)

(Continued)

SEG	PT	CURVE POINTS		SEG	PT	CURVE POINTS	
		XT	YT			XT	YT
2	29	9.3847	3.8474	3	25	10.7302	1.1971
2	30	9.4248	3.8656	3	26	10.7914	1.1747
2	31	9.4644	3.8809	3	27	10.8578	1.1559
2	32	9.5034	3.8930	3	28	10.9294	1.1411
2	33	9.5417	3.9019	3	29	11.0066	1.1302
2	34	9.5795	3.9073	3	30	11.0896	1.1236
2	35	9.6165	3.9091	4	1	11.1786	1.1214

SEG Current segment of the spline being analyzed.

PT Counter of the sequential point number in the current segment.

CURVE POINTS Points that are xt,yt positions on the spline. These points are evaluated along the spline such that the maximum error of the chord between two sequential points is less than or equal to the tolerance specified.

Figure 2-14. Example and Description of Chart 5

The charts displayed (refer to figures 2-10 through 2-14) may take several pages to complete, depending on the number of points in the spline and the tolerance entered previously. If more than one page needs to be displayed, the system displays:

CONTINUE?

Enter:

Y To continue.

N To return to the SELECT ANALYSIS prompt.

18.2 Analytic Area/Perimeter

With this choice, you can calculate and display the area and perimeter of a single closed figure. (The perimeter calculation is accurate; however, for greater accuracy in calculating the area, use 18.3 2-D SECTION ANALYSIS.)

ENTITY SELECTION

- 1. SCREEN SELECT
- 2. CHAIN

3. REGION IN

:

Use the Entity Selection menu to select a set of entities that form a single closed figure. A maximum of 100 curves can be selected.

If the selected figure contains segments that are not connected, the system displays:

DOESN'T CONNECT, TRY AGAIN?

Enter:

Y To select the figure again.

N To exit from this operation.

If you select a connected figure, the system displays:

1. PERIMETER = n.nn

2. AREA = n.nn

If the system cannot solve the problem analytically, the area is displayed as -1.0. If you have selected only two arcs to define the region, the system displays:

MUST USE SECTION ANAL TO SOLVE

Enter] to display the area in acres:

ACRES = n.nnnn

The acreage displayed assumes feet as the unit of measure.

18.3 2-D Section Analysis

With this choice, you can calculate the following for any two-dimensional figure:

- Perimeter
- Area
- Center of gravity
- Axial moments of area
- Axial moments of inertia
- Principal moments of inertia
- Principal axes angles
- Products of inertia
- Axial radii of gyration
- Polar moment of inertia
- Polar radius of gyration

You specify the location of the origin to be used in the analysis, select the figure on which to perform the analysis, and choose the type of analysis.

Select the method to specify the location of the origin of a coordinate system. This coordinate system is used in the analysis.

LOCAL ORIGIN FOR ANALYSIS
1.ORIGIN XT=0, YT=0
2.KEY IN COORDINATES
3.EXISTING POINT
4.CENTER OF GRAVITY OF THE FIGURE

Enter:

- 1 To use the origin (0,0) in the coordinate system.
- 2 To enter the coordinates of the origin.

1.XT ORIGIN = 0.0
2.YT ORIGIN = 0.0

Enter the coordinates of the origin.

- 3 To select an existing point as the origin.

INDICATE POINT

Use the graphics cursor to select an existing point as the origin of the coordinate system.

- 4 To use the computed location of the center of gravity for the figure you select.

You then select the figure for which you want to compute the analysis information.

INDICATE CURVE

Use the graphics cursor to select a set of up to 250 curves that form a single closed figure. These curves must be selected in order; that is, end to end. Selectable entities include lines, circles, conics, 2-D splines, 3-D splines, and point-sets.

If you want to use another method of selection, enter E or CTRL-E to receive the following menu:

ENTITY SELECTION

- 1. SCREEN SELECT
- 2. CHAIN
- 3. REGION IN
- :

Use the Entity Selection menu to select the curves. For more information, refer to the ICEM Design/Drafting Introduction and System Controls manual.

If you select a set of curves that does not form a single closed figure, the system displays the following prompt:

THE SELECTED CURVE(S) DO NOT
FORM A CLOSED FIGURE; ACKNOWLEDGE

If you select a set of curves containing one or more 3-D splines, the system displays the following prompt:

YOU HAVE SELECTED ONE OR MORE 3-D SPLINES
NOTE THAT 2-D ANALYSIS IS VIEW APPARENT, ACKNOWLEDGE

After you select the figure, enter the tolerances for the analysis:

- 1. INTOL = n.nnnn
- 2. OUTTOL = n.nnnn

Enter the maximum distance that the lines drawn by the system may be short of (INTOL) or extend across (OUTTOL) the edge curves. The preset value is 0.2540 mm (0.0100 in). Normally, INTOL and OUTTOL are equal. If they are not, the system displays the INDICATE OUTTOL SIDE prompt and you select the outside of the figure.

Enter:

- [To return to the INDICATE CURVE prompt.
-] To accept the preset values.

Select the type of analysis to be performed.

SELECT ANALYSIS

- 1. PERIMETER
- 2. AREA
- 3. CENTER OF GRAVITY
- 4. AXIAL MOMENTS OF AREA
- 5. AXIAL MOMENTS OF INERTIA
- 6. PRINCIPAL MOMENTS OF INERTIA
- 7. PRINCIPAL AXES ANGLES
- 8. PRODUCTS OF INERTIA
- 9. AXIAL RADII OF GYRATION
- 10. POLAR MOMENT OF INERTIA
- 11. POLAR RADIUS OF GYRATION

Enter:

- 1 To determine the perimeter of the figure.
- 2 To determine the area of the figure.
- 3 To determine the center of gravity of the figure.
- 4 To determine the axial moments of the area of the figure with respect to each coordinate axis.
- 5 To determine the axial moments of inertia (second moments of area) of the figure with respect to each coordinate axis.
- 6 To determine the principal moments of inertia of the figure with respect to the principal axes.
- 7 To determine the angles between the coordinate axes and the principal axes (in degrees).
- 8 To determine the products of inertia of the figure.
- 9 To determine the axial radii of gyration of the figure with respect to each coordinate axis.
- 10 To determine the polar moment of inertia with respect to the local origin of the figure.
- 11 To determine the polar radius of gyration of the figure.

The origin used in response to selections 3 through 11 is the origin indicated in response to the LOCAL ORIGIN FOR ANALYSIS prompt. The coordinate axes used in response to selections 4, 5, 6, and 9 are those that pass through this origin and are parallel to the bottom and left edges of the screen.

18.4 3-D Analysis

With this choice, you can calculate specific analytical properties of certain three-dimensional figures. These figures should be two-dimensional shapes that have been projected to a depth, rotated about the xt-axis, rotated about the yt-axis, or rotated about a given line. Shapes in three-dimensional space may also be used, but only their projection onto the XT,YT plane will be considered. The following analyses can be performed:

- Surface area
- Volume
- Weight and/or weight per unit length
- First moment and center of mass
- Moment of inertia
- Radius of gyration
- Spherical moment of inertia
- Spherical radius of gyration

The system displays the following menu choices:

3-D ANALYSIS**1.PERFORM ANALYSIS****2.DISPLAY ACCUMULATED RESULTS****Enter:**

- 1 To perform analysis on a projected or rotated object.
- 2 To display the accumulated results of three-dimensional analysis on two or more distinct objects.

18.4.1 Perform Analysis

With this choice, you can specify the type of three-dimensional figures to be analyzed.

SELECT ANALYSIS TYPE

- 1.PROJECTED
- 2.ROTATED ABOUT THE XT-AXIS
- 3.ROTATED ABOUT THE YT-AXIS
- 4.ROTATED ABOUT A LINE

Enter:

- 1 To select projected two-dimensional entities.

1.ZT BOTTOM = n.nnnn
2.ZT TOP = n.nnnn

Enter the bottom and top zt values of the projected figure to be analyzed. The default values are:

1.ZT BOTTOM = -1.0 in (-25.4 mm)
2.ZT TOP = 1.0 in (25.4 mm)

Enter:

- [To return to the Select Analysis Type menu.

-] To accept the default values and continue to the DENSITY prompt.

- 2 To select a two-dimensional figure to be rotated 360° about the xt-axis.
- 3 To select a two-dimensional figure to be rotated 360° about the yt-axis.
- 4 To select a two-dimensional figure to be rotated 360° about a line in the XT,YT plane.

INDICATE LINE

Use the graphics cursor to select an existing line as the axis of rotation for the two-dimensional figure. The line should be coplanar with the figure and both should be in the XT,YT plane. If they are not, the system projects them onto the XT,YT plane and uses the projections for analysis.

Enter:

- [To return to the Select Analysis Type menu.

-] To return to 18.4 3-D ANALYSIS.

For all types of figures, the system next displays:

1.DENSITY = n.n

Enter the density of the figure in grams per cubic millimeter (pounds per cubic inch). The default value is 1.0.

Enter:

[To return to the Select Analysis Type menu.

] To accept the default density.

Now select the entity you want to analyze.

INDICATE 2-D CURVE

Use the graphics cursor to screen select the edges of the entity. A maximum of 250 curves can be selected. For a projected solid, the generating two-dimensional figure must be a closed curve.

If you want to use another method of selection, enter E or CTRL-E to receive the following menu:

ENTITY SELECTION

1.SCREEN SELECT

2.CHAIN

3.REGION IN

:

Use the Entity Selection menu to identify the edges. For more information, refer to the ICEM Design/Drafting Introduction and Systems Controls manual.

Enter:

[To return to the DENSITY prompt.

] To return to the Select Analysis Type menu if you made no selections. If you made the selections, entering J sends you to the INDICATE 2-D CURVE prompt. In the case of a rotated figure, if the edges do not form a closed curve, perpendiculars from the ends of the curve are dropped onto the axis of rotation, thereby closing the curve.

Enter the tolerances for the analysis:

1.INTOL = n.nnnn
2.OUTTOL = n.nnnn

Enter the maximum distance that the lines drawn by the system may be short of (INTOL) or extend across (OUTTOL) the edge curves. The preset value is 0.2540 mm (0.0100 in). Normally, INTOL and OUTTOL are equal. If they are not, the system displays the INDICATE OUTTOL SIDE prompt and you select the outside of the figure.

Enter:

- [] To return to the Entity Selection menu.
- [] To accept the preset values.

Now select the analysis mode you want to use.

SELECT ANALYSIS MODE
1.SURFACE AREA
2.VOLUME
3.WEIGHT
4.WEIGHT PER UNIT LENGTH
5.FIRST MOMENTS
6.CENTER OF MASS
7.MOMENTS OF INERTIA
8.RADIUS OF GYRATION
9.SPHERICAL MOMENT OF INERTIA
10.SPHERICAL RADIUS OF GYRATION
11.DISPLAY ALL

18.4.1 Perform Analysis

The system calculates and displays the selected results of the three-dimensional analysis. If you choose 11.DISPLAY ALL, you receive the following information. (The values given are for a specific 2-D curve.)

3-D ANALYSIS RESULTS

1.SURFACE AREA ----- A=64.0000
2.VOLUME ----- V=32.0000
3.WEIGHT ----- W=32.0000
4.WEIGHT PER UNIT LENGTH WT/U=16.0000
5.FIRST MOMENTS OF MASS X-MM=0.0000
Y-MM=96.0000
Z-MM=96.0000
6.CENTER OF MASS X-CM=3.0000
Y-CM=3.0000
Z-CM=0.0000
7.MOMENTS OF INERTIA X MI=341.3333
Y MI=341.3333
Z MI=661.3333
XY MI=2811.5000
XZ MI=96.0000
YZ MI=96.0000
8.RADIUS OF GYRATION X-RADIUS=3.2660
Y-RADIUS=3.2660
Z-RADIUS=4.5461
9.SPHERICAL MOMENT OF INERTIA =672.0000
10.SPHERICAL RAD OF GYRATION =4.5826

Enter:

- [To return to the Entity Selection menu.
-] To acknowledge the results and continue to the ACCUMULATE RESULTS prompt.

The system can accumulate the results of the three-dimensional analysis when two or more nonintersecting objects are being analyzed.

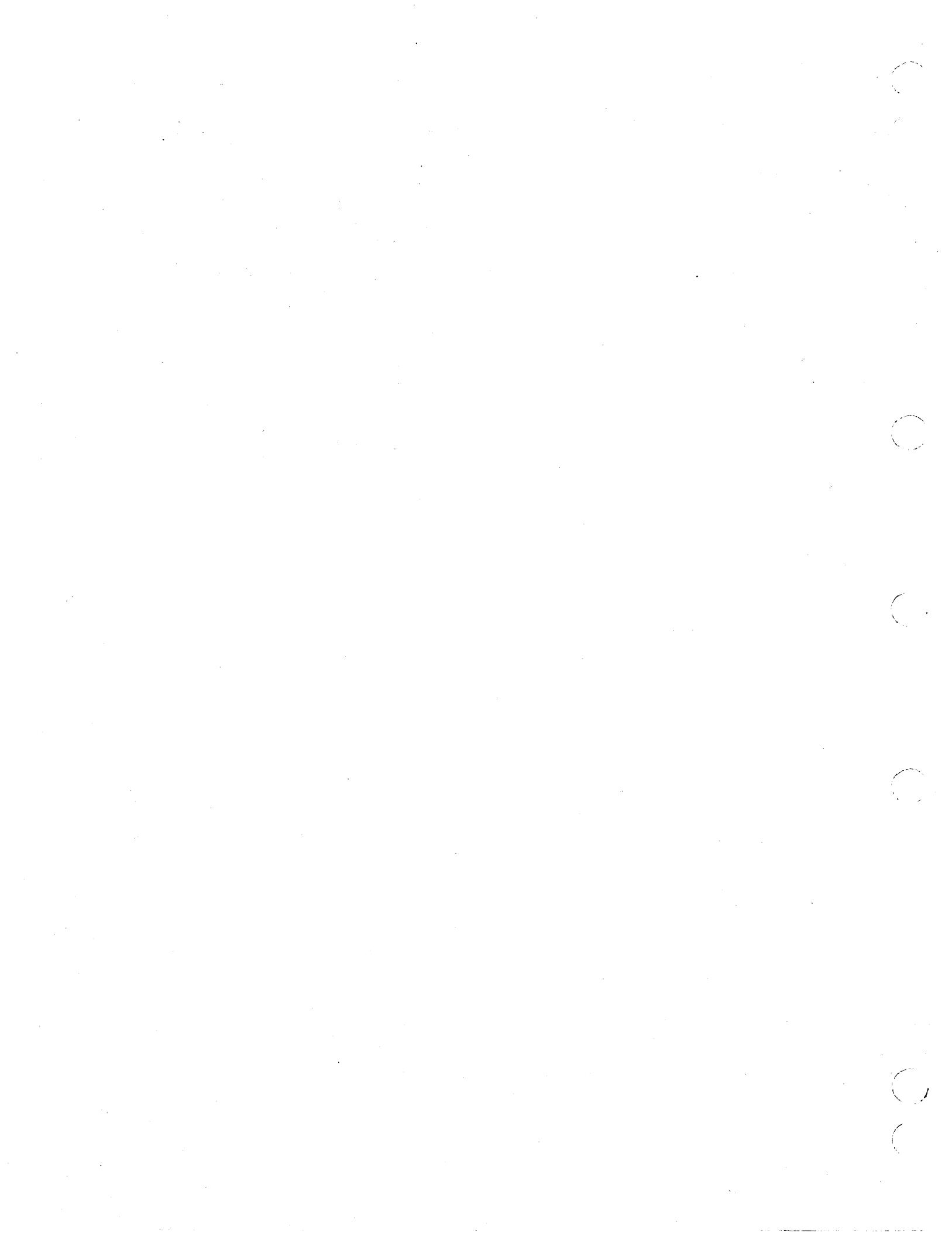
ACCUMULATE RESULTS
1.INITIALIZE
2.ADD RESULTS
3.SUBTRACT RESULTS
4.SELECT NEW ENTITY

Enter:

- 1 To initialize the accumulator (zeros all the values).
- 2 To add the results of the three-dimensional analysis of the latest figure to those already in the accumulator.
- 3 To subtract the results of the three-dimensional analysis of the latest figure from those already in the accumulator.
- 4 To return to the Entity Selection menu so you can select another figure for analysis.
 - [To return the system to the Select Analysis Mode menu.
 -] To return the system to the 3-D Analysis menu where 2.DISPLAY ACCUMULATED RESULTS displays the accumulated three-dimensional analysis results of two or more figures.

18.4.2 Display Accumulated Results

With this choice, you can display the accumulated results in the form shown in the 3-D ANALYSIS RESULTS example.



Menu 19: SI/US/Resize

3

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Menu 19: SI/US/Resize

3

Using Menu 19

With this menu, you can convert parts from SI units to U.S. customary units or from U.S. customary units to SI units. You can also resize the part (display it again using another scale factor).

If the system currently in effect uses SI units, the menu is:

- SELECT CONVERSION MODE
- 1. MILLIMETERS TO INCHES
- 2. RESIZE

If the system currently in effect uses U.S. customary units, the menu is:

- SELECT CONVERSION MODE
- 1. INCHES TO MILLIMETERS
- 2. RESIZE

19.1 Millimeters to Inches

19.1 Inches to Millimeters

With this menu choice, you can convert the units of measure from the system currently in effect to the desired system. The system returns to the top-level ICEM DDN menu after repainting the part.

If the dimensions in the part meet the criteria listed for scaling, the system displays:

DO YOU WANT TEXT CONVERTED?

Enter:

- Y To change the dimensions to the units of measure from the system currently in effect to the desired system.
- N To return to the top-level ICEM DDN menu without converting the text. The part is converted.

If you select Y, the system displays:

DECIMAL PLACES = n

Enter the number of decimal places to be displayed.

The scaling logic used here is identical to the scaling logic used in 13.ENTITY MANIPULATION in the ICEM Design/Drafting Basic Construction manual. Refer to the comments on moving entities for restrictions and guidelines.

19.2 Resize

With this menu choice, you can redisplay the part using another scale factor.

RESIZE FACTOR = 1.0000

Enter the desired scale factor. Default is 1.0000. The factor must be greater than 0. The system redisplays the part. The dimensions are changed according to the new scale factor.

The resize operation applies the factor specified to the part but the origin remains the same. Some or all of the part that is on the screen may be transformed off-screen. In these cases, there will be no request for a rescale operation.

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Glossary

A

A

Above the Line Dimensioning

Dimension text written above and usually parallel to the dimension line.

Alphanumeric

The letters of the alphabet (A through Z) and the digits (0 through 9).

Angular Dimension

The entity type showing the angular distance between two lines.

ANSI

American National Standards Institute. English (U.S. Customary) dimensions are specified in feet and/or inches. Metric dimensions are specified in millimeters.

Arc Length Dimension

A measure of the length of an arc's circumference.

Aspect Ratio

The ratio between the character width and the character height in dimensions.

Attention Indicator

A small oval that the system displays on an entity to indicate that the entity has been selected.

Automatic Dimensioning

The system capability that calculates the distance or angle of a dimension and automatically writes the text for that distance or angle in the dimension.

B

Balloon

An identifier symbol containing the detail number (part number) and the sheet number.

Blanking

A process applied to entities in the current part where the entities remain in the part, but are not displayed on the screen. Contrast with Deleting.

C

Centerline

1. A font type.
2. A drafting entity shown only in the view of definition, used to refer to center locations.

Chain Select

The process of selecting a series of contiguous curves.

Chamfer Dimension

A measure of length along the longitudinal direction of a 45 degree chamfer.

Composite Geometric Tolerance

A single geometric characteristic symbol in a frame, followed by the tolerance and datum requirements contained in two frames, one above the other.

Conical Taper Dimension

The entity type showing the ratio of the difference between the diameters of two sections of a cone (that are perpendicular to the axis) to the distance between these two sections.

Contiguous Curves

Adjacent curves that share a common endpoint or that have endpoints within a short distance of each other.

Coordinate Axes

The lines that form a right-handed, three-dimensional frame of reference. The model axes (x, y, and z) and the transform axes (xt, yt, and zt) are referenced from a shared origin point in which x=0, y=0, and z=0.

Coordinate Space

A space created by the position (rotation and translation) of the coordinate axes. There are two types of coordinate space: model and transform.

Coordinates

The position of a point in relation to the x-, y-, and z- or xt-, yt-, and zt-axes.

Curve

A line, arc, conic, or two-dimensional spline. In some cases, it also includes points, three-dimensional splines, strings, composite curves, machining curves, and Bezier curves.

D**Data Base**

An integrated set of files, tables, arrays, and other data structures.

Data Capture

The system capability to save as a variable either a default parameter value or a previously entered parameter value displayed on the screen.

Datum Feature Symbol

One or two characters in a rectangular frame that identifies a part feature.

Datum Points, Lines, Edges

Features assumed to be exact for purposes of computation or reference, and from which to establish the position of other features.

Datum Target

A specific point, line, or area identified on a drawing by a datum target symbol. It establishes datum planes commonly used for manufacturing or inspection repeatability.

Default Value

A value used by the system if] is entered in response to a data entry prompt. It is also the initial value assigned to a modal until it is changed.

Deleting

The removal of entities from the data base. Contrast with Blanking.

Depth

The current value of the workplane along the z- or zt-axis of the workspace.

Dimension

An entity used to show a geometric characteristic of a part, such as the diameter, length, angle, or center distance.

Dimension Lines

Arrows from dimension text to extension lines that indicate the dimension being measured.

Dimension Not to Scale

A line drawn under a dimension, indicating that the dimension is not to scale.

DIN

Deutsche Industrie Norme (German industry standards).

Displacement

A specified distance in the x, y, and z directions from a selected base point.

Dormant Entity

An entity created by the system for the purpose of defining another entity. Dormant entities cannot be displayed or manipulated. All dormant entities have sequence numbers.

Downspace Ratio

The distance from the bottom of one line of dimension text to the bottom of the next line.

Dragging

The movement of selected entities around the screen in order to view their position at any given point. The entities are drawn in the accepted location.

Dual Dimensioning

The system capability to produce dimensions containing both SI and U.S. customary units.

E**English (U. S. Customary) Mode**

The condition in which all entities are defined in English (U. S. customary) units (inches and/or feet).

Entity

The representation of a geometric construction in the ICEM DDN data base. Examples are points, lines, arcs, and spheres.

Entity Type Numbers

System-assigned code numbers used to identify specific entity types such as line, group, and copious data.

Extension Lines

Lines extending from entities between which dimensions are given and indicating the distance measured.

F**Feature**

A specific component of a part such as a notch, the sides of a hole, or the flat side.

Feature Frame

The entity type used for datum feature, geometric tolerance, or composite geometric tolerance symbols.

Flat Taper Dimension

A measure specifying a toleranced slope and a toleranced height at one end of a drawing.

Font

The method used to represent a line or curve in a display. Examples of fonts are solid, dashed, phantom, and centerline.

G**Geometric Tolerance**

A geometric characteristic symbol, followed by a tolerance and datums, contained in a single frame and separated by vertical bars.

Graphics Cursor

The cursor symbol used to locate or define entities by screen position. The graphics cursor can be crosshairs or some other cursor symbol.

Group

A set of entities defined and treated as a unit such as a balloon or a surface texture entity.

I**ICEM DDN**

Control Data's integrated computer-aided engineering and manufacturing software application for design, drafting, and numerical control.

ICEMDDN

The name of the direct access file that contains ICEM DDN.

Implicit Point

A point automatically generated by the system at curve endpoints, midpoints, intersections, and circle centers.

Island

A closed region that bounds section lining inside or outside of the region.

Isometric Drafting

The use of standard two-dimensional drafting entities on a three-dimensional model in a three-dimensional view.

L**Label**

Text and a leader with an arrow from the text to an entity on the drawing.

Leader

A line leading from a dimension value or an explanatory note to a selected entity.

Least Material Condition (LMC)

The minimum size limit of an external dimension, or the maximum size limit of an internal dimension.

LMC

See Least Material Condition.

Local File

Any file that is currently associated with a job.

M**Maximum Material Condition (MMC)**

The maximum size limit of an external dimension, or the minimum size limit of an internal dimension.

Menu

A list of options used to perform operations in ICEM DDN.

Message

Text written by the system to display information, errors, or warnings. See also Prompt.

Modal

A user-assigned status or value that controls the operation of ICEM DDN. Examples are modals for setting system decimal places and for activating the grid.

Model Coordinate Space

The space created by the position of the x-, y-, and z-axes. This space is displayed as the front view, view 1. See Transform Coordinate Space.

N

Normal

Perpendicular.

NOS

Network Operating System.

Note

An entity that contains graphics text for drawings.

O

Operating System

The set of system programs that controls the execution of user programs and commands.

Operation Complete --]

A keyboard/tablet operation that completes the current operation.

Operation Reject -- [

A keyboard/tablet operation that rejects the previous operation. The system returns to the preceding prompt.

P

Parallel Dimension

A measure of the perpendicular or two-dimensional distance between the ends of two entities.

Parameter

A variable whose values determine the operation or characteristics of a system.

Part Name

A series of alphanumeric characters that identify a part or drawing used in conjunction with a sheet number.

Permanent File

A file saved by the system between terminal sessions.

Point

A geometric construction located by x-, y-, and z-coordinates.

Prompt

A screen display requiring user action. See also Message.

R**Rectangular Coordinate Dimension**

A dimension specifying the distance between a datum and a selected entity in either a horizontal or vertical direction.

Regardless of Feature Size (RFS)

A dimension, within the specified positional tolerance, that must be maintained regardless of the actual size of the features.

Region Select

A process of selecting all the entities within or outside a specified region.

Repetitive Feature Dimension

A dimension with text added that refers to a number of identically dimensioned features.

Rescale

To change the scale factor and redraw a part using the new scale so all entities in the part are displayed.

RFS

Regardless of Feature Size.

Rotate

To revolve a construction about an axis.

S**Scalar**

A quantity that has magnitude only. Contrast with Vector.

Scale Factor

Ratio of the current display with respect to the data base.

Section Lining

An entity type defining patterned representations for material types including iron, steel, brass/copper, rubber/plastic, refractory, glass/slate, lead, or aluminum/magnesium. Section lining was formerly called cross hatching.

Sequence Number

A unique sequential number associated with each entity.

Sheet Number

A subdivision of a part name. Sheets can be numbered from 0 to 9,999 with a part name.

SI

Système International d'Unités (international system of units). Dimensions are calculated in metric units. Lengths have units of millimeters. Areas have units of millimeters squared unless otherwise specified.

Single Select

A process of selecting individual entities one at a time.

Spacing

The distance between the centers of consecutive characters.

Spherical Feature Dimension

A diameter or radius dimension with the spherical feature symbol S added, indicating that it refers to a spherical feature.

Standard Views

The system-generated coordinate systems from which the user can view a display of the part surfaces from these view points: front, back, top, bottom, right, left, isometric, and auxiliary.

Surface Texture

Specification of the smoothness quality of a part's surface.

System

The ICEM DDN software system.

T**TAPE3**

A local file on which parts are filed and from which parts are retrieved.

Temporary File

A nonpermanent file associated with a job only during job processing.

Text

Alphanumeric characters and symbols used in notes, labels, and dimensions.

Thickness Dimension

1. The entity type that indicates the measure of thickness between two curves.
2. A measure of the distance between two curves from a point on the first curve, along an imaginary line normal to the second curve, and to the position where this line intersects the second curve.

Tolerance

The range of variation allowed in maintaining a specified dimension.

Tolerance Ratio

The ratio between the character size for tolerance characters and the character size for main characters in dimensions.

Transform Coordinate Space

The space created by the position of the xt-, yt-, and zt-axes. This space is displayed in any view other than view 1. See Model Coordinate Space.

Transformation of Coordinates

The mapping of the xt-, yt-, and zt-coordinates into the x-, y-, and z-coordinates.

U**Unblank**

A procedure for making visible entities that have been blanked by the user.

V**Vector**

A quantity that has magnitude and direction. It is commonly represented by a line segment with directionality. The line's length represents the magnitude and its orientation in space represents the direction. Contrast with Scalar.

View

A display of coordinate space.

W**Work Plane**

A specific plane in workspace on which two-dimensional entities are constructed.

Work View

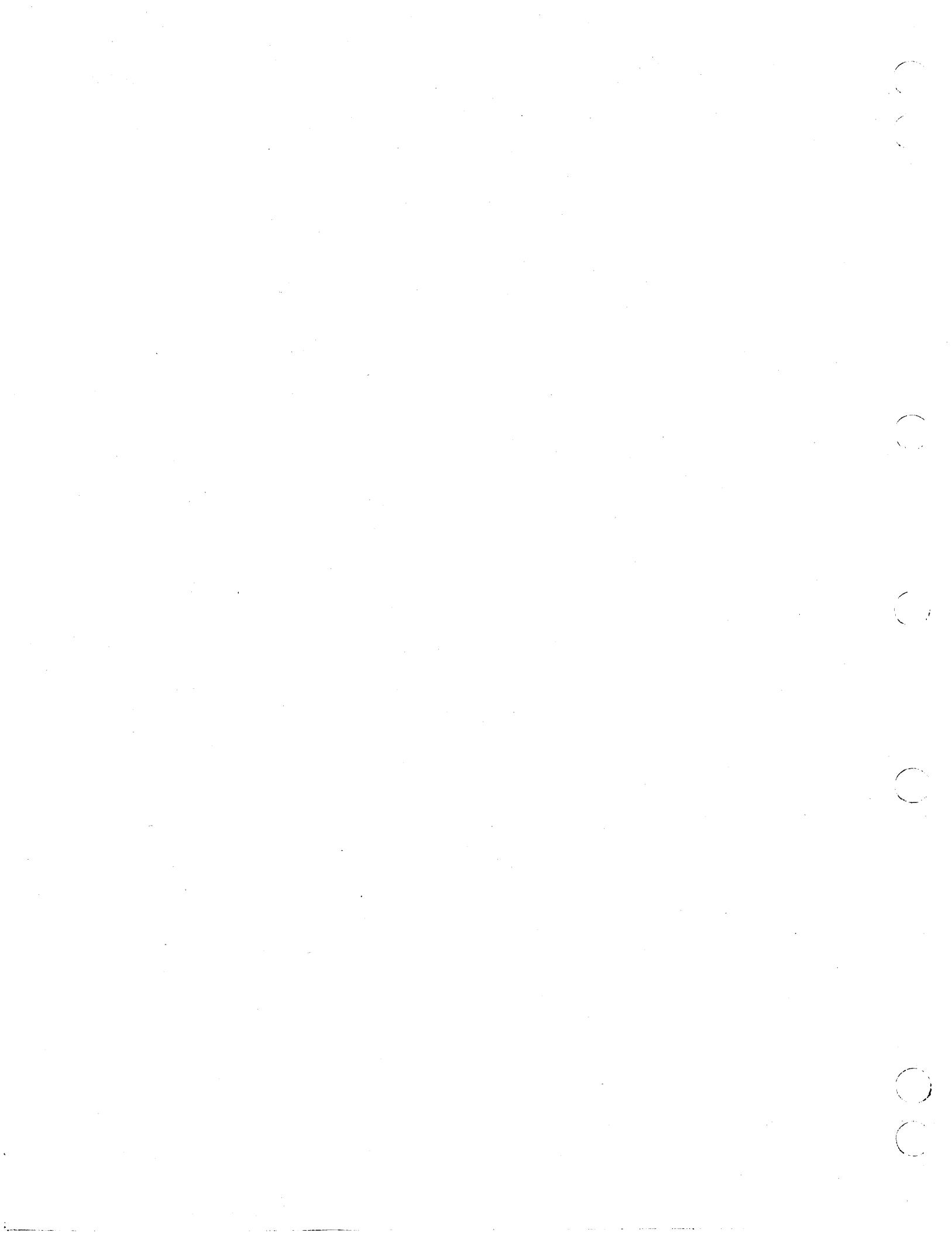
The view in which screen input is accepted.

Workspace

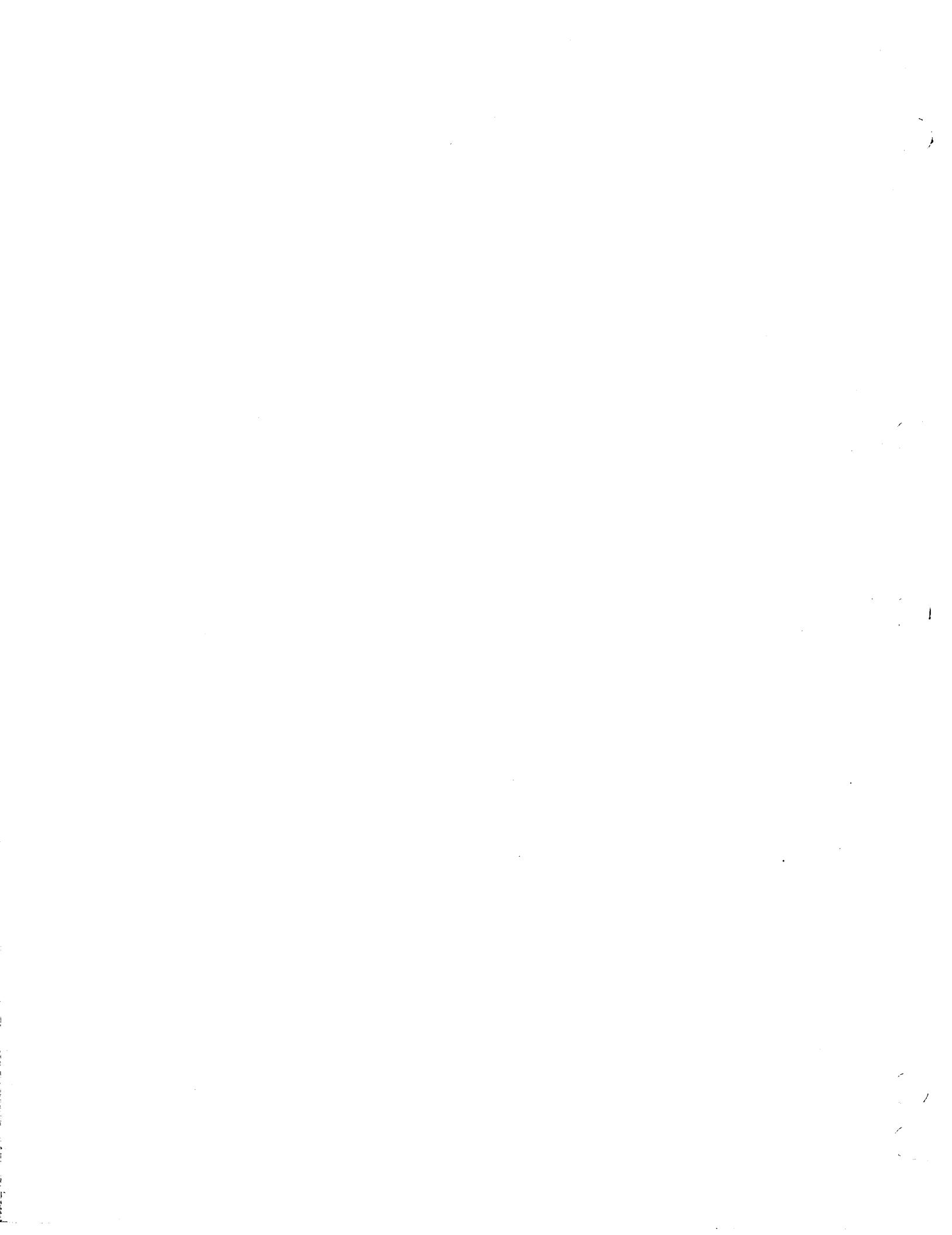
The space in which entity construction and specification occurs.

Z**Zoom**

To enlarge or decrease the size of the display proportionately.



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